

MANUAL

FOR

MAPE CNC 1 OG CNC 2000XL

TABLE OF CONTENTS:

Introduction	page	4.
Layout	page	5.
Installation	page	6.
Unpacking	page	6.
Placing the machine	page	6.
Alignment	page	7.
Services required	page	7.
Cooling system	page	8.
Acclimatization	page	9.

OPERATION:

Operating the machine	page	10.
Tool loading	page	11.
Collar-setting	page	12.

EDITING:

How the computer works	page	13.
Hardware	page	13.
Memory	page	13.
Deleting/Correcting , illustration	page	14.

PROGRAMMING:

Digitizing ("manual" programming)	page	15.
Programming from film	page	15.
Programming from PCB	page	15.
Programming with CAD floppy and serial data	page	16.
Program structuring	page	18.
Drilling & Routing program example	page	19.
Numeric capacity	page	19.
Millimeter or Inch software setting	page	19.

PRODUCTION:

Warm up	page	20.
Single stack drilling	page	20.
Offloading tools	page	20.
Loading new tools	page	21.
Board size	page	21.
Reminder	page	22.

KEYBOARD-FUNCTIONS:

Functions and control elements	page	23.
--------------------------------------	------	-----

TABLE OF CONTENTS (continued):

WORKING CODES:

Summary of Function codes	page 41.
Description and examples	page 49.
Circular interpolation	page 49.
Program number	page 51.
Spindle speed	page 57.
Arc routing	page 58.
Step & Repeat, sub sequence	page 59.
Super-repeat (code A13)	page 63.
Tool change, routing	page 65.
Linear routing	page 66.
Z-axis infeed	page 67.
Program end	page 68.
Mirror imaging	page 69.
Step & Repeat, main sequence	page 74.
Drill cycle	page 79.
Routing feed	page 81.
Scaling	page 83.
Displacement, floating zero	page 87.
Z axis lower limit	page 88.
Tool change, drilling	page 90.
Z axis upper stroke limit	page 91.
Rotation	page 92.
IC pin spacing	page 95.
IC SIL number programming	page 97.
Position preset	page 98.
SIL/DIL programming, axis parallel	page 100.
Z-retraction feed	page 103.
Dwell timer	page 104.
SIL-programming, rotated	page 104.
Tool life time counter	page 107.
APSS enable	page 110.
Automatic corner round-off function	page 111.
Automatic arc-routing	page 114.
Router radius compensation	page 117.
Programming examples, routing	page 123.
Guidelines for routing	page 128.

MAINTENANCE:

Cooling system	page 131.
Collet and HF-spindles	page 133.
Vacuum extraction system	page 136.
Daily	page 134.
Weekly	page 137.
Monthly	page 138.
Annual	page 140.
General	page 141.

INTERFACE:
Interfaceability page 142.
PC "ProgramBank" Front End system page 145.

GENERAL:
Autoparameter-tables (APSS) page 149.
Autoparameter-tables (APSS graphic) appx. #1.
Tool lifetime vs hole quality page 152.
Operating instructions for HF-Spindles appx. #2.

WARRANTY:
Warranty and restrictions appx. #3.

INTRODUCTION:

Mape CNC-1000 and CNC-2000/L are fully automatic drilling- and routing machines, with linear and circular interpolation and automatic router-radius compensation.

A PC can be connected to the interface of the CNC-1000/2000 and Mape can supply a special PC-based front end system as optional equipment.

The memory capacity equals 6500 blocks (i.e. instruction-lines or "holes"), but when using one or more of the machines STEP & REPEAT-functions, the memory capacity is considerably increased, because these functions multiply the memory capacity almost infinitely.

For programming purposes, the CNC-1000/2000 is equipped with a video-system, consisting of a TV-monitor and a CCD-camera.

The camera has a special optical system with built-in reticle. This makes the system independent of thermal and magnetic influences, providing very high accuracy even when the machine is operated in extreme environments. The magnification of the optical system is approx. 15X.

The machine is provided with a special tooling-system for programming purposes, located between the two tooling-systems for the production. In the CNC-1000 single spindle machine, the programming pilot hole, is offset approx. 150mm and is located in-line with the production-tooling system.

When programming, a film or a PCB is placed in the programming tooling-system, and the program is entered (i.e. "digitized") into the memory by means of the video-system, joystick and the 'module'-keys.

After programming and testing is carried out, the stack(s) of PCB's are placed under the spindle(s) tooling-system(s).

The drilling and routing operations are now automatically carried out by the machine, controlled by the MAPE UNITROL SYSTEM-2 CNC which is a combined programming- and production controller.

All operating elements are concentrated on the front panel of the Mape "Unitrol-2" computer, including a high-brightness information display.

On the rear side of the CNC-system, there is a standard "RS-232C" serial interface-system for connection of a PC-based Front End System, or an external (peripheral) device.

The mains switch and the spindle hour-counter is located on the rear side of the CNC-system.

INTRODUCTION....

PHYSICAL LAYOUT:

All the control electronics, such as the hi-power drives for the X-, Y- and Z-positioning motors, the CPU- and the power supply unit, are also located in the computer box, that is, in a 19"-rack system, which is easily accessible from the rear side of the box. The spindle converter is located just under the keyboard for easy access and convenient supervision of the converter status.

Due to the modular and highly compact layout, and the fact, that all the control electronics consists of only 6 boards (of which the three X,Y and Z-modules even are identical and interchangeable), makes electronic servicing very simple.

INSTALLATION:

UNPACKING:

Normally the machine is delivered in a big wooden crate. The top of the crate is removed by means of a crowbar, and the sides can be removed in a similar way.

NOTE: Do not use tools which might cause damage to either the machine or the standard accessories of the machine.

Lift the machine approx. 10 cm / 4" by using the built-in slots in the lower end of the pedestal and a fork-lift with sufficient capacity.

The machine must be lifted vertically approx. 10 cm (4") in order to get it free from the base support. When lifted, the bottom of the crate may be removed, and the machine can be driven to its place of production.

The lifting must be performed by a fork-lift with a capacity of minimum 2.0 ton (CNC-1000) or 2.5 ton (CNC-2000) or 3.0 ton (CNC-2000-L).

Computer-box, TV-monitor or the protective shield may not be exposed to any direct pressure, and of cause it is strictly forbidden to lift in any of these parts as well as any other projecting parts of the machine. You should especially **avoid to use other lifting gear such as wires, chains or straps.**

PLACING THE MACHINE:

When placing the machine, it is important to remember, that some service is performed from the rear side. Therefore it is appropriate, that easy acces to the rear side of the machine and the vacuum system and computer is possible.

Hence, allow minimum 50cm/20" of free access behind the machine.

Due to the extremely solid construction of the machine, which involves granite, special concrete and steel details, the machine is designed with only 3 feet. This means among others, that it is no longer necessary to perform any special alignment procedure in order to bring the machine performance within the factory adjusted tolerances.

ALIGNMENT:

The machine must be situated on a hard and even surface (preferably on a concrete floor), and it is important, that the floor is not slippery in any way, as the machine will tend to move due to the 'hammer-like' forces of the X/Y-movement. If the floor is slippery, please place some very thin (>1mm) soft rubber on the floor.

As described above, no special adjustments are necessary, but it is advisable not to place the machine on a tilting surface or on a surface which is not stable (e.g. wooden floors). However, the machine performance would not suffer due to its extreme constructional stiffness and surplus of axis-movement power, but the building may be damaged or the machine may tend to move on a tilting surface.

SERVICES REQUIRED:

When the machine is perfectly sited, electricity and compressed air can be connected, but **NOT** turned on yet!
The machine is connected to 220V 50-60Hz and **GROUND (!)**, single phase; current consumption is approx. 10 Amperes.

The ground-connection MUST be of excellent quality in order to avoid operational interruptions due to electrical interference. Also make the connection-wires as short as possible and make sure, **that no other machinery is connected to the same phase as the drilling machine.** Connections to the same group of phases loaded with machines having large inrush-currents, such as big electric motors or heaters, as well as spark-generating equipment like welding machines etc., **must be avoided!**

The compressed air must be absolutely dry and oil-free and set **exactly at a pressure of 5.0-6.0 Bars** (use a service-unit with a 5 micron main filter). The air pressure must be kept within this range at any time!

The central connection for the compressed air is situated and clearly marked "AIR" on the upper right side of the machine. The dimension of the air-inlet branch is 1/4" (Ø 6,35mm).

COOLING SYSTEM:

After having obtained the room temperature, you may now connect the enclosed cooling unit. It is a re-circulating temperature controlled system, that provides optimum temperature conditions for the HF-spindle(s) of the machine. The cooling unit is factory adjusted at a temperature of approx. 20 degrees Centigrade.

The water container must be filled 3/4 (=30 ltrs) of tap water with a temperature of approx. 20 deg. Centigrade. This can be obtained by filling the container with 20 liters of cold water and 10 liters of hot water and simultaneously add a half bottle (0.5 liter) of the enclosed coolant additive "NALCO 43-73". Stir well, but don't taste!

The Nalco 43-73 coolant additive should be mixed 60:1, meaning 1 liter Nalco to 60 liters of tap water.

Observe that the temperature gauge on the front of the cooling unit, remains in the green-area of the dial during after the tank has been filled. If not, adjust the temperature.

NOTE: If the water is too cold (below 15 degree C) or too warm (above 36 degree C), the machine will refuse to start the spindle(s). Instead, the display will show a temperature alarm.

Connect the two water hoses to the machine's water inlet and water outlet (marked "WATER" I I) just beside the air-inlet. Follow the arrows of the water in/outlet block and connect to the cooling unit's corresponding in/outlets. It is not critical if the flow of water is reversed.

Finally, connect the cooling unit's electrical plug to the mains socket on the rear side of the CNC-box and turn-on the cooling unit by rotate the big, mains rotary switch until it shows "ON" (i.e. turn to position "I" or "ON").

NOTE: Never switch off the cooling unit. The cooling unit is automatically operated via the CNC-system's power outlet.

TOOL LOADING:

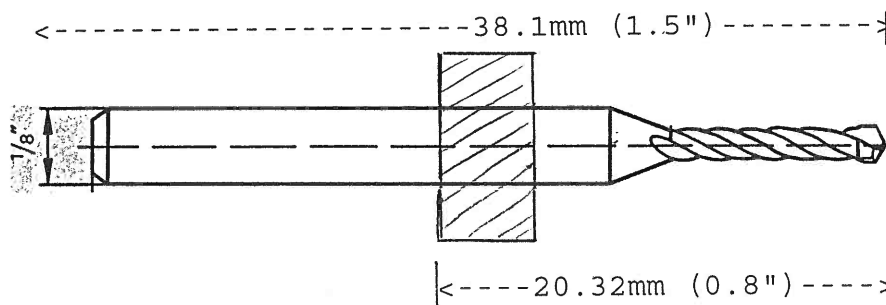
Before mounting the tools in the tool-magazines, they must be fitted with a collar, i.e. a plastic ring that fit exactly into the tool-pods.

The tool-length should preferably be the industry standard of 1,5" (38,1mm) measured from the drill point to the shaft-end. However the shorter 1 1/4"- version might be used, but is not recommended.

The enclosed "collar setter" is for adjusting the collar rings on the tools.

The collar-rings must be adjusted exactly to the industry standard of 20,32mm (or 0,8") measured from the TOP END of the ring to the POINT of the tool.

ILL.:



The measure of 20,32mm must be closely observed, as deviations might cause either damage to the machine's workplane and the HF-spindle bearings or (if less than 20,32mm) insufficient drilling or routing results.

NOTE:

The collar setter will only work properly with collar-rings with the following dimensions: Ø7,50mm x H 4,50 mm (tol.: -0/+0,1mm).

If the height of the collar-ring is less than 4,5mm the resulting distance from the the spindle nose to the tool-point will be less than the specified 20,32mm (0,8"), hence the holes will not be drilled all the way through the workpiece.

Alternatively, if the ring height is more than 4,5mm, the result will be, that the tools might damage the workplane and spindles as the distance from the spindle-nose to the tool-point will exceed the specified distance of 20,32mm (0,8").

Most of today's suppliers of tools for PCB-production can supply tools already fitted with collar-rings to your exact specifications, and this is greatly preferable, especially if the tools are returned to the manufacturer for re-grinding. Only collar-rings with smooth edges and of proven quality should be used!

"ACCLIMATIZATION":

It is absolutely **imperative**, that the machine-temperature equals the room-temperature (=20 Centigrades +/-2 Centigrades) **BEFORE** the machine is switched on and before ANY operation is performed! This is especially important to remember, when the machine has been stored outdoors or recently been under transport. If not, the formation of condensed water as well as changes in measurements can cause damage to the machine! Especially the HF-spindles are sensitive to too low temperatures because of the small internal clearances and tolerances.

Note: The acclimatization-period can take more than 24 hours because of the massive masses of the machine. Please consider how long time, 1 cubic meter of granite stone would take to obtain the room-temperature after having been moved from a freezer directly to your factory floor.

OPERATION:

OPERATING THE MACHINE:

Having connected the machine to the mains supply (220V +EARTH), compressed air (5-6.0 Bar) and water cooling unit and when the internal machine-body has obtained the working-temperature (20 degrees Centigrades), it can be switched on by means of the mains switch located on the rear side of the computer (top/right).

Just after turning the machine on, it must be observed that the cooling unit starts (red light goes on) and that it starts to circulate the water.

IMPORTANT: Please check that the water actually begins to flow in the transparent plastic pipes, especially watching the return pipe, in order to check that the entire flow-path is OK.

Just after switching on the machine, the CNC will produce four short tones ("Beeps") followed by an interval, telling the operator that a command is expected.

If the shield is open, the display will read:

```
***** MAPE CNC-2000 (No.145) *****  
>> CLOSE SHIELD TO OPERATE <<
```

or, if the shield was -or now is- closed the display will read:

```
***** MAPE CNC-2000 (No.145) *****  
>> PRESS REFERENCE-KEY <<
```

When the operator presses the REF!-key, the tones will stop, and the machine will travel to the datum point, and the TV-monitor will show, that the camera is "parked" exactly over the datum-point. The display will now read:

```
MODE:00      GRID:00200      UNIT:0,01MM  
A 00 X 13000 Y 00500      BLOCK:00000
```

Now the machine is ready for operation and tools must be adjusted and loaded into the tool magazine(s). After that, programming and memory editing may begin.

TOOL LOADING:

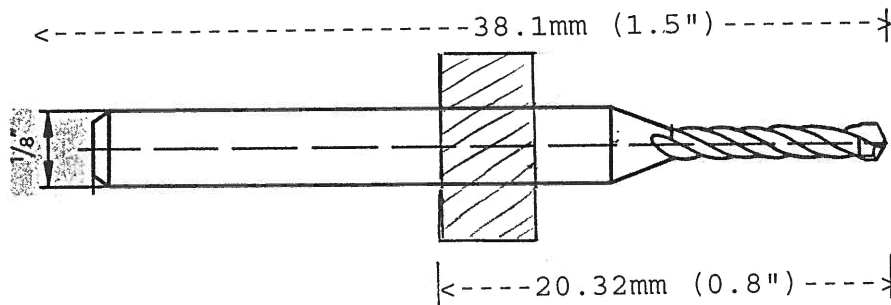
Before mounting the tools in the tool-magazines, they must be fitted with a collar, i.e. a plastic ring that fit exactly into the tool-pods.

The tool-length should preferably be the industry standard of 1,5" (38,1mm) measured from the drill point to the shaft-end. However the shorter 1 1/4"- version might be used, but is not recommended.

The enclosed "collar setter" is for adjusting the collar rings on the tools.

The collar-rings must be adjusted exactly to the industry standard of 20,32mm (or 0,8") measured from the TOP END of the ring to the POINT of the tool.

ILL. :



The measure of 20,32mm must be closely observed, as deviations might cause either damage to the machine's workplane and the HF-spindle bearings or (if less than 20,32mm) insufficient drilling or routing results.

NOTE:

The collar setter will only work properly with collar-rings with the following dimensions: Ø7,50mm x H 4,50 mm (tol.:-0/+0,1mm).

If the height of the collar-ring is less than 4,5mm the resulting distance from the the spindle nose to the tool-point will be less than the specified 20,32mm (0,8"), hence the holes will not be drilled all the way through the workpiece.

Alternatively, if the ring height is more than 4,5mm, the result will be, that the tools might damage the workplane and spindles as the distance from the spindle-nose to the tool-point will exceed the specified distance of 20,32mm (0,8").

Most of today's suppliers of tools for PCB-production can supply tools already fitted with collar-rings to your exact specifications, and this is greatly preferable, especially if the tools are returned to the manufacturer for re-grinding. Only collar-rings with smooth edges and of proven quality should be used!

COLLAR SETTING :

The hole over the dial micrometer is used for setting the tools to the specified distance of 20,32mm (0,8") measured from the top of the collar ring to the bottom, i.e. the drill point.

The collar-ring is placed in the countersink hole over the dial micrometer, and the tool is put through the ring and the hole.

Then the rocking lever is moved down until the micrometer is just activated and the reading is zero (0).

For checking of the micrometer adjustment please insert the testing-mandrel (test gauge) in the same countersink hole over the micrometer and press down the rocking lever firmly.

Now, the micrometer reading must be zero. If not, please adjust the micrometer setting by turning the dial or by loosening the allen-screw and move the micrometer up or down to the reading is exactly zero.

The collar setter can be used for all tool-diameters. However, for tools larger than $\sqrt{3,175}$ mm it is necessary to perform an additional operation before final adjustment is carried out according to the above mentioned.

- 1) First place the collar ring in the countersink hole over the micrometer and then insert the tool in the ring but upside down, i.e. with the tool-point pointing upwards and the tool-shaft pointing downwards through the collar-ring. Then press the lever down until the tool's cutting edge is just touching the collar-ring.
- 2) Then turn the tool with the collar-ring and insert the tool in the same countersink hole and now again press the lever while observing the micrometer reading which must be exactly zero.

As the collar setting device is not equipped with any 'end-stop' for automatic adjustment of the tools, one must closely observe the micrometer reading for each tool, as any deviations may cause severe damage to the machines vital parts or insufficient drill-performance.

When removing the collar-ring from the tool, it is necessary to use the left-side countersink hole.

Place the tool with the collar-ring in the left side and operate the lever until the tool has been 'punched' through the collar-ring and drops down.

EDITING:

HOW THE COMPUTER WORKS.

HARDWARE:

The "Unitrol-2" computer is equipped with a keyboard, an alpha-numeric display and a joy-stick. These elements are used when the operator creates the CNC-programs, that are necessary for the production of a PCB.

A CNC-program consists of various numeric codes, called "working-codes", each of which will control the machine to perform a certain operation.

The Mape machines use a unique blend of "working-codes" and hardware keys to perform the various desired machine operations.

The most frequently used functions or commands (e.g. "FETCH TOOL!", "SEARCH REFERENCE", "TEST PROGRAM" etc.) are directly accessible via the keyboard, as these functions are used very frequently during production and programming.

THE MEMORY:

The total memory-capacity is 6500 "blocks" (= programming lines). You can imagine the memory of the computer as one, long "streamer" from a calculator, where each line equals a "block" which is the name of the command-lines.

To put the computer on the first block in the memory, you simply activate the " MEMO"-key. This brings the computer to a place in the memory, just before the first block accessible to the user (i.e. block no. "0"). This position in the memory can very well be considered a "parking-place" for the computer.

But why does the computer go to block 0, and not to block no. 1? This is because the computer always shows the block just BEFORE the memory-block, where can insert a possible new block. *Therefore: A new block will always be inserted on the next line in the memory!*

This is because, it is most convenient that the computer shows the actual block, when you want to insert a new block in the memory.

But what will happen, when you remove a block in the middle of an existing program? Will the memory have "empty spaces" left over? No!

The memory is arranged in a way, that it will never produce "empty blocks" when deleting blocks from a program.

In the same way, it is possible to add ("insert") new blocks to an existing program, without having the risc, to write into existing blocks.

THE MEMORY....

The computer will always create the necessary space in the memory whenever adding new block, simply by "pushing" the following blocks one line ahead every time a new, additional block is inserted in the program.

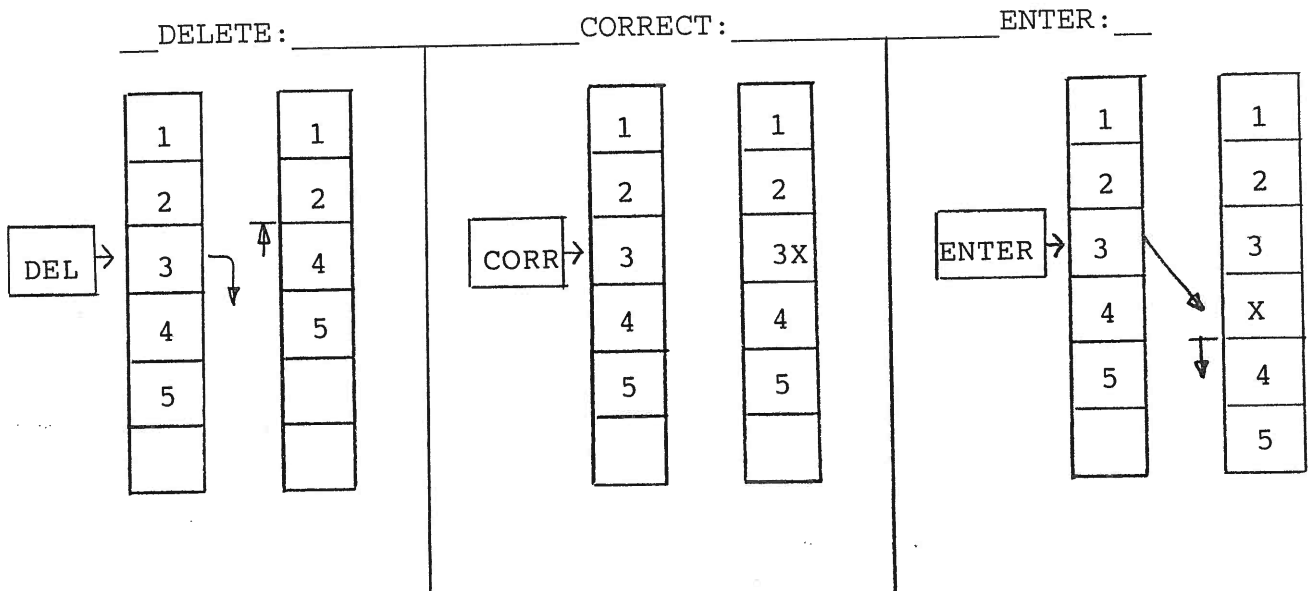
In the same way, the computer will automatically move the following blocks one line backwards each time a block is deleted from an existing program, thereby eliminating the possibility for any "empty blocks" in the memory.

When using the "CORR"-key (which purpose is to correct or "modify" an existing block), the block is just changed to the desired value, and since only the content of the block is changed (not the size), no change in the memory will occur in this case.

This is due to the fact, that all block in the memory are of equal size, regardless what the content of the blocks are.

In the example below is shown, how the editing-keys of the computer works.

EX.:



PROGRAMMING:

Basically, there are two ways of programming the machine:

- 1) By manual programming, i.e. using the joystick and the module-keys in combination with the video programming system.
- 2) By entering a ready-made program (CAD-program) from a PC e.g. the Mape 'ProgramBank' front-end system connected to the machine's serial interface (RS-232C).

1) MANUAL PROGRAMMING ('DIGITIZING'):

For manual programming it is recommended to use either a 2-stack PCB with blank copper surface (i.e. less tin and solder-mask) OR a positive film placed on a white, transparent acrylic board of 3mm thickness, which is provided with two pilot holes to suit the pinning system of the machine, i.e. typically $\varnothing 3.00\text{mm}$ or 1/8".

1A:) PROGRAMMING FROM FILM:

The film is aligned to match the axes of the machine and attached to the acrylic board with tape.

The module-keys are especially well suited for the physical alignment of the film, because the operator needs to travel along one of the sides of the film or between two fiducial marks on the film in order to align it with respect to the machine's axes.

For this purpose, the operator may easily enter any desired distance between two marks, enabling him to use the machine's module-keys for convenient and speedy positioning of the machine camera in order to align the film to the machine axis.

The alignment of the film is especially important when utilizing the automatic programming features of the machine, such as the module-keys and the joystick auto-centring facility, because these functions are in parallel with the machine-axes.

Programming by means of film provides maximum brightness on the TV-screen, completely free of disturbing reflexions from the pattern of the circuit board.

1B:) PROGRAMMING FROM PCB:

Programming can also be carried out by using a printed circuit board. The maximum contrast is achieved by using a non-tinned PCB without solder-mask.

In both cases it is preferably to draw a working-route on the surface before programming is started.

PROGRAMMING.....
1B) FROM PCB.....

Programming directly from a PCB is recommended in most cases, because it compensates for any deviation between the film and the actual pattern of the PCB-surface.

The main advantage is, that the PCB is self-aligning to match the machine's axes due to the pre-made tooling holes of the PCB which always are (should be) in-line with the PCB-pattern.

The major drawback compared to the film-method is, that one can obtain better contrasts with a film mounted on a white acrylic board.

2) PROGRAMMING FROM CAD/FLOPPY DISK:

Finally, it is possible to enter a CNC-program directly via the interface system of the machine. The interface system is a standard "RS-232"bidirectional interface for the use with peripheral equipment such as a PC (Personal Computer) or any other external device having a serial interface with the following standard specifications:

INTERFACE PROTOCOL FOR MAPE CNC MACHINES:

SIGNAL CODING : ISO / ASCII / EVEN PARITY.
TRANSMISSION RATE : 9600 BAUD (selectable from 1200-19200 baud)
SIGNAL COMPOSITION: 7 DATABITS, EVEN PARITY + 2 STOPBITS.

If the machine is connected to the Mape PC-ProgramBank Front-End System, the way of program transfer is simply carried out according to the menu's of the Mape PC.

Please refer to the operating-manual for the Mape ProgramBank which is included in the PC-software. The menu's of the system will also guide you through the operation of the input/output of the machine.

By means of the Mape ProgramBank it is possible to enter CAD-programs directly from a floppy-disk via a PC to the Mape machine, saving most of the tedious programming work.

ProgramBank Front End Systems supplied as an integral part of a Mape machine are factory set to 19200 Baud, in order to maintain a minimum machine down time.

PROGRAMMING FROM CAD.....

If you are using another PC with your own software or you want to communicate with other CNC-equipment, a brief explanation of the way of operation shall be given here:

INPUT:

To download a CNC-program from an external device to the machine, you must first find a place in the memory of the machine where you want the downloaded program to begin. Normally, this would be from block 0, i.e. the very beginning of the machine's memory.

To find this block simply press the " MEMO"-key, after which the display will show >>BLOCK: 0000<<.

Now, press the "DATA IN" key of the machine followed by an "transmit"-command of the external device, whereby transmission takes place.

However, using the "DATA IN" / "DATA OUT" keys means that the programs must be in "Mape Format" i.e. consisting of Mape acknowledged codes and formats only.

If you want to receive drill programs in other formats, you must add a 'shift code' before activating the "DATA IN" key.

For detailed information about the various possibilities, please refer to the interface section of this manual.

Please observe, that the interface protocol of the external device must be in strict compliance with the set-up of the Mape machine as described above. If not, the result will be incorrect or incomplete data transfer.

OUTPUT:

To download a program from the Mape machine to an external device, please do as described above, but using the "DATA OUT" key instead of the DATA IN key.

GENERAL:

A more technical discussion the interface will take place elsewhere in this manual. Please refer to this section.

PROGRAM STRUCTURING:

The item you want to program (PCB or film) is placed in the programming pin/slot system on the machine (below the camera), and the "REF!"-key is activated.

The brightness and intensity of the TV-monitor can be adjusted according to the requirements.

Now locate the "block" (=program line) in the memory where you want the program to start (normally the first line in the memory), by using the keys: " MEMO", "FWD" or "BWD".

You might as well search any arbitrary location in the memory by means of the " CODE"-key or the "SHIFT"-function on the "FWD" "BWD"-keys, that allows you to respectively search any arbitrary code, e.g. "17" (=program-end), or "03" (=program identification number), or to "jump" any arbitrary number of blocks forward or backward in the memory.

The jump-facility is activated by keying:

"Y"/200/"SHIFT"/"JUMP FWD"

Then the computer will jump 200 blocks forward **-calculated from the present block number.**

Entering is carried out by means of the "ENTER"-key and you don't have to repeat codes which are identical to the preceding code, and you don't have to key "Y" each time you want to write in the "Y"-field, as the computer will do this automatically, unless the operator activates one of the "A", "X" or "Y" keys.

Each time the "ENTER"-key is activated, the computer will write an equation-sign ("=") on the display and sound a short "Beep". In this way, the operator is always informed, whether he has activated "ENTER" or not, hence avoiding double-programming of holes etc.

At the same time the block-counter will write the location of the memory (i.e. the block-number) on the display, as an information to the operator.

PROGRAMMING.....

EXAMPLE OF A DRILLING AND ROUTING PROGRAM:

A program for production of 3 drill sizes and one contour-routing might look this way:

```
A03 X00000 Y00001 = Program identification number 1.
A33 X00001 Y00080 = Tool pod nr. 1 is loaded with a 0,80mm drill.
A25 X..... Y..... = First canned drill-cycle co-ordinate.
A25 X..... Y..... = Last ditto.
A33 X00002 Y00100 = Tool pod nr. 2 is loaded with a 1,00mm drill.
A25 X..... Y..... = First drill co-ordinate in 1,00mm program.
A25 X..... Y..... = Last ditto.
A33 X00003 Y00330 = Tool pod nr. 3 is loaded with a 3,30mm drill.
A25 X..... Y..... = First drill co-ordinate in 3,30mm program.
A25 X..... Y..... = Last ditto.
A14 X00004 Y00240 = Tool pod nr. 4 is loaded with a 2,4mm router.
A55 X00120 Y00000 = Router radius compensation, left 1.20mm.
A04 X..... Y..... = First routing co-ordinate in 2,40mm program.
A15 X..... Y..... = Last ditto.
A17 X..... Y..... = Program end/Clear following memory.
```

Of course, drilling /routing parameters might still be inserted in the memory, but optimum performance is achieved when using one of the three built-in speed-tables.

If the operator want to insert his own speed and feed-rates in the program, these commands (e.g. codes A06/A16/A26) must be placed just after code A33 and code A14.

NUMERIC CAPACITY:

The numeric capacity of the CNC-2000 machines is five digits and signs, that is: X-99999 to X+99999 and Y-99999 to Y+99999. However, only the minus sign is allowed, as X99999 and Y99999 always are considered as positive values by the computer. No decimal points are allowed.

MILLIMETER SETTING:

The machine can accept both millimeters (mm) and inches ("). The millimeter setting of the machine is in 1/100mm, i.e. X12000 means X120.00mm.

INCH SETTING:

The inch-setting of the machine is in 1/2000", i.e. X12000 means $X12000/2000 = 6$ ". The reason for using 1/2000" instead of 1/1000" is, that it doubles the axis-resolution, resulting in higher accuracy.

PRODUCTION:

WARM UP:

To ensure the longest possible working-life of the the spindle-bearings, the HF-spindles must be run for 10-15 minutes at 25% of full speed, (=20.000 RPM) every morning, or whenever the machine has been switched off for more than two hours.

The warm-up function is activated by pressing the " WARM"-key.

This will cause the spindles to run at the factory set speed of 25% of full speed, and after 10-15 minutes the operator activates the "STOP"-key.

NOTE: *The procedure should be used immediately prior to the start of the production.*

This procedure is only necessary for machines supplied with ball-bearing spindles. For machines supplied with air-bearing spindles, no warm-up is needed.

WARNING:

Always leave the HF-spindles with the last used tool in the collet!

If not, the collet will be damaged (over-tightened), and this will affect dependability of the tool change as well as causing spindle run-out failures.

Please refer to the **SPINDLE INSTRUCTION MANUAL** supplied with the machine for thorough explanations and maintenance schedule.

SINGLE STACK DRILLING:

When only one spindle is needed of a multispindle machine, (drilling a single stack of PCB's), it is a must, that the other spindle is performing "blind-drilling", that is: **with tools mounted equal to the spindle in operation!!**

Also, it is recommended to use tools of 1,5 inch length (=38,1mm), as the use of shorter tool might cause problems. The huge pressure on the tool-shaft created by the spindle's clamping force, may cause a deformation of the collet, if the drill shaft is too short.

OFFLOADING TOOLS:

Before starting a new program, the tools left over in the spindle(s) from the previous day, must be returned to the tool-magazine.

The tool-pod used to receive the tool are defined by the operator, and of cause **MUST BE EMPTY!**

The setting of the tools is carried out by the operator by entering e.g. "10" / "TOOL ", and the machine will place the tools in tool-pod no. 10 (which must be empty before the execution)!

LOADING NEW TOOLS:

Now the tools used for the previous production may be removed, and the new tools presupposed by the program are mounted in the magazines. For this purpose, the computer-display will act as an easy reminder during the mounting, telling the operator which tool-sizes should be mounted in which tool-magazine number:

Example:

Following the reading or programming of the new program, code "A33" is searched by keying: "A33" / " CODE".

The display will first read the last code "A33" in the program, e.g.: **A33 X00003 Y00250**, which means, that tool magazine no. 3 must be loaded with a 2,50mm drill-bit.

When re-activating the " CODE"-key, the display will read the next to last code "A33", e.g.: **A33 X00002 Y00130**, informing the operator, that tool-magazine no. 2 must be loaded with a 1,30mm drill-bit.

This procedure is repeated, until all codes "A33" have been shown on the display, and the corresponding magazines have been loaded with drill-bits.

In case you also have a routing part in your program, the procedure can be repeated using code "A14", and the display will show, if any routers are used in the program.

MAXIMUM BOARD SIZE:

Before the start of production, one **MUST** make sure, that the size of the PCB's, which have to be processed, **DO NOT** exceed the size of the working-plane, as this might cause severe damage to the pinning system of the machine!

CAUTION: *The PCB's may hit the machine's rear steel side, thereby applying the full force of the axis drive to the small pilot hole in the pinning system.*

R E M I N D E R :

- 1) Run ballbearing HF-spindles for 15 min. at 25% of max. speed (20.000 RPM) prior to production start. (Air-bearing spindles need no warm-up).
- 2) Tools left in the spindles are returned into an empty magazine, before the tools for the new production are loaded.
- 3) Never run a HF-spindle without tool in the collet.
- 4) Pressure of compressed air must be controlled prior to start.
- 5) Clean the machine daily by means of a vaccum cleaner, especially the tool-magazines and the machine interior.
- 6) Clean the collets frequently, especially when much routing is performed by the machine.
- 7) NEVER use compressed air for cleaning!
- 8) CAUTION: NEVER obstruct the safety switch!
- 9) Keep temperature variations as small as possible (20 deg. C).
- 10) Control liquid level of the cooling system. Refill with 1/60 mixture of "NALCO 43-73" coolant and 59/60 tap water.
- 11) Control water-flow in cooling system frequently. The total flow should not be below approx. 1 Liter/minute when measured directly on the return flow to the cooling unit.
- 12) Empty and clean the cooling system of the machine every 6 months and refill the system with 30 liters of tap water plus 0.5 liters of Nalco 43-73 coolant.

IF IN DOUBT: ASK! You are always welcome to contact us by
phone: +45/86/199 811 or by fax: +45/86/188 188.
We are eager to answer your questions.

KEYBOARD-FUNCTIONS:

CONTROL-ELEMENTS ON MAPE UNITROL-2:

WARM \odot : This function will start the spindles at 25% of maximum speed, typically 20.000 RPM, and is intended for easy warming-up of the spindle(s) before production. It will not affect any other function of the machine. The vacuum unit will remain switched off, in order to keep the noise level as low as possible.

In order to maintain a good economy, it is possible to start programming work while the spindles are running.
(INSP) : This shift function will perform a test of the tool-change facility. All tool change commands (i.e. the A33- and A14-commands) are automatically recognized by the computer and each tool is picked-up and downloaded in the corresponding tool pods according to the program. The function can be used before a program is started in order to test, if all tools are properly inserted in the tool-pods.

99+INSP: Service-function only.
If a number '99' is entered before the activation of the SHIFT/INSP-function the machine will perform unlimited tool-change operations until stopped by the STOP-key. This function is mainly used for service-purposes, and is used for testing/run-in of new tool-pods or testing of the spindle tool change facility. When using this function it is necessary to fetch only one tool from an arbitrary tool-pod and make sure, that all remaining tool-pods are empty!

HITS ? : Displays the total number of hits performed by the HF-spindles, calculated from the last "reset" of the counter. The counter is reset to zero by using the shift-(RESET): function of the key: "RESET".

TOOL \uparrow : This function means "Fetch tool indicated in the A-field of the display". That is, when the operator enters a number from 1-10 (up to 20 for machines with 20-tool magazines) in the A-field and then presses the TOOL key, the machine will travel to the desired tool-magazine, and pick up the tool.

NOTE: The Collet must be empty before using this command!

(LEVEL): This shift function will lower the Z-axis (i.e. the HF-spindle) to the value indicated in the in the "Y"-digits in 1/100mm.

E.g.: If the operator enters "Y"/ "300" and activates "SHIFT"/"LEVEL", the Z-axis will travel from its top-position to a level 3,00mm above the lowest position.

Ex.:

If you enter Y0+SHIFT+LEVEL, the Z-axis will travel to the lowest position (which normally leaves a clearance from the drill-point to the workplane of approx. 0.9 to 1.0mm).

TOOL ↓ : This function means "Place tool in the tool-pod number indicated in the A-field of the display". The function is activated similarly to the TOOL -function described above.

WARNING: Remember that the tool pod in which the tool should be placed, MUST BE EMPTY before activating this function! Otherwise, the machine's automatic tool-change monitoring system will be triggered and cause a fault-message on the display.

(TABLE) : This Shift-function causes the workplane to travel to a position, where the tool-magazines are just uncovered, providing easy access to the tools and the workplane. It is especially useful when the operator wants to change the tools for a new production. Also, it can be useful when you need to perform manual work on the workplane.

START : This function starts the automatic execution of the program. The machine starts by carrying out the next instruction in the program, i.e.: the instruction displayed will not be executed!
When START is activated, the computer will read all the introductory instructions entered by the operator (i.e. tool-size and -location, feed and speed commands etc.) after which the computer will start the HF spindle(s) and the vacuum unit and move to the first X,Y-position of the program, after having changed to a tool according to the instructions.

(CHECK) : This Shift-function allows the operator to have only one (that is: the next) program-step carried out by the machine. Normally this function is useful for "checking" the position of the first hole drilled, in order to adjust the drill-position for maximum accuracy (utilizing one of the "Floating Zero"-facilities).

STOP : This function causes an unconditional STOP in the machine operations. The key also acts as an EMERGENCY STOP!

The machine can be stopped during ALL operations:
The machine will remember the exact situation where it was stopped, and can be started again without drawbacks, even during complicated contour routing or Step&Repeat-

operations!

The STOP key also cancels the automatic start/stop-feature of the protection shield, used during the production phase.

MEMO ↵ : This function puts the computer on memory location no. zero. I.e. a "park position" just before the first block accessible in the memory.

(END) : This Shift-function allows the operator to search the last block of the program(s) stored in the memory. This final block found by the computer is indicated in the display.

CODE Ⓞ : This function searches for a code/coordinate entered by the operator. If the code or block were found in the memory, it will be shown in the display. If not, the display will show:

>>IDENTICAL BLOCK NOT PRESENT<<

or

>>A-CODE NOT PRESENT<<

If only the two-digit A-code is entered, the computer will search (backwards) in the memory to the nearest identical A-code. If the operator is not satisfied with the first proposal from the computer, he just activates the "CODE" -key again, until the right A-code is shown in the display.

If the operator wants to search for a specific program-number, e.g. no. 12345, the entering is carried out this way:

"A03 Y1235" and then press "CODE".

(COORD): This Shift-function makes it possible to search an arbitrary position on the worktable. The function is especially useful, when a broken drill has to be relocated after having changed to a new drillbit. By means of the joystick, simply travel the machine to a position close to where the drill is broken, and activates the "SHIFT" and "COORD"-keys, after which the computer internally will calculate all positions in the program, and automatically travel to the nearest position in the program (calculated from the present position!). This means, that the operator doesn't have to localize the spot exactly, as the computer will do this for him.

NOTE: It is even possible to use the function in complicated STEP & REPEAT patterns!

REF! : This key causes the machine to travel to it's datum-point, that is, the machine's "reference point", in proportion to which the programs are normally made. The reference coordinates of the machine is normally X13000 Y00500. In the datum point position, the camera will be sighting exactly on the pilot-hole of the central pinning system, i.e. the "programming" pin-system.

However, the coordinates of the pilot hole may differ from one customer to another, as the value can be set in the software to match other CNC-machines datum-coordinates or when measuring system of the CNC-2000 is set to millimeters instead of inches (adjustable via the machine's INTERNAL REGISTERS).

(HOME) : This shift-function is used whenever the operator wants the workplane to travel to the original datum-point regardless of any floating zeroes etc. When using this function it is possible to see, if a floating zero is activated: If the display shows any other values than the standard datum-coordinates (i.e. X13000Y500 for machines adjusted for inch-operation), then the floating zero is activated, and it is possible to calculate the floating distance which is the difference between the standard datum coordinate values and the values now being displayed. The computer also utilizes the HOME-function when the "oφ ADJ"-function is used (see this).

φ SET : This key "presets" the position counter of the machine, to a value determined by the operator. The function is also called "FLOATING ZERO", and is normally used to "synchronize" the position counter of the machine to match a program made externally, i.e. from a CAD-system or another CNC-machine having datum-points of different X,Y-values compared to the CNC-2000/1000.

Evidently, this function is able to displace the "0"-point (datum-position) of the machine, and can operate with this displacement until changed by the operator or the "REF!"-key is activated, after which all displacement values are neutralized.

This function is widely used, when the operator needs to "adjust" the position to fit the pattern of a previously produced PCB, e.g. when the pinning-holes in the PCB's have been changed for some reason.

The 'adjustment' is carried out by pressing the 4 incremental-keys around the joy-stick until the desired position is reached. When the "oo ADJ"-key is released, the machine will operate with the displacement entered by the operator via the module keys.

(XY's) : This shift-function allows the operator, to add or subtract an arbitrary X- and/or Y-value to all the X,Y-positioning codes of an existing program. The function is used, when a definitive shift in position is desired by the operator, e.g. when the pilot-holes of a PCB has been shifted in position for some reason.

NOTE: As the X,Y-values is changed in the memory (and not only during execution), the function should be used with care, as the original program is changed!

To add a certain X,Y-value, enter the value in the X and/or Y-digits of the display and press "SHIFT" and "XY's".

Immediately, all X,Y-positions of the program will be changed according to the value entered by the operator. To subtract a value, just enter a minus sign before the value is entered and then activate the function as described above.

AUTO-ADD However there is also an automatic feature under this function, named "AUTO-ADD": When a program is processed utilizing the floating-zero feature of the machine, and the operator wants to eliminate the floating-zero adjustment procedure for future program executions, he simply adjusts the machine zero to match the PCB-pattern the first time the PCB is processed in the machine. Then he activates the "SHIFT / HOME"-function, whereupon the machine travels to the datum-point and shows the datum co-ordinates including the operator defined 'offset' on the display. Without pressing any other keys or the joy-stick, the operator now presses the "SHIFT" and the "XY's "-keys. The computer will now automatically recalculate the entire program to match the machines original datum point. This means, that the program is now converted and matches the machine zero, so that no 'offset' or floating zero-adjustments are needed for this program, which will save time and possible errors.

SHIFT : The "SHIFT"-key is always used in combination with one of the other keys on the control panel. The purpose of this key is to double the number of keys on the control panel, without affecting the general view of it.

When activating the SHIFT-key, all the functions printed in **blue** are valid.

The procedure to make the floating zero-adjustment is:

- 1) Put the PCB in the programming pin/slot-system.
- 2) Let the machine position to the first drill-position by activating the "TEST"-key.
- 3) Travel the machine (by means of the joystick or the incremental module-keys) to the correct position.
- 4) Press "FWD"-key and "BWD"-key.
- 5) Press "o SET"-key.
Now, the machine will be "adjusted" to the new, correct position.

The same procedure can be used, when the operator wants to correct the position slightly during production. However it is also possible to change the position by

- 1) Put the PCB in the programming pinning system.
- 2) Travel to a known and specified position, (e.g. the datum-mark of a CAD-pattern with specified X,Y-value).
- 3) Then enter the X,Y-value specified in the CAD-pattern, and press the "⊕ SET"-key.

Now the machine will be in accordance with the CAD-pattern (or any other pattern) and the production can start.

(FIXED): This Shift-function allows the operator to make a new machine datum-point with the machines original datum point coordinates (normally X13000,Y00500 for machines with standard inch-software) whenever desireable. Just travel to the point where you want the new datum-point to be, and enter "SHIFT" & "FIXED".

Then all machine positions are now synchronized with respect to this new datum-point, and will remain so, until the "REF!"-key is pressed or another floating-zero is entered in the computer.

⊕ ADJ : When activating this key, the machine will automatically set the module-keys to 1-step (appr. 0,01mm) increments, and will "freeze" the X,Y-values of the position-counter (shown on the display) until the key is released. While the key is activated, the operator can adjust the position of the machine to fit the pattern of the PCB drilled, in order to obtain maximum accuracy.

If an invalid Shift-function is activated, the display will show:

>>UNDEFINED SHIFT-FUNCTION<<.

In this manual, the SHIFT-FUNCTIONS are indicated in brackets.

TEST : This function reads the next instruction in the memory, and activates the positioning system of the machine: If the next instruction is considered a valid X,Y-position of the computer, (that is, if the A-code is a positioning-code such as A25, A04, A15, A01 etc.), the machine will travel to the X,Y-value connected to this code.

No positioning occurs if the A-code is considered an information-code such as program-no. or feed- and speed-parameters etc.

(POS) : This Shift-function enables the user to travel to an absolute X,Y-value entered by the operator. This function is normally used, when a certain reference-point is known (i.e. according to a drawing) and one wants to locate this point on a PCB-pattern. The X,Y-value is simply entered via the "X" and "Y"-keys followed by the activation of "SHIFT" and "POS"-keys. Then the machine immediately will travel to the value entered in the display by the operator. If the entered value is beyond the machine position-limits, the display will show:

>>POSITION OUT OF TABLE<<.

FWD : This function causes the computer to advance one block (one line) in the memory, and simultaneously show the block in the display.

This function is widely used for programming and editing work. No machine operations (i.e. movement of the X, Y or Z-axes) will be carried out.

(JUMP) : This shift-function causes the computer to advance an operator-defined number of blocks in the memory. The number of blocks is calculated from the present block, that is, if you decide to advance 500 blocks in the memory, and your present position in the memory is block 200, the resulting block-no. will be 700.

Please, refer to the chapter: "Programming and program structuring".

BWD : This function causes the computer to go back one block through the memory, and simultaneously show the block in the display. The function is, in other words, the opposite to the FWD-function described above.

(JUMP) : This Shift-function causes the computer to jump any arbitrary number of blocks backward in the memory, calculated from the present block-number. The function is the opposite of the "JUMP-FWD" described above.

DEL : This key DELETES or "erases" the present block shown on the display. Simultaneously, all the following blocks will be moved one block backwards, so no "empty spaces" will occur in the memory. That is, all following block-numbers will be reduced with one, after having deleted one block in a programme. Of course, deleting 10 blocks, will result in a reduction of 10 in all following block-numbers. For further informations, please refer to the separate section "How does the computer work?".

(PROG) : This Shift-function causes the computer to delete the entire program indicated in the display! To delete a program, the operator localizes the program in the memory utilizing e.g. the "CODE"-function or the "FWD" or "BWD"-keys. Once the display shows the desired program number, the "SHIFT" & "PROG"-keys are activated, and the program is immediately deleted.

WARNING: This function may delete programs unnoticed !!

CORR : This function corrects the block shown on the display to another code or X,Y-value defined by the operator. The function is widely used when a position has to be corrected, or an A-code or parameter must be changed. If an invalid "A"-code is entered by the operator, the display will show:

>>UNDEFINED A-CODE<<.

(PROG) : This Shift-function allows a total correction of a PCB-pattern regardless of the coordinate-values and X/Y-directions. The function is normally used when a CAD-program is entered into the computer via the interface system or via the Mape PC-system.

In general it is quite time-consuming to correct a CAD-program to make it run on a CNC-machine. Normally the X,Y-values are not in correspondance with the machine zero and frequently, the X and Y axes are interchanged in the CAD-program, compared to the machine-axes.

With this shift-function it is made possible to adjust the CAD-program to the machine automatically, saving both time and efforts.

(PROG) : The way of operation is as follows:

First the CAD-coordinates are centered around the machine's central tooling slot and then any dislocation is calculated with respect to the machine's datum-point and the program corrected to match exactly the machine-zero.

If the program cannot be located inside the working area of the machine, the program will be rotated 90 degrees before the program is corrected to match the machine-zero.

If the program still cannot be executed due to the size of the PCB, the computer will display the following error-message:

>>PCB TOO LARGE<<

In the case of this error-message, please note, that all program-coordinates will remain in their original values i.e. the computer will perform no correction of the CAD-program.

However, this error-message can also occur when the operator have entered the CAD-program according to a wrong scale, i.e. if the operator has informed the computer, that a specific program is in millimeters, but the program in fact was in inches. In this case, the program would appear to be 27% larger than actual, and the computer may give an error-message although the machine is able to process the PCB if the operator had entered the correct informations regarding the measuring-unit.

ENTER : The "ENTER"-function causes the computer to move the present A-code and X,Y-value indicated on the display, into the memory on the following block-number. Simultaneously, all following blocks are advanced one block in the memory, in order to "create space" for the new block just entered. The function is also used, when adding data to an existing program. For further informations, please refer to the section "How the computer works".

< ^ v > : "MODULE KEYS" (or incremental-keys). These four keys are placed on each side of the joy stick, and are fully programmable via the "GRID"-key, or via various SHIFT-functions (described in the corresponding paragraphs).

The module-keys can be programmed as follows:

- 1) Fixed incremental distance 1/10" or 2.54mm.
- 2) Fixed incremental distance 1/20" or 1.27mm.
- 3) Fixed incremental distance 1/2000" or 0.01mm.
- 4) Fixed arbitrary incremental distance in units of 1/100mm.
- 5) Momentary (arbitrary) incremental distance in units of 1/100mm.
- 6) Fixed router-radius compensation distance.

(1+2+3): The fixed incremental distances 1/10", 1/20" and 1-step, are programmed by activating the "GRID"-key (see the description below). The "GRID"-key is activated the necessary number of times by the operator, to the desired distance is shown in the "GRID"-field of the display.

When the "GRID"-key has been activated three times, it will start all over again, i.e. with the first proposal. In machines which are set up to inch-software the three proposals will be displayed like this:

- 1) >>GRID: 00200<< which means $200/2000" = 1/10"$,
- 2) >>GRID: 00100<< which means $100/2000" = 1/20"$,
- 3) >>GRID: 00001<< which means $1/2000" = 0.0005"$

In machines which are set up to millimeter-software the three proposals will be displayed like this:

- 1) >>GRID: 00254<< which means $254/100\text{mm} = 2.54\text{mm}$,
- 2) >>GRID: 00127<< which means $127/100\text{mm} = 1.27\text{mm}$,
- 3) >>GRID: 00001<< which means $1/100\text{mm} = 0.01\text{mm}$.

(4:) If a fixed arbitrary distance is desired, the "SHIFT/2"-function must be used. This function allows the operator to program the "MODULE-KEYS" in any arbitrary mm-distance in 1/100mm units. The desired distance must be entered in the "Y"-digits of the display and in 1/100mm (i.e. the actual distance multiplied by a factor 100).

That is, if you want to program a distance of 162,40mm, this is entered as follows:

"Y" / "16240" / "SHIFT" / "2".

Simultaneously, the "GRID"-field of the display will change to a corresponding value but converted from mm to 1/2000".

However, on machines with mm-measuring system, the programming of the incremental-keys is simpler, as it is

only necessary to enter the distance in the Y-field and then press the 'GRID'-key, e.g.: "Y" / 16240" / "GRID". Now, the incremental-keys are programmed to a distance of 162,40mm, and each time any of the incremental-keys are activated, the machine planes will travel the desired distance, i.e. up, down, left or right.

- (5:) If a momentary arbitrary distance is desired, that is, if you need a distance which is different to the one programmed via the GRID- or SHIFT-2 functions, it is possible to program a new distance momentary WITHOUT changing or deleting the fixed value programmed first. This is done very similarly to the SHIFT-2 function, as you just enter: "Y" / "16240" / ">", and then the machine will travel 162.40mm to the right. The distance just travelled, is then "forgotten" by the computer, and the original, programmed distance is again valid. In other words, the momentary distance can override the programmed distance for only one time, and then need to be re-programmed if desired.

- (6:) The router-radius compensation is carried out by means of the MODULE-keys as well. Once having programmed the A-code for router-dimension (=code "A14"), the incremental module-keys may be programmed to match the router-radius by entering:

"SHIFT" / "3".

The computer uses the last entered router-dimension as calculating basis; that is, if the last programmed code "A14" is a 2.4mm router, the computer will program the incremental module-keys to 1.20mm travel.

GRID : This key is used for switching between three different values of: 1/10", 1/20" and 1/2000" when the machine is delivered with inch-software. If the machine software is changed to millimeter-measuring system, the three fixed values are 2.54mm, 1.27mm and 0.01mm respectively. The desired grid-distance is observed in the "GRID"-field of the display, where all distances are shown in either 1/2000" or 1/100mm (according to the measuring system desired by the customer). The "GRID"-key is used for programming the incremental "MODULE"-keys, as described above.

- (A.S.C): "SHIFT" + "GRID", **Automatic Self Centering** of joystick. This function is activated by entering "SHIFT"/"GRID". This Shift-function provides an automatic self-centering of the joy-stick on the grid-position indicated in the "GRID"-field of the display.

To indicate when the ASC-function is active, a small circular ring with a dot inside, is shown in the display:

>> © <<.

To switch the ASC-function off, it is necessary to re-activate the "SHIFT"- and "GRID"-keys again, after which the small ASC-sign will disappear from the display.

DATA IN: Please refer to the "INTERFACE" section of this manual if peripheral equipment such as printer, CAD-systems, Computers or CNC-machines should be connected to and operated via the CNC-machine.

DATA OUT: Please refer to the above mentioned regarding the connection of peripheral equipment.

SYSTEM RESET: This is the most significant key of the machine, because it interrupts any machine operation in progress! The function sets all internal counters to zero, nullifies all distances and constants concerning STEP&REPEATS, SUPER REPEATS, MIRROR IMAGING, ROTATIONS, MAGNIFICATION CONSTANTS ETC! Also, it forces the Z-axis to it's top position, puts the computer on the first line in the memory and forces the operator to search reference point before any further machine operation is allowed.

The memory contents is not affected by activation of the "SYSTEM RESET".

Power-failures or disruptions of the mains supply are closely watched by an internal "watch-dog" in the power supply unit, and when even the shortest disruption of the mains supply occurs, the "watch-dog" will automatically activate the SYSTEM RESET-key internally.

MODE &: This function is a "STORE & RECALL"-feature, i.e. a (MODE) function that allows the operator to store and recall a single block (program-line) in the memory. The function is used when it is considered helpful to store and recall a block which is frequently used in a program. The function is used as follows:

"SHIFT"/"MODE" = Store the line shown in the display.
"MODE" = Recall the hidden line from the memory.

Please note, that the block stored in the memory is not deleted when the power is switched off.

SHIFT+1: This shift-function is for service-purposes only, and it is activated by entering "SHIFT" and the "1"-key. The function serves dual purposes:

- 1) It will close the HF-spindle collet in case of the collets should be open, e.g. after a tool has been loaded into a tool-pod.
- 2) It will also show the actual mains-voltage value on the display in volts a.c.
- 3) It will make it possible to open the HF-spindle's collet. This can be used when it is necessary to open the collet for service or repair, and the function is activated by entering: "99" / "SHIFT" / "1".

WARNING: *If a tool is stuck into the collet even after the "SHIFT/1"-function (open collet) is activated, the tool MUST BE REMOVED MANUALLY before any other operation is carried out!*

CAUTION: The collets must never be closed without a tool inserted, and the machine must not be switched-off when the collets are open!

SHIFT+9: This shift-function is for service-purposes only, and it is activated by pressing the "SHIFT" and the "9"-key. This function will perform a total RAM-test of the memory-section of the machine. If the memory-test is correct, the computer will transmit 4 short tones (Beeps). If the computer discovers any failures in the RAM-sections, the computer will write the following error-message on the display:

>>WARNING: FATAL ERROR<<.

*1-*6 : Undefined functions. Options.

- : Minus sign.

A : A-field of the display. Working code.
Used for assigned entering of codes in the A-field.

X : X-field of the display. X-value of coordinate or codes.
Used for assigned entering of values in the X-field.

SHIFT X: Swap X/Y's. This shift function performs an interchange of all X and Y co-ordinates and can be combined with a minus-sign to provide total control of a CNC-program, i.e. whether it should be rotated and/or mirrored after it has been entered into the machine's memory. This feature is especially useful when CAD-programs are entered in the memory, because many CAD programs are not in correspondance with the machine's X/Y-axis location. During this operation, the X,Y-values may be greatly affected i.e. that the program cannot be executed on the machine's workplane due to too large or negative co-ordinate values which is a possible result of the computer-calculations. However this can be overcome and automatically corrected by using the unique function 'SHIFT/CORR' which will automatically correct and adjust the program to suit the machines zero-point!

So, after using the 'SHIFT/X'-function the operator should utilize the 'SHIFT/CORR'-function immediately to prevent time-consuming 'manual' adjustments of the program's mechanical position on the workplane.

Please refer to the example below 'ROTATION OF CNC PROGRAMS FOR CNC-2000 / CNC-1000'.

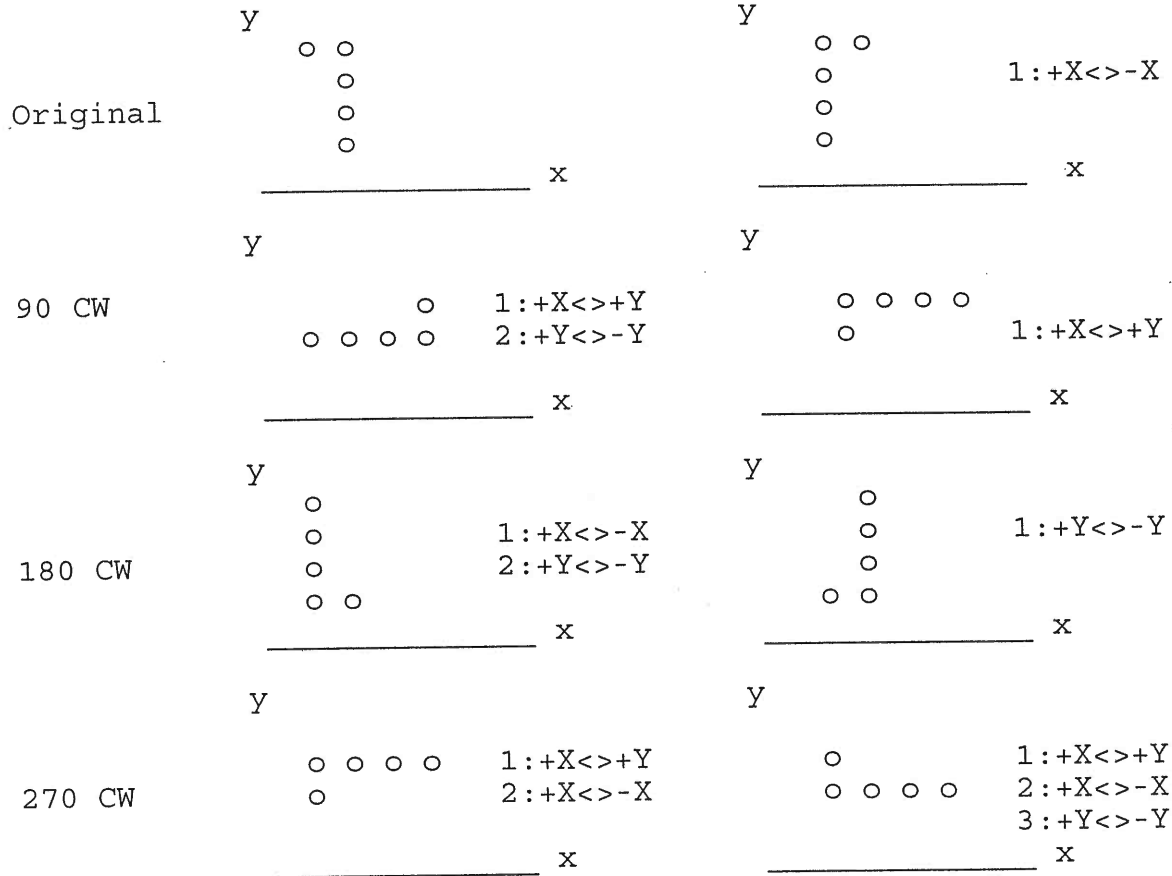
Y : Y-field of the display. Y-value of coordinates or codes.
Used for assigned entering of values in the Y-field.

SHIFT Y: Swap Y/X's. This shift function is the same as the shift-function described above ('SHIFT X') and is used in combination with the above function and in combination with the minus sign ('-') to provide full and automatic adjustability of e.g. CAD-programs or programs from other CNC-machine brands. The 'SHIFT Y'-function should also be used in combination with the special function 'SHIFT/CORR' for fully automatic positioning of the CNC-program on the machines workplane (centrering around the center-line and around the machine's reference-point).

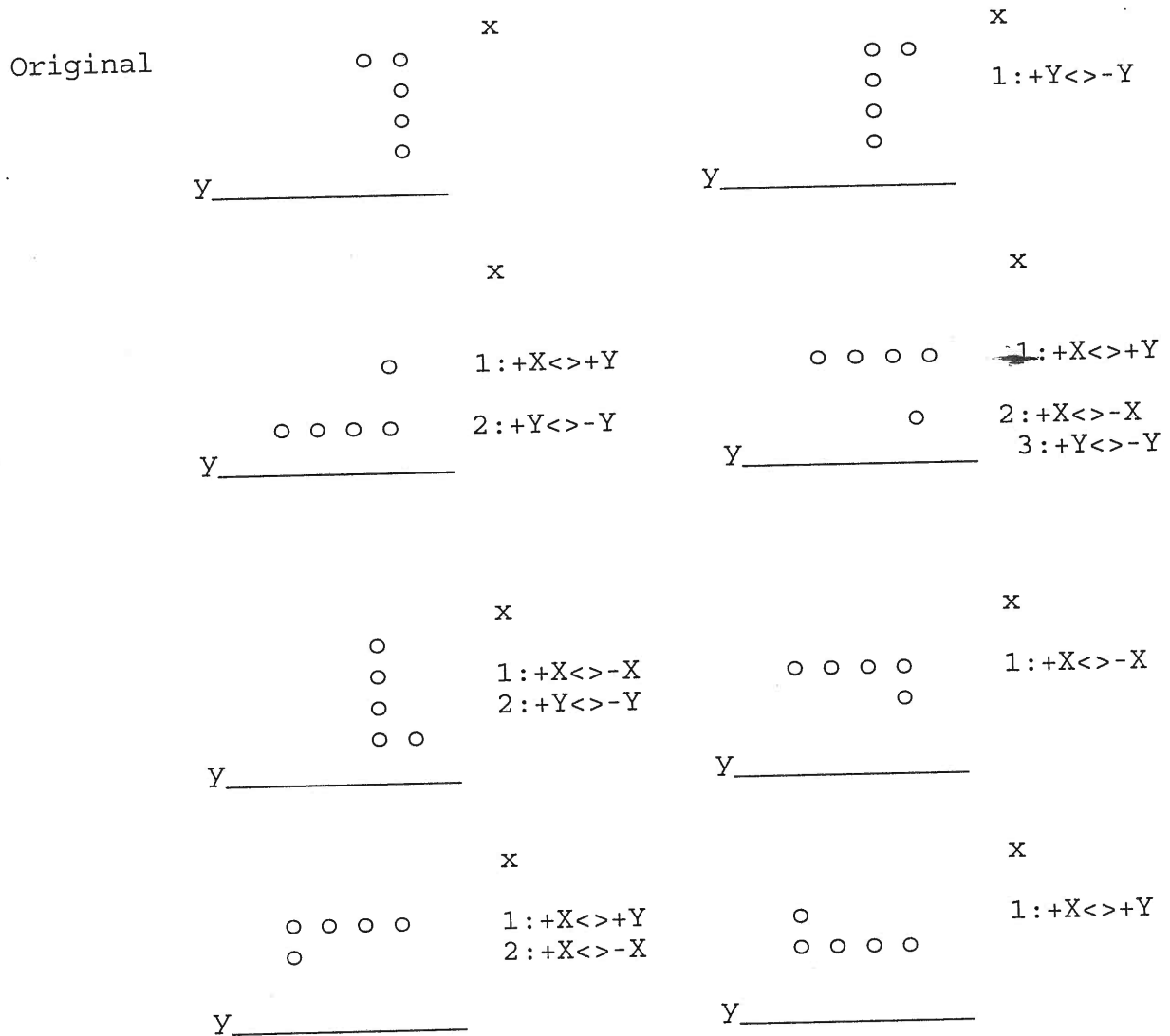
Please, refer to the examples on next side.

EXAMPLE OF AUT. ROTATION/MIRRORING OF CAD PROGRAMS:
 (THE "SHIFT/X", "-/SHIFT/Y" and "-/SHIFT/X" functions):

FOR CNC-2000:



FOR CNC-1000:



NOTE:

+X<>+Y = SWAP XY = SHIFT/X.

+X<>-X = NEG X = -/SHIFT/X.

+Y<>-Y = NEG Y = -/SHIFT/Y.

These functions should be followed by the SHIFT/CORR-function ("CORRECT PROGRAM") or the manually operated function "ADD XY's" which is activated by SHIFT/ooADJ.

0-9 : Numeric keys of the keyboard.

-/SHIFT : "Sign-function".

This shift-function enables interchanging of the signs for both X and Y coordinates, i.e. reversing of the numeric values from minus (-) to plus (+) or plus to minus of both X and Y coordinates.

In reality, this function performs a 'mirroring' of the pattern-coordinates compared to the original program. The function is widely used when the machine receives a CAD-program which is 'mirrored' with respect to the machine. In this case, it is most convenient to perform a mirroring of the axe(s) until the program is in conformity with the machine layout.

If both X and Y coordinates needs mirroring to obtain conformity between the machine and the program, the result is actually a 180 degree turning of the pattern.

The function is activated like this:

"-" / "SHIFT" / "X" This will change all X-coordinates from "+X" to "-X" and all "-X" to "+X". In fact this means a mirroring of the program in the X-axis.

"-" / "SHIFT" / "Y" This will perform negation of all the Y-coordinates as described just above. The result will be a mirror image of the original pattern.

SHIFT+2 : For machines with software in INCH-setting:
Module keys programmed to entered value in Y-digits in 1/100mm. An incremental distance of 65 mm is entered as:
"Y" / "6500" / "SHIFT" / "2".
Then the display will read the value in the "GRID"-digits, converted to 1/2000" which is the basic machine unit.

(Please refer to the paragraph "MODULE-KEYS" above).

For machines with the software in MILLIMETER-setting:
Refer to the paragraph "MODULE-KEYS" described above.

SHIFT+3 : Module keys programmed to router-radius compensation. The last entered router-diameter in the program is the point of reference for the computer when calculating the router-radius compensation.

(Please refer to the paragraph "MODULE-KEYS" above).

CLEAR : This key clears ("erases") the last entered value in the A-, Y- or X-fields of the display.
That is: If you have entered "A" and a value which is incorrect, then the activation of the "CLEAR"-key, will only erase the A-digits, and not the values in the X- and Y-digits.
If no assigned entering has been done, the activation of the "CLEAR"-key will erase all three fields in the display; that is, both the A, X and Y-field.

99+SHIFT+CLEAR: This function ERASES THE TOTAL MEMORY CONTENTS OF THE COMPUTER!!
The function is used when you want to delete the total memory contents.
As the computer automatically saves all programs entered via the keyboard or the RS-232 interface, that is, without deleting any of the previous programs entered, one will sooner or later exceed the maximum memory limit, and the display will show:

>>MEMORY LIMIT<<

Therefore, it is necessary to delete the memory from time to time. It is recommended, that you only store one program in the memory for each job in order to avoid confusion.
The memory erase function is deliberately "cryptisized" to avoid unnoticed loss of programs, and the function is activated by entering:
"99" + "SHIFT" + "CLEAR", after which the memory will be deleted, and the computer is returned to block no. 0, ready for programming.

WORKING CODES:

SUMMARY:

This is a summary of all working codes of the Mape CNC-machines. For more informations, please refer to the paragraph "DESCRIPTION OF THE WORKING CODES".

FUNCTION	CODE	X-VALUE	Y-VALUE	COMMENTS
Stop-command	A00			Unlimited break in machine-operation.
Center for circular routings & IC-functions.	A01			Center for all arcs and circles. Also startpoint for aut. DIL/SIL-pattern programming.
Dwell-point	A02			Programmable break-point in operation.
Program-number	A03	0-99999	0-99999	Should be first block in any program.
Positioning code (routing)	A04			Lift Z-axis (retract spindle) and travel to entered X,Y-pos.
Circular routing (diameter)	A05	Number of turns and aut.cross routing.	0-65000 x0,01mm	Circle diameter w/ aut. router-radius compensation. X=0 means 1 turn; X=1 means 1 turn+cross; X=2 means 2 turns+cross 'Y' indicates the inner diameter in 1/100mm.
Spindle speed	A06		0-60 x 1000 RPM	Aut. calculated if not programmed.
Change before sub-sequence.	A07			Enables changes between subsequence-S&R.
Change before main-sequence.	A08			Enables changes between mainsequence S&R.
Route counter-clockwise.	A09			Lower Z-axis and route to X,Y-position, with a radius calculated from the A01-position.

FUNCTION	CODE	X-VALUE	Y-VALUE	COMMENTS
Route clockwise.	A10			Lower Z-axis and route to X,Y-position, with a radius calculated from the A01-position.
Step&Repeat-subseq. follows	A11			Starts S&R-subsequence: repeat pattern follows.
Repeat sub-sequence.	A12			Repeats pattern between preceding A11 in XY.
Automatic S&R, "Super-repeat"	A13			Two-point aut. S&R-function with rotation and tool separation. Used for uncritical programming of mis-aligned patterns.
Fetch router (dia in 0,01mm)	A14	Magazine no. 1-10	Diamet. 10-600	Always first block in new router-size.
Linear interpolation.	A15			Z-axis down and route to X,Y-pos. indicated.
Z-feed	A16		1-125mm per sec.	Aut. calculated if not programmed.
Tape and memory end code.	A17			Indicates last block in memory. Deletes succeeding parts of the memory.
Mirror-imaging over X-axis.	A18			Mirror X-coordinates over an arbitrary point on the Y-axis. Result: Rear-side up on PCB.
Mirror-imaging	A19			Mirror Y-coordinates over the X-axis (see above).
Mirror-imaging	A20			Combination of X and Y-mirroring. Result: 180 degree Rotation.

FUNCTION	CODE	X-VALUE	Y-VALUE	COMMENTS
Step&Repeat-mainseq. follows	A21			Starts S&R-mainsequence repeat-pattern follows.
Repeat main-sequence.	A22			The pattern between the proceeding code 21 and the code 22, is repeated in X,Y (even if including A11/A12-S&R's).
Program bracket	A23			Program limit for repeat-functions
Cancel Mirror-image	A24			Cancels mirror-function codes A18/A19/A20.
Aut. "canned" drill-cycle.	A25			Used for drilling of holes in X,Y.
Router feed	A26		1-45mm per sec	Aut. calculated if not programmed.
Center for 'scaling'.	A27			Scale-adaptation center used for calculating the magnification/reduction-factor by the computer.
Scale-adaptation 'from/to'.	A28			First code A28 indicates present size. Second code A28 indicates new size.
Cancel scale-adaptation.	A29			Cancels scale-function codes A27/A28.
Position-displacement, 'from/to'.		A30		All coordinates are shifted with the difference between the two codes A30.
Z-bottom position	A31		0-4500 (x0,01mm)	Aut. set to 0 if not programmed.
Program bracket	A32			Not active/FORMAT-1.

FUNCTION	CODE	X-VALUE	Y-VALUE	COMMENTS
Fetch drill (dia in 0,01mm)	A33	Magazine no. 1-10	Diamet. 10-600	Always first block in new drill-size.
Cancel displacement.	A34			Cancels displacement- codes A30.
Stop spindle & travel to X,Y.	A35			No longer active. (Format-1 only).
Z-top position	A36		0-4500 (x0,01mm)	Aut. set to 8 mm if not programmed.
Lift router.	A37			Not active/format-1.
Rotation center.	A38			Rotational center used by the computer to calculate the rotation.
Aut. rotation 'from/to'.	A39			First A39 indicates present position. Second A39 indicates desired rotated pos.
Cancel Rotation.	A40			Cancels rotation- codes A38/A39.
IC-pin spacing.	A41		0,01mm	Aut. set to 2,54mm if Y=0 or not programmed. Must be entered in this sequence: A41, A01, A48 or A41, A01, A45.... Additional A45 (or A48) codes will repeat the pattern as a parallel displacement. Code A41 needs only to be programmed once, then it remains active until changed to a new value.

FUNCTION	CODE	X-VALUE	Y-VALUE	COMMENTS
SIL-pin number.	A42			Aut. calculated on the basis of a 2,54mm-grid if not programmed or Y=0. Must be entered in this sequence: A42, A01, A48 (A48...); additional A48 codes will repeat the last pattern as a parallel displacement. The A42-code must be entered each time a SIL-pattern is desired (contrary to code A41).
Position preset.	A43			Presets (synchronizes) the position-counters to the indicated X,Y-value.
Preset position off.	A44			Cancels all offsets, (e.g. A43 offset).
Last position in SIL- or DIL-patterns.	A45			Last hole in SIL/DIL-patterns which are in parallel to the machine axis. Entering sequence: A41(or A42), A01, A45.. Several A45's may be entered for repeats of first entered pattern.
Z-return feed	A46		0-125mm per sec	Aut. calculated if not programmed.
Dwell-timer	A47		0-65.000 millisc.	Programmable delay in machine operation.
Last position in arbitrary SIL-patterns.	A48			Last position in any arbitrary (rotated) SIL-pattern. Entering sequence: A41 (or A42), A01, A48 (A48..). More A48's may be entered as a parallel displacement of the first entered SIL-pattern.

FUNCTION	CODE	X-VALUE	Y-VALUE	COMMENTS
Lifetime counter.	A49		0-99999	Indicates the highest desirable hit-count of a drill-bit. Y=Lifetime limit, X=Remaining lifetime. A49 is entered after A33 (Tool-change).
APSS-table.	A50		Y=1,2,3	Automatic speed selection system: A50Y1=Slow/hi quality. A50Y2=Standard setting. Quality/speed. A50Y3=Hi speed setting. A50 is entered after A03 and remains active for all tools.
Aut. 'corner round-off'.	A51		Rx0,01mm	Single corner round-off function with compensation.
Aut. 'corner round-off'.	A52		Rx0,01mm	Multi-corner round-off function with compensation.
Aut.arc-routing linear/circle	A53	X=CW.	Y=CCW.	Routing of arc between linear tracks. '-X' or '-Y' means routing of the larger arc.
Router	A55	X=left	Y=right	Aut. router-radius compensation to the left or right of the routing track. The value in X or Y is the router-radius in units of 0,01mm.
Reserved	A56			For new function.
Reserved	A57			For new function.
Reserved	A58			For new function.
Reserved	A59			For new function.

FUNCTION	CODE	X-VALUE	Y-VALUE	COMMENTS
Router compensation OFF.	A60			Cancels router radius compensation-codes A61 or A62 (Sieb+Meyer mode equals G40).
Router compensation 'LEFT'.	A61			Aut. router radius compensation to the LEFT of the track. The calculation is based on the last A14 code entered (Y-value/2). Equals Sieb+Meyer G41.
Router compensation 'RIGHT'.	A62			Aut. router radius compensation to the RIGHT of the routing track. Calculated on the last entered A14 Y-value/2. Equals Sieb+Meyer G42.
Check-drill.	A63			Drilling of check-hole outside S&R-patterns. A63 must be entered after last A25/before next A33.
Reserved.	A64			For new function.
Outside circular routing.	A65	X=number of tabs.	Y=diameter x0,01mm.	Full-circle routing on the outside with aut. fixpoint selection. Code A65 is used with A01 (as center-pos.). X-value indicates the desired number of fixpoints on the circle, while Y indicates the desired outer diameter in 1/100mm. FROM SOFTWARES > V5.0.

DESCRIPTION OF WORKING-CODES:

This section of the manual describes in details the function codes of the Mape machines.

A00: STOP-CODE.

This code means "unconditional stop" in the machine-operations. When you activate the "START"-key, the machine will proceed with the program again.

The A00-code can be used, if you are uncertain about some details of a program, and would like to stop the machine on a certain place in the operation. Then you can enter a A00-code temporarily in the program. When the computer "meets" the A00-code during the operation, it will stop all further program executions, just until you activates the "START"-key.

A01: CENTER FOR CIRCULAR INTERPOLATIONS AND DIL/SIL PATTERNS.

The A01-code indicates the XY-center for all circular routing-commands, such as codes A05, A09 and A10.

The A01-code is also used for indicating the first hole XY-position of the automatic DIL/SIL-pattern generating codes A45 and A48.

Always consider the A01 X,Y-position as pin no. 1 of any Dual-In-Line (DIL) pattern.

When programming DIL-IC-patterns, it is mandatory, that the code A01 indicates pin no. 1 according to the standard rules of IC-pinout. Otherwise, the IC may be drilled in the opposite direction.

A01...

When using the code A01 in combination with the A41 ("GRID DISTANCE") code, the A01 X,Y-position must be located exactly at the right spot according to the A41-value. If not, the computer may automatically adjust the start-point (X,Y-value) of the A01-code slightly, because the A45 or A48 end-point position is always fixed!

EX.1:

```

(A45#2)
A01 X,Y (#1)      o o o o o o o o
A45 X,Y (#2)      .                               correct.
                   o o o o o o o o
(A01#1)
```

If pos. #1 and pos. #2 accidentally were programmed in the opposite corners of the IC, the result would have been:

```

(A01#1) o
wrong:   o
         o
         o (A45#2)
```

In the example above, a standard 100 mil spacing is assumed.

A02: INSPECTION-STOP.

This code is used whenever it is desirable to have a temporary stop during the execution of a program. The code is especially usefull when routing operations are performed, and it is desirable that the machine stops during the execution and wait for the operator to remove possible loose parts of a PCB or to remove the "entrance-boards" of the PCB-stacks before the routing program is started.

The code remains active (i.e. the delay is unlimited) until the operator opens and closes the protection-shield of the machine ("AUTO START"-facility) or until the operator activates the "START"-key, after which the machine will continue the execution of the program.

A02...

The A02-code is inserted in the program just before the place, where it is desired to stop temporary, e.g. just before a routing-operation is executed:

EX.: A14 X..... Y..... (tool-change to router).
 A02 X..... Y..... (inspection-stop).
 A04 X..... Y..... (first routing position).
 A15 X..... Y..... (next routing position).
 etc.

In the example above, it is possible for the operator to remove the aluminium-entrance boards before the routing-operation takes place.

One could also imagine, that it was necessary for the operator to remove a large piece of PCB-material, that has been routed loose during the operation, before the machine continues with the next routing sequence:

EX.: A14 X..... Y..... (tool-change to router).
 A01 X..... Y..... (center for circular routing).
 A05 X..... Y..... (routing of circle & diameter).
 A02 X..... Y..... (inspection-stop).
 A04 X..... Y..... (next routing sequence....)

A03: PROGRAM-NUMBER.

This code indicates the beginning of every program. Because the computer is able to contain several programs in the memory at the same time, it is necessary for the computer to have a code for separation of the individual programs.

However, the A03-code is not only used for the purpose of separating the programs, but is also used for numerous other applications by the computer.

The A03-code is also used as a "limitation" for the computer so that it remains in the program under execution. One can also say, that the A03 code is some sort of a paranthesis, that limits the working-area of the computer, where the first A03-code (i.e. the program-number for the actual program) should be considered as the beginning-paranthesis "(", while the possible following A03-code or a code A17 is considered as the ending-paranthesis ")" or limit by the computer.

On the next page it is shown scematically how the computer

A03...

works in the memory, i.e. how you should begin and end your CNC-programs.

Additionally, code A03 also serves as a "lower limit" for the step & repeat commands A12, A22 and A13, provided no other "limit-codes" such as A11, A21 or A23 are entered.

Code A03 also serves as a "cancellation-code" for all the mathematical constants which are calculated by the computer each time it executes a program.

Whenever the computer "meets" a A03-code in the memory, it will automatically cancel the constants for the following codes: A27/28 (scaling-factor), A30 (displacement), A38/39 (rotation), A13 (super-repeat), A18/19/20 (mirror-imaging) as well as many other functions, such as step & repeats, the automatic corner round-off function and the router radius compensation function.

The cancellation of all these constants etc. are most important, because if not, the computer would accidentally influence other (new) programs with the constants from previous programs.

The cancellations takes place in the very moment, where the A03-code is shown in the display. Therefore, when a new program is started, the computer will automatically be "cleaned" for possible constants deriving from previously executed programs.

When the " MEMO"-key is activated, this will also cancel the constants in the computer, just as if the computer has observed a code A03.

When the computer meets a code A03 it will finally also cancel all possible variations from the initial setting of the machine (the setting it becomes when power is switched on).

The computer will automatically choose the following default values:

APSS table-value:	TABLE-2.
Z top-position:	8,00mm.
Z low position:	0,00mm.
Spindle-speed:	25.000 RPM.
X/Y/Z-feed rate:	16mm/sec.
Z return feed:	125mm/sec.

A03...

Schematic illustration of the memory structure, assuming max. memory size of 6500 blocks and two CNC-programs:

BLOCK-number	BLOCK-content	COMMENTS:
0	xxxxxxxxxxxxxxxxxxxx	Reserved block for computer.
1	A03 X00000 Y00001	Program-no. 1.
2	A33 X00001 Y00080	Tool-change #1, drilling 0,8mm.
3	A25 X..... Y.....	First drill-position/ tool #1.
+n	etc.	+ 'n' drill-positions/ tool #1.
110	A33 X00002 Y00110	Tool-change #2, drilling 1,10mm.
111	A25 X..... Y.....	First drill-position/ tool #2.
+n	etc.	+ 'n' drill-positions/ tool #2.
279	A14 X00003 Y00240	Tool-change #3, routing 2,40mm.
280	A04 X..... Y.....	First routing position/ tool #3.
+n	etc.	+ 'n' routing positions/ tool #3.
316	A03 X00000 Y00002	Program-no. 2.
317	A33 X00001 Y00090	Tool-change #1, drilling 0,90mm.
318	A25 X..... Y.....	First drill-position/ tool #1.
+n	etc.	+ 'n' drill-positions/ tool #1.
468	A33 X00002 Y00160	Tool-change #2, drilling 1,60mm.
469	A25 X..... Y.....	First drill-position/ tool #2.
+n	etc.	+ 'n' drill-positions/ tool #2.
519	A25 X..... Y.....	Final drill-position of tool #2.
520	A17 X..... Y.....	Final block in memory/ end of program/ clear rest of memory.
521	A00 X00000 Y00000	
522	A00 X00000 Y00000	
+n	-"- --"---- --"----	+5977 blocks with "0" 's.
6499	A00 X00000 Y00000	Final memory block available.

A04: POSITIONING CODE.

The A04-code is most frequently used to indicate the first position of a routing-program.

The code serves more purposes:

- 1: The machine is lifting the Z-axis (the HF-spindles) to the upper working-position (i.e. the position which is controlled by code A36, normally 8 mm).
- 2: The work-planes of the machine are travelled to the X,Y-position indicated in the A04-code.

Code A04 is a pure positioning-code only, i.e. Z-axis movement will NOT be activated, when using this code. It is always the following code (normally code A15), that determines whether Z-axis movement should be activated, i.e. pulled down.

Codes with this ability are e.g. A25 (drilling), A15 (routing), A05 (circular routing) or A02 (inspection-stop).

Finally, code A04 can also be used for special purposes, e.g. when using the "floating zero"-function ("oo SET"-key), and the code A43 "preset position" -function.

A05: CIRCULAR ROUTING.

The desired diameter is entered in the Y-field of the display in 1/100mm, and the computer compensates automatically for the router-diameter on the basis of the last entered code A14 Y-value.

EX.:

```
A14 X00006 Y00240 (fetch router 2.4mm in magazine #6).
A01 X..... Y..... (center for the circular routing).
A05 X00000 Y00950 (routing of 9.50mm circular hole).
```

— This example means, that around the X,Y-coordinate in code A01 there will be routed a circular hole having a diameter of 9,50mm. The routing will be executed with a 2,4mm router which is picked up from tool-magazine no.6.

Furthermore, the computer will automatically compensate for the router-diameter with the value specified in the A14-code in this way:

$$240/100\text{mm} : 2 = 1.20\text{mm}.$$

A05...

If this automatic compensation was not carried out, the resultant diameter of the routed hole would have been 11.9mm.

However, there are some limitations for the A05-code:

- 1) The maximum diameter of the circle may never exceed 655,35 mm.

If one enters a larger diameter than 655,35mm, the computer will write the following error-message in the display:

>>RADIUS OUT OF RANGE<<.

NOTE: If a part of the circle is exceeding the max. working area of the machine, this will be noticed by the computer during the actual routing operation (i.e. not during a test of the program by means of the "TEST"-key). In this case, the machine will stop and the computer will show the A05-code that causes the problem, as well as writing the following error-message:

>>POSITION OUT OF TABLE<<.

- 2) The desired circle-diameter **MUST** be larger than the router diameter. If this is not the case (e.g. on account of a faulty entering), the computer will write:

>>PARAMETER OUT OF RANGE<<.

- 3) When performing circular routing, the feedrate is limited to 50mm/sec (=3 mtrs/minute). This limitation is due to the fact, that increased feedrates will result in unacceptable high inaccuracies in the workpiece, due to the high pressure on the router.

The deviation of the routed track may be as large as 1 mm if excessive feedrates are used during the routing operation!

If one has entered a higher feedrate than the maximum available (= 50mm/sec.), the computer will automatically limit the max.feed to the said 50mm/sec.

During the routing operations, one will frequently have pieces of loose materials which might cause problems, as these can get stucked between the workpiece and the Z-axis etc. (especially if the loose pieces of PCB are of a certain shape and size).

A05...

If this creates problems, it might be advantageous to use the A02-code ("INSPECTION-STOP") after each A05-code, as this would make it possible for the operator to remove the "floating" materials during the execution of the program. (Please refer also to the paragraph "A02").

Another possibility is to use the A05-code with a special feature for cross-sectioning of the superflous material:

If you add a number in the X-field of the display for the A05 code, the machine can perform an automated cross-sectioning (division) of the loose piece of the circular holes routed as well as adding a second turn ~~of~~ the router to obtain a higher precision.

The way of entering is e.g.: A05 X1 Y2500,
where X1 means "1 turn" and "aut. cross routing".

A05 X2 Y2500 means "2 turns" and "aut. cross routing", i.e. the machine will make two turns (of which the first will be slightly under tolerance) and the second turn will be a high-accuracy turn with almost no router deviation.

When entering either 1 or 2 in the X-field under code A05, the automatic cross-routing feature will be activated, and the computer automatically calculates how many cross-sections are necessary to obtain the maximum size of the superflous material, that can be 'swallowed' by the extraction system.

This means, that small circles will not be divided at all, whereas medium sized pieces will be divided in two, larger sizes in 4 and very large pieces will be divided in 6 or 8 smaller pieces according to the following calculations:

CALCULATION:

Between 0-7,99mmØ there will be no aut. cross cutting of the superflous material. From 8,00 to 11,99mmØ the material will be cut in two only. From 12,00 to 14,99mmØ, a cross cutting will be generated by the machine. From 15,00 to 18,99mmØ the machine will perform a 6-cross cut and from 19,00 to 21,99mmØ the machine will make a 8-cross cut.

Above 22 mmØ the pieces will anyway be so large, that the function is bypassed.

NOTE: To the above mentioned diameters (Ø) one should add the diameter of the tool!

A05...

If X=0, the machine will not utilize the automatic cross-routing feature that automatically reduces the loose pieces to a size that can be absorbed by the machine's vacuum extraction system.

However, when the loose piece becomes too large, the computer will refrain from making any cross-sectioning at all, because this would take too much time and lead to unacceptable wear of the router. In this case, one should remove the pieces manually, e.g. utilizing the A02 inspection stop command.

A06: SPEED.

Code A06 sets the speed of the HF-spindle(s). The desired speed is entered in the Y-field in thousands of RPM, e.g.:

A06 X00000 Y00050

which means 50.000 RPM.

The code is only used, if you want to change the speed proposed by the computer (via the "APSS"-function). Code A06 must be inserted AFTER the tool-information (i.e. code A33 or A14) and AFTER a possible APSS-information (i.e. code A50).

EX.:

A33 X00001 Y00080 (tool-change/0,8mm drill/magazine #1)
A50 X00000 Y00003 (APSS-table #3 = hi speed level).
> A06 X00000 Y00045 (HF-spindle speed: 50.000 RPM).
A25 X..... Y..... (drilling....)
etc.

Machines equipped with "Precise SC-53-H" spindles, have a maximum speed-limitation of 80.000 RPM.

If the entered speed exceeds the max. allowable of 80.000, the computer will automatically limit the speed to the said 80.000 RPM, and the computer will write the following error-message on the display:

>>PARAMETER OUT OF RANGE<<.

A07: CHANGE BEFORE SUBSEQUENCE.

Code A07 is used in combination with code A11/12-STEP & REPEATS, where a change of one or more parameters is desired before the Step & Repeat is carried out, e.g.: "repeat with mirroring", "repeat with rotation" or "repeat with increased depth" etc.

The code is operated as in the following way:

When the computer meets a code A07 first time in a program, it will notice and store the following code that indicates the desired change (e.g. a code A31: 'Z-level'). The next code A12 ('repeat from code A11') will send the computer back to the A11-code. The program-part located between code A11 and A07 will now be repeated with the desired change. When the computer meets the very same code A07 (=the first) again, it will exclude this code as well as the code to be changed and the A12-code just executed, and instead it will jump to the next block in the program (which of course might be a second code A07 with a new change and a second code A12).

A08: CHANGE BEFORE MAIN-SEQUENCE.

This code has exactly the same function as code A07, but is used in combination with the codes A21 and A22 (STEP & REPEAT / MAIN-SEQUENCE).
(Refer to the paragraph "A07" above).

A09: CIRCULAR ROUTING, COUNTER-CLOCKWISE.

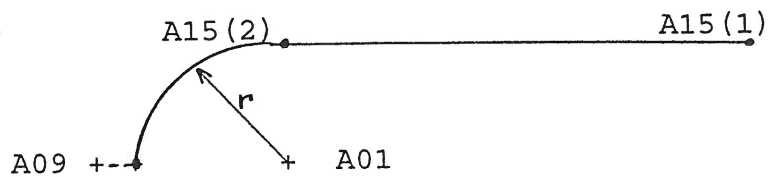
Code A09 performs counter-clockwise routing from any actual X,Y-position to the X,Y-position indicated in the A09-code. The code is used for routing of arc's and contours.

Code A09 is always used in combination with code A01, as this code indicates the center of the arc (and hence the arc-radius).

When using code A09, one should note, that both the entered center of the arc (A01) and the ending-point of the arc (A09) MUST be accurately entered (i.e. equal distances from the center). If not, one can expect large inaccuracies in the workpiece, due to calculation-errors.

A09...

EX.:



If the ending-point of the small example above (A09) is not accurately indicated, the computer will automatically displace this point to match a position on a circle around the A01-position and with a radius equal to the distance between the A01 and the A15(2)-position.

A10: CIRCULAR ROUTING, CLOCKWISE.

Code A10 is a routing code with exactly the same function as code A09 with the exemption, that it performs clockwise routing.

A11: STEP & REPEAT SUBSEQUENCE FOLLOWS.

The A11-code indicates to the computer, that a part of the program following the A11-code, should be repeated on a position indicated in the following code A12.

The STEP & REPEAT-codes A11/12 are primarily used for the programming of details on a PCB-pattern, such as IC-patterns and the programming of multi-connectors etc.

A12: REPEAT "S&R-SUBSEQUENCE" FROM CODE 11.

Code A12 indicates to the computer, that the program-part between the preceding code A11 and the code A12, must be repeated in the X,Y-position indicated in the A12-code.

In order to calculate the distance between the "original" pattern and the "repeated" pattern(s), the computer uses the distance between the last programmed positional code in the original pattern and the A12-position(s).

In other words, the computer uses the last entered hole or routing-position in the original pattern as the basis of its calculations. When placing the A12 code(s) it is mandatory to use the VERY SAME position(s) for the repeats as the last programmed position of the original pattern.

EX.:	IC-1			IC-2	
				A12	
A03 X..... Y.....	1	o	ø 8	o	ø
A33 X..... Y.....	2	o	o 7	o	o
A11 X..... Y.....	3	o	o 6	o	o
A25 X..... Y..... (1)	4	o	o 5	o	o
etc.					
A25 X..... Y..... (8)					
A12 X..... Y..... (REPEAT-POINT).					

In the examples below it is shown, how the computer will "understand" the different codes when using the A11/12 Step & Repeats, i.e. which "flow" the computer will choose when executing the examples below and on the next page:

EX-1:
 >>A03 (program-no).
 A33 (tool-change).
 A25 (drilling).
 --- (etc).
 A25 (drilling).
 A12 (#1. S&R).
 A12 (#2. S&R).
 etc.....

EX-2:
 A03 (program-no).
 A33 (tool-change).
 A25 (drilling).
 >>A11 (S&R SUB follows).
 A25 (drilling).
 --- (etc).
 A25 (drilling).
 A12 (#1. S&R).
 A12 (#2. S&R).
 etc..

"A12" PROGRAMMING EXAMPLES:

The next example will show the most simple form for a STEP & REPEAT-program:

```

A.. ..... (BLOCK: 0)
A25 X..... Y..... (BLOCK: 1) hole-1 ---> o
A25 X..... Y..... (BLOCK: 2) hole-2 -----> o
A25 X..... Y..... (BLOCK: 3) hole-3 --> o
A12 X..... Y..... (BLOCK: 4) S&R -----> o
A17 X..... Y..... (BLOCK: 5)
  
```

The example also includes a 3-hole-pattern drilled by an arbitrary drill size, and one S&R-pattern (similar to the original 3 holes) which is repeated in a new position desired by the operator.

Then, we complete the example from above to include both tool- and identification-numbers as well as two S&R-patterns:

```

A03 X..... Y..... (BLOCK: 0001)
A33 X....1 Y..100
A25 X..... Y..... -> o
A25 X..... Y..... ---> o
A25 X..... Y..... --> o
A12 X..... Y..... -----> o
A12 X..... Y..... -----> o
  
```

Now it is possible to add a second S&R-level. Consider the above program as a PCB-pattern consisting of 3 holes placed 3 times on the same PCB.

It can be a great advantage to STEP & REPEAT the whole PCB-pattern several times on a large PCB, especially if the original PCB-pattern is relatively small, in order to reduce processing time.

For this purpose, the code "A22" (S&R-Mainsequence) is used:

```

A03 X..... Y.....
A33 X....1 Y..100
A25 X..... Y..... o o o o
A25 X..... Y..... oo oo oo oo
A25 X..... Y.....
A12 X..... Y..... o o o o
A12 X..... Y..... oo oo oo oo
A22 X..... Y..... (A12#1)
A22 X..... Y..... o o o o
A22 X..... Y..... oo oo oo oo
A17 X..... Y..... (A12#2) (A22#1) (A22#2) (A22#3)
  
```

A12...

As can be seen from the example above, this program consists of a 3-hole-pattern which is repeated twice on the PCB, and then the entire PCB is repeated three times, making a total of 4 small PCB's mounted together on one large PCB.

If some holes are needed outside the Step&Repeat-patterns, or the Step&Repeat patterns varies during the program, the computer must be told where the repeat-pattern begins.

The operator must insert a "bracket" (e.g. code 11 or 21) in the program that defines the repeat-patterns. Each time a new pattern (e.g. an IC) is desired, the operator enters a code 11 (or 21 if main-sequence-S&R is used) just before the new pattern is entered.

Please refer to the description of code A22 (main sequence) later in the manual.

A13: SUPER-REPEAT.

This universal code is normally used with great advantage when programming artworks which have been mounted manually, typically when individual artworks have been copied without using sophisticated step- and repeat camera, i.e. when the individual films have been manually aligned and therefore are not completely in alignment with the machine's axes.

Also, code A13 can be used with great advantage when programming multi-pattern boards, i.e. boards containing small, identical patterns, especially when the individual patterns includes several tool-changes.

- Normally, it is very time consuming to program all the tool-changes for each individual pattern of a multi-pattern board.

However, when utilizing the A13-function, it is only necessary to program one original pattern completely as if only one boards was existing, and then add the A13-codes for repeating and automatically aligning the succeeding repeat-patterns.

After having programmed the first (original) pattern including all desirable tool changes, conventional step and repeats, IC-patterns, routings etc., it is necessary to enter two diagonally placed "checkpoints" (A13-positions) on the original pattern and two identically placed A13-positions for each repeat-pattern of the multi pattern board.

The first pair of A13-positions are used to indicate the original pattern and the following pair's of A13-positions are used to indicate the position and alignment of each of the desired repeat-patterns.

- For obvious reasons, it is imperative to program the original pattern very accurately and the same applies for each of the succeeding A13-pair of coordinates; otherwise the repeated patterns will not be correctly drilled or routed.

Please note, that there is a few restrictions in using the A13 SUPER REPEAT-function:

- A) In a program which is using the A13 SuperRepeat function, it is not possible to use the normal rotation-function, because the A13-function already utilizes the same function.
- B) The functions "change before subsequence" and "change before mainsequence" cannot be used when utilizing the A13-SuperRepeat function, which is in contrary to the normal step- and repeat functions (A11/A12 and A21/A22).

A13...

The way of operation is:

The computer uses the first A13 coordinate value as a check-point to calculate the step and repeat difference. The next A13 coordinate value is used for determining the rotation-value.

Hence, the order of programming is utmost important in order not to introduce faults in the program.

If the order of programming is changed, the computer will rotate the specific pattern 180 degrees.

All A13-codes must be programmed according to the following example:

EX.:

```
A03X.....Y.....
A33X.....Y.....
A11X.....Y.....
A25X.....Y.....
A25X.....Y.....
A25X.....Y.....
A25X.....Y.....
A12X.....Y.....
A12X.....Y.....
A33X.....Y.....
A25X.....Y.....
A25X.....Y.....
A25X.....Y.....
- "-
A25X.....Y.....
A14X.....Y.....
A04X.....Y.....
A15X.....Y.....
A15X.....Y.....
A15X.....Y.....
A11X.....Y.....
A01X.....Y.....
A05X.....Y.....
A12X.....Y.....
A13X.....Y..... (original, first corner)
A13X.....Y..... (original, second corner)
A13X.....Y..... (#1 repeat, first corner)
A13X.....Y..... (#1 repeat, second corner)
A13X.....Y..... (#2 repeat, first corner)
A13X.....Y..... (#2 repeat, second corner)
A13X.....Y..... (#3 repeat, first corner)
A13X.....Y..... (#3 repeat, second corner)
etc etc. up to (#N' repeat, first corner)
                (#N' repeat, second corner)
```

A13...

The program indicated above contains 2 drill sizes and one routing program with linear and circular interpolation. Also, there is involved conventional step- and repeats for both the drilling and the routing parts.

The example shown is maybe not realistic, but it gives an illustration of the flexibility of this universal function.

The computer calculates the rotations relative to the original pattern, and at the same time it automatically separates the tool-sizes, i.e. first all patterns are drilled with tool number one, next all patterns are drilled with tool number two and finally all patterns are routed with the router.

A14: ROUTER TOOL CHANGE.

This code performs a tool-change cycle in a similar way as code A33 (see this).

The entering sequence is:

A14 X5 Y240

which means that the machine will pick up the tool located in tool-pod number 5, which is a router with a diameter of 2.40mm.

The actual machine operation is, that the machine will first lay down the present tool (i.e. the tool which is in the spindle) in the tool-pod where it was originally taken from. Then it will travel to tool pod nr. 5 and take up the tool, and assume that it is a router, with a diameter of 2.40mm.

Because the computer actively uses the router diameter entered in the Y-field when calculating the router radius compensation, it is very important that the operator enters the correct value, i.e. the right diameter of the tool. If not this will lead to inaccurate hole-diameters and dimensions (if codes A61/A62 are used for the compensation).

The tool-change command A14 is normally the first code in a routing program, and if you use more routers of different diameter, you must enter a new A14-code for each new size.

The computer also uses the diameter indicated in the Y-field for calculating the optimum speed and feed for a given router size. (Refer to the APSS-function code A50 for specific informations re. speed and feed rates).

A15: LINEAR ROUTING.

This is a position code used for linear routing (linear interpolation).

The function of this code is to lower the Z-axis to the predetermined level and route to the X,Y-position given in the code.

The predetermined level is normally "Z0" i.e. full stroke down, but it can be set at any height (level) by the operator by means of the code A31 (see this).

The Z-axis is lowered to the desired level (height) at the actual XY-position and then the spindle is moved to the XY-position given in the A15-code, with the feed desired by the operator or given by the machine's automatic speed/feed selection table (APSS-function, see this).

EX:

```
A14 X7      Y200    (=tool change to T7 = router ø2.00mm)
A04 X0100   Y1000   (travel to X100,Y1000 with z-axis lifted)
A15 X1000   Y1000   (z-axis down, and route to X1000,Y1000)
```

This is an example of a simple linear routing along the Y-axis from X100 to X1000, i.e. 9mm routing distance.

The feed of the routing and the speed of the spindle is in this example determined by the machine's APSS-system, because there is no speed and feed command before the routing codes. Normally, you should also use the APSS system of the machine, because it gives optimum results, and simple programming.

In the following example, you will see a small program that takes a Ø2.00mm router from tool-pod no. 7, and where the operator has selected a spindle speed of 30.000 RPM, an in-feed rate of 16mm/sec and a routing-feed of 15mm/sec.:

EX:

```
A14 X7      Y200    (T7=Ø2.00mm ROUTER)
A06 X0      Y30     (SPINDLE SPEED 30.000 RPM)
A16 X0      Y16     (Z-AXIS IN-FEED 16mm/SEC.)
A26 X0      Y15     (XY-AXIS ROUTING FEED 15mm/SEC.)
A04 X1000   Y1000   (TRAVEL TO X10,Y10; Z-AXIS IN TOP POS.)
A15 X4000   Y1000   (Z-AXIS DOWN, ROUTE TO X40MM Y10MM)
A15 X4000   Y8000   (ROUTE TO X40MM Y80MM)
A15 X1000   Y8000   (DO, X10MM Y80MM)
A15 X1000   Y1000   (DO, X10MM Y10MM)
A17 X....   Y....   (END)
```

In the example above, the machine will route a rectangle of 30x70mm less the router diameter, meaning that the resulting measure of the rectangle will be 28x68mm.

A16...

The A16-code only determines the in-feed, i.e. the down-feed direction of the Z-axis. The retraction-speed (up-feed) is determined by another code, called A46 (see this).

EX:

```
A33 X7      Y200    (T7=Ø2.00mm DRILL)
>> A16 X0      Y16    (Z-AXIS IN-FEED 16mm/SEC.)
A25 X..... Y..... (TRAVEL TO X,Y and drill a hole)
A25 X..... Y..... (do. 2nd hole)
A25 X..... Y..... (do. 3rd hole)
- A25 X..... Y..... (do. 4th hole)
A25 X..... Y..... (do. 5th hole)
A17 X..... Y..... (END of program)
```

A17: PROGRAM END.

The A17-code is normally the very last code of any program.

It is an "END OF PROGRAM"-code, telling the computer to go back to the program-start and cancel all constants that might have been used during a specific program.

Such constants is made by the computer if the operator has utilized some of the machine's special features, such as 'rotation' or 'stretching' or 'offset' of a pattern during the production.

Please also refer to the A03-code for details about the mathematical constants built-up during the program-execution by the computer.

The A17 code also deletes succeeding programs that resides after the A17-code in the memory.

NOTE: Please observe, that this can lead to accidental loss of programs if a code A17 is entered between two programs.

A18/A19/A20/A24: MIRROR IMAGE FUNCTIONS.

A18 MIRROR X:

Code A18 mirrors the X-coordinates over the Y-axis in the indicated coordinate value.
All X-coordinates are mirrored over an arbitrary point along the Y-axis.
This means, that only the X-coordinate value is used, whereas the Y-coordinate can be any arbitrary value when using the A18-code, because the Y-value is not recognized by the computer.

After mirroring is performed, the back-side of a PCB pattern is turned upside.

The A18 mirror code for the x-coordinates must always be used in combination with the A11/A12 step & repeat function.

EX:

```
A03 X00000 Y12345 (PROGRAM NO. "12345")
A33 X00001 Y00100 (TOOLCHANGE T1=Ø1.00MM DRILL)
A11 X00000 Y00000 (S&R-PATTERN START)
A25 X01000 Y01000 (DRILL CYCLE, #1ST HOLE)
A25 X02000 Y02000 (DO, NEXT ....)
A25 X03000 Y03000 (DO, LAST HOLE)
A18 X00000 Y..... (MIRROR IN X0, y don't care)
A12 X03000 Y03000 (REPEAT IN LAST DRILLED POSITION)
A17 X..... Y..... (END OF PROGRAM)
```

The above example would drill a row of holes in a 45 degree turned position (compared to the machine axes).
The mirror-pattern would be repeated in a position starting with X-1000 Y1000 and ending with X-3000 Y3000, because we have used X0 as the mirror-position.

NOTE:

If the A12 code were omitted, the A18 mirrored pattern would not actually have been drilled in the new position.

A19 MIRROR Y:

Code A19 mirrors the Y-coordinates over the X-axis in the indicated coordinate value.
All Y-coordinates are mirrored over an arbitrary point along the X-axis.
This means, that only the Y-coordinate value is used, whereas the X-coordinate can be any arbitrary value when using the A19-code, because the X-value is not recognized by the computer.

A19...

After mirroring is performed, the back-side of a PCB pattern is turned upside.

The A19 mirror code for the x-coordinates must always be used in combination with the A11/A12 step & repeat function.

EX:

```
A03 X00000 Y12345 (PROGRAM NO. "12345")
A33 X00001 Y00100 (TOOLCHANGE T1=Ø1.00MM DRILL)
A11 X00000 Y00000 (S&R-PATTERN START)
A25 X01000 Y01000 (DRILL CYCLE, #1ST HOLE)
A25 X02000 Y02000 (DO, NEXT ....)
- A25 X03000 Y03000 (DO, LAST HOLE)
A19 X..... Y00000 (MIRROR IN Y0, x don't care)
A12 X03000 Y03000 (REPEAT IN LAST DRILLED POSITION)
A17 X..... Y..... (END OF PROGRAM)
```

The above example would drill a row of holes in a 45 degree turned position (compared to the machine axes). The mirror-pattern would be repeated in a position starting with X1000 Y-1000 and ending with X3000 Y-3000, because we have used Y0 as the mirror-position.

As in the example for the A18-code, the A19-pattern would not actually be drilled in the new position, if the A12-code were omitted.

A20 MIRROR X AND Y:

Code A20 mirrors both the X- and Y-coordinates over a point indicated by the XY-coordinate value of code A20.

All X-coordinates are mirrored over an arbitrary point along the Y-axis, while all Y-coordinates are mirrored over an arbitrary point along the X-axis.

This means, that both the X- and Y-coordinate values are used by the computer, when using the A20-code.

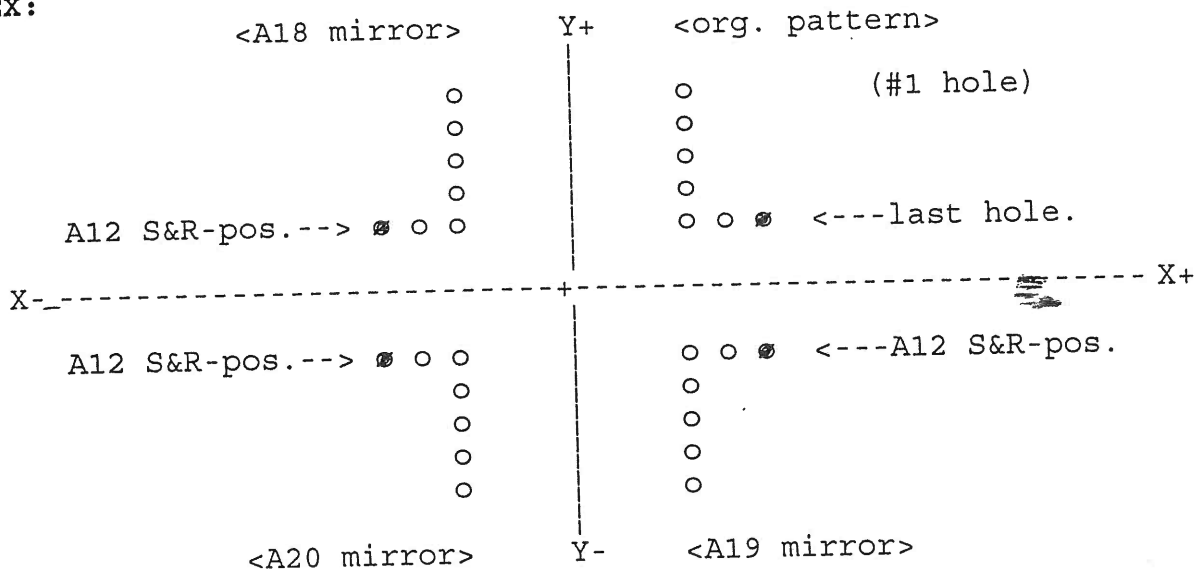
The A20-mirror function actually perform a 180 degrees turning of the original pattern. The turning point is the XY-coordinate values given in the A20-code.

As can be seen, the result is not a mirroring but only a turning of the pattern.

A20...

You can imagine the 3 mirror codes A18, A19 and A20 in the following example:

EX:



As for the A18 and A19 mirror codes, the A20 must always be used in combination with the A11/A12 step & repeat function.

Otherwise the A20 mirror will not actually be drilled on the desired mirror location.

As can be seen from the example above, the most convenient position for the X,Y-coordinate values of all the mirror commands, is X0,Y0 because this point is the exact center between all the 4 patterns.

However, the X0,Y0 can only be used in this example, because in real life, you can expect the center-point to be anywhere in the coordinate system.

But you can calculate this center-point if you know the corner coordinates of your original board. This corner coordinate is used as a reference for calculating the A18-, A19- and A20-mirror positions, and

the corner can be found by simply calculating it from a drawing, a CAD-layout or by using the machine's built-in video programming system. Using the video system, will actually give you the answer in seconds by simply putting the board in the machine and synchronize the drill program to the pattern and finally go to the 'reference-corner' and read the X,Y-coordinate directly from the display.

A20...

Let us assume the 'reference corner'-coordinate of the original PCB-edges is located in the position X1150,Y2100. Furthermore, we assume that you want a 2.40mm track between the boards, for convenient de-panelization.

This means that your 'mirror-line' should be placed 1.20mm to the left of the original PCB-edge (in the X-direction) and 1.20mm down under the original PCB-edge (in the Y-direction) in order to obtain a total distance of 2.40mm between the edges.

The reason for this is, that placing the mirror-line 1.2mm from the original edge, will mirror the repeated patterns $2 \times 1.2\text{mm} = 2.4\text{mm}$ away from the original position:

EX:

```
          ++++++++          y+1.2mm
----- y0
          ++++++++          y-1.2mm
```

As can be seen from the example above, you can now calculate the A20-coordinate value relative to the reference-edge coordinate of the original PCB which was X1150,Y2100. The A20 X-value should be $X1150 - 120 = X1030$. The A20 Y-value should be $Y2100 - 120 = Y1980$.

The center-point that should be used for all the mirror-codes, i.e. A18, A19 and A20 then is:

```
X1030,Y1980
=====
```

EX:

```
A03 X00000 Y12345 (PROGRAM NO. "12345")
A33 X00001 Y00100 (TOOLCHANGE T1=Ø1.00MM DRILL)
A11 X00000 Y00000 (S&R-PATTERN START)
A25 X02500 Y02500 (DRILL CYCLE, #1ST HOLE)
A25 X04000 Y04000 (DO, NEXT ....)
A25 X05000 Y05000 (DO, LAST HOLE)
A20 X01030 Y01980 (MIRROR IN X0,Y0 = TURN 180 degrees)
A12 X05000 Y05000 (REPEAT IN LAST DRILLED POSITION)
A17 X..... Y..... (END OF PROGRAM)
```

The above example would drill a row of holes in a 45 degree turned position (compared to the machine axes). The mirror-pattern would be repeated in a position starting with X-440 Y1460 and ending with X-2940 Y-1040, because we have used X1030,Y1980 as the mirror-position, i.e. all coordinate values are referenced to this position.

A20...

As for the codes A18 and A19, if we assume that the A12 code were omitted, the A20-pattern would not actually have been drilled in the new position.

A24: CANCEL MIRROR IMAGING.

The A24 code is used for cancelling the mirror-image codes A18, A19 and A20.

The A24 code can be used in special program sequences, that includes internal program brackets (e.g. code A23) which prevents the computer from going back to its normal starting line (A03) after a mirror-imaging has been utilized.

In such cases, or in cases where the operator wants to use several mirror-image functions in one program, it may be necessary to cancel the first mirror-image values, in order to avoid a build-up of mirror-image offsets every time the computer passes through the program.

The point is, that the computer normally is returned to its initial starting block (code A03) when it has reached the end of a program.

When it meets the original starting block, the inherent A03-code will automatically cancel all the calculated offsets etc which may have been used during the execution of a given program. So, in cases where special codes (like A23) prevents the computer from going back to its normal starting block, the computer does not cancel the build-up offsets or rotations etc.

In such cases, the computer may multiply the desired offset or rotation etc. which will lead to larger and larger errors of the machined patterns, every time the machine runs through the program.

A21: STEP & REPEAT MAIN-SEQUENCE FOLLOWS.

The A21-code indicates to the computer, that a part of the program following the A21-code, should be repeated on a position indicated in the following code A22.

The STEP & REPEAT-codes A21/22 are primarily used for the repeat of entire PCB-patterns, contrary to the A11/A12 sub-sequence step & repeat function which is primarily used to details of a single pattern, such as IC's and multi-connectors etc.

The A21/A22 main-sequence Step & Repat can contain several A11/A12 S&R-patterns.

A22: REPEAT "STEP & REPEAT MAIN-SEQUENCE" FROM CODE 21.

Code A22 indicates to the computer, that the program-part between the preceding code A21 and the code A22, must be repeated in the X,Y-position indicated in the A22-code.

In order to calculate the distance between the "original" pattern and the "repeated" pattern(s), the computer uses the distance between the last programmed positional code in the original pattern and the A22-position(s).

In other words, the computer uses the last entered hole or routing-position in the original pattern as the basis of its calculations. When placing the A22 code(s) it is mandatory to use the VERY SAME position(s) for the repeats as the last programmed position of the original pattern.

The last programmed position may well be a A12 sub sequence step & repeat code.

A22...

EX.:		pattern #1		pattern #2
A03 X..... Y.....	1	o	ø 8	o ø A22
A33 X..... Y.....	2	o	o 7	o o
A21 X..... Y.....	3	o	o 6	o o
A25 X..... Y..... (1)	4	o	o 5	o o
etc.				
A25 X..... Y..... (8)				
A22 X..... Y..... (REPEAT-POINT).				

In the examples below it is shown, how the computer will "understand" the different codes when using the A21/22 Step & Repeats, i.e. which "flow" the computer will choose when executing the examples below and on the next page:

EX-1:
>>A03 (program-no).
A33 (tool-change).
A25 (drilling).
--- (etc).
A25 (drilling).
A22 (#1. S&R).
A22 (#2. S&R).

EX-2:
A03 (program-no).
A33 (tool-change).
A25 (drilling).
>>A21 (S&R MAIN follows).
A11 (s&r sub follows).
A25 (drilling)
--- (etc.)
A25 (last drilling)
A12 (#1. s&r sub.)
A12 (#2. s&r sub.)
A22 (#1. MAIN S&R).
A22 (#2. MAIN S&R).
etc...

In the example "EX-2" above, the last code A12 (#2) is used as the offset-point for the following A22 codes. This means that the offset-distance is calculated as the distance between the A12XY-position and the A22XY-position(s).

"A22" PROGRAMMING EXAMPLES:

It is possible to add a second S&R-level to a program consisting of drill codes and A12 sub-sequence S&R's. Consider the program below as a PCB-pattern consisting of 3 holes placed 3 times on the same PCB.

It can be a great advantage to STEP & REPEAT the whole PCB-pattern several times on a large PCB, especially if the original PCB-pattern is relatively small, in order to reduce processing time.

For this purpose, the code "A22" (S&R-Mainsequence) is suitable:

```

A03 X..... Y.....
A33 X....1 Y..100
A25 X..... Y.....      o      o      o      o
A25 X..... Y.....      oo     oo     oo     oo
A25 X..... Y.....
A12 X..... Y.....      o      o      o      o
A12 X..... Y.....      oo     oo     oo     oo
A22 X..... Y.....      (A12#1)
A22 X..... Y.....      o      o      o      o
A22 X..... Y.....      oo     oo     oo     oo
A17 X..... Y.....      (A12#2) (A22#1) (A22#2) (A22#3)

```

As can be seen from the example above, this program consists of a 3-hole-pattern which is repeated twice on the PCB, and then the entire PCB is repeated three times, making a total of 4 small PCB's mounted together on one large PCB.

If some holes are needed outside the Step&Repeat-patterns, or the Step&Repeat patterns varies during the program, the computer must be informed where the repeat-pattern begins.

The operator must insert a "bracket" (e.g. code 11 or 21) in the program that defines the repeat-pattern. Each time a new pattern (e.g. an IC or greater sections of a pattern) is desired, the operator enters a code 11 or 21 just before the new pattern, which must be repeated later in the program is entered.

EX.:

Assume a layout consisting of two identical boards each with 3 IC's 16-pin DIL-patterns and some additional holes in arbitrary positions.

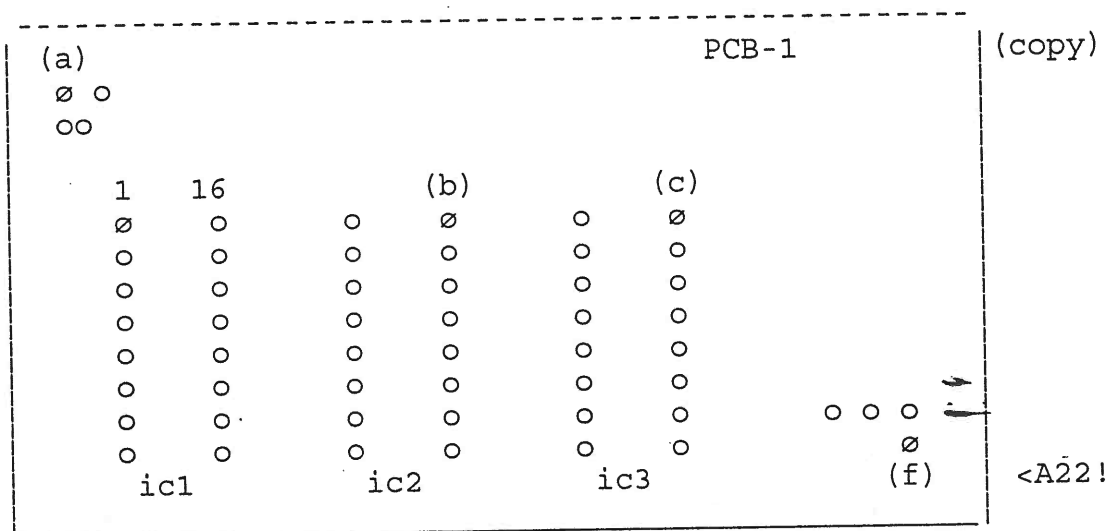
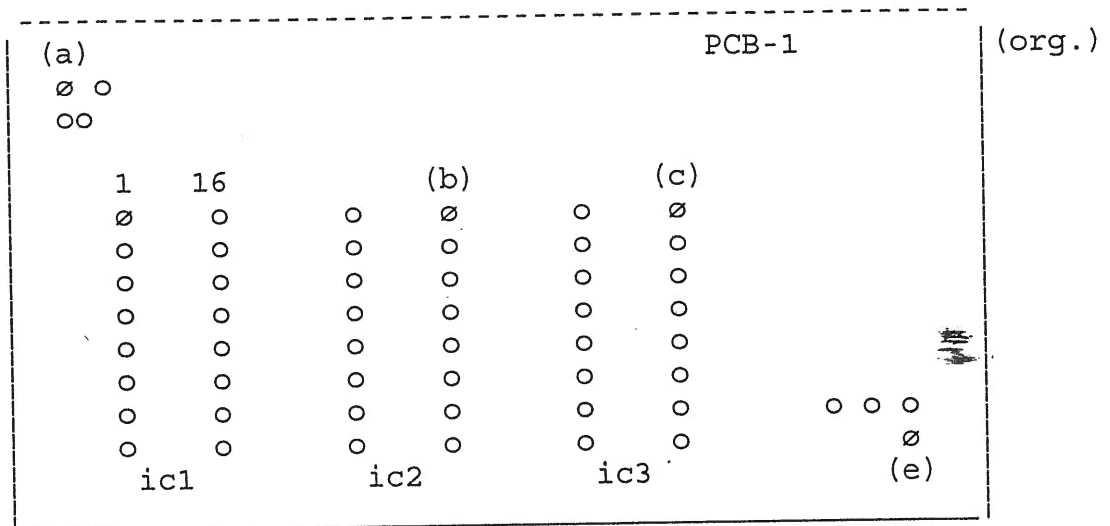
We will use both the 'subsequence-step&repeat' function as well as the 'mainsequence-step&repeat' function in this example.

The mainsequence-S&R command (A21) is entered first in order to tell the CNC that we want to repeat the entire pattern of the original PCB one (or several) times.

Then some of the arbitrary placed holes are entered into the memory using the standard A25-codes, after which we now want to enter the IC (DIL)-patterns. At this point we must tell the CNC that we now want to make some subsequence-S&R's, by using code A11, before we enter the drill-positions of the first IC-pattern.

A22...

The original pattern may look like this:



A22 PROGRAMMING EXAMPLES

The actual CNC-program of the a.m. example should look this way:

```

A03 X..... Y.....   Program number.
A33 X00001 Y00100     Tool-change, Ø1,00mm).
A21 X..... Y.....     Mainsequence S&R start.
A25 X..... Y.....     First drill position marked 'a' on drawing.
---
A25 X..... Y.....     Next drill position etc etc.
A25 X..... Y.....     Last drill position of the four arbitrary
                        holes in the upper left corner.
A11 X..... Y.....     Subsequence S&R START (of DIL patterns).
A25 X..... Y.....     First hole in DIL-pattern #1, 'pin 1'.
---
                        next 14 holes of first IC.
A25 X..... Y.....     Last hole in DIL-pattern #1, 'pin16'.
A12 X..... Y.....     First Step&Repeat of DIL-pattern, i.e. IC no.
                        2. The repeat-position must be the last hole,
                        i.e. in position marked 'b' on the drawing.
A12 X..... Y.....     Second S&R, i.e. IC no. 3. Again the repeat-
                        position is the last hole, marked 'c'.
A25 X..... Y.....     Now the 4 remaining holes in the lower right
---
                        corner are entered.
A25 X..... Y.....     Last hole in the board, marked 'e'. The first
                        board is now programmed and should be re-
                        peated in position 'f' of the repeated board.

A22 X..... Y.....     Mainsequence Step&Repeat (position 'f').
                        The original board will now be repeated in-
                        cluding the subsequence S&R's (A11/A12) with
                        an offset which is calculated as the differ-
                        ence between the last A25 position of the
                        original board ('e') and the position of the
                        A22 X,Y-value ('f').
                        The reference for a code A22 S&R can be any
                        positional code such as A25 or A12, A04 or
                        A15. So you may use a subsequence S&R as the
                        reference for the A22 mainsequence S&R.
                        It is possible to use the S&R-functions as
                        required. There is no limits in the number
                        of S&R's.

A17 X..... Y.....     Stop program.

```

NOTE:

Code A11 is a "bracket" or a limit that tells the computer from where to repeat, that is, code A11 and code A12 form a complete set of "brackets", where code A11 is the 'starting bracket' "(" and code A12 is the 'end-bracket' i.e. ")".

Exactly the same applies for code A21 and A22, except that A21 is the start-bracket in MAIN SEQUENCE S&R's and A22 is the end-bracket in main sequence S&R's.

Assuming you failed to program code 11 in the example above,

A22...

the computer would have repeated not only the DIL-patterns but also the 4-hole-pattern in the upper left corner throughout all the repeat patterns.

A25: DRILL CYCLE.

This code is always used for drilling of holes, hence it is - the most comonly used code in the entire code-system.

The meaning of the code is "travel to the x,y-coordinate indicated and drill a hole".

The depth of the z-axis movement, the in-feed and return-feed can all be controlled by using the corresponding codes before the A25-code is entered.

A typical drill program might look like this:

EX-1:

```
A03 X00000 Y12345 (PROGRAM NUMBER)
A33 X00001 Y00120 (TOOL-CHANGE T1=1.2MM DRILL)
A25 X..... Y..... (DRILL HOLE IN POSITION X,Y)
A25 X..... Y..... (DO.)
A25 X..... Y..... (DO. last hole in T1-part)
A33 X00002 Y00150 (TOOL-CHANGE T2=1.5MM DRILL)
A25 X..... Y..... (DRILL HOLE IN POSITION X,Y)
A25 X..... Y..... (DO, last hole in T2-part)
A17 X..... Y..... (END OF PROGRAM)
```

The program example "EX-1" above, utilizes the machine's automatic speed and feed selection system (APSS), and therefore it is not necessary to enter any codes for the control of in-feed or return-feed or spindle speed.

Also, the machine's standard settings for z-axis depth and stroke height is used. These settings are designed for a 8mm Z-axis lift and full penetration of a 3-layer of 1.6mm PCB-stack of boards including 1.6-2mm of backing board.

However, if the operator want to override the machine's standard setting, this is done according to the next example:

A25...

EX-2:

```
A03 X00000 Y12345 (PROGRAM NUMBER).
A33 X00001 Y00120 (TOOL-CHANGE T1=1.2MM DRILL).
A06 X00000 Y00042 (SPINDLE-SPEED 42.000 RPM).
A16 X00000 Y00034 (Z AXIS IN-FEED 34MM/SEC).
A46 X00000 Y00120 (Z AXIS RETRACTION FEED 120MM/SEC).
A31 X00000 Y00060 (Z BOTTOM POSITION, 0.60MM ABOVE ZERO)
A36 X00000 Y01000 (Z TOP POSITION 10.0MM ABOVE ZERO)
A25 X..... Y..... (DRILL HOLE IN POSITION X,Y)
A25 X..... Y..... (DO.)
A25 X..... Y..... (DO. last hole in T1-part)
A33 X00002 Y00150 (TOOL-CHANGE T2=1.5MM DRILL)
- A25 X..... Y..... (DRILL HOLE IN POSITION X,Y)
A25 X..... Y..... (DO, last hole in T2-part)
A17 X..... Y..... (END OF PROGRAM)
```

In the example above, the operator has overridden the machine's automatic speed and feed selections as well as the standard level settings for tool-1.

However, for tool #2, the operator has utilized the APSS-function of the machine, but the Z-level settings are still changed to the operator's specifications, because the Z-level setting codes are 'globally' active in a program and remains active until erased from the program.

This is for security reasons, and because a level change is normally used when there is a general change in the height of a workpiece (e.g. the stack of boards are increased from 3 boards to say 4 boards).

"A25" PROGRAMMING EXAMPLES:

```
○ ○ ○ ○ ○ ○
○ ○ ○ ○ ○ ○
○ ○ ○ ○ ○ ○
○ ○ ○ ○ ○ ○
```

The most simple way of preparing a drilling program is to enter:

```
0: ..... BLOCK: 0000 (Memory first line)
1: A25 X..... Y..... BLOCK: 0001 ○ hole-1
2: A25 X..... Y..... BLOCK: 0002 ○ hole-2
3: A25 X..... Y..... BLOCK: 0003 ○ hole-3
4: A17 X..... Y..... BLOCK: 0004
```

In this example, the programming was started at the zero line of the memory (Block no. 0), that is, the first block

A25...

is then entered in block no. 1 as the zero line is considered a "park-position" for the computer, and hence is not available by the operator.

The machine will drill 3 holes on the given X,Y-positions, and then return to the starting point, waiting for the exchange of PCB's to be drilled next.

Obviously this program is not a "normal" program, among others because the program does not specify which tool-number to use, and because one also might expect a program identification-number in the very beginning of the program. The example only illustrates the most simple way of programming the machine.

The next example illustrates a more likely appearance of a program:

```
A03 X..... Y....1 (=Program no. "1").
A33 X....1 Y..100 (=Fetch Tool-1 which is Ø 1,00mm drill).
A25 X..... Y..... (=1st drilling position).
A25 X..... Y..... (=2nd drilling position).
A25 X..... Y..... (=3rd drilling position).
A17 X..... Y..... (=End of program/STOP).
```

In this example the machine will also drill 3 holes, just like in the example above, but now with a toolsize and a tool-location desired by the operator (i.e. code "33" = Ø1,00mm drill from tool pod no. 1) and with a program identification code in the beginning of the program (i.e. the code "03").

Of course it is now possible to extend the program with all sorts of codes: Several tool-changes, Routing-programs, STEP&REPEAT 1 & 2 (=subsequence/mainsequence resp.), SUPERREPEAT (including automatic alignment and tool separation of the circuit patterns) and various other refinements.

A26: ROUTING FEED.

The routing feed rate is normally taken from the machine's APSS-system, and is therefore automatically calculated by the computer if not programmed.

The A26 code works in a similar way as the A16 code and is also inserted after the tool change command A14, i.e. before the first routing command.

A26...

However the A26 code is used for feed (routing movement) in the X and Y directions, whereas the A16 code refers to the Z-axis movement only (i.e. Z-axis down direction).

The code is entered as:

A26 Y45

The computer will store the block as: A26 X00000 Y00045, which means routing feed is 45mm/sec. The routing feed value can be entered at any interval between 1mm/sec and 45mm/sec, which is the highest feedrate allowed by the software.

- If you enter higher values, the computer will write the following error-message in the display:

>>PARAMETER OUT OF RANGE<<

The A26 code is inserted like the A16 code:

EX.:

```
A14 X00018 Y00240 (TOOL-CHANGE T18=2.40MM ROUTER).
> A26 X00000 Y00022 (ROUTING FEED 22MM/SEC.)
A16 X00000 Y00016 (Z-AXIS IN-FEED 16MM/SEC.)
A01 X..... Y..... (ROUTE CIRCULAR HOLE IN X,Y POSITION)
A05 X00000 Y01750 (HOLE DIAMETER 17.50MM, ONE TURN ONLY)
> A26 X00000 Y00015 (ROUTING FEED 16MM/SEC.)
A04 X..... Y..... (GO TO X,Y POSITION)
A15 X..... Y..... (Z-AXIS DOWN + LINEAR ROUTING TO X,Y-POS.)
A15 X..... Y..... (ROUTE LINEAR TO X,Y-POSITION)
A17 X..... Y..... (END OF PROGRAM)
```

In the example above, the operator has decided to override the machine's APSS-values for in-feed and routing feed.

In addition the operator has used two different routing-feeds for circular routing and for linear routing.

The circular routing feed is set to 22mm/sec while the routing feed for the linear routing part is reduced to 15mm/sec. The z-axis in-feed value remains constant through the entire program, as this is defined by the operator to be 16mm/sec (code A16).

A27/A28/A29: SCALING.

A27 SCALING CENTER.

The A27 code is used as the center for scaling a PCB-pattern up or down. The A27 XY-value is used by the computer to calculate the magnification or reduction-factor. This 'scaling-factor' is calculated on the basis of the distance between the A27 code and two consecutive A28-codes.

The distance from the A27 to the first A28 is considered the present size of the pattern. The distance from the A27 to the second A28-code is considered to be the new, desired size of the pattern.

The two results are then calculated by the computer and the difference (i.e. the 'scaling factor') is then added or subtracted to each XY-position of the program under execution.

The A27 code is entered in the program as follows:

EX.: ORIGINAL PROGRAM:	MODIFIED PROGRAM (SCALED):
A03 X00000 Y00001	A03 X00000 Y00001 (program nbr)
A33 X00020 Y00115	A33 X00020 Y00115 (T20=Ø1.15mm drill)
A25 X..... Y.....	> A27 X01000 Y01000 (scaling center)
A25 X..... Y.....	> A28 X11000 Y10000 (scale from x,y)
etc.....	> A28 X12000 Y10000 (scale to X,Y)
A25 X..... Y.....	A25 X..... Y..... (#1 drill pos.)
A17 X..... Y.....	A25 X..... Y..... (#2 drill pos.)
	etc.....
	A25 X..... Y..... (last drill pos.)
	A17 X..... Y..... (end of program).

From the example above, the scaling is initiated in the beginning of the program, i.e. just before the first drill position, in order to make sure, that all the holes of the pattern are scaled correctly.

If the scaling codes A27-A28 were inserted at a later stage in the program, the scaling were zero for the first part of the program, i.e. before the A27/A28 codes.

A27...

EX.: The calculation of the scaling factor can be made like this:

X-factor:

$$\begin{aligned} & (A28\#2X - A27X) / (A28\#1X - A27X), \\ \Rightarrow & (X12000 - X1000) / (X11000 - X1000), \\ \Rightarrow & (X11000) / (X10000) \\ \Rightarrow & X1.1 \text{ (scaling factor is 1.10000 i.e. +10\%)} \\ & ===== \end{aligned}$$

Y-factor:

$$\begin{aligned} & (A28\#2Y - A27Y) / (A28\#1Y - A27Y), \\ \Rightarrow & (Y10000 - Y1000) / (Y10000 - Y1000), \\ \Rightarrow & (Y9000) / (Y9000) \\ \Rightarrow & Y1.0 \text{ (scaling factor is 1.0000 i.e. 0\%)} \\ & ===== \end{aligned}$$

In the example above is shown how the operator can calculate a certain scaling-percentage. In real life, the way of entering the scaling factor can be much simpler.

Let us assume, that you have started drilling a pattern, and you discover, that the drilled pattern does not fit exactly to the actual pattern printed on the PCB-surface (or the layout or film).

In this case, just move the PCB from the production-tooling system to the programming tooling system and view the PCB on the monitor.

You now find an end of the PCB where the holes seems to fit exactly to the pattern. In this end (preferably near a corner), you enter the A27-code in the right place of the memory, i.e. before the first drilling code A25. Now you look for a diamentrally opposite end of the PCB where the misfit between the actual drilled holes and the PCB-pattern seems to be worst.

Here you select a hole-position by the camera and centers on the drilled hole. In this position you enter the first code A28 (i.e. 'scale from') and finally you move the camera to the point where you want it to be drilled in order to match the pattern perfectly, and in this position you enter the second A28-code (i.e. 'scale to').

According to the examples given above, the three codes A27, A28, A28 must be entered in succession and just prior to the first drilling code (A25) or for routing, the first routing code (e.g. A04 or A01 etc).

A28 SCALING FROM/TO:

The A28 code **must** always be used in pairs and in combination with the A27-code for scaling a PCB-pattern.

The A28 XY-values are used by the computer to calculate the magnification or reduction-factor.

This 'scaling-factor' is calculated on the basis of the distance between the A27 code and two consecutive A28-codes.

The distance from the A27 to the first A28 is considered the present size of the pattern. The distance from the A27 to the second A28-code is considered to be the new, desired size of the pattern.

The two results are then calculated by the computer and the difference (i.e. the 'scaling factor') is then added or subtracted to each XY-position of the program under execution.

The set of A28 codes are entered in the program as follows:

EX.:	ORIGINAL PROGRAM:	MODIFIED PROGRAM (SCALED):
	A03 X00000 Y00001	A03 X00000 Y00001 (program nbr)
	A33 X00020 Y00115	A33 X00020 Y00115 (T20=Ø1.15 drill)
	A25 X..... Y.....	A27 X01000 Y01000 (scaling center)
	A25 X..... Y.....	> A28 X11000 Y10000 (scale from x,y)
	etc.....	> A28 X12000 Y10000 (scale to X,Y)
	A25 X..... Y.....	A25 X..... Y..... (#1 drill pos.)
	A17 X..... Y.....	A25 X..... Y..... (#2 drill pos.)
		etc.....
		A25 X..... Y..... (last drill pos.)
		A17 X..... Y..... (end of program).

Please refer to the paragraph above "A27 SCALING CENTER".

A29 CANCEL SCALING:

The A29 code is used to cancel the scaling factor initialized by code A27/A28.

In normal programs, the cancelling of the scaling factor is automatically made by the computer when it returns to the A03-code after having executed a program.

However, it is possible to use special programming techniques, that prevents the computer from going back to the A03-code, and in such cases, it is necessary to insert a A29 code in order to avoid repeated calculations of the scaling factors, because this would lead to continuously greater scaling factors for every time, the machine goes through the program.

NOTE:

Please refer to the paragraph "A24 CANCEL MIRROR IMAGE" for further information regarding the use of code A29, because the same rules applies for both the A24 and A29 codes.

A30/34 DISPLACEMENT:

Code A30 is always used in pairs of two A30 codes, and the function is, to offset or displace an existing pattern to a new position.

Such offset or displacement can also be obtained in a manual way, by using the machines special 'preset' function (see the paragraph "Keyboard Functions: "o SET"-key).

However, when it is desirable that an existing program is displaced a certain distance, and the displacement should take place automatically, it is possible to use the A30-code.

When using the A30-codes, all program-coordinates are displaced with the 'displacement-offset' which is calculated on the basis of the two A30 codes position.

The two A30 codes are entered the following way: —

EX.: ORIGINAL PROGRAM:

A03 X00000 Y00001
A33 X00020 Y00115
A25 X..... Y.....
A25 X..... Y.....
etc.....
A25 X..... Y.....
A17 X..... Y.....

MODIFIED PROGRAM (DISPLACED):

A03 X00000 Y00001 (program nbr)
A33 X00020 Y00115 (T20=Ø1.15 drill)
A30 X01000 Y01000 (displace fm x,y)
A30 X11000 Y10000 (displace to x,y)
A25 X..... Y..... (#1 drill pos.)
A25 X..... Y..... (#2 drill pos.)
A25 X..... Y..... (#3 drill pos.)
etc.....
A25 X..... Y..... (last drill pos.)
A17 X..... Y..... (end of program).

A30...

From the example above, the scaling is initiated in the beginning of the program, i.e. just before the first drill position, in order to make sure, that all the holes of the pattern are displaced evenly.

If the two A30 displacement codes were inserted at a later stage in the program, the first part of the program, i.e. before the A30 codes, would be drilled without displacement-offset.

This means that the first part would be drilled according to the X,Y-positions shown in the program, whereas the second part of the program (i.e. after the A30 codes), would be drilled on a displaced position.

The new position would be the sum of the original X,Y-position indicated in the program and the 'displacement-offset' as calculated by the computer.

EX.:

The calculation of the "displacement-offset" can be made in the following way:

X-coordinate offset:	Y-coordinate offset:
(A30#2 - A30#1)	(A30#2 - A30#1),
=> (X11000 - X1000)	=> (Y10000 - Y1000),
=> X-offset is X+10000,	=> Y-offset is Y+9000
=====	=====

In the example above is shown how the operator can calculate a certain displacement distance. In this example, to all X-coordinates will be added 10000 and 9000 to all Y-coordinates.

However, this addition will not be visible in the actual CNC-program, because the calculated offset will be added to the coordinate values during the program execution only.

A34: CANCEL DISPLACEMENT.

The A34 code cancels the displacement-offset created by the A30-codes.

The code might be used for the same reasons as described for code A24 and A29 (cancel-codes for mirror-imaging and scaling).

Please refer to the paragraphs regarding these codes for more details.

A31: Z DOWN-STROKE LIMIT.

- The A31 code is used whenever it is considered necessary to set a certain limit for the Z-axis depth, i.e. the lower end of the Z-axis stroke.

The A31 can be used for routing in various Z-levels, or for making special drill-cycles, such as 'pecking' for drilling high aspect-ratio holes (see below).

If code A31 is not programmed, the machine will always use the standard setting of the machine which is 0 (zero).

Zero means, that the Z down travel will stop, when the tool-point is approx. 1mm above the worktable surface (provided the tool-length is correctly adjusted).

The minimum adjustable increment for the A31 Z-level is 0.01mm, and the value is entered in 1/100mm in the Y-field.

It is NOT possible to enter negative values, because at the value 0 (zero), which is the standard setting of the machine, the Z-axis movement is very near it's mechanical end-stop.

The entering sequence is:

EX-2:

```
A03 X00000 Y12345 (PROGRAM NUMBER).
A33 X00001 Y00120 (TOOL-CHANGE T1=1.2MM DRILL).
>> A31 X00000 Y00150 (Z BOTTOM POSITION, Z=1.50MM ABOVE ZERO)
A25 X..... Y..... (DRILL HOLE IN POSITION X,Y)
A25 X..... Y..... (DO.)
A25 X..... Y..... (DO. last hole in T1-part)
A33 X00002 Y00150 (TOOL-CHANGE T2=1.5MM DRILL)
A25 X..... Y..... (DRILL HOLE IN POSITION X,Y)
A25 X..... Y..... (DO, last hole in T2-part)
A17 X..... Y..... (END OF PROGRAM)
```

In the example above, the operator has overridden the machines standard level-settings (=0) for TOOL#1.

For TOOL#2, the operator has not entered any A31-code after the tool-change command (A33), but the entered Z-level setting remains at the level specified in the first entered A31-code because the Z-level setting codes are 'globally' active in a program and remains active until erased from the program.

The reason for this is security, and because a level change is normally used when there is a general change in the height of a workpiece (e.g. the exit board thickness has been increased from 2mm to say 3.6mm). In such a case, it would also be necessary to increase the Z-top stroke limit, i.e. code A36 (refer to the description of this code below).

It may also be desirable to insert more values of the A31 code, especially in 'canned pecking cycles', such as drill cycles for drilling very deep holes with small diameter drills. This is also called "drilling with high ASPECT-RATIO".

If you assume that holes of 0.3mmØ must be drilled in a stack of 3 PCB's, the drill must penetrate 3x1.6mm plus the entrance board of 0.5mm as well as 0.3mm down in the exit-board, making a total travel of min. 5.6mm.

The aspect-ratio is the "Z-travel / drill-diameter" , i.e. 5.6mm / 0.3mm = 18.66:1, or simply put: "The aspect ratio is 19".

EX.:

```
A03 X00000 Y00123
A33 X00017 Y00025
A11 X00000 Y00000 (Step&Repeat begins:)
A31 X00000 Y00390 (Drill depth 3.9mm above 0)
A25 X01000 Y01000 (First hole in the program)
A31 X00000 Y00230 (Drill depth 2.3mm above 0)
A25 X01000 Y01000 (First hole...)
A31 X00000 Y00070 (Drill depth 0.7mm above 0)
A25 X01000 Y01000 (First hole...)
A12 X..... Y..... (Repeat the 'pecking' at the 2nd hole)
A12 X..... Y..... (Do., at 3rd hole)
etc..... (etc..)
A12 X..... Y..... (Do., at last hole in the program)
A17 X00000 Y00000 (End of program)
```

As can be seen from the example above, only the first hole must be programmed with the Z-axis limitations.

All the following drill positions are 'copied' from the first hole by using the Step & Repeat function (A11/A12..).

In this way, the first hole serves just like a 'sub-program' that can be copied for the remaining holes of a program.

A33: DRILL TOOL CHANGE.

The A33 code is the tool-change command for drills.

A33 is entered in the following way:

"A33 X11 Y200".

A33 means "tool-change",
X indicates the tool-pod number and
Y indicates the tool-size (i.e. diameter) in 1/100mm.

The computer use the information of drill-size for the automatic calculation of the optimum speed and feed-rate for a certain drill size (APSS). Also, the tool-number information is used by the computer to localize ~~the~~ desired tool-pod.

The A33 code only applies to tool-changes for DRILLS, not for routers. The reason for distinguishing between drills and routers is, that the computer needs the information to calculate the APSS-values as well as the router compensation etc.

The actual machine operation is, that the machine will first lay down the present tool (i.e. the tool which is in the spindle) into the tool-pod where the present tool was originally picked up from. Then it will travel to tool pod nr. 11 and pick up the tool, and assume that it is a drill, with a diameter of 2.00mm.

Because the computer actively uses the tool diameter entered in the Y-field when calculating the spindle speed and Z axis down feed *it is very important that the operator enters the correct value, i.e the right diameter of the tool.*

If not, this may lead to inaccurate hole-diameters and miscalculation of speed and feed rates, which again might lead to reduced hole quality due to epoxy-smearing effects or reduced tool-life time.

The tool-change command A33 is normally the first code in a drilling program, and if the program contains several tool-sizes, a A33-code must be entered for each new size.

Please refer to the description of code A14 for additional information. The A33 code works similiary to the A14 code.

A36: Z UP-STROKE LIMIT.

The A36 code is used whenever it is considered necessary to set a certain limit for the upper Z-axis level, i.e. the top level of the Z-axis stroke.

The A36 is normally used to adjust the general upper level of the Z-stroke in a program, where this may be relevant.

Assume that you in some cases use a higher backing-board of say 6mm instead of the normal 2mm thickness. In such a case it will be necessary to increase the Z-lift between the drill strokes from 8 to 12mm in order to ensure, that the drill-bit is retracted from the stack before travelling to the next drill position.

If the Z-axis height were unchanged, the drill bit would break.

Another example would be, that you should drill some test-jigs, which were fitted with a high frame, making the actual board surface to be e.g. 20mm above the machine's workplane. In this case you would have to adjust both the A36 and the A31 codes in order to move your entire "work-plane" up to higher level.

If code A36 is not programmed, the machine will always use the standard setting of the machine which is 8.00mm. This means, that the upper Z-axis stroke level, is where the tool-point is approx. 1mm+8mm=9mm above the worktable surface (provided the tool-length is correctly adjusted).

The minimum adjustable increment for the A36 Z-level is 0.01mm, and the value is entered in 1/100mm in the Y-field.

It is NOT possible to enter negative values, because at the value 0 (zero), the Z-axis movement is very near it's lower mechanical end-stop.

The maximum value for code A36 is Y4500 (i.e. 45mm lift), because this is very close to the mechanical top end-stop of the Z-axis movement.

The entering sequence is:

EX-2:

```
A03 X00000 Y12345 (PROGRAM NUMBER).
A33 X00001 Y00120 (TOOL-CHANGE T1=1.2MM DRILL).
>> A36 X00000 Y01250 (Z TOP POSITION, Z=12.50MM ABOVE ZERO)
A25 X..... Y..... (DRILL HOLE IN POSITION X,Y)
A25 X..... Y..... (DO.)
A25 X..... Y..... (DO. last hole in T1-part)
A33 X00002 Y00150 (TOOL-CHANGE T2=1.5MM DRILL)
A25 X..... Y..... (DRILL HOLE IN POSITION X,Y)
A25 X..... Y..... (DO, last hole in T2-part)
A17 X..... Y..... (END OF PROGRAM)
```

A36...

In the example above, the operator has overridden the machines standard level-settings (=8.0mm) for TOOL#1.

For TOOL#2, the operator has not entered any A36-code after the tool-change command (A33), but the entered Z-level setting remains at the level specified in the first entered A36-code because the Z-level setting codes are 'globally' active in a program and remains active until erased from the program.

The reason for this is security, and because a level change is normally used when there is a general change in the height of a workpiece (e.g. the exit board thickness has been increased from 2mm to say 5mm). In such a case, it would also be necessary to increase the Z-top stroke limit, i.e. code A36.

NOTE: The A36 code is frequently used in combination with the A31 code (please refer to the description of this code for further information).

A38/A39/A40: ROTATION.

A38 ROTATION CENTER.

The A38 code is used as the center for rotating a PCB-pattern at any angle between 0 and 360 degrees.

The A38 XY-value is considered to be the rotation center by the computer.

The rotation angle is calculated on the basis of the distance between a fictive line going from the A38 XY-position to the first A39 XY-position compared to a fictive line from the A38-position to the second A39-position.

The distance from the A38 to the first A39 is considered the present place of the pattern. The distance from the A38 to the second A39-code is considered to be the new, desired rotation of the pattern.

The angle between the two lines are then calculated by the computer and the difference (i.e. the rotation angle) is finally calculated for each XY-position of the program under execution.

A38...

The A38 code is entered in the program as follows:

EX.:	ORIGINAL PROGRAM:	MODIFIED PROGRAM (ROTATED):	
	A03 X00000 Y00001	A03 X00000 Y00001	(program nbr)
	A33 X00020 Y00115	A33 X00020 Y00115	(T20=Ø1.15 drill)
	A25 X..... Y.....	> A38 X01000 Y01000	(rotation center)
	A25 X..... Y.....	> A39 X11000 Y10000	(rotate from x,y)
	etc.....	> A39 X11050 Y09950	(rotate to X,Y)
	A25 X..... Y.....	A25 X..... Y.....	(#1 drill pos.)
	A17 X..... Y.....	A25 X..... Y.....	(#2 drill pos.)
		etc.....	
		A25 X..... Y.....	(last drill pos.)
		A17 X..... Y.....	(end of program).

From the example above, the rotation is initiated in the beginning of the program, i.e. just before the first drill position, in order to make sure, that all the holes of the pattern are rotated.

If the rotation codes A38-A39 were inserted at a later stage in the program, the rotation would be zero for the first part of the program, i.e. before the A38/A39 codes.

In real life, the way of entering the rotation angle can be made like this:

Let us assume, that you have started drilling a pattern, and you discover, that the drilled pattern does not fit exactly to the actual pattern printed on the PCB-surface (or the layout or film).

In this case, just move the PCB from the production-tooling system to the programming tooling system and view the PCB on the monitor.

You now find an end of the PCB where the holes seems to fit exactly to the pattern. In this end (preferably near a corner), you enter the A38-code in the right place of the memory, i.e. before the first drilling code A25.

Now you look for a diamentrally opposite end of the PCB where the misfit between the actual drilled holes and the PCB-pattern seems to be worst.

Here you select a hole-position by the camera and centers on a drilled hole. In this position you enter the first code A39 (i.e. 'rotate from') and finally you move the camera to the point where you want it to be drilled in order to match the pattern perfectly, and in this position you enter the second A39-code (i.e. 'rotate to').

A38...

According to the examples given above, the three codes A38, A39, A39 must be entered in succession and just prior to the first drilling code (A25) or routing code (e.g. A04 or A01), in order to ensure that all parts of the program are evenly rotated.

A39 ROTATION FROM/TO:

The A39 code must always be used in pairs and in combination with the A38-code for rotating a PCB-pattern.

The A39 XY-values are used by the computer to calculate the rotation-factor (angle). The rotation angle is calculated on the basis of the distance between the first A38 code and two consecutive A39-codes.

The angle from the A38 to the first A39 is considered the present angle of the pattern. The distance from the A38 to the second A39-code is considered to be the new, desired rotation of the pattern.

The two results are then calculated by the computer and the difference (i.e. the rotation angle) is then added or subtracted to each XY-position of the program under execution.

The set of A39 codes are entered in the program as shown below:

EX.:	ORIGINAL PROGRAM:		MODIFIED PROGRAM (ROTATED):	
	A03 X00000 Y00001		A03 X00000 Y00001	(program nbr)
	A33 X00020 Y00115		A33 X00020 Y00115	(T20=Ø1.15 drill)
	A25 X..... Y.....		A38 X00000 Y00000	(rotation center)
	A25 X..... Y.....	>	A39 X00000 Y10000	(rotate from x,y)
	etc.....	>	A39 X10000 Y10000	(rotate to X,Y)
	A25 X..... Y.....		A25 X..... Y.....	(#1 drill pos.)
	A17 X..... Y.....		A25 X..... Y.....	(#2 drill pos.)
			etc.....	
			A25 X..... Y.....	(last drill pos.)
			A17 X..... Y.....	(end of program).

In the example above, the machine will perform a 45 degree rotation clockwise.
Please refer to the description of code A38 above.

A40 CANCEL ROTATION:

The A40 code is used to cancel the rotation initialized by codes A38/A39.

In normal programs, the cancelling of the rotation codes is automatically made by the computer when it returns to the A03-code after having executed a program.

However, it is possible to use special programming techniques, that prevents the computer from going back to the A03-code, and in such cases, it is necessary to insert a A40 code in order to avoid repeated calculations of the scaling factors, because this would lead to continuously greater scaling factors for every time, ~~the~~ machine goes through the program.

NOTE:

Please refer to the paragraph "A24 CANCEL MIRROR IMAGE" for further information regarding the use of code A40, because the same rules applies for the A24 and A40-codes.

A41: IC-PIN SPACING.

The A41-code is always used in combination with the automatic SIL-pattern code (A48) or DIL-pattern code (A45).

The A41-code determines the spacing between the pins of a SIL- or DIL-pattern in 1/100mm.

NOTE: "SIL" means "Single In-Line", i.e. a line of holes evenly distributed along one straight line. The SIL-pattern is most commonly used for programming single rows or columns of hole-patterns or when programming in-line holes on an arbitrary (rotated) angle compared to the machine's axes.

"DIL" means "Dual In-Line", i.e. two rows of holes evenly distributed along two parallel, straight lines. The DIL-pattern is most commonly used for programming dual rows or columns of hole-patterns, typically IC-patterns or patterns for headers, edge-connectors etc.

A41...

The entering is:

A41 X00000 Y00250 (pattern pin-spacing 2.50mm)

and the A41 code should be entered before the A45 or A48 codes. The entering sequence for making two 16-pin IC-patterns with a grid spacing of 1.27mm, is:

EX.1:

```
A03 X00000 Y00125 (PROGRAM NUMBER #125).
A33 X00001 Y00020 (TOOL-CHANGE: T1=0.2MM DRILL).
A25 X..... Y..... (DRILL HOLE IN POSITION X,Y).
A25 X..... Y..... (DO.).
A25 X..... Y..... (DO.).
>> A41 X00000 Y00127 (IC PIN SPACING 1.27mm).
A01 X..... Y..... (FIRST IC-PIN POSITION, pin-1, #1st IC).
A45 X..... Y..... (LAST IC-PIN POS., e.g. pin-9, #1st IC).
A45 X..... Y..... (LAST IC-PIN POS., e.g. pin-8, #2nd IC).
A45 X..... Y..... (LAST IC-PIN POS., e.g. pin-9, #2nd IC).
A17 X..... Y..... (END OF PROGRAM)
```

In the example above, the operator has overridden the machines standard grid spacing of 2.54mm and made a 1.27mm spacing instead.

If code A41 were not programmed at all, the machine would use a grid-spacing of 2.54mm (0.1") as default value. Also, if the A41 were programmed as A41 X0 Y0, the computer would use the grid spacing default value of 2.54mm, simply because a grid spacing of 0 does not give any reasonable meaning.

NOTE:

Once programmed in a program sequence, the A41 code works globally, i.e. remains active until it is deleted or changed to another value. In other words, A41 needs only to be programmed once in a program in order to change the grid value.

In the example below (EX-2), the operator has programmed an A41 X00000 Y00000 which is the same as not entering the A41-code at all or program it as A41 X00000 Y00254, i.e. the standard setting of the machine:

EX.2:

```
A41 X00000 Y00000 (A45#2)
A01 X..... Y..... (#1)      o o o o o o o
A45 X..... Y..... (#2)      o o o o o o o
(A01#1)
```

A41...

In example #3 below, the operator has programmed a DIL-pattern with a 1.27mm spacing:

EX.3:

```
A41 X00000 Y00127 (A45#2)
A01 X..... Y..... (#1) oooooooooooooo
A45 X..... Y..... (#2)  oooooooooooooo
                        (A01#1)
```

Please also refer to the section discussing the "A01"-code of this manual.

A42: SIL NUMBER.

The A42-code is always used in combination with the automatic SIL-pattern code (A48) and/or the DIL-pattern code A45.

The A42-code determines the number of holes between two end-positions of a SIL-pattern. DIL-patterns can be programmed by using the double number of SIL-patterns.

The entering is:

```
A42 X00000 Y00064 (number of pattern pins, = 64 pins)
```

The A42-code must always be entered in the end of an IC-pattern group, i.e. after the last A45 or A48 in that particular group.

NOTE: A42 is only valid in the IC-pattern group to which it is attached.

Code A42 is a so-called 'one-shot' command which does not work globally in a program (opposite to the A41-code for pin spacing).

The A42 code may also be used together with code A45 (IC).

A42...

The entering sequence for code A42 is:

EX.1:

```
A03 X00000 Y00125 (PROGRAM NUMBER #125).
A33 X00001 Y00020 (TOOL-CHANGE: T1=0.2MM DRILL).
A25 X..... Y..... (DRILL HOLE IN POSITION X,Y).
A25 X..... Y..... (DO.).
A25 X..... Y..... (DO.).
A01 X..... Y..... (FIRST SIL-POSITION, typically pin-1).
A48 X..... Y..... (LAST SIL-PIN POS., = pin 32-position).
A48 X..... Y..... (LAST SIL-PIN POS., = pin 32-position).
>> A42 X00000 Y00032 (IC PIN NUMBER: 32 PINS).
A17 X..... Y..... (END OF PROGRAM)
```

If code A42 were omitted or programmed as A42 X0Y0, the machine would use the standard grid-spacing of 2.54mm (0.1" or 100 mil) as default value and make a line of holes with this standard spacing.

NOTE: The maximum number of holes in a row is 32.767 holes.

A43/A44: POSITION PRESET.

Code A43 presets the position counter of the machine to a predetermined X,Y-value.

The A43 code can be used for introducing a floating zero (offset) of a program.

There are several ways of making a floating zero or position offset of a program. This can be achieved in a 'manual' way, by showing a certain hole or known position on the video-system, enter the desired X,Y-value and then press the " ⌀ SET"-key ('preset'-key).

It is also possible to use the A43 code for making an 'automatic' preset of a desired X,Y-value, e.g. X0,Y0 or any other arbitrary X,Y value desired by the operator.

Let us assume, that the programmer knows the coordinate-value of a fiducial mark of a specific PCB-pattern (e.g. the X0,Y0 datum-point of a CAD-program).

It is possible to enter a code A43 with the desired X,Y-position of the fiducial mark, and the machine will go to the A43XY-position where the position-counters of the machine will be preset to the desired value, and the program executed with the desired offset.

A43...

It is important to remember to cancel the offset made by the A43 code. Otherwise, the machine will be adding the desired offset every time it goes through the program.

This is normally not desirable, but in some instances it may be used actively during the programming as a special form of repetitive programming sequence.

The cancelling code is A44. Please refer to the description of this code below.

The A43 code is entered in the program as shown below:

EX.: ORIGINAL PROGRAM:	MODIFIED PROGRAM (offset):
A03 X00000 Y00001	A03 X00000 Y00001 (program number)
A33 X00020 Y00115	A33 X00004 Y00085 (T4=Ø0.85 drill)
A25 X..... Y.....	A04 X04000 Y01000 (go to X4000Y1000)
A25 X..... Y.....	> A43 X00000 Y00000 (preset to X0,Y0)
etc.....	A25 X..... Y..... (#1 drill pos.)
A25 X..... Y.....	A25 X..... Y..... (#2 drill pos.)
A17 X..... Y.....	A25 X..... Y..... (#3 drill pos.)
	etc.....
	A25 X..... Y..... (last drill pos.)
	> A44 X..... Y..... (cancel A43-preset)
	A17 X..... Y..... (end of program).

NOTE:

The A43-code is a 'globally' active code, i.e. once it is initiated in a program, it remains active until cancelled by the A44-code (see below).

A44: CANCEL PRESET.

The A44-code is used to cancel the position preset introduced by the A43-code.

The A44 should be inserted after the last drilling or routing position, i.e. at a place in the memory where the computer passes through before returning to the beginning of the program (to the A03-code).

A45: SIL/DIL PROGRAMMING (AXIS-PARALLEL).

The A45-code is normally used for creating an IC-pattern drill-sequence (DIL-pattern). However, it can also be used for making a SIL-pattern, but code A48 is normally used for creating SIL-patterns.

The A45 code must always be used in combination with the A01-code, which indicates the first hole position of an IC-pattern, whereas the A45 code indicates the diametrically opposite hole position of an IC-pattern.

EX.: If we assume the pattern of a 16-pin IC, code A01 must be placed at pin #1 and code A45 at pin #9.

The A45 may also be used in combination with the code for IC-pin spacing (A41) if non-standard grid-distances are desired. If no A41-code is used in combination with the A45 code, the computer assumes a standard grid spacing of 2.54mm (0.1") or 100 mil.

NOTE: "SIL" means "Single In-Line", i.e. a line of holes evenly distributed along a straight line. The SIL-pattern is most commonly used for programming single rows or columns of hole-patterns or when programming in-line holes which is parallel with the machine's axes.

"DIL" means "Dual In-Line", i.e. two rows of holes evenly distributed along two parallel, straight lines. The DIL-pattern is most commonly used for programming dual rows or columns of hole-patterns, typically IC-patterns or patterns for headers, edge-connectors etc., which are parallel to the machine's axes. If programming of non-parallel rows of holes are desired, please use the A48-code for this purpose (refer to the A48 paragraph below).

The entering is:

A01x..... Y..... (X,Y-position of first hole in IC-pattern)
A45X..... Y..... (X,Y-position of diametrically opposite hole in the IC-pattern).

The A01-code is used for indicating the first hole XY-position of the automatic DIL/SIL-pattern generating codes A45 and A48.

Always consider the A01 X,Y-position as pin no. 1 of any Dual-In-Line (DIL) pattern.

A45...

When programming DIL-IC-patterns, it is mandatory, that the code A01 indicates pin no. 1 according to the standard rules of IC-pinout. Otherwise, the IC may be drilled in the opposite direction.

When using the code A01 in combination with the A41 ("IC-PIN SPACING") code, the A01 X,Y-position must be located exactly at the right spot according to the A41-value.

If not, the computer may automatically adjust the start-point (X,Y-value) of the A01-code slightly, because the A45 or A48 end-point position is always fixed!

EX.1:

```

(A45#2)
A01 X,Y (#1)      o o o o o o o
A45 X,Y (#2)      o o o o o o o
(A01#1)

```

correct!

If pos. #1 and pos. #2 accidentally were programmed in the opposite corners of the IC, the actual drilled result would be:

```

(A01#1) o          o
          o          o
          o          o
(A45#2)

```

incorrect!

If just a SIL-pattern is desired, it can be made by placing the end-point code A45 in-line with the A01 position.

If the Y-coordinates for the two codes A01 and A45 are identical, the machine will only produce one row of holes, i.e. a SIL-pattern. Please refer to the example below:

EX.:

```

(A01#1)          (A45#2)
o o o o o o o o

```

In the examples above, a standard 100 mil spacing is assumed.

A46: Z-RETRACTION FEED.

The A46 code determines the Z-axis return feed-rate i.e. the 'speed' by which the tool retracts from the workpiece.

Normally, the machine calculates the optimum feed-rate automatically, but under special circumstances, it is necessary to use other feed-rates, especially when drilling/routing in special, non-PCB, materials like acrylic etc.

The A46-code is inserted after the tool-change command (i.e. A14 or A33) prior to the first position code (e.g. A25, A04, A15 etc) where the desired retraction-feed should apply.

It is possible to insert the A46 code at any place in a drilling or routing program, but the desired A46 feed-rate will apply only from the following block and onwards through the program until the next tool-change command is executed. After the tool change, the computer's automatic feed/speed selection system (APSS) will again be in action.

The A46-code only determines the return-feed, i.e. the up-feed direction of the Z-axis. The down-feed is determined by a separate code, called A16 (see this).

EX:

```
A33 X00007 Y00200 (T7=Ø2.00mm DRILL)
A16 X00000 Y00016 (Z-AXIS IN-FEED 16MM/SEC.)
>> A46 X00000 Y00125 (Z-AXIS RETURN-FEED 125MM/SEC.)
A25 X..... Y..... (TRAVEL TO X,Y and drill a hole)
A25 X..... Y..... (do. 2nd hole)
A25 X..... Y..... (do. 3rd hole)
A25 X..... Y..... (do. 4th hole)
A25 X..... Y..... (do. 5th hole)
A17 X..... Y..... (END of program)
```

A47: DWELL TIMER.

The dwell timer function is used to make a programmable delay (or time controlled 'break') in the machine operation.

The A47 dwell time is entered in the Y-field in milliseconds, whereas the X-field is inactive.

The entering is shown below:

EX:

```
A33 X00007 Y00200 (T7=Ø2.00mm DRILL)
A01 X..... Y..... (travel to X,Y and route circular hole)
A05 X..... Y06500 (hole diameter Ø65mm)
>> A47 X00000 Y05000 (delay: 5000mS = 5 seconds delay)
A01 X..... Y..... (travel to next xy-position and route)
A05 X00000 Y06500 (hole diameter Ø65mm)
A17 X..... Y..... (END of program)
```

The example above shows how to make a 5 seconds break in the machine operation (e.g. in order to view the progress of the circular routing of a Ø65mm hole).

A48: SIL PROGRAMMING WITH ROTATION.

The A48-code is normally used for creating an SIL-pattern drill-sequence.

However, it can also be used for making DIL-patterns, by using the double number of SIL-patterns.

The A48 code must always be used in combination with the A01-code, which indicates the first hole position of a SIL-pattern, whereas the A48 code indicates the end-position of a SIL-pattern.

EX.: *If we assume the pattern of a 40-pin SIL, code A01 must be placed at pin #1 and code A48 at pin #40.*

The A48 may also be used in combination with the code for IC-pin spacing (A41) if non-standard grid-distances are desired. If no A41-code is used in combination with the

A48...

A48 code, the computer assumes a standard grid spacing of 2.54mm (0.1") or 100 mil.

NOTE: "SIL" means "Single In-Line", i.e. a line of holes evenly distributed along a straight line. The SIL-pattern is most commonly used for programming single rows or columns of hole-patterns or when programming in-line holes in an arbitrary (rotated) angle compared to the machine's axes.

"DIL" means "Dual In-Line", i.e. two rows of holes evenly distributed along two parallel, straight lines.

The DIL-pattern is most commonly used for programming dual rows or columns of hole-patterns, typically IC-patterns or patterns for headers, edge-connectors etc. Programming of DIL-patterns are possible with the A48 code, by doubling the numbers of SIL-patterns.

However, code A48 should only be used if the desired pattern is non axis parallel (i.e. rotated compared to the machine's axes).

The entering is:

A01 x..... y..... (X,Y-position of first hole in IC-pattern)
A48 X..... Y..... (X,Y-position of the last hole position of the SIL-pattern).

The A01-code is used for indicating the first hole XY-position of the automatic SIL-pattern generating code A48. Always consider the A01 X,Y-position as pin no. 1 of any Single-In-Line (SIL) pattern.

When using the code A01 in combination with the A41 ("IC-PIN SPACING") code, the A01 X,Y-position must be located exactly at the right spot according to the A41-value.

If not, the computer may automatically adjust the start-point (X,Y-value) of the A01-code slightly, because the A48 end-point position is always fixed!

EX.1:

A01 X,Y (#1)
A48 X,Y (#2)

(A01#1) (A45#2)
o o o o o o o o

In the examples above, a standard 100 mil spacing is assumed.

A48...

A typical entering sequence for a 3 pcs of 32 pin SIL-patterns with a hole grid spacing of 2.00mm is:

EX.:

```
A03 X00000 Y00125 (PROGRAM NUMBER #125).
A33 X00001 Y00020 (TOOL-CHANGE: T1=0.2MM DRILL).
A25 X..... Y..... (DRILL HOLE IN POSITION X,Y).
A25 X..... Y..... (DO.).
A25 X..... Y..... (DO.).
A41 X00000 Y00200 (PIN SPACING 2.00mm).
A01 X..... Y..... (FIRST POSITION, pin-1 OF #1 SIL-PATTERN).
> A48 X..... Y..... (LAST POSITION, pin-32 OF #1 SIL-PATTERN).
> A48 X..... Y..... (LAST POSITION OF #2 SIL-PATTERN)
> A48 X..... Y..... (LAST POSITION OF #3 SIL-PATTERN)
A17 X..... Y..... (END OF PROGRAM).
```

The program includes 3 identical SIL-patterns placed at the positions determined by the X,Y-values of the A48-codes.

In the example above, the operator has overridden the machines standard grid spacing of 2.54mm and made a 2.00mm spacing instead.

NOTE:

Once programmed in a program sequence, the A41 code works globally, i.e. remains active until it is deleted or changed to another value. In other words, A41 needs only to be programmed once in a program in order to change the grid value.

Example of a rotated SIL-pattern using the A48 code:

EX.:

```
A41 X00000 Y00200 (=2mm grid).
A01 X..... Y..... (pin-1/SIL#1).
A48 X..... Y..... (pin-4/SIL#1).
A48 X..... Y..... (pin-4/SIL#2).
A48 X..... Y..... (pin-4/SIL#3).
```

SIL#1: SIL#2: SIL#3:

```
(A01) 0                    0                    0
      0                    0                    0
         0                    0                    0
             0                    0                    0
                 (A48#1)      (A48#2)      (A48#3)
```

A48...

Please also refer to the section discussing the "A01"-code of this manual.

A49: TOOL LIFE COUNTER.

The A49 code is used to limit the tool-life to a value determined by the operator.

The tool life is commonly limited to between 2500 and 5000 hits for PTH-technology, where the hole quality is a critical factor. As every hit normally means 3 drilled holes in a 3-stack PCB package, the maximum expectable tool life is approx. between 7500-15000 holes.

The A49 code must be entered just after the tool-change command, code A33.

The X-field specifies the remaining life time (i.e. the down-count) while the Y-field specifies the maximum tool hit specified by the programmer.

A typical entering sequence for the A49 hit counter is:

EX.:

```
A03 X00000 Y00125 (PROGRAM NUMBER #125).
A33 X00001 Y00085 (TOOL-CHANGE: T1=0.85MM DRILL).
>> A49 X00000 Y03000 (MAX TOOL LIFE 3000 HITS).
A25 X..... Y..... (DRILL HOLE IN POSITION X,Y).
A25 X..... Y..... (DO.).
etc.....
A25 X..... Y..... (DO., last hole).
A33 X00002 Y00115 (TOOL-CHANGE: T2=1.15MM DRILL).
>> A49 X00000 Y03500 (MAX TOOL LIFE 3500 HITS).
A25 X..... Y..... (DRILL HOLE IN POSITION X,Y).
A25 X..... Y..... (DO.).
etc.....
A25 X..... Y..... (DO., last hole).
A17 X..... Y..... (END OF PROGRAM).
```

In the example above, we assume that T1 drills 950 hits, and T2 drills 625 hits.

A49...

After the program has been executed one time (i.e. drilling one stack with T1 and T2), the memory would look like this:

EX.2:

```
A03 X00000 Y00125 (PROGRAM NUMBER #125).
A33 X00001 Y00085 (TOOL-CHANGE: T1=0.85MM DRILL).
>> A49 X02050 Y03000 (MAX TOOL LIFE 3000 HITS).
A25 X..... Y..... (DRILL HOLE IN POSITION X,Y).
A25 X..... Y..... (DO.).
etc.....
A25 X..... Y..... (DO., last hole).
A33 X00002 Y00115 (TOOL-CHANGE: T2=1.15MM DRILL).
>> A49 X02875 Y03500 (MAX TOOL LIFE 3500 HITS).
A25 X..... Y..... (DRILL HOLE IN POSITION X,Y).
A25 X..... Y..... (DO.).
etc.....
A25 X..... Y..... (DO., last hole).
A17 X..... Y..... (END OF PROGRAM).
```

In the EX-2 example above, the remaining tool-life is only 2050 for T1 and 2875 for T2 just prior to the 2nd execution.

For each program execution ("run"), the X-value of the two A49 life time counters would be reduced with 950 and 625 respectively.

When any of the counters becomes 0, the machine will stop and download the used tool in the corresponding tool-pod and write the following message on the display:

>>RENEW TOOL 01<<

When the above message appears on the display, the operator opens the shield, removes the worn-out drill bit and insert a new one in the same tool-pod (= T1 in this example).

When the operator closes the shield again, the machine will automatically pick-up the tool and continue the program-execution.

A49...

If the machine stops and ask for a new tool while being very near the end of the job, the operator may desire to override the machine's request for renewing the tool.

This can be done in either of two ways:

- 1) *The operator can just open and close the shield swiftly, and the machine will believe that a new tool is inserted, In this case the residual life time field of code A49 will show X3000, i.e. pretending that 3000 holes could be drilled with the worn out drill.*

This might lead to expensive error in the production and hence it is recommended instead to change the residual life time counter from 0 to e.g. 300 hits (provided the operator estimates that less than 300 hits are sufficient for drilling the remaining number of holes):

- 2) *The operator just inserts an A49-code in the memory at the position where the machine stopped and requested a new tool. The A49 code should only be entered as A49Y300.*

A50: APSS ENABLE.

The A50 code is used if the programmer wants to override the standard speed- and feed-settings of the machine.

The machine utilizes the tool-size informations of the A33- and A14-codes to calculate the optimum values for rotation-speed (A06), in-feed (A16), routing-feed (A26) and return-feed (A46).

Basically, the programmer can choose between 3 different speed-settings of the machine.

A50 Y1: *Lowest productivity, but very high quality, typically used for drilling multilayer PCB's.*

A50 Y2: *Medium productivity, still maintaining high quality, suitable for all PTH PCB's. This is the standard setting of the machine (i.e. default value).*

A50 Y3: *High speed performance for high volume, uncritical single-sided PCB's.*

A50...

If the A50-code is omitted, the machine always select the APSS-table #2 as default setting.

If the programmer desires to select a high speed setting, an A50XY3 block (i.e. APSS table #3) is entered in the program according to the example below:

EX.:

```
A03 X00000 Y01125 (PROGRAM NUMBER #1125).
A33 X00001 Y00085 (TOOL-CHANGE: T1=0.85MM DRILL).
>> A50 X00000 Y00003 (APSS table #3= high speed setting).
A25 X..... Y..... (DRILL HOLE IN POSITION X,Y).
A25 X..... Y..... (DO.).
etc.....
A25 X..... Y..... (DO., last hole).
A33 X00002 Y00115 (TOOL-CHANGE: T2=1.15MM DRILL).
A25 X..... Y..... (DRILL HOLE IN POSITION X,Y).
A25 X..... Y..... (DO.).
etc.....
A25 X..... Y..... (DO., last hole).
A17 X..... Y..... (END OF PROGRAM).
```

The A50 code is 'globally' active, which means that once the A50 code is inserted in a program, it remains active until it is deleted or changed to another value.

NOTE: *The A50 code must be entered after the tool-change command/prior to the first drill or route command.*

For further informations regarding feed/speed and chiploads, please refer to the chapter "APSS": "Automatic Parameter Selection System".

A51: AUTOMATIC CORNER ROUND-OFF (SINGLE CORNER):

This code involves automatic corner round-off of a single corner in a routing-pattern.

The desired rounding-radius is entered in the Y-field, and in 1/100mm. The X-field is not used by the computer in this case.

IMPORTANT:

The A51-code utilizes the router radius entered in code A55 as the basis for the calculations.

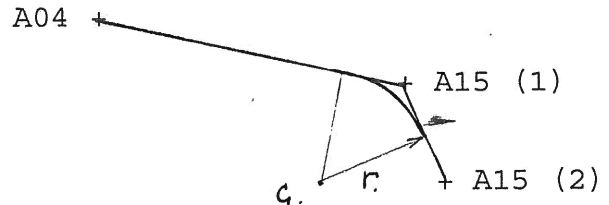
A51...

If code A61/A62 is used instead of code A55 for the aut. router radius compensation, code A51 (and A52) will use the router-size entered in code A14's "Y"-field, as the basis for the calculations.

Code A51 is always placed in the program just **AFTER** the A15-code which should be rounded-off:

EX.:

```
A04 X..... Y.....  
A15 X..... Y..... (1)  
A51 X00000 Y02500  
A15 X..... Y..... (2)  
etc.
```



If you have several corners on a pattern that need to be rounded-off, one must use the A51-code for each rounding. Therefore the code is mainly used when the pattern has several different rounding-radii.

Otherwise, one should use the A52-code to round-off all the corners with the same radius. Please refer to the next paragraph (A52).

A52: AUT. CORNER ROUNDING, GLOBAL.

Routing with automatic rounding-off of **ALL** corners in a pattern. The "Y"-field of the display indicates the desired corner-radius in 1/100mm, while the X-field is not used.

The A52-function remains active until it is cancelled with a code A52 Y00000!

It is also possible to have all corners rounded, except one or two.

In this case, simply put a code A51 just after the corner(s) to be rounded with another radius.

Then all corners will be rounded-off with the radius-value indicated in code A52 except the corner-positions followed by the A51-code.

Hence, the A51 code overrides the globally active code A52.

A52...

Code A52 is using the same basis of calculation as code A51 (see this).

If ALL corners need to be rounded-off, code A52 must be placed **BEFORE** the first routing command (normally A04).

If the rounding function is only desired from a given point in a pattern, code A52 must be entered just **AFTER** the first position to be rounded (i.e. the same guidelines as for code A51).

If all corners but one must be rounded-off with a specific corner-radius, this is easily carried out by entering a code A51 just after the corner with the different corner-radius. All following corners will continue to have the same rounding-radius as entered in code A52.

In this way, it is possible to make one or more "exceptions" to the rounding-radius specified in code A52.

IMPORTANT: CODE A52 REMAINS ACTIVE UNTILL IT IS CANCELLED WITH A "A52X0Y0" COMMAND!

EX.:

```

A14 X00012 Y00240 (T12=Ø2.4R)
A55 X00120 Y00000 (1.2 LEFT COMP)      A15 (1)      A15 (2)
A52 X00000 Y00500 (r=5mm)
A04 X..... Y.....
A15 X..... Y..... (1)
A15 X..... Y..... (2)      A04/A15 (7) *
A15 X..... Y..... (3)
A51 X00000 Y00000 (r=0/no rounding)
A15 X..... Y..... (4)
A15 X..... Y..... (5)
A15 X..... Y..... (6)      A15 (6)
A15 X..... Y..... (7)
A17 X..... Y..... (END)

```

NOTES:

- Code A51 and code A52 can be used as a round-off function only between two linear routing-paths, e.g.:

```

A04 X..... Y.....
A15 X..... Y..... (1)
A51 X..... Y.....
A15 X..... Y..... (2)
etc.

```

A52...

- 2) At the crossing between two circular routings (i.e. circle/circle), and between a circular and a straight line (i.e. straight/circle and circle/straight), the codes A51 and A52 cannot be used!
- 3) The numeric capabilities of the A51- and A52-codes are limited to +1 to +65.535 in the Y-field, which equals a corner-radius of 655,35mm.
- 4) When routing **INNER** contours of a pattern, the following guidelines should be noted:
If the desired round-off radius is less than or equals the router diameter, the machine will not perform any round-off, because it is the tool-diameter, that determines the smallest possible rounding radius.
- 5) The calculation of the automatic router radius compensation for code A51/52 is based on the entered Y-value in code A55.
If code A61 or A62 is used for the router radius compensation, then the calculation is based on the entered Y-value in the last code A14 (router-size).

A53: AUT. ARC ROUTING.

Routing of an arc between two linear routing-positions. X and Y indicates the the arc-radius in 1/100mm and the routing-direction (clockwise or counter-clockwise) as well as the arc size (as there are two possibilities: large or small, depending on the location of the center of the arc).

To use the code A53-function, one must determine the following:

- 1) On which side of the routing-path the arc should be placed.
- 2) On which side of the routing-path the arc-center should be placed (i.e. the size of the arc: large or small).

In total, there are four possibilities, and therefore it is necessary to use both the X- and the Y-field of the display, in order to determine both the size and the position of the arc.

However, one may only use either the X- or the Y-field,
and never use both in the same instruction!

A53...

The following possibilities can be used for the programming of code A53:

POSITIVE X-VALUE: The smaller arc to the left of the routing-direction (clockwise routing). Refer to the arc marked *1 on the drawing below.

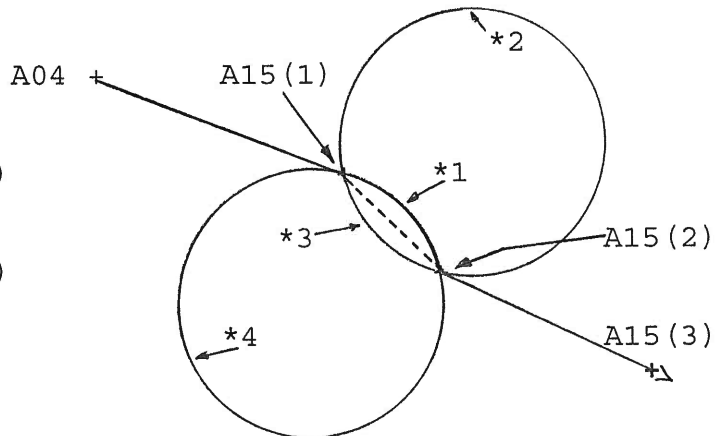
NEGATIVE X-VALUE: The larger arc to the left of the routing-direction CW (clockwise routing). Refer to the arc marked *2 on the drawing below.

POSITIVE Y-VALUE: The smaller arc to the right of the routing-direction CCW (counter clockwise routing). Refer to the arc marked *3 on the drawing below.

NEGATIVE Y-VALUE: The larger arc to the right of the routing-direction (counter-clockwise routing). Refer to the arc marked *4 on the drawing below:

EX. CODE A53:

A14 X..... Y.....
A04 X..... Y.....
A15 X..... Y..... (1)
A53 X..... Y.....
A15 X..... Y..... (2)
A15 X..... Y..... (3)
etc.



CANCELLATION OF A53:

If both the X- and Y-value is '0' when entering code A53, the automatic 'arc routing'-function will be cancelled. In the previous example, the machine will just route a linear path between the two codes A15(1) and A15(2), which is indicated as a dotted line on the drawing.

A53...

SEMI-CIRCLES:

If the X- or Y-radius value is too small compared to the distance between the two codes A15, the machine will perform a precise semi-circle (180 degrees).

This can be useful, if you want to make a precise semi-circle between two points. In this case, you just enter one of the following two commands: A53X1 or A53Y1. With this command, the machine will route a semi-circle

between the two points to the left or to the right of the straight line between the points (depending on the routing direction).

NOTE: Please, note that the X- or Y-value must be '1' and not '0'. If the value is '0', the A53-function will be cancelled.

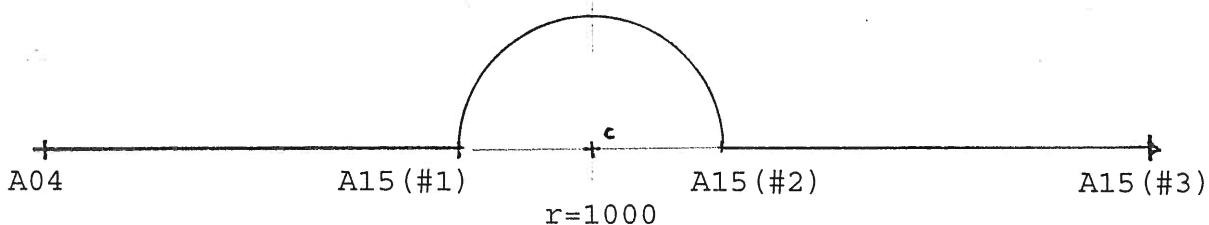
ERROR-SITUATIONS WHEN USING THE CODE A-53:

In the next example (below) it is shown, what will happen if the distance between the two linear routing codes "A15(#1)" and "A15(#2)", does not equal the entered arc-radius in code A53.

In fact, if the distance between the two A15-codes differs only 0,03mm compared to the entered arc-diameter, this will result in a routing-error of approx. 1mm compared to the "ideal" arc-size!

To avoid such problems, one should let the computer calculate the optimum radii etc, and simply set X=1 or Y=1 in code A53. In this case, the computer will automatically calculate the most correct semi-circle between two linear routing-positions.

EX.:



A55: ROUTER RADIUS COMPENSATION LEFT/RIGHT.

Automatic router radius compensation. The desired compensation-value (normally 1/2 of the router diameter) is entered in the X- or the Y-field of the display, and is specified in 1/100mm.

If the desired value is specified in the X-field only, the machine will compensate to the **LEFT** of the routing direction. Alternatively, if the value is entered in the Y-field only, then the automatic compensation will be to the **RIGHT** of the routing-track.

NOTE:

*You can memorize the A55-function this way:
The X-value is always given in the LEFT side of the display,
and the Y-value is always entered in the RIGHT side.*

OBSERVE:

When a value is entered in the X-field, the Y-field **MUST** be set to zero (Y=0), and vice versa.

Thus, code A55 serves several purposes:

- 1) Code A55 activates or cancels the automatic router radius compensation function.
- 2) Code A55 determines the direction of the compensation, (i.e. left/right compared to the routing direction).
- 3) Code A55 determines the value of the compensation.

CANCELLATION:

If the X- and Y-value are both zero (X=Y=0), the code A55-function is cancelled, i.e. no radius compensation active.

EX.1:

Entering A55 X00120 Y00000 means, that the machine will compensate 1.20mm to the left of the routing direction.

Apart from the alternatives mentioned above, there is another option:

If only the Y-field is used for entering the desired compensation-value, the compensation-side can be changed from right to left, simply by adding a minus-sign to the Y-value.

A55...

The same procedure can be used if the desired compensation-value is specified in the X-field. In this case, the compensation will be changed from the left to the right side of the routing-track:

EX.2: **A55 X00000 Y-000120**

This command means, that the machine will perform automatic router radius compensation to the **LEFT** side of the routing-direction.

However, the normal entering sequence is:

EX.3:

```
A14 X00018 Y00240 (TOOL-CHANGE T18=2.40MM ROUTER).
A01 X..... Y..... (ROUTE CIRCULAR HOLE IN X,Y POSITION)
A05 X00000 Y01750 (HOLE DIAMETER 17.50MM, ONE TURN ONLY)
> A55 X00000 Y00120 (COMPENSATE TO THE RIGHT: 1.20MM).
A04 X..... Y..... (GO TO X,Y POSITION)
A15 X..... Y..... (Z-AXIS DOWN + LINEAR ROUTING TO X,Y-POS.)
A15 X..... Y..... (ROUTE LINEAR TO X,Y-POSITION)
A17 X..... Y..... (END OF PROGRAM)
```

From the example above, it is shown, that the circular routing code A01/A05 does not need the router radius compensation code, because it is already built into the circular routing code.

On the other hand, it would NOT influence the circular routing to place the A55 router-radius compensation code before the A01/A05 codes, simply because the A01/A05 code does not utilize the radius compensations codes.

The A55 (or A61/62, see below) code should be entered just before the first linear- or arc-routing code in which it should apply:

EX.4:

```
A14 X00018 Y00240 (TOOL-CHANGE T18=2.40MM ROUTER).
A04 X..... Y..... (GO TO X,Y POSITION)
A15 X..... Y..... (Z-AXIS DOWN + LINEAR ROUTING TO X,Y-POS.)
A15 X..... Y..... (ROUTE LINEAR TO X,Y-POSITION)
A15 X..... Y..... (ROUTE LINEAR TO X,Y-POSITION)
A15 X..... Y..... (ROUTE LINEAR TO X,Y-POSITION)
> A55 X00000 Y00120 (COMPENSATE TO THE RIGHT: 1.20MM).
A04 X..... Y..... (GO TO X,Y POSITION)
A15 X..... Y..... (Z-AXIS DOWN + LINEAR ROUTING TO X,Y-POS.)
A15 X..... Y..... (ROUTE LINEAR TO X,Y-POSITION)
A15 X..... Y..... (ROUTE LINEAR TO X,Y-POSITION)
A15 X..... Y..... (ROUTE LINEAR TO X,Y-POSITION)
A17 X..... Y..... (END OF PROGRAM).
```

A55...

In the example above is shown, that the machine first will route a figure (e.g. a square hole) without the router radius compensation activated, after which it will go to a second figure where the endges are routed with the router radius compensation activated.

The router radius compensation does not need to be cancelled in the example above, because the computer is returned to code A03 by the A17 code, and the A03-code will automatically cancel the A55 compensation value.

If the A55 compensation value were not cancelled, it would lead to a gradually greater offset of the machine for each time the program were executed.

However, it may be necessary in some programs to **insert a cancellation block** in order to cancel the latest activated A55 compensation, e.g. especially in 'nested' programs, where the computer is deliberately re-circulated inside a program in order to perform e.g. certain repeated operations.

To cancel the A55 compensation, simply add a new A55-code in the program with the XY-values 0, i.e.:

A55X00000Y00000.

NOTE:

It is strongly recommended to use the A55 code for the automatic router radius compensation (opposite to the codes A61 and A62).

This is because, it may frequently be necessary to adjust the compensation value to match the desired tool-diameter and to adjust the correct size of the workiece.

Because the codes A61/62 uses the A14-tool diameter as the basis for the calculation of the compensation, it would be necessary to adjust the value of the router diamter to match a desired change in compensation value.

However, the tool-diamter is also used by the operator of the machine to load the tool-magazines, each time a new job is started.

A55...

The operator would then get confused to see a router diameter of e.g. "A14 X00012 Y00230", which means that tool-number T12 is a router with a diameter of 2.30mmØ, which he may accidentally load into the machine, instead of the originally programmed 2.40mm router.

Instead it is recommended to leave the correct tooling information in the program unchanged, and use the A55-code for the compensation of the routing programs.

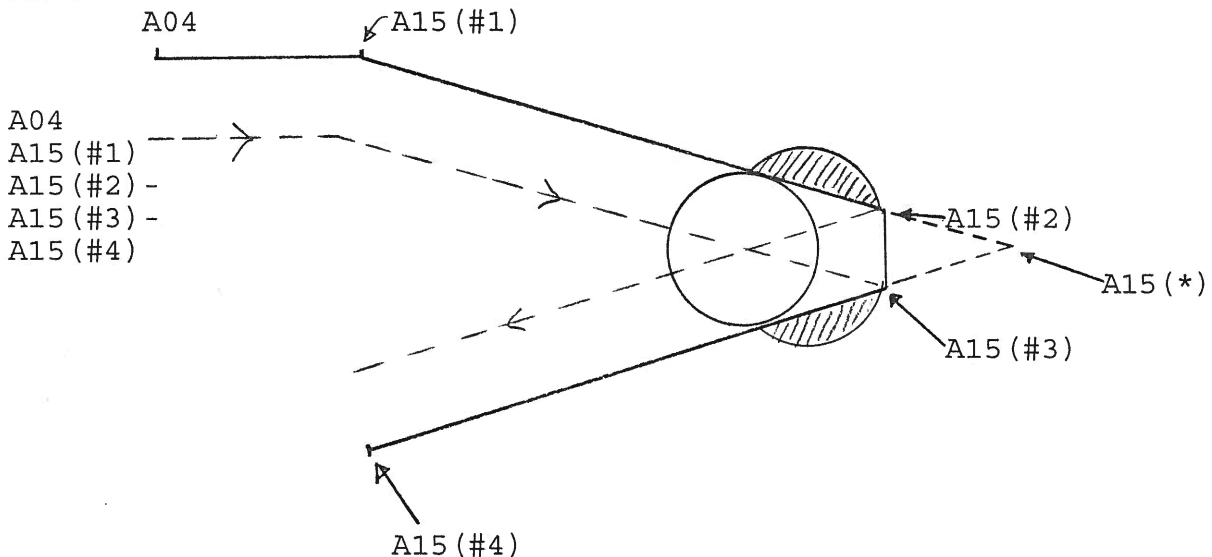
POSSIBLE ERRORS WHEN USING AUT. ROUTER RADIUS COMPENSATION:

In the example below you will find a typical error-situation (magnified approx. 10 times), which is caused by the following problem:

The distance between kode A15(2) and A15(3) is too small compared to the radius of the router.

In cases, where this situation can be predicted, the operator should substitute code A15(2) and A15(3) with a new code "A15*" which is shown in the example below.

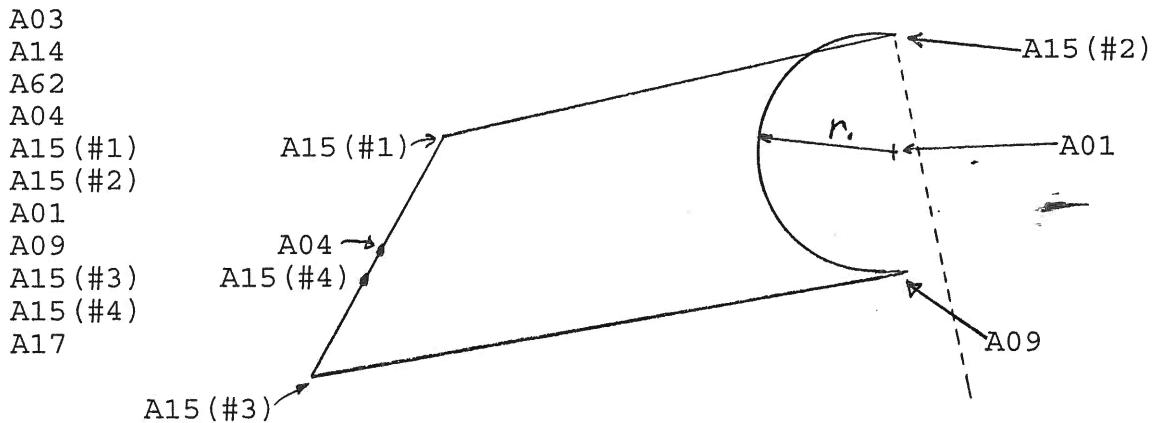
EX.:



The result of the programming-error described above would be, that the two arcs (marked as shaded areas) would be routed, though this were not intended.

A55...

In the example below, one can see another typical error-situation, where an incorrect placement of a code A01 results in an "uncontrolled" router radius compensation during the operation of the machine.



The error-situation described above can also arise, if the arc is programmed by means of the codes A53 and A15, because a slightly incorrect radius-value might result in a "false" position of the arc-center.

A60: ROUTER RADIUS COMPENSATION OFF.

Cancellation of the code A61 and A62 (=automatic router radius compensation).

Code A60 cancels the aut. router radius compensation introduced by code A61 (left) or A62 (right).

The codes are used similary to the ISO-codes G40, G41 and G42.

Refer to the paragraphs below concerning code A61 and A62.

A61: ROUTER RADIUS COMPENSATION LEFT.

Automatic router radius compensation to the LEFT of the router direction on the programmed contour.

The compensation is automatically calculated on the basis of the last entered tool-size (i.e. code A14).

The compensation is calculated this way:

A14 X00000 Y00240 (tool change to a router with 2.40mm dia).
A61 X..... Y..... (compensate to the LEFT with $240:2=1.2\text{mm}$).

NOTE: *The dimensional accuracy of the workpiece is highly dependent on the precision of the code A14-input.*

If code A14 is missing, code A61/62 will NOT enable the automatic router radius compensation-function.

The X- and Y-fields are not used by the computer when utilizing code A61/62.

Code A61 corresponds to the ISO-code G41 and it is cancelled by the code A60 (see above).

A62: ROUTER RADIUS COMPENSATION RIGHT.

Automatic router radius compensation to the RIGHT of the router direction.

The compensation is aut. calculated on the basis of the last entered tool-size (i.e. code A14).

The aut. compensation is carried out exactly as for code A61 (see above).

A65: OUTSIDE CIRCULAR ROUTING W/TAPS.

This code is only valid from software-versions V5.0 or higher.

PROGRAMMING EXAMPLES, ROUTING:

PLACING THE START/STOP-POINT:

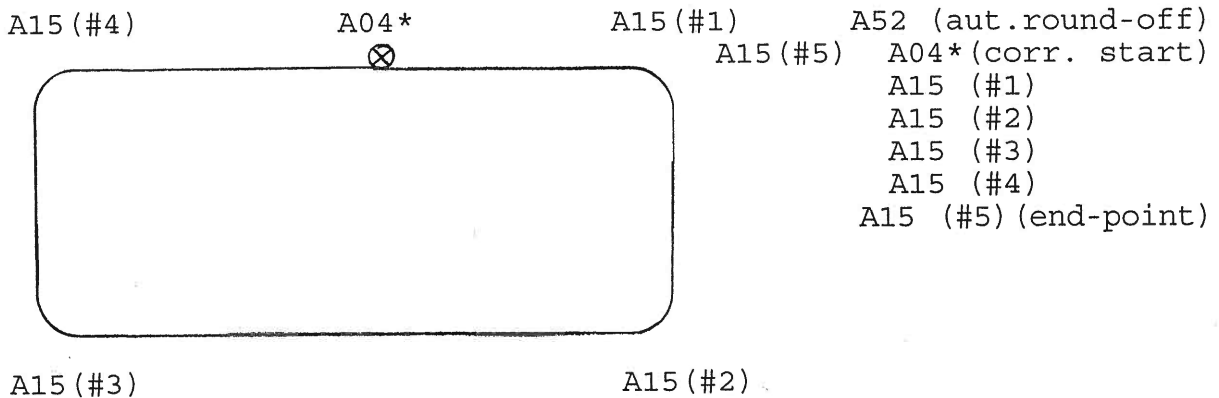
When routing, it is important to determine the most suitable placement of the start/stop-point of the pattern.

This is especially important, because the machine is equipped with the fully automatic corner round-off function, and therefore it is not possible to use a corner as the start/stop-point for the routing-operations.

On the other hand, if you are certain, that the routing-pattern does not contain any corners that need to be rounded-off (now or in the future), then you might still use one of the corners of the pattern as a start/stop-point; but it is highly recommended NOT to use this way of programming, as it might create some "bad" practise, which, on a later stage, could lead to problems when the programs might be modified.

In the occasions where at least one of the routing-paths are fully parallel to the X- or the Y-axis of the machine, it is possible to create a new "corrected" start/stop-point, as shown in the example below:

EX.:



On the next page, you will find an example of a routing-pattern, where it is more complicated to find a new, "corrected" start-point for the routing.

As can be clearly seen in the example below, none of the sides of the pattern are parallel to the machine X/Y-axes.

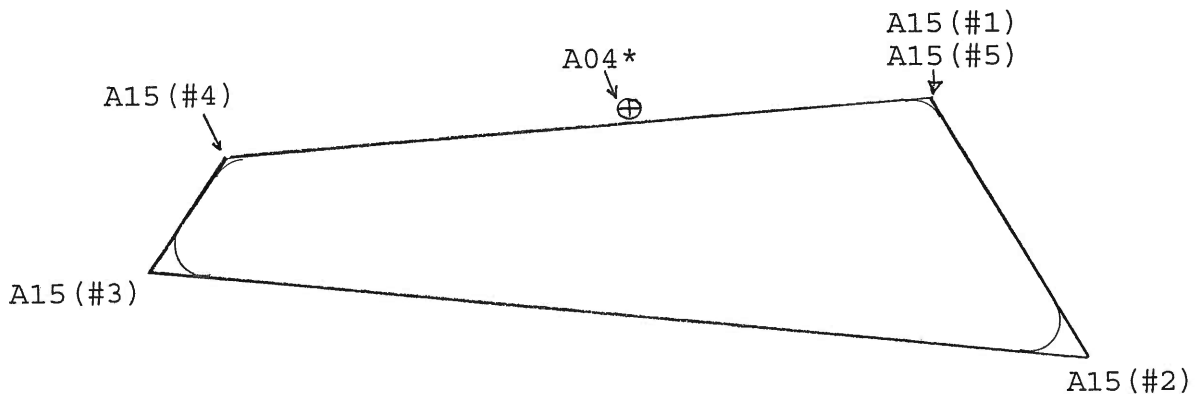
Additionally, let us assume that both automatic router radius compensation and automatic corner round-off are desired on the pattern. Only the coordinates for the corner-points A15(#1) through A15(#5) are known.

The following procedure should be used for programming the pattern in the example:

```

A03 X..... Y..... (program-number)
A14 X..... Y..... (router size)
A55 X..... Y..... (aut. router compensation)
A52 X..... Y..... (aut. corner round-off)
  <<INSERT HERE THE NEW CORRECTED START-POINT "A04*".
A15 (#1)
A15 (#2)
A15 (#3)
A15 (#4)
A15 (#5)
A17 X..... Y..... (end)

```



WAY OF OPERATION:

Enter the routing codes for the four corners (i.e. five "A15" codes in total).

Then, let the machine perform a test-run of the program **WITH THE AUTOMATIC ROUTER RADIUS COMPENSATION SWITCHED OFF!** (i.e. with code A55X0Y0).

IMPORTANT: *This test-run must be activated by means of the "START"-key (i.e. NOT the "TEST"-key)!*

While the machine is performing the routing program, one stops the machine (by means of the "STOP"-key) when it reaches a position approximately in the middle of the positions between code A15(#4) and code A15(#5).

However, the start/stop-points should always be selected in a way, that these do not interfere with the arcs of the round-off's whenever these are activated.

This means, that the start/stop-points always must be located at a distance which is greater than or equals the corner-radius!

OBSERVE: *Assuming the corner radius is 10mm, the start/stop-points must be located at least 10mm from any of corner.*

After the machine has been stopped at a place between code A15(#4) and A15(#5), one search code A52 in the program **WITHOUT POSITIONING THE MACHINE**, and insert a code A04 between code A52 and code A15(#1) in the memory (by means of the "ENTER"-key).

Finally the correct router radius compensation-value is entered in code A55, and the program is ready for use.

START/STOP POINTS IN INTERNAL CONTOURS:

When routing internal contours, one cannot use ANY of the corners as start- or stop-points. Therefore it is necessary for the operator to find a new and "corrected" start/stop-point in these cases (refer to the example on the preceding page).

The reason for not using any of the corners in an internal contour is, that the router would remove a bit (equalling the router-radius) of the surrounding material both at the start- and the stop-position. This is because the router only compensates for the router radius in one direction, namely 90 degrees angular to the routing path.

However, when the router plunges down into the material exactly in the corner, half of its diameter will be behind the entering position because it only compensates to either side, not forward/backward.

The same situation occurs when the router goes to the last position. Here, half of the router will be in front of the actual end-position, hence a small cross will be generated in the start/stop-corner, if you are not moving the start/stop-positions at least a distance equalling the router-radius both at the start position and the end position.

EX.:

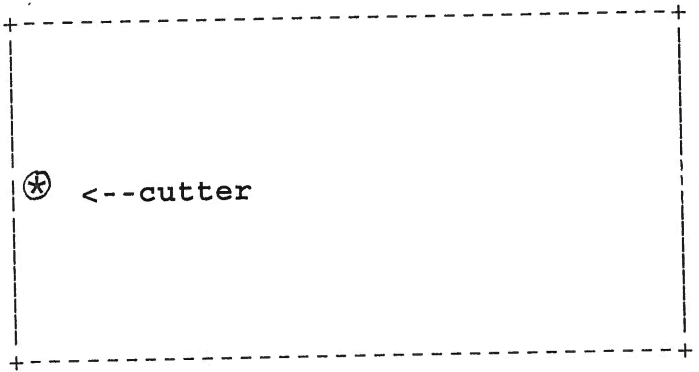
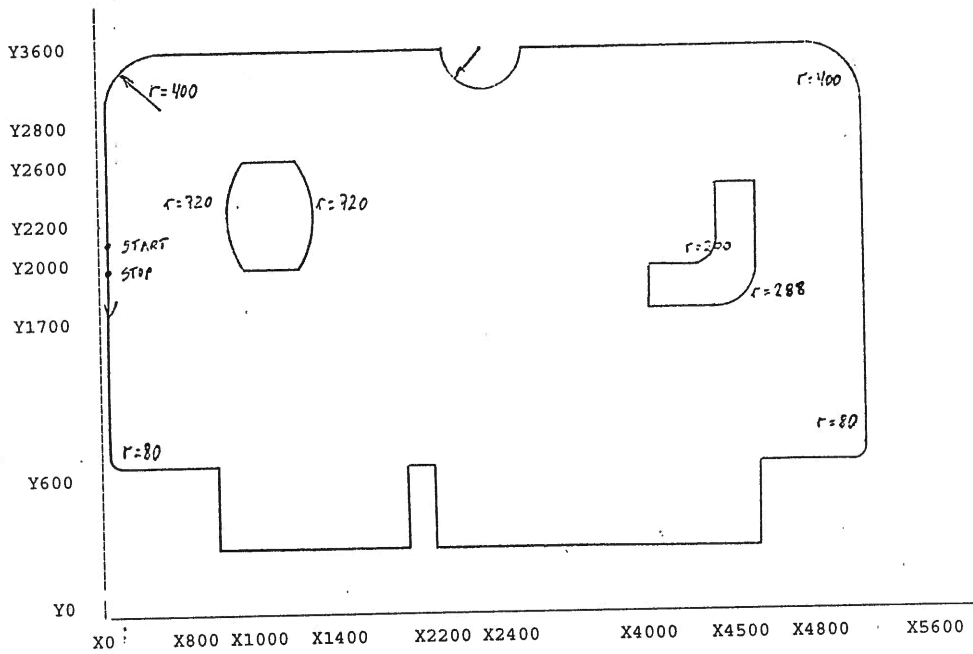


ILLUSTRATION TO THE EXAMPLE WITH AUT. ROUTER RADIUS COMPENSATION:



PROGRAMMING EXAMPLES, ROUTING:

Example of routing with the automatic router radius-compensation, (please refer to the illustration on the preceding page).

```
A03 X00000 Y00001 Program number 1.
A14 X00001 Y00200 Tool-change to router 2,00mm in magazine no.1.
A55 X00000 Y00100 Start of aut. router radius compensation, 1mm
                    compensation to the right side of the track.
A04 X00000 Y02200 Start-point.
A15 X00000 Y00600
A51 X00000 Y00080 Single corner round-off / 0,8mm radius.
A15 X00800 Y00600
A15 X00800 Y00000
A15 X02200 Y00000
A15 X02200 Y00600
A15 X02400 Y00600
A15 X02400 Y00000
A15 X04800 Y00000
A15 X04800 Y00600
A15 X05600 Y00600
A51 X00000 Y00080 Single corner round-off / 0,8mm radius.
A15 X05600 Y03600
A52 X00000 Y00400 Multi-corner round-off / 4,00mm radius.
A15 X03100 Y03600
A53 X00001 Y00000 Arc-function / Semi-circle (because X=1).
A15 X02500 Y03600
A15 X00000 Y03600
A15 X00000 Y02000
A52 X00000 Y00000 Cancel aut. router radius compensation (X=Y=0).
A04 X01200 Y02000 Start-point of "()" -shaped hole.
A15 X01000 Y02000
A53 X00720 Y00000 Arc-function/ smallest arc/ counter-clockwise.
A15 X01000 Y02800
A15 X01400 Y02800
A53 X00720 Y00000 Arc-function/ smallest arc/ counter-clockwise.
A15 X01400 Y02000
A15 X01200 Y02000 Stop-point of "()" -shaped hole.

A04 X04500 Y01700 Start-point of "]" -shaped, angular figure.
A15 X04000 Y01700
A15 X04000 Y02000
A15 X04500 Y02000
A51 X00000 Y00200 Single corner round-off / 2,00mm radius.
A15 X04500 Y02600
A15 X04800 Y02600
A15 X04800 Y01700
A51 X00000 Y00288 Single corner round-off / 2,88mm radius.
A15 X04500 Y01700 Stop-point of "]" -shaped, angular figure.
A17 X..... Y..... End of program/end of memory/tape stop/etc.
```

The program-example described above, refers to the drawing on the preceding page.

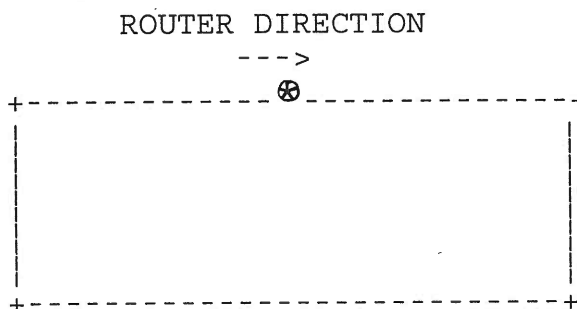
COMMENTS ON THE ROUTING-CODES:

GUIDELINES:

To get the best result when routing PCB's, one should note the following fundamental guidelines:

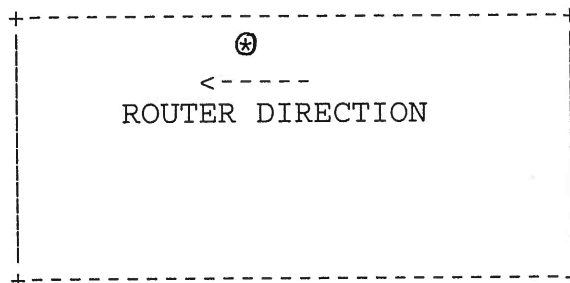
ROUTING OF THE EXTERIOR:

When routing the outer side of a PCB, one should always use the clockwise routing in order to get the best result:



ROUTING OF THE INTERIOR:

The inner contour should always be routed counter-clockwise, as shown in the example below:



If these basic guidelines are followed, one will get the best results in terms of clean cutting surfaces and the highest precision of the routing.

STOPPING THE ROUTING-OPERATION:

If a routing operation is stopped by means of the "STOP"-key or by means of the built-in safety-switch in the shield, it is very important NOT to change the position of the work-tables, e.g. by activating the joystick!

This must be avoided, because the machine will continue the routing operation from the actual position.

Also, it is not allowed to change the block-number in the memory, if the routing-operation should be continued from the same position, where the "STOP"-function was activated.

If the machine-position has been changed accidentally, one can search the nearest block and/or position which is known BEFORE the place where it was stopped.

This is carried out by means of the "⊕ COORD"-shift-function. Afterwards, the routing-operation can be continued from this (preceding) place.

NOTE: *Searching of a coordinate position is possible in all circumstances, even in "STEP & REPEAT's", "MIRRORING", "SCALING" etc.*

SUMMARY OF ROUTING COMMANDS:

- A01: Center for all circles and arcs (partial circles). The code is used in combination with the codes A05, A09 and A10. However, the A01-code is also used when programming DIL and SIL IC-patterns to indicate the first IC or SIL pin-position.
- A04: Positional code. Travels the machine to the X,Y-position without any Z-axis movement.
- A05: Routing of a full circle. Y-value is circle-diameter in 1/100mm. Code A05 is used in combination with code A01.
- A09: Counter-clockwise routing of an arc (part-circle) to X,Y. Code A09 is used in combination with code A01 (center).
- A10: Clockwise routing of an arc to X,Y-position. Code A10 is used in combination with code A10 (center of the arc).
- A15: Pull Z-axis down and perform linear routing to X,Y.
- A51: Routing with automatic corner-rounding of a SINGLE corner. The "Y"-field of the display indicates the desired corner-radius in 1/100mm.
- A52: Routing with automatic corner-rounding of ALL corners on a pattern. The "Y"-field of the display indicates the desired corner-radius in 1/100mm.
- A53: Routing of an arc between two positions (i.e. straight-circle or circle-straight). X and Y indicates the arc radius in 1/100mm and also the routing direction as well as the arc-size (large or small).
- A55: Automatic router radius compensation. The desired compensation is indicated in the X (LEFT) or in the Y (RIGHT) in 1/100mm, where 'LEFT' and 'RIGHT' is compared to the routing direction.
- A60: Code A60 cancels the automatic router radius compensation, programmed with code A61 and A62.
- A61: Automatic router radius compensation to the LEFT of the routing-path direction. Code A61 compensates the router radius on the basis of the last code A14 (=router size).
- A62: Automatic router radius compensation to the RIGHT of the routing-path direction. Code A62 compensates the router radius on the basis of the last code A14 (=router size).

MAINTENANCE:

COOLING SYSTEM:

CONNECTION AND MAINTENANCE OF COOLING UNIT FOR CNC-1000/2000:

The "PHK-450" cooling-unit must be connected to the machine before it is switched on or in any way put into operation.

ELECTRICAL CONNECTION:

The cooling-unit must be connected to the mains outlet on the rear side of the Unitrol-2 CNC-cabinet.

Use the rightmost mains-outlet (viewed from the rear side), of the special double mains distributor, as these two rightmost outlets are used for the cooling-unit and the video-system.

Using only this electrical outlet of the machine will ensure, that the cooling-unit is always switched on simultaneously with the machine!

WARNING!

Make sure, that you never switch off the cooling unit for any reason whatsoever, as this may cause severe damage to the sensitive HF spindle bearings!

NOTE: *Be careful NOT to use the special, single mains outlet for the vacuum extraction system!*

The vacuum extraction outlet is START/STOP-controlled by the computer, whereas the double outlet is continuously ON whenever the machine is switched ON.

CAUTION: Always check, that the control lamp on the front-panel of the cooling-unit is illuminated whenever the machine is switched on.

Check frequently during operation, that the temperature is within green-range of the thermometer on the front.

MAINTENANCE:

CONNECTION OF THE COOLING-FLUID PIPES:

On the side of the machine stand, you will find two 8mm pipe-branches for the connection of the cooling-fluid pipes. The leftmost of the pipes, marked 'IN' is connected to the cooling-unit's outlet marked "VORLAUF" and the other, marked 'OUT' is connected to the cooling-unit inlet marked "RÜCKLAUF".

NOTE: Make quite sure, that both pipes is connected properly and that the connections are completely tight, so absolutely no fluid is leaking from the system.

Now fill the cooling-tank (*WHILE YOU CONSTANTLY WATCH THE LEVEL OF THE COOLING-UNIT*) with approx. 30 liters of ordinary tap-water to which first is added 1/2 liter of the enclosed "N-4373" cooling-fluid (red coloured).

To fill the tank of the cooling-unit, remove the top-section by lifting it approx. 50 cm upwards in the two handles. The cooling-unit top must be put on a **clean** floor while filling the water-container, to which you now have easy access.

The "N43-73" cooling fluid serves more purposes, but primarily it preserves the water and also add some red colour to the water, which makes it easier to observe the level of cooling-fluid in the system.

CAUTION: The "N43-73" fluid is a strong alkaline solution which can be harmful and cause severe injuries. Therefore protect your eyes and your skin, and wear protective glasses and rubber gloves when working with the cooling-unit!

CHEMISTRY:

The chemical composition of the N-4373 cooling-fluid is:

Concentrated alkaline mixture of Natriumnitrite as well as in-organic and organic corrosion-inhibitors. Contains Nitrite.

MAINTENANCE:

GENERAL MAINTENANCE SCHEME:

DAILY:

CLEANING OF THE MACHINE PARTS:

Clean the machine every day at the end of the working hours.
Use a dry cloth and a vacuum cleaner.

If the machine is operated in 3 or 5 shifts, it is recommended that the cleaning procedure is made during one of the job-shifts every 24 or at least 48 hours.

Especially the tool-change magazines, the tooling-systems, the workplanes and the bottom of the machine should be cleaned at regular intervals.

CLEANING OF THE HF-SPINDLES:

The spindles are normally service free, except for cleaning of the spindle-collet.

Especially when using the machine for heavy-duty routing applications, it is recommended to remove the collets for cleaning up to 2 times per shift.

So, in 3 shift applications, the collet(s) may need cleaning 6 times per day.

Otherwise, the spindle accuracy and/or the clamping of the toolshaft may be severely affected.

COLLETS:

The dismantling of the collet is carried out by means of the small knurled black knob, kept in the SPINDLE MAINTENANCE KIT supplied together with the machine.

COLLET DISMANTLING:

To remove the collet, you must first ask the machine to download the present tool which is in the collet.

This is done by using the "TOOL |"-key (see this), after which the machine puts the tool down in the desired tool-pod, which of course MUST BE EMPTY!

After downloading the tool, the collet is OPEN!

You can now position the Z-axis to an appropriate position of the workplane by means of the joystick.

Remove the clamping foot and get full access to the spindle's nose end.

Insert the knurled "collet remover"-tool into the collet and turn it the same way as a normal screw, i.e. the same way as the spindle's rotational direction (=Clockwise, seen from above).

JAMMED COLLETS:

Sometimes a collet is slightly blocked, and it can be necessary to apply a little extra force when turning it out.

However, it is important that you **DO NOT USE EXCESSIVE FORCE**, and certainly not additional equipment to loosen the collet.

OBSERVE: *If a total blocking occurs, please consult your local dealer or the manufacturer. We can advice you, how to proceed. Do NOT try to use special tools or sheer power!*

COLLET CLEANING:

When the collet has been removed from the spindle, it can be cleaned by means of the tools included in the spindle maintenance kit and compressed air.

NOTE: *In case of high contamination, it may be necessary to use acetone to dissolve the stamped laminate-dust. However, the rubber and plastic sealings may suffer from this chemical treatment, so these must be removed from the collet before chemical treatment or better: Replaced with new sealing-elements. Be careful not to damage any of the seals, because this will lead to premature bearing break-down.*

WARNING: NEVER USE COMPRESSED AIR TO CLEAN THE HF-SPINDLE!

CLEANING THE SPINDLE CONE:

Before refitting the collet, the spindle's taper section must be cleaned. Use the two enclosed wooden tools for cleaning of the conical and cylindrical interior respectively of the spindle's taper section.

Be careful to clean the wooden tools before inserting them into the spindle cone. Especially it must be avoided that the cleaning tools touch the spindle exterior during the insertion process, because the spindle nose normally is highly contaminated with dust.

GREASING THE COLLET:

Before refitting the cleaned collet, it must be greased slightly with the enclosed "NBU"-grease. The NBU-grease is applied in very small amounts to the internal thread of the collet and on the upper, cylindrical, external shaft-end of the collet.

Please, use a toothpick to apply the grease in the internal thread.

IF YOU FAIL TO GREASE THE COLLET BEFORE REFITTING, IT MAY JAM TOTALLY. This means a complete, costly repair of the spindle!

NOTE: *Be careful that no grease gets into the collet, as this will imply that the spindle cannot clamp the tools sufficiently, especially during routing operations.*

REFITTING THE COLLET:

When refitting the collet, make sure that it does not get in contact with the spindle's nose end (the external) because this is extremely contaminated with PCB-dust. This most closely observed, otherwise you will accidentally introduce a severe contamination of the sensitive spindle-bearings!

Now turn (CCW) the collet to it's mechanical end-stop. Make sure that it is really the mechanically end of travel of the collet.

Sometimes, the collet is momentarily jammed during the insertion because the internal thread of the collet and the screw inside the spindle-body are both made of hardened steel. Two parts of hardened steel can easily lock, thereby making the operator believe, that the mechanical end stop is reached.

If possible make a visual inspection of the collet position and/or try to loosen the collet again when it is at the end stop and tighten it again.

REFITTING THE COLLET....

This procedure will give you a feeling for whether the collet has actually reached the end or not, because if you after loosening the collet e.g. 1/2 turn can tighten it maybe 1 turn, this clearly indicates that the collet was not at the correct end-stop. In this case, retry again.

CAUTION: *Please follow the spindle-manufacturers guide-lines for the cleaning and general maintenance of the HF-spindle. If in doubt, refer only to the spindle manufacturers instructions, which supersede the comments made here.*

VACUUM EXTRACTION EMPTYING:

The vacuum container of the extraction unit must be checked a couple of times every day, if working in one shift. When the machine is operated in 3-5 shifts, it may need emptying several times a day.

In this case, it must be stressed, that the standard vacuum unit is not intended for heavy duty applications. It is strongly recommended to connect the machine to a central vacuum system. The mains outlet for the vacuum unit can in this case be used to control a vacuum-throttle, so that the vacuum debris is still controlled by the machine.

However, if using the standard vacuum unit, it is important to check the dust-level frequently in order to avoid that it is over-filled.

CAUTION: *Over-filling would cause the extraction to be blocked, and this might severely harm the HF-spindle(s) of the machine, because the extraction cup/clamping foot will be filled with dust and swarfs. When the rotating parts of the spindle nose grind the swarfs it will overheat the entire spindle nose end, and cause fatal damage to the spindle(s).*

CLEANING OF THE ELECTRONICS:

The CNC-system should be cleaned in the air-inlet for the fan. This air-inlet is found on the rear side of the CNC-system. It is covered by a protective shield, and it must be vacuum-cleaned at regular intervals.

CAUTION: Do not try to open the CNC-rack for internal cleaning! There is no user's serviceable parts inside, and it is highly DANGEROUS to touch any of the internal electrical parts of the machine!

WARNING: Voltages of 400V inside the CNC-system are present, even after having switched the machine OFF!

CHECK-POINTS:

Prior to all operations on the machine make sure, that the cooling-unit is switched on (observe red light on the unit), and that the air-pressure is in the range of 5.5-6bars.

Especially, it is important to check, that the air-pressure does not exceed 6 bars.

Higher pressures may damage the HF-spindles, whereas lower pressures may lead to problems with the tool-change reliability.

WEEKLY:

WATER LEVEL:

Once a week, you must check the water level of the system, because experience shows, that up to 1 ltr. of the cooling-fluid can evaporate from the system per week!

Be careful when determining the water-level:

Small particles of dust and chalk on the water-surface can create a coloured "ring" on the walls of the water-container making it difficult to determine the actual water level.

The most efficient way of controlling the water level is to rock the cooling unit on the floor and watch the lapping waterlevel.

MONTHLY:

COOLING UNIT:

Once a month, you must clean the segments of the condenser air inlet, as this **MUST BE FREE OF DUST**. The air-inlet for the cooling-unit is located on the front-side.

Too much dust on the segments will cause insufficient cooling capacity, which might lead to damage of the ball bearings of the HF-spindles or the cooling unit itself.

For cleaning of the fragile cooling-segments of the system, use only a vacuum-cleaner and a little compressed-air (with limited pressure).

LEADSCREWS:

The leadscrews for the X, Y and Z-axes must be greased at regular intervals. Again, the grease intervals depends on the use of the machine. If the machine is used in 3 or 5 shifts operation, it should be greased at least once a month.

Use a grease gun with flexible nose and grease of good quality,

e.g. **SKF "LGMT-2/1"**

or a similar high quality bearing grease.

Before applying the grease to the grease nipples, make sure that the end of the grease gun as well as all the grease nipples are cleaned completely. Then press a small amount of grease out of the grease gun before the actual greasing takes place.

To get access to all the grease nipples it is recommended to remove the bonnet of the machine. The X-axis as well as the Z-axis grease nipples are located on the upper workplane (Z-plane) whereas the grease nipple for the table (i.e. lower workplane) is located below the workplane, approx. 10cm (4") towards the center of the workplane and to the left of the leadscrew assy.

Place the upper workplane approx. in the center of it's travel and the lower workplane to the rear and switch off the machine to avoid accidents while the greasing procedure is carried out.

GUIDE RAILS:

The machine's axes travels on precision grinded, hardened guide-rails which should be greased in the same way as described above. There are grease nipples at the end-of travel positions on the workplanes and these should be greased according to the rules set forth above (see 'LEADSCREWS').

The round guides are life-time greased internally, and should not be greased additionally. Only apply a thin layer of oil-film on the round guides to prevent rust. Only use a thin, acid-free oil for this preventive measure, and only if the surrounding air is aggressive (i.e. having high acid density or humidity levels).

ANNUAL :

COOLING UNIT:

Every 6-12th month, you should empty the cooling-unit totally, and clean it with fresh water.

To empty the cooling unit tank, remove the pipe connected to the "RÜCKLAUF" pipe-branch, and use this pipe for emptying the fluid into a suitable pail or container (with a capacity of approx. 40 liters).

When the cooling-tank is empty, fill some new, fresh water into the system by using the "ZULAUF"-inlet branch which is located on the front-plate of the cooling-unit, or by simply removing the cooling-unit top (as described above).

Add to the fresh water 1% of a special cleaning mixture called "BODROB-REINIGER", which is available from your local Mape-dealer.

Continued

The "Bodrob-Reiniger" will clean the cooling-system pipes as well as the tank and the interior channels in the spindles.

When having added the 1% of "Bodrob-Reiniger" let the cooling-unit run for 24 hours before you empty the system again.

NOTE: Be sure, that the pump in the cooling unit is actually active, after having drained the system.

The problem is, that the pump can be filled with air, and therefore is not able to produce any flow!

To test the system for this severe malfunction, one should switch-on the cooling-unit before refitting the "RÜCKLAUF"-pipe-branch, and actually watch, that the fluid is pumped through the pipes.

Use a pail or another suitable container to catch the fluid from the pipe.

If the pump fails to produce any flow, this can be cured by starting and stopping the cooling-system with intervals of 10 seconds, until the air is sucked away and the pump suddenly starts producing sufficient flow.

After having operated the cooling-unit with the 'Bodrob-Reiniger' for approx. 24 hours, this "cleaning-mixture" MUST be evacuated from the tank, before the 'real' cooling-fluid is filled into the tank.

The new cooling fluid is prepared of 30 liters of ordinary tap-water to which added 1/2 liter of "N-4373" fluid (see above).

NOTE: Do not use demineralized or distilled water in the cooling fluid mixture, as this will lead to formation of biological "slime" and other slimy particles in the system, hence severely degrading the cooling-performance!
Only if your water supply is very "hard" or "chalky", one should use up to 1/2 demineralized water and 1/2 tap-water to make the right cooling mixture.

After having filled the system with the right mixture, again make sure, that the pump is actually pumping, by performing the same test as described above.

Finally make sure, that all the branches are tight so no fluid is wasted in the system.

GENERAL:

- 1) Prior to all operations on the machine make sure, that the cooling-unit is switched on (observe red light on the unit).
- 2) Check that the air-pressure does not exceed 6 bars.
Higher pressures may damage the HF-spindles.
- 3) Never run the cooling-unit without water in the tank.

=====

INTERFACE.

INTERFACE TO MAPE CNC-MACHINES:

This section of the manual discusses the interfaceability between Mape machines and various CAD systems and CAD-generated programs.

Also, the interfaceability between other popular CNC-formats such as **SIEB+MEYER*** and **EXCELLON**** has been of great interest to our distributors and customers.

Therefore we shall try to explain about the interface possibilities to and from the Mape machines.

NOTE:

- *) **SIEB & MEYER** is the trademark of Sieb & Meyer GmbH, Germany.
- **) **EXCELLON** is the trademark of EXCELLON INDUSTRIES INC, An ESTERLINE CO, USA.

CAD FLOPPY:

Re. CAD-programs contained on floppy diskettes:

If you want to enter a CAD drilling program contained on a floppy disk into the Mape machine, you must use a PC that includes a software program for the data transmission of the CAD-datas via the serial interface port of the PC ("RS-232"). The PC must also include a suitable floppy-disk drive (either 5,25", 3,5" or both).

In this case, you enter the floppy disk into the PC and prepare the PC for the transmission to the Mape machine.

The Mape machine is then commanded to receive CAD-data in either 1/100mm or 1/1000" respectively (depending on the format contained in the floppy disk).

It is possible for the Mape machine to receive serial data transmission in the following formats:

- 1) CAD data in 1/100mm ("METRIC") with leading- or trailing zeroes (which is automatically detected by the CNC), and which is consisting of max. 5 digits and signs (+ or -).
- 2) CAD-data in 1/1000" ("IMPERIAL") with leading- or trailing zeroes (which is automatically detected by the CNC), and with max. 5 digits in X and Y and signs (+ or -).
- 3) CAD-data in 1/1000" with "leading zeroes" which is a typical EXCELLON format.

If the program is a leading-zero format, whether in INCH or MM, it is necessary to enter the correct data transmission code before the data transmission from the PC is received.

However programs in MM-format with leading-zeroes, are not an industry-standard.

- 4) During the serial data transmission to the Mape machine, it is possible to swop (exchange) the X and Y co-ordinates during the data transmission by entering the corresponding code before transmission is carried out.

If the customer does not have a PC with a special software package for the data transmission, we have developed a special PC-based system for the storage and bi-directional transfer of all sorts of CAD- and CNC-programs, called the "MAPE ProgramBank".

NOTE: Please refer to the section below, "Mape ProgramBank Front End System".

When customers are asking whether Mape machines can interface to their CAD-system, the answer is **YES** (provided the CAD system has a RS-232 serial interface can generate an industry standard drilling program in 1/100MM or 1/1000 inch)!

According to our experience, any of the numerous CAD programs we have seen through the years, will be able to produce the desired drill file for data exchange with the Mape machines.

We have kept the CAD data requirements and the hardware requirement at the lowest possible level for safe data transfer, thereby ensuring that the machines can interface to the majority of the CAD systems in the world market.

EXAMPLES:

For easy understanding, the normal Excellon CAD format is made in 1/1000", leading zeroes, 5 digits (and signs +/-).

Leading zeroes means, that the normal trailing zeroes has been removed. In order to maintain the 'decade'-information for the machines, it was necessary to add leading zeroes instead.

However, it was considered important to save as much memory-space and paper tape as possible, and the leading zero format indeed saved some bytes. The drawback of the leading zero format is, that it is difficult to read for operators only familiar with the conventional (european) trailing zero format.

Examples of a 1/1000", 5-digit Excellon leading zero format:

X001 means 'X100' or $X100/1000 = X0.1$ " (=2.54mm),
X01 means X1000 or 1" (=25.4mm) in european standard,
X015 means X1500 or 1.5" (=38.1mm) do.

In order to enter a standard Excellon format CAD program into the Mape machine, the operator should choose the following commands in the CNC prior to the data transmission:

- 1) Set-up the PC for transmission of the CAD program according to the transmission program and the operators manual. The standard factory setting of the RS-232 transmission protocol is:

9600 baud, 7 data bits, 2 stopbits.

However, when a Mape ProgramBank system is supplied as a part of a machine, **19200 baud** is the standard setting used in order to reduce transmission time by 50%.

- 2) Set-up the Mape machine for receiving a CAD-program in inch-format ("IMPERIAL") by activating the following keys:

"8" + "2" + "SHIFT" and finally "DATA IN".

So, before the transmission is carried out, the machine's display should show:

>>A82 X..... Y.....<<

after which you press the "SHIFT"-key followed by the "DATA IN"-key.

"8" means serial reception of 1/1000"-program, and
"2" means leading zeroes.

- 3) Now, the display changes to: **"INPUT OPEN/STOP KEY ACTIVE"**, and the RS-232 serial input of the machine is ready for receiving the CAD data. During transmission, the built-in beep'er will produce small tick'ing sound according to the signals received in order to create an audible warning of data transmission in progress, in addition to the display, that will show:

>>INPUT RECEIVING<<.

- 4) After succesful transmission, the display will show the last block received. For conventional CAD-programs, the last block received will be "M30" which will be translated to the corresponding Mape code "A17" which is displayed on the display. The M30=A17 means "program end, rewind tape".



RECEIVE S+M FORMATS:

A60 X..... Y..... where X..... is not used (should be X00000) and where the last 3 digits in the Y-field determines the following options:

3rd DIGIT: "ZERÖES/DECADE":

- "0"= trailing zeroes or fixed decimal point (xxx.xxx).
- "1"= leading zeroes in 5 decades (xxxxx), typ. 1/1000".
- "2"= leading zeroes in 6 decades (xxxxxx).
- "3"= leading zeroes in 7 decades (xxxxxxx).

2nd DIGIT: "UNIT/RESOLUTION":

- "0"= 1/100mm.
- "1"= 1/1000mm.
- "2"= 1/1000".
- "3"= 1/2000".
- "4"= 1/10.000".

1st DIGIT: "MACHINE FORMAT":

- "0"= MAPE format and CAD-format.
- "2"= Sieb+Meyer format-1000.
- "3"= Sieb+Meyer format-3000.
- "4"= Sieb+Meyer format-5000.

EXAMPLE:

A60 X00000 Y 00213 means: Receive serial data (=A60),
leading zeroes/6-decades (=Y00213),
1/1000mm=0.001mm resolution (Y00213),
Sieb+Meyer format-3000 (Y00213).

The way of entering the command is:

Press "A" then "60" then "X0" (not used) then "Y00213" and finally the "DATA-IN"-key.

Now the machine's serial interface will start receiving the data from the PC-system, provided the PC is set-up correctly.

MAPE PROGRAM-BANK

PC FRONT-END SYSTEM:

The no-nonsense, front end system.

- The Mape machines and front end systems can interface to all CAD formats in both MM and inches (0,01mm and 0,001" respectively).
- It can also interface to existing CNC production programs in SIEB & MEYER format 1000, 3000 and 5000 (from software version V5.0 or higher).
- The 'ProgramBank' front-end data storage and data-conditioning system is capable of storing large amounts of CNC production programs. It has powerful facilities for editing of CAD-files prior to production and serves as a central, off-line distribution system for the data flow control in a PCB-production plant.

The system runs on any standard IBM-compatible PC (right from XT to '486) with the ProgramBank software for carrying out the following tasks:

- *PROGRAM TRANSMISSION VIA DIRECT RS-232 LINE LINK,
- *PROGRAM DELETE FUNCTION,
- *ALPHABETICAL SURVEY OF CUSTOMERS AND JOB NUMBERS,
- *AUTOMATIC SEARCH FUNCTION,
- *READING OF CAD-PROGRAMS DIRECTLY FROM FLOPPY DISKS,
- *INPUTTING AND OUTPUTTING OF ANY FILE DIRECTLY ONTO USER DEFINED FLOPPY DRIVES (A and B).
- *POWERFUL EDITING FACILITIES,
- *NETWORK OPERATION FOR MULTI MACHINE-INSTALLATIONS,
- *NEW BACK-UP AND FORMATTING SYSTEM,
- *NEW EASY TO USE SET-UP MENU FOR THE INTERFACE.

- *NEW FILTER FUNCTION ALLOWS FOR OPERATOR MESSAGES OR COMMENTS/HEADERS DIRECTLY IN THE CNC-PROGRAMS.
- *NEW 'SHORTCUT' FEATURE BETWEEN ANY MENUS.
- *NEW DRILL ROUTE OPTIMIZER, SAVES UP TO 75% OF DRILLING-ROUTE IN CNC-MACHINES.
- *NEW "MACHINE DOCTOR" TROUBLE SHOOTING SYSTEM FOR DOWNLOADING OF MACHINE SOFTWARES AND REGISTERS & DIAGNOSTIC SYSTEM FOR MACHINE TESTING.

The ProgramBank-system forms a low cost and versatile way of saving CNC- and CAD-programs.

The system is totally menu-operated, and contains a new integrated HELP-menu system, which makes the system virtually self-explanatory to the operators.

PC FRONT END SYSTEM....

The ProgramBank allow CNC-programmers to save the production-programs in a PC, thereby avoiding all the problems with digital-tapes or paper-tapes which are sensitive, expensive and give rise to safety concerns.

HARDWARE REQUIREMENTS:

Any IBM-compatible PC (XT, AT) works with the Mape-software, and the Mape ProgramBank software works on any DOS-version 3.21 to DOS 6.0.

There should be minimum one floppy disk drive (preferably 3.5") and a harddisk with minimum 20MB (preferably <80MB) capacity.

Also, serial interface RS-232 (standard 25- or 9-pin connector) must be included in the PC.

INSTALLATION PROCEDURE:

The installation procedure only applies for installations of the Mape software in a PC, which has not been supplied by Mape:

- 1) Switch off your PC.
- 2) Insert the software protection device ("KEY-LOCK") in the PC's parallel connector (LPT1=printer port). If you have a printer connected to this port, please remove the connector, insert the protection device and connect the printer cable to the other end of the protection device. The parallel port is not affected by the key-lock device, and parallel operated devices can still be used.
- 3) Insert the software floppy disk in the 'A'-drive of the PC, and write "A:" and press <ENTER>.
- 4) When the prompt "A:>" appears on the screen, please enter:

"UKBANK" and press <ENTER>.
- 5) The PC will now go through the starting path and finally show the main menu of the Program-Bank software.

CABLE CONNECTION:

Now the cable to the CNC-machine should be installed in the serial I/O-port of the PC. Please make sure, that you chose the correct (serial) connector, normally "COM-1".

The connector must be a 25-pole CANON standard RS-232 connector. The cable name is "25-POLE ALL-THROUGH" and this can be purchased from all major PC suppliers.

NOTE: *If your PC is fitted with a 9-pole connector, you can insert a 9-to-25 pole adaptor between the PC and the cable, or you can use a 25-to-9 pole serial cable.*

WIRING SCHEME:

The wiring scheme is shown below:

PC-I/O PIN NO:	PC-SIGNAL & DIRECTION:	CNC-I/O PIN-NO:	CNC-SIGNAL & DIRECTION:	FUNCTION:
2	XMIT (output)	2	RECEIVE (input)	DATA.
3	RECEIVE (input)	3	XMIT (output)	DATA.
7	GND (ground)	7	GND (ground)	GROUND.
8	DSR (input)	8	RTS (output)	HANDSHAKE.
20	DTR (output)	20	DTR (input)	HANDSHAKE.

NOTE: *Use preferably a standard "25-pole ALL-THROUGH" PC-cable.*

HELP FILE:

The PC system is now ready and can be used together with the Mape CNC-machine(s), just follow the screen-instructions of the PC!

However, the Mape ProgramBank system can also be used successfully with machines of other manufacture. We have made several installations for other machines during the years. We shall be happy to inform you, how you can link various machines together and control your production flow conveniently via the Mape ProgramBank system.

For further information, please use the built-in service menus of the PC ProgramBank system, which is accessible during all operations, so you get an immediate explanation of the menus and options wherever you may be in the system.

NOTE: *A full manual is implemented in the software, and it is possible to call an "instant help file" for each individual menu- and submenu-point.*

BAUD-RATE:

However, it may be necessary to set-up the transmission speed etc., and this is done by selecting the internal "SET-UP" -menu of the Mape ProgramBank (press "S" for "SET-UP").

Select the desired baud-rate from the following table:

BAUDRATE:
110 Baud
150 Baud
300 Baud
600 Baud
1200 Baud
2400 Baud
4800 Baud
9600 Baud
19200 Baud (standard setting from 1995 onwards)
38400 Baud
56000 Baud

COMPORT SETTING:

Also the correct port must be selected, i.e. "COM-1" or "COM-2", which is also carried out by a simple selection from the SET-UP menu.

DELAY:

It is possible to set the values for number of stop-bits and dwell-timers to suit any application.

Sometimes, it is necessary to implement a delay after each character during the transmission to meet the requirements for certain machines, which may have problems in the receiver input/handshake output's reaction time.

To implement a delay, you simply select the "DELAY" field of the SETUP-menu which is normally set to 0, and change the value (in mS), but please make it as small as possible, in order to maintain a high transmission speed.

NOTE: *Just 1 mS delay will double the transmission time at 9.600 Baud!*

AUTOPARAMETERTABLE NO. 1:

DRILLING: CUTTING SPEED 150M/min, CHIPLOAD 45 uM/rev.

Ø mm	RPMx 1000	INFEEED mm/sec	UPFEED mm/sec	Ø mm	RPMx 1000	INFEEED mm/sec.	UPFEED mm/sec.
0,1	60	6	42	3,1	16	12	125
0,2	60	11	85	3,2	15	11	125
0,3	60	17	125	3,3	15	11	125
0,4	60	23	125	3,4	14	11	125
0,5	60	28	125	3,5	14	11	125
0,6	60	34	125	3,6	13	10	125
0,7	60	39	125	3,7	13	10	125
0,8	60	45	125	3,8	13	10	125
0,9	53	40	125	3,9	12	9	125
1,0	48	36	125	4,0	12	9	125
1,1	43	32	125	4,1	12	9	125
1,2	40	30	125	4,2	12	9	125
1,3	37	28	125	4,3	12	9	125
1,4	34	26	125	4,4	13	10	125
1,5	32	24	125	4,5	13	10	125
1,6	30	22	125	4,6	13	10	125
1,7	28	21	125	4,7	13	10	125
1,8	26	20	125	4,8	13	10	125
1,9	25	19	125	4,9	13	10	125
2,0	24	18	125	5,0	14	11	125
2,1	23	17	125	5,1	14	11	125
2,2	22	16	125	5,2	14	11	125
2,3	21	16	125	5,3	14	11	125
2,4	20	15	125	5,4	14	11	125
2,5	19	14	125	5,5	14	11	125
2,6	18	14	125	5,6	14	11	125
2,7	18	13	125	5,7	14	11	125
2,8	17	13	125	5,8	15	11	125
2,9	17	13	125	5,9	15	11	125
3,0	16	12	125	6,0	15	11	125

ROUTING, CUTTING SPEED 150 M/min., CHIPLOAD: LOW.

Ø mm	RPMx 1000	INFEEED mm/sec	FEEED mm/sec	UPFEED mm/sec	Ø mm	RPMx 1000	INFEEED mm/sec	FEEED mm/sec	UPFEED mm/sec
0,6	60	5	5	5	2,4	20	12	12	12
0,8	60	5	5	5	2,6	18	12	12	12
1,0	48	6	5	5	2,8	17	12	12	12
1,2	40	8	8	8	3,0	16	12	12	12
1,4	34	9	9	9	3,2	15	12	12	12
1,6	30	11	11	11	3,4	14	12	12	12
1,8	26	12	12	12	3,6	13	12	12	12
2,0	24	12	12	12	3,8	13	12	12	12
2,2	22	12	12	12	4,0	12	12	12	12

AUTOPARAMETERTABLE NO. 2:

DRILLING: CUTTING SPEED 150M/min, CHIPLOAD 60 uM/rev.

Ø mm	RPMx 1000	INFEEED mm/sec	UPFEED mm/sec	Ø mm	RPMx 1000	INFEEED mm/sec.	UPFEED mm/sec.
0,1	60	8	42	3,1	16	16	125
0,2	60	15	85	3,2	15	15	125
0,3	60	23	125	3,3	15	15	125
0,4	60	30	125	3,4	14	14	125
0,5	60	38	125	3,5	14	14	125
0,6	60	45	125	3,6	13	13	125
0,7	60	53	125	3,7	13	13	125
0,8	60	60	125	3,8	13	13	125
0,9	53	53	125	3,9	12	12	125
1,0	48	48	125	4,0	12	12	125
1,1	43	43	125	4,1	12	12	125
1,2	40	40	125	4,2	12	12	125
1,3	37	37	125	4,3	12	12	125
1,4	34	34	125	4,4	13	13	125
1,5	32	32	125	4,5	13	13	125
1,6	30	30	125	4,6	13	13	125
1,7	28	28	125	4,7	13	13	125
1,8	26	26	125	4,8	13	13	125
1,9	25	25	125	4,9	13	13	125
2,0	24	24	125	5,0	14	14	125
2,1	23	23	125	5,1	14	14	125
2,2	22	22	125	5,2	14	14	125
2,3	21	21	125	5,3	14	14	125
2,4	20	20	125	5,4	14	14	125
2,5	19	19	125	5,5	14	14	125
2,6	18	18	125	5,6	14	14	125
2,7	18	18	125	5,7	14	14	125
2,8	17	17	125	5,8	15	15	125
2,9	17	17	125	5,9	15	15	125
3,0	16	16	125	6,0	15	15	125

ROUTING, CUTTING SPEED 175 M/min., CHIPLOAD: MIDDLE.

Ø mm	RPMx 1000	INFEEED mm/sec	FEED mm/sec	UPFEED mm/sec	Ø mm	RPMx 1000	INFEEED mm/sec	FEED mm/sec	UPFEED mm/sec
0,6	60	6	6	6	2,4	23	16	16	16
0,8	60	6	6	6	2,6	21	16	16	16
1,0	56	8	8	8	2,8	20	16	16	16
1,2	46	10	10	10	3,0	19	16	16	16
1,4	40	12	12	12	3,2	17	16	16	16
1,6	35	15	15	15	3,4	16	16	16	16
1,8	31	16	16	16	3,6	15	16	16	16
2,0	28	16	16	16	3,8	15	16	16	16
2,2	25	16	16	16	4,0	14	16	16	16

AUTOPARAMETERTABLE NO. 3:

DRILLING: CUTTING SPEED 150M/min, CHIPLOAD 80 uM/rev.

Ø mm	RPMx 1000	INFEEED mm/sec	UPFEED mm/sec	Ø mm	RPMx 1000	INFEEED mm/sec.	UPFEED mm/sec.
0,1	60	10	42	3,1	16	21	125
0,2	60	20	85	3,2	15	20	125
0,3	60	30	125	3,3	15	20	125
0,4	60	40	125	3,4	14	19	125
0,5	60	50	125	3,5	14	19	125
0,6	60	60	125	3,6	13	17	125
0,7	60	70	125	3,7	13	17	125
0,8	60	80	125	3,8	13	17	125
0,9	53	71	125	3,9	12	16	125
1,0	48	64	125	4,0	12	16	125
1,1	43	57	125	4,1	12	16	125
1,2	40	53	125	4,2	12	16	125
1,3	37	49	125	4,3	12	16	125
1,4	34	45	125	4,4	13	17	125
1,5	32	43	125	4,5	13	17	125
1,6	30	40	125	4,6	13	17	125
1,7	28	37	125	4,7	13	17	125
1,8	26	35	125	4,8	13	17	125
1,9	25	33	125	4,9	13	17	125
2,0	24	32	125	5,0	14	19	125
2,1	23	31	125	5,1	14	19	125
2,2	22	29	125	5,2	14	19	125
2,3	21	28	125	5,3	14	19	125
2,4	20	27	125	5,4	14	19	125
2,5	19	25	125	5,5	14	19	125
2,6	18	24	125	5,6	14	19	125
2,7	18	24	125	5,7	14	19	125
2,8	17	23	125	5,8	15	20	125
2,9	17	23	125	5,9	15	20	125
3,0	16	21	125	6,0	15	20	125

ROUTING, CUTTING SPEED 200 M/min., CHIPLOAD: HIGH.

Ø mm	RPMx 1000	INFEEED mm/sec	FEED mm/sec	UPFEED mm/sec	Ø mm	RPMx 1000	INFEEED mm/sec	FEED mm/sec	UPFEED mm/sec
0,6	60	8	8	8	2,4	27	20	20	20
0,8	60	8	8	8	2,6	25	20	20	20
1,0	60	10	10	10	2,8	23	20	20	20
1,2	53	13	13	13	3,0	21	20	20	20
1,4	46	16	16	16	3,2	20	20	20	20
1,6	40	19	19	19	3,4	19	20	20	20
1,8	35	20	20	20	3,6	18	20	20	20
2,0	32	20	20	20	3,8	17	20	20	20
2,2	29	20	20	20	4,0	16	20	20	20

DRILLING, HOLE QUALITY AND DRILL LIFETIME:

It is of the greatest importance when producing PTH- and multilayer boards to control the drill bit lifetime.

There are no specific rules for the life time calculation, hence it is recommended to make some in-house tests.

There are certain factors which influence the tool life time:

- The drill-parameters (in-feed and RPM).
- The desired PCB material and quality.
- The drill bit manufacture and -type.
- The entrance board material.
- Run-out errors of the HF-spindles.
- The Z-axis stability.

When changing just one of the above factors, the resulting drill life time will change accordingly. Therefore, it is recommended to make a life time test procedure each time one of the factors or the combinations are altered.

Of course, such tests can be entirely omitted if the acceptable drill life time is set to a very low limit.

However, this is certainly not advisable due to the involved economical aspects. It is very important to know exactly when the drill bits MUST be exchanged or re-grinded because the toolcosts are quite an important factor in modern PCB-manufacture.

HIGH-SPEED DRILLING:

One of the most decisive factors for the drill life time is the revolution speed.

An excessive speed reduces the drill life time due to too much heat generated during the drilling sequence.

However, the drills can be used in a so-called 'high-speed range' i.e. with a peripheral speed of approx. 200M/min and a corresponding increased feedrate (in order to maintain the same hole quality).

The greatest advantage of high speed drilling is the 25% increased feedrate, which will give approx. 25% increased yield in an existing production without sacrificing hole quality.

The drawback is the reduced drill life time, which will be reduced approx. 50%.

A simple calculation can reveal, if high speed drilling is a good business or not

Continued...

The revolution speed of the HF-spindles is the single most decisive factor, because it is most important to get the most frequently used drill-sizes into the high speed range.

If the maximum revolution speed of the spindle is 60.000RPM, it is impossible to achieve high speed drilling at drill sizes less than 1.0mmØ, because a Ø0.8 drill bit would demand 80.000 RPM, when running with high speed parameters.

DRILL LIFE TEST:

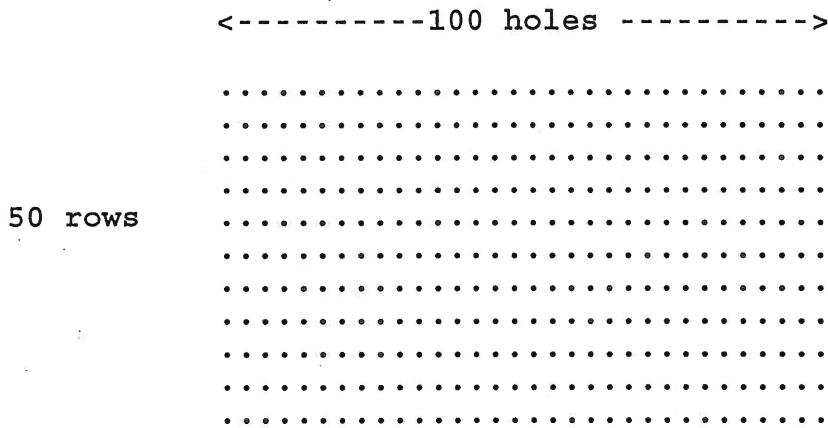
The life time test is carried out by making a simple drilling program in which 5000 holes are drilled in a 0.05" (1.27mm) grid.

Use a standard stack of PCB's as normally used in the factory.

NOTE: ALL materials and equipment, i.e. drillbits, parameters and CNC-machine(s) must be the normally used.

The program might be made like this:

EX.:



If you choose 1/20" (=1.27mm) grid, the resulting stack size is approx. 115x210mm (=4.5"x8.25") if we assume that we want 25mm (=1") unused material along the left/right sides and approx. 40mm (=2") in each end for placing the tooling pins.

The run time will be approx. 20-25 minutes, depending on the selected drill size and speed table. We recommend to use a 0.8mm drill bit because this is one of the most common sizes in the industry.

Continued...

2.

After the test has been carried out, the result will clearly show the degrading hole quality as an inverse function of the increasing hole quantity.

It is now possible to examine the hole quality by using the standard casting and grinding process in order to have a microscopic view of the hole walls.

Hence it can be established at which number of holes the quality is still acceptable for the following PTH-process.

The test result should show a number between 3000- and 5000 holes in FR-4 material. If this number is lower than 3000 holes, it is strongly recommended to examine the case closer in order to find the reason for the excessive drill wear.

Normally, you should find that the hole quality is still acceptable at around 5000 holes.

=====

TABLE no. 1, Chipload 45 $\mu\text{m}/\text{RPM}$ (Low)

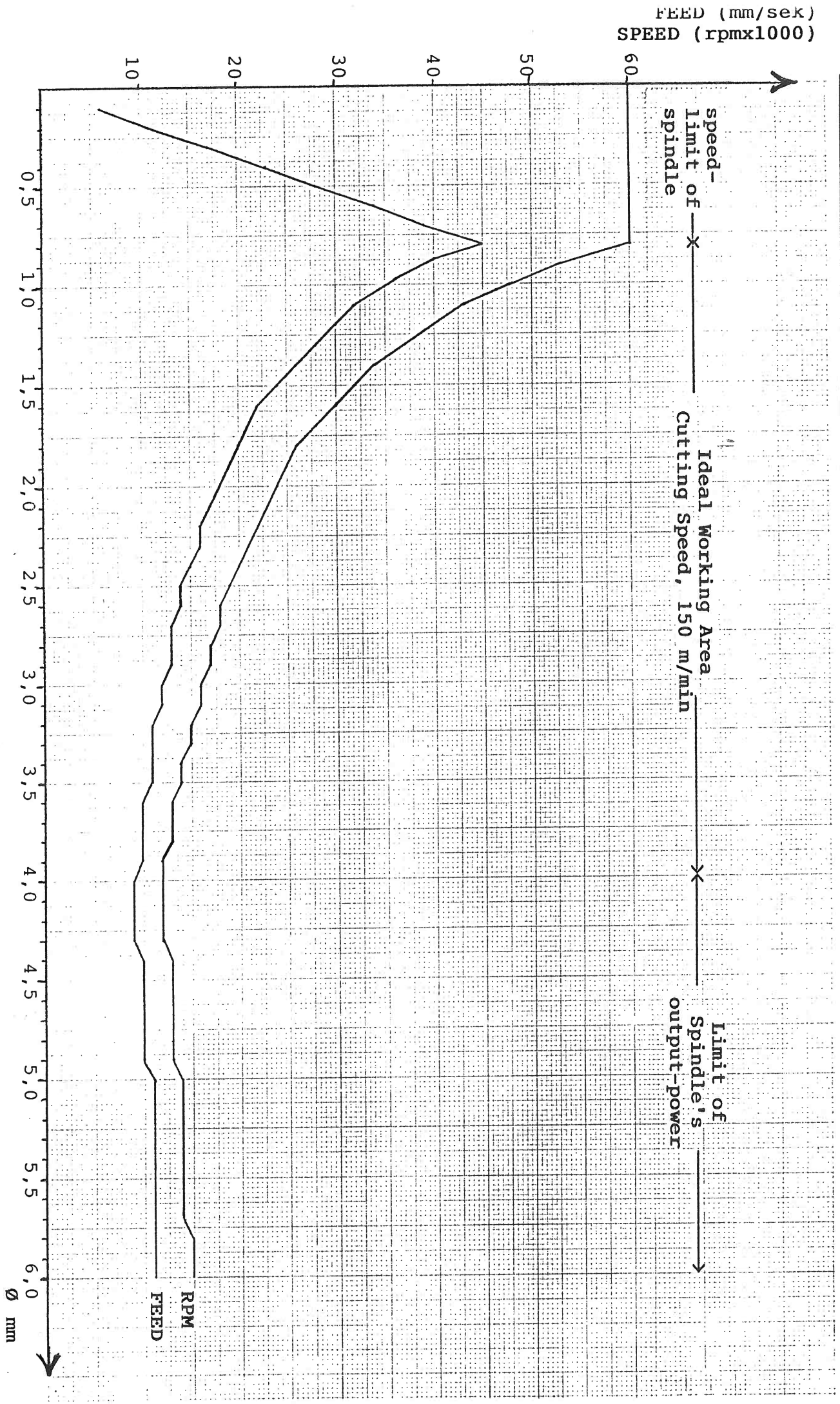


TABLE no. 2, Chipload 60 μm / RPM (medium)

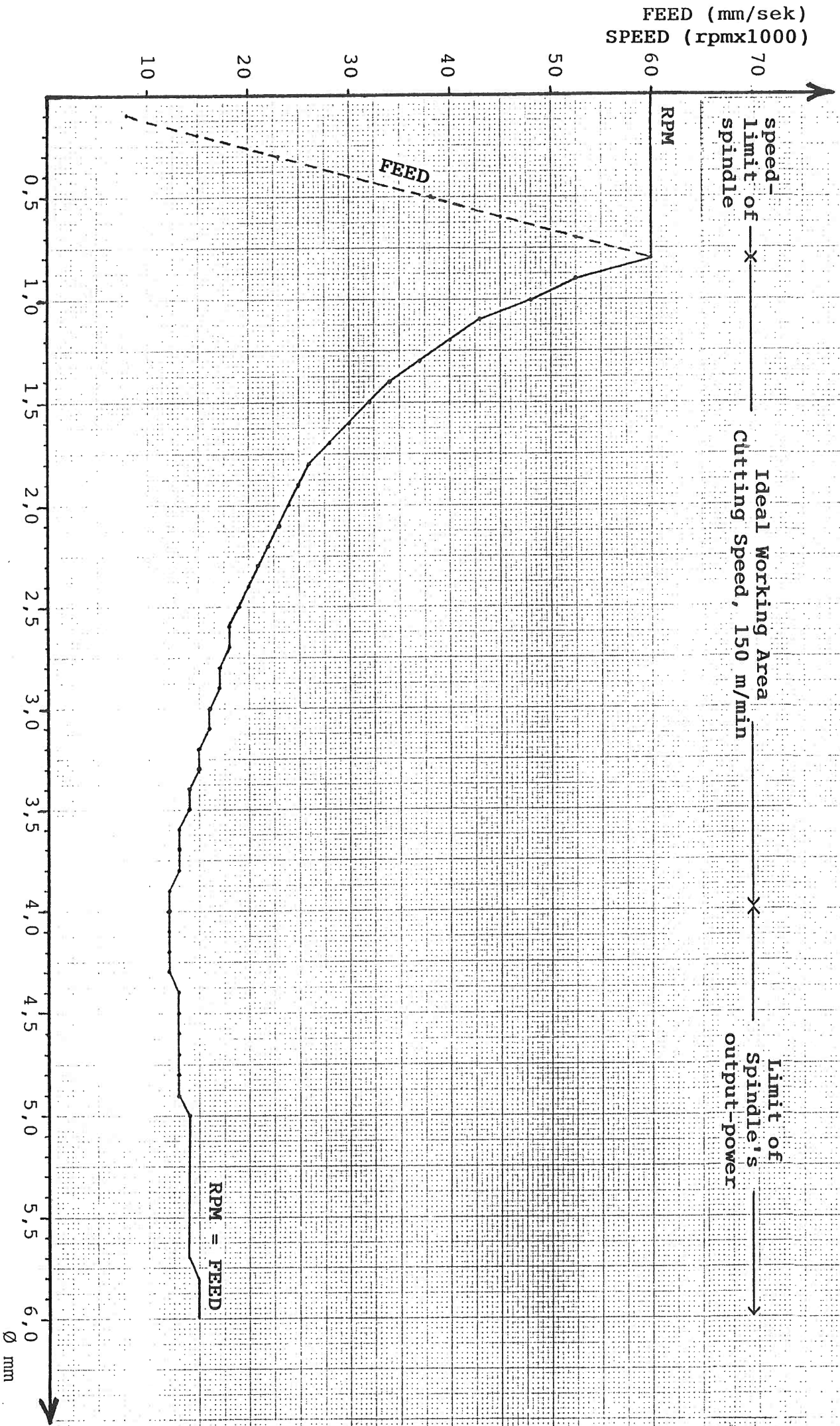
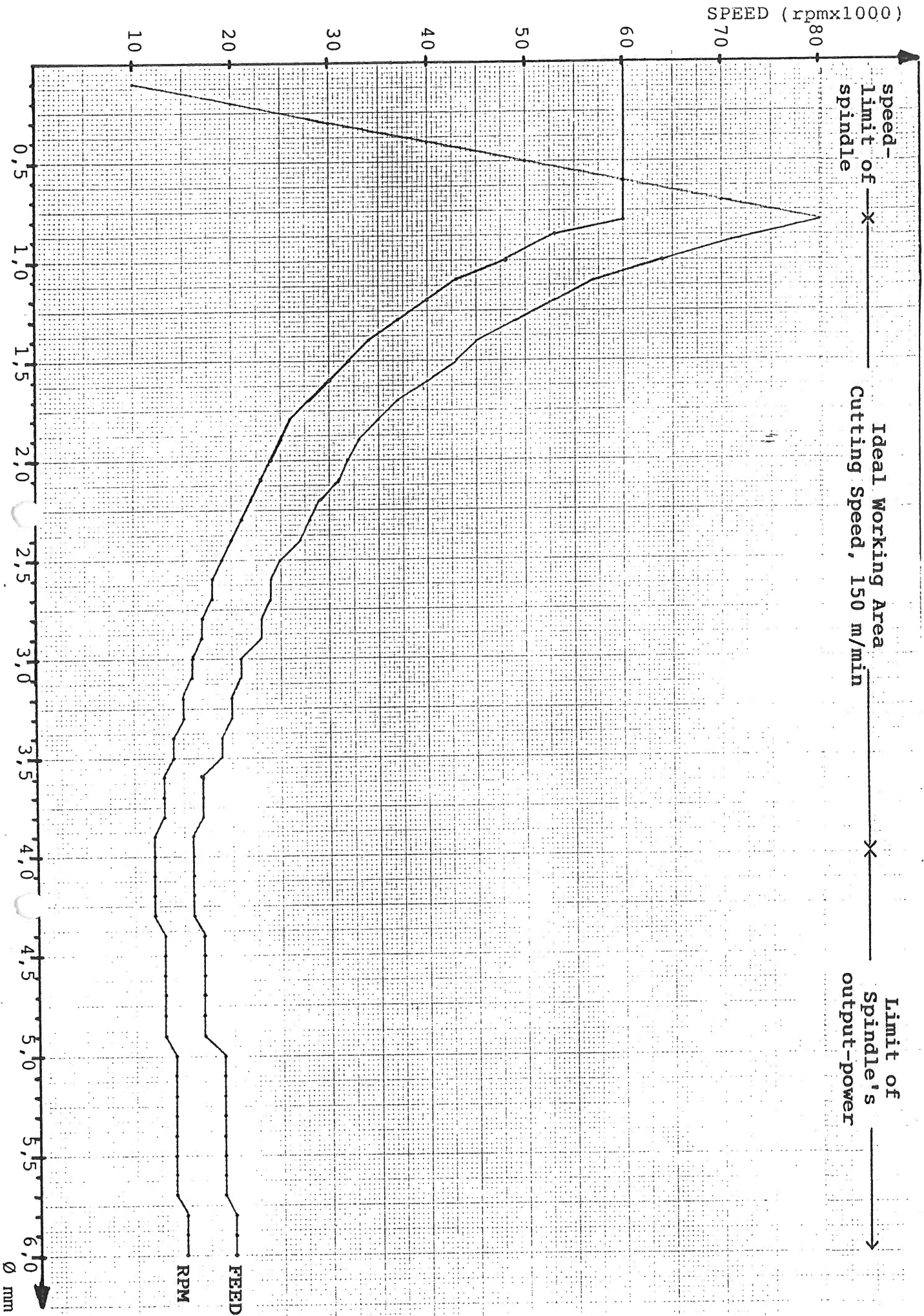
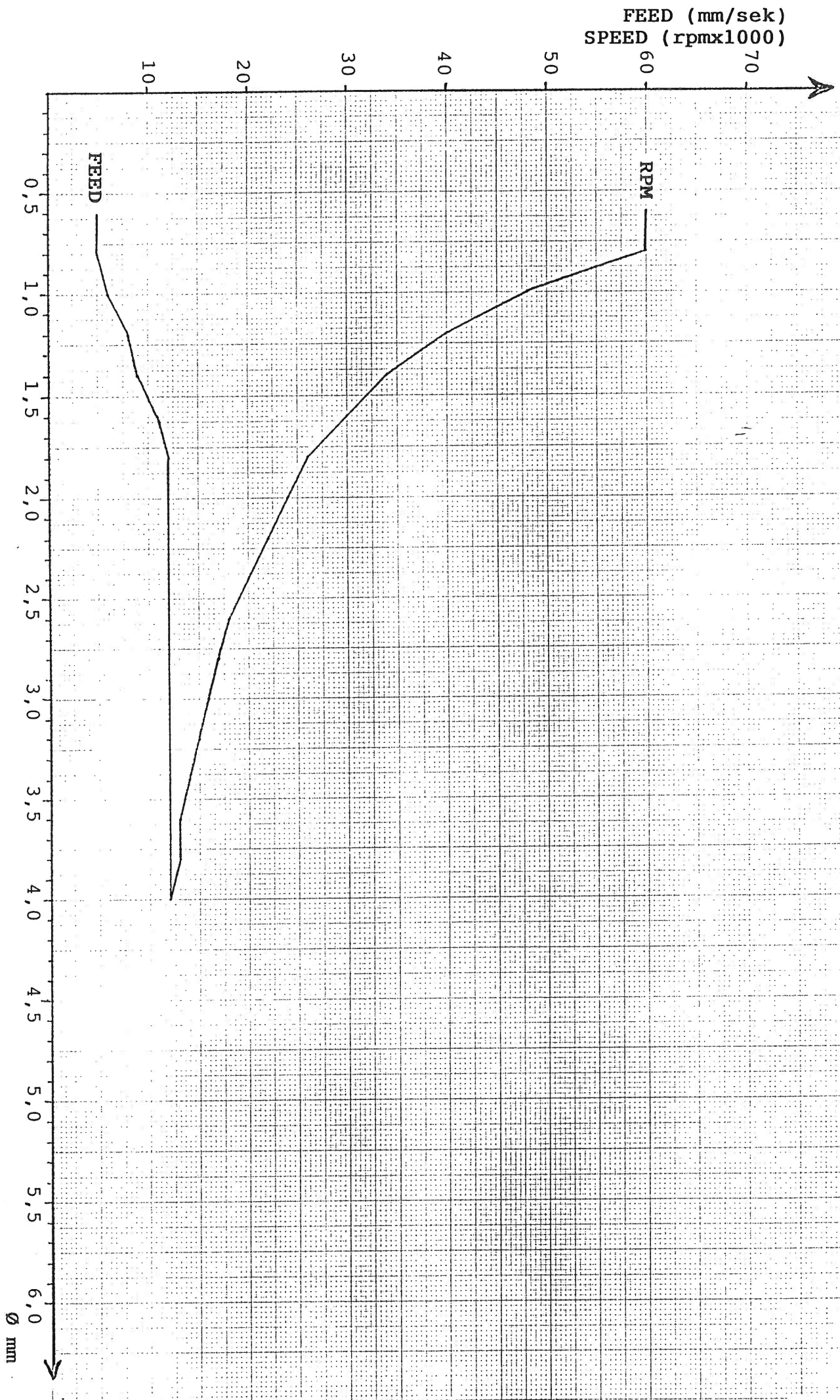


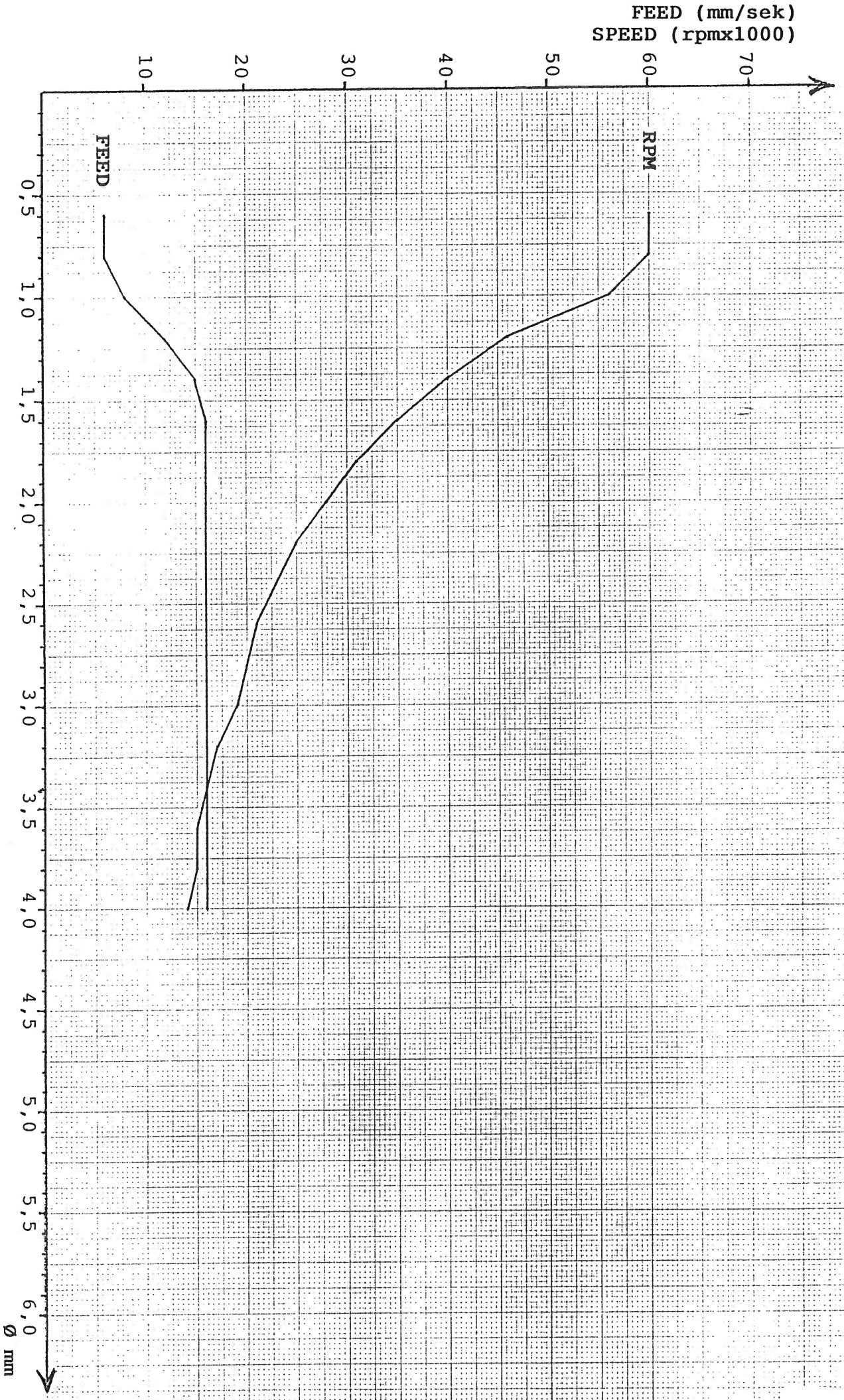
TABLE no. 3, Chipload 80 $\mu\text{m}/\text{RPM}$. (high)



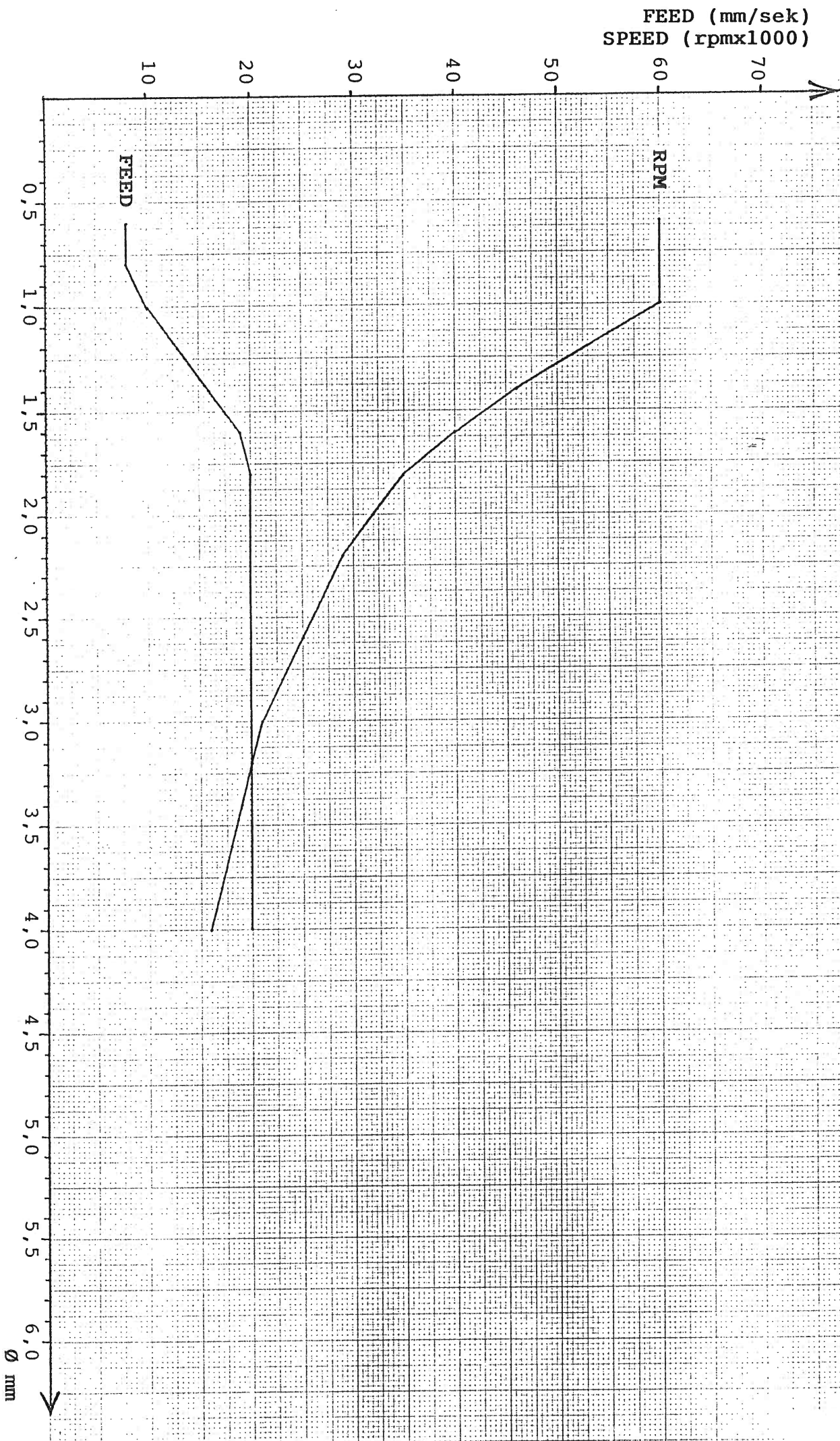
TABEL 1: 150 m/min, LOW FEED



TABEL 2: 175 m/min, MEDIUM FEED



TABEL 3: 200 m/min, HIGH FEED



The use of the automatic Laser-measuring system.

1.0 AUTOMATIC/MANUAL OPERATION:

During normal production, i.e. in "automatic" mode, the laser system will automatically check the diameter and length of the tool in operation.

Sections 2.0 and 3.0 describe the other ways of (manually controlled) operations.

- 1.1 The machine has, as the standard setting "Automatic" mode. The machine picks the desired tool in the normal way, and then travels to the position for laser measuring. From this position, the machine will make one run through the laser device in order to measure the diameter of the tool in the spindle, and then reverse the direction and run back through the laser device while lifting the Z-axis (spindle) to check the length of the tool..

If the desired tool's diameter and length complies with the size specified in the program, and if it is within the selected tolerances specified by the operator, the machine will proceed with the tool in the other spindle (2-spindle machines only), and carry out exactly the same procedure for the second spindle.

1.2 ERROR MESSAGES:

If the laser system detects a tool that is beyond the desired tolerance, the machine will stop and display one of the following error messages on the control display:

- >> **DIAMETER TOO LARGE** << (Tool diameter is too large according to the CNC-program)
- >> **DIAMETER TOO SMALL** << (Tool diameter is too small according to the CNC-program)
- >> **NO TOOL DETECTED** << (The tool is missing in the magazine or it is broken)
- >> **TOOL TOO LONG** << (The tool is longer than +0.5mm over tolerance)
- >> **TOOL TOO SHORT** << (The tool is shorter than -0.5mm under tolerance)

1.3 MEASURING RANGE:

The machine is factory adjusted to accept +/- 40uM (i.e. +/-0.04mm) as the maximum acceptable deviation from the diameter indicated in the CNC-program.

However, it is possible for the operator to change the factory setting of +/- 40uM to another desired value by changing two internal registers:

“INTERNAL REG A23 X (- tolerance) Y (+ tolerance)”.

The tool-length tolerance is factory selected to +/- 0.5mm, which ensures, that it is impossible to penetrate the work table of the machine. On the other hand, it also ensures that the PCB-stack is fully penetrated, provided the correct exit board thickness of 2mm is used.

The factory selected tool-length tolerance of +/- 0.5mm is not programmable by the operator, because it is laid down in the machine's software for security reasons.

The machine's nominal tool-length is in accordance with the industry norm of 0.8" (=20.32mm). Hence the machine will accept tool-length deviations in the interval from 19.82mm to 20.82mm.

If the machine display the following error messages:

>>TOOL TOO LONG<< or **>>TOOL TOO SHORT<<**

there is only one cure:

The operator must put the tool back into the tool magazine, remove it and adjust the tool in the ring-setter to the correct length, and finally insert it again in the machine's tool-magazine .

1.4 MEASURING RESOLUTION:

The machine's unique laser system makes it possible to measure the tool-diameter within 1uM (0.001mm) resolution.

This is achieved by using an entirely new type of laser device, which is constructed to work closely together with the machine's computer.

The computer accurately analyses the result of the measuring procedure of the laser device via the unique "AUTOTRACKING" system, while the tool is passed through the laserbeam.

In addition, the computer is automatically compensating for the ageing of the laser and for decrease of the laser intensity due to dust-contamination of the laser lens-system etc.

This means, that the accuracy of the Mape laser system is maintained over a long period of time.

1.5 MEASURING RESULTS:

It should be noted, that all measuring results includes run-out failures of the spindles. If a drill-bit shows too large diameter, it can sometimes cure the problem, to put the tool down in the magazine, then turn it 90 degrees and ask the machine to pick the tool up again and let it make a new measuring procedure. In many cases, the tool can now be accepted within the tolerance.

However, if this happens frequently, it suggests that the collet needs cleaning or renewal. This phenomenon might as well be caused by an awry drill bit (factory defect) or when the cutting edges are contaminated with copper or cutting-particles.

Especially, thin tools with extended spirals will have a tendency to vibrate slightly, and this often leads to "unsteady" measuring results. A thin drill bit with a short spiral will not suffer from such vibrational problems. Hence the short type is preferred because it will produce steady and reliable measuring results.

These phenomenonons are physical determined, and has nothing to do with the laser device or the machine.

If the machine display the error message: **>>DIAMTER TOO LARGE<<** during the normal production (i.e. auto-mode), it is desirable to check the tool-diameter "manually" using the machine's laser-system. For this purpose, please refer to the next section "2.0".

1.6 BROKEN TOOLS:

After using each tool, the machine will automatically check that the tool has its full length before it is returned to the tool-magazine. If this is not the case, the machine will display the following error message on the control display:

>>TOOL BROKEN<<

In this case, the operator must check the reason for the broken tool, remove the broken tool(s) and insert new tool(s) in the corresponding tool magazine(s). Also, he must find the place where the tool broke and remove the remaining tool-fragments and continue the work again.

1.7 ENABLING/DISABLING OF THE AUTOMATIC LASER MEASURING:

It is possible to disable the automatic laser measuring sequence by entering a code "A23" anywhere in a program. The code for disabling is "A23 X..... Y0", after which the laser measuring will be cancelled.

If it is desired to enable the laser measuring again later in a program, a new code A23 must be entered: "A23 X..... Y1". This means, A23XY1=ON and A23XY0=OFF.

2.0 MANUAL OPERATION:

NEW KEYBOARD FUNCTIONS:

2.1 TOOL UP / TOOL DOWN:

The machine is now equipped with new useful functions. It has been necessary to implement an automatic pneumatically operated retraction system for the clamping feet.

This function is utilized by the laser device during the measuring sequence, because it is necessary to retract the clamping feet due to the accuracy.

This function can be activated manually by means of the **TOOL-UP** and the **TOOL-DOWN** keys.

If no tool-number is entered prior to the activation of the **TOOL-UP** key, this means that the clamping feet are retracted to their top position. Activating the **TOOL-DOWN** key, means that the clamping feet are lowered to their (normal) bottom position.

The function described above, can be used for inspection purpose, i.e. to check a tool rotating in the spindle or to check whether or not there are tools in the spindles.

2.2 DIA? -KEY:

Pressing the "DIA?" -key means that the machine will start measuring the 1st (left) spindle's tool and display the result in the display in 0.01mm resolution. Immediately after the read-out of the first tool's result, the machine continues with the 2nd (right) spindle's tool, and finally displays the result of the 2nd tool's measuring result.

If there are no tools in the spindles, the machine will stop and display the following error message in the control display:

>>NO TOOLS DETECTED<<

After this message, the tool(s) must be inserted in the spindle(s) by the operator in the normal way.

If the operator enters a digit (either 1 or 2, for left or right respectively) prior to the activation of the "DIA?"-key, the machine will measure only the corresponding spindle's tool and display the result (both diameter and length) in the control display.

E.g. if the operator activates "1" + "DIA?", the machine will measure the 1st (= the left) spindle's tool and display the resulting diameter and length in the display.

2.3 DISPLAY READ OUT:

The machine is normally showing the measuring results (diameter and length) in 1/100mm (0.01mm). Even though the diameter is always measured in μM (0.001mm), the results are presented in 0.01mm in order to ease the reading of the measuring results.

However, it is possible to change the read-out from 0.01mm to 0.001 (μM) by entering "SHIFT" + "DIA?". Now the machine will display the results in μM (0.001mm) until the "SHIFT" + "DIA?"-keys are activated again or until the SYSTEM RESET key is pressed or the machine is switched off.

During the μM (0.001mm) read-out mode, the actual laser modulation signal amplitude is also shown in the display (as described in section 3.0 below).

LENGTH:

When the machine displays e.g. "LENGTH +0.05" in the control display, this means that the tool-point is 0.05mm lower than the theoretically correct measure of 20.32mm (0.8"). In the example given above, the tool is 20.37mm measured from the top of the tool-ring to the tool-point.

3.0 SERVICE FUNCTIONS:

3.1 SHIFT + DIA?:

This function serves two purposes. Firstly it changes the read-out of the measuring results from 0.01mm to 0.001mm (uM).

Secondly, it displays the actual value of the laser modulation signal in Volts. The laser modulation level can be used to determine the present status of the laser system, e.g. if the system is contaminated or prematurely aged or the like.

3.2 MAINTENANCE/SERVICE:

No special maintenance is required for the laser system. It is only necessary to clean the lenses from time to time, using a soft cotton wool sticker moistured with a little window cleaner fluid. Do not use any solvents or strong fluids. If in doubt, use no fluid at all..

NOTE: Do not -under any circumstances- try to disassemble the laser unit or use compressed air during the cleaning. It is not allowed to adjust the trimming potentiometer on the laser-board assy, because this will lead to immediate destruction of the laser system!

3.3 SERVICE FUNCTIONS:

“99” + “DIA?”: Updates automatically the X-position of the laserbeam into the internal register. This function is normally only used by the manufacturer for the initial adjustment of the laser system. However, it might be necessary for a customer to use this function, if an exchange of the laser-head should become necessary.

“91” + “DIA?”: Causes the machine to measure the 1st spindle (left) in an infinite loop until the “STOP” -key is pressed. This function is used by the manufacturer for the initial adjustment the laser system.

“92” + “DIA?”: Causes the machine to measure the 2nd spindle (right) in an infinite loop until the “STOP”-key is pressed.

3.4 TROUBLESHOOTING:

If the decimal read-out becomes gradually more unstable (random decimals), this is most likely due to contaminated lenses, which should be cleaned as described above.

However, the machine will not be very sensitive to contamination because of the special software controlled system used in the machine. This system ensures full control over the signal levels and makes the machine capable of automatically adjusting and updating the laser system within relatively wide limits, which automatically compensates for possible changes in the signal levels during the years.

The laser light is modulated with a very high frequency which makes the system insensitive to other light sources, sun-light or reflections etc.

The laser light is harmless, because the emitted laser power is less than 1mW @670nm, and complies with the safety norms for CLASS-2 laser systems.

However, do not stare directly into the laser-beam!

4.0 SPLIT AXIS DRILLING:

New feature for processing of giant size boards.

From May 1998 onwards, the Mape CNC-2000L machines have been equipped with a new software, that enables you to drill extremely large panels.

It is possible to set up the machine to act like a giant single spindle machine. In this “split-axis”-mode, the software automatically detects in which half area the hole positions are located, and the left spindle only drills the holes in the left area, whereas the right spindle automatically detects and drills the holes in the right area of the machine.

The operator does not have to worry about anything, since the detection and execution is fully automatic and tightly controlled by the CNC.

The codes for enabling/disabling the automatic split axis drilling action (or “single spindle mode”) is:

A37 X.....Y1 (split axis drilling action ON),
A37 X.....Y0 (split axis drilling action OFF)

NOTE: Routing with split axis operation enabled:

Split axis routing operation is only possible for routing patterns that does not pass through the centerline between the two work-areas!

5.0 DISABLING SPINDLES:

It is also possible to control in the program, which spindle should be active or deactivated. This can be desirable if only one stack of boards is processed in the machine.

The code for disabling/enabling a spindle is A32, and this code may be inserted anywhere in a program, e.g.:

“A32 X1Y0” means left spindle active, right spindle disabled.

“A32 X0Y1” means right spindle active, left spindle disabled.

”A32X1Y1” means both spindles active, which is the default value if no code A32 is entered in the program.

(End of file Manual97.doc 5.8.98...)

Program-parts can also be made temporarily "invisible" by the use of brackets and FILTER.

The FILTER-function also removes all unnecessary spaces, and thereby increases the transmission speed.

If FILTER is active, it is displayed on the screen when transmission starts.

Technical:

When the computer is put into transmission mode BEFORE the machine is prepared to receive, it is important that the computer goes into waiting-position. This can be performed as a test, that the wiring is OK. If the computer starts to send "out in nowhere", it is an indication that the wiring is incorrect.

Normally it will then be the HANDSHAKE-signal that fails. The transmission software uses DTR-input only as handshake.

The symptoms of an incorrect wiring are lost characters, causing program-blocks to be corrupted randomly in the memory of the machine after the program has been transferred.

Reception of programs is carried out via the COMPORT, and with the settings selected under the SETUP menu.

The reception of data can be stopped by means of the ESC-key. If the ESC-key is activated during the transmission, the program will NOT be saved in the ProgramBank, because the system consider this a cancellation of the transfer in progress.

If any transmission errors are detected during the transfer of data, the transmission in progress will be stopped and cancelled. The system displays a corresponding fault message on the screen. The program will not be saved in this situation.

If a switch-box for data distribution is connected between the ProgramBank and one -or more machines, the switch-box has to be set in the right position, BEFORE the ProgramBank or the machine is made ready to exchange data.

When the machine is put into transmission mode BEFORE the ProgramBank is put into reception mode, the machine MUST be waiting for the clear signal from the PC.

If the machine starts transmitting immediately, before the PC is ready to receive, it is a clear evidence, that the cable-connection is incorrect or faulty.

A faulty connection results in loss of characters. However, this will normally be detected by the ProgramBank, and a corresponding alarm message will be displayed.

The delete-function is equipped with a safety-text, to avoid accidental deletion of programs.

Once a program has been deleted, it cannot be restored again.

The Program Survey has 3 functions:

- 1) If a COMPANY NAME is entered, or the Enter-key is activated at the proposal given by the ProgramBank, all PROGRAMS referring to that COMPANY will be displayed together with the total number. If all the programs cannot be displayed at one time, the next page appears by activating the ENTER-key.
- 2) If the letter "C" is entered as company name, the ProgramBank will display all company names, which has been established in the system. The number of names will also appear on the screen.
- 3) If the letter "P" is entered as company name, the ProgramBank will start a printing procedure, which will print all COMPANY NAMES in the system with their corresponding PROGRAMS. However, the demand for large amount of hard copy paper does not tempt to use this function very often.

There is a maximum limit of 1000 COMPANY NAMES in the system, and a limit of 1000 PROGRAMS under each company name.

The OPTIMIZER is a quite harmless function, that does not touch anything, that it does not understand.

Consequently, it does no harm to let the optimizer "give it a try", just to see if a program could be optimized.

If the program contains anything, that is unknown to the optimizer, it will automatically stop. The first unaccepted program-line will then be displayed, including the line number. This makes it easy to find the line in the EDITOR, where the correction can be made.

The optimizer is able to optimize a complete MAPE program with everything, whereas new programs (e.g. CAD-programs) in general, must be cleaned-up by means of the EDITOR, before the optimizer accepts them.

The program contents accepted by the optimizer is limited to %-sign, A-codes, X/Y coordinates and T-numbers.

The optimizer adds automatically %-sign, A03 and A17 codes to the program, if these are not already present.

If the optimizer was able to reduce the drill-route, the improvement will be displayed in percent, and the optimized program saved in the system.

If no improvement was possible, this will be displayed instead, but then the original program will remain unchanged in the system.

The optimizer is capable of handling programs up to 10.000 program lines. If the program-size exceeds this maximum, it results in an error-message on the screen, and a cancellation of the OPTIMIZING procedure.

The OPTIMIZER in particular benefits from a fast PC, because of the complexity involved by this kind of function.

It will soon be possible to install a TIGA-GRAPHIC card, which will be able to draw the programs on a colour screen with very high speed and resolution. The program job can be displayed in full including all step & repeats, routing and so on.

The new card offers a lot of useful and easy functions for fast and reliable adaptation of e.g. CAD-programs. The system is able to show the operator exactly what he is up against and what he is doing while preparing the job.

The TIGA-GRAPHIC card must have a separate SVGA-screen, and then the Program-Bank will constitute a dual-screen system, with separate monitors for text and drawing.

Installation of this OPTION will more than double the performance of the ProgramBank system.

READING OF FLOPPY DISK can be accomplished from either the A -or the B drive.

DIRECTORIES may appear on a diskette, and therefore the system is designed to cope with this situation.

If DIRECTORIES exist, they will appear on the screen as white bars, specially highlighted. They have been given a number that can be selected, just like the ordinary files.

If a DIRECTORY is selected, the content of that directory will be displayed on the screen. This directory may also have SUB-DIRECTORIES, which again are specially highlighted etc.

At the same time, it is indicated, that 0 can be selected if you want to go one step backwards in the DIRECTORY-system to the previous DIRECTORY.

The EDITOR is a highly advanced tool, which at a first glance, does not appear to be anything special.

However, the EDITOR is in fact so powerful, that it would be too much to describe all the details in this text, therefore we shall just mention a few:

It can search for codes (CTRL-Q F),
It can convert codes (CTRL-Q A),
It can jump to a specified line-number (CTRL-J L+number).
It can copy a marked program-section to other places in the program.
It can swap X- and Y-values.

Please refer to the manual for the EDITOR for more details.

This function converts a program from CNC 100-R to CNC 2000 format.

All coordinates are re-calculated from 1/2000" to 1/100mm and the zero-point is adjusted to fit at X13000 Y500.

At the same time, the codes 06,16,26,23,37,35 are removed from the program, and codes 14 and 33 will get a toolpod-number added to the X-field.

The code 36, which is the Z-axis lifting level, will be multiplied by 10, and the codes 00,07,08,11,17,21,24,29,34,40 will be zeroed in X and Y.

The toolsizes will NOT be multiplied by 10. They have to be checked for correct sizes.

After conversion the file shows the text: (CONVERTED FROM 1/2000" TO 1/100mm) in the very beginning.

The baudrate is the transmission speed, by which the program-transfer is made. It is stated and entered as data-bits per second.

The ProgramBank can be set-up for the following transmission speeds:

110
150
300
600
1200
2400
4800
9600 < is the normal setting.
19200

If all the connected machines can accept a transmission speed of 19200 baud, it is highly recommendable to use this setting, due to the decrease in transmission time, which can be reduced by fifty percent compared to the standard setting of 9600 baud.

All MAPE models CNC-1000 and CNC-2000 can be set for 19200 baud.

This is accomplished by adjusting the internal register 26, in which the X-value is set to 1 and the Y-value likewise 1.

The parity-bit is an additional bit, which is transmitted after the 5, 6, 7 or 8 bits which form the transmitted character.

ODD, EVEN or NO PARITY can be selected.
The normal setting is EVEN.

The additional parity-bit forms together with the normal 7 data-bits, a 8-bit data set, which always has even parity.

The number of databits can be selected to 5, 6, 7 or 8.

The normal setting is 7 databits.

The number of stop-bits can be selected to 1 or 2.

The normal setting is 2.

The stop-bit is the last bit which is transferred in a data, and can be considered as a pause between the individual data-characters.

2 stop-bits instead of 1, results in a larger pause between the individual characters, thereby decreasing the transmission speed. However, it will increase the safety margin during the transmission.

All MAPE machines accepts 1 stop-bit without any problems.
The 8th-bit value can be selected as continuously '0', continuously '1' or variable.

The normal setting is variable.

The 8th-bit is the most significant bit in the transmitted data, and the value of the 8th-bit, only has significance if the data length are set-up to 8 bits.

The delay-time is selectable from 0 to 128 milli-seconds (mS).
The normal setting for MAPE machines is 0.

The delay-time is a pause or delay in milliseconds between the TRANSMITTED characters, and should only be used if there is a transmission speed problem in the RECEIVING END.

If the HANDSHAKE signal at the receiving end does not produce a STOP-level in due time, an over-run of characters will occur, which again leads to loss of transmitted data.

Based on transmission tests, the delay must be selected to the smallest possible value. The reason for this is, that even a small delay-value, drastically increases the the transmission time: A delay-time of just 1mS will double the transmission time at 9600 baud!

The FILTER can be selected to active (ON) or inactive (OFF).

The normal setting is inactive (OFF).

If FILTER is "ON", it is active only during "SEND TO MACHINE" operation, and all that has been placed in brackets () in the program, will not be sent to the machine in this case. Space-characters are also ignored.

This function makes it possible to place comments directly in the programs, e.g. operator messages, tooling informations etc., without increasing the transmission time or disturbing the machine.

Also program-parts that temporarily have to be "invisible" to the machine, can be so using brackets and filter "ON".

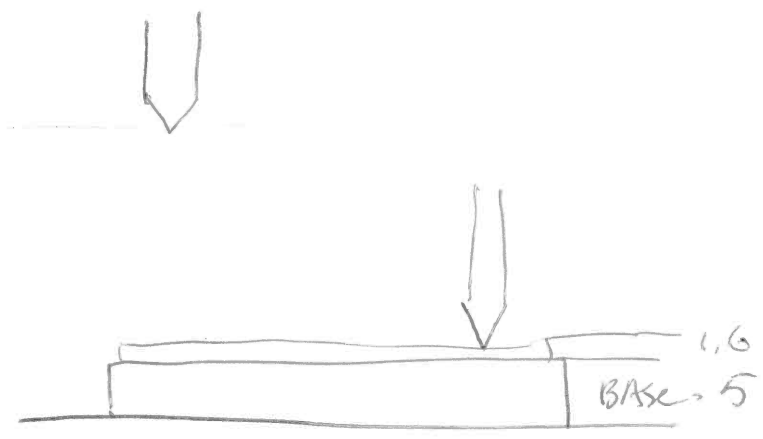
The COMPORT can be selected to either COM1 or COM2.

The normal setting is COM1.

If the data transmission does not take place when specifying COM1, please try COM2.

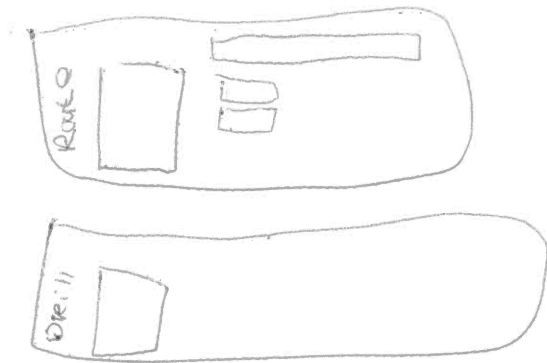
Normally, COM2 is a 9-pole male connector, whereas COM1 is a 25-pole male connector. In PC's with only one COMPORT, this is always COM1.

The networking to a MAPE machine is easily done by purchasing a standard, shielded 25-pole interface cable with one male and one female connector, a so-called "25-ALL-THROUGH" cable.

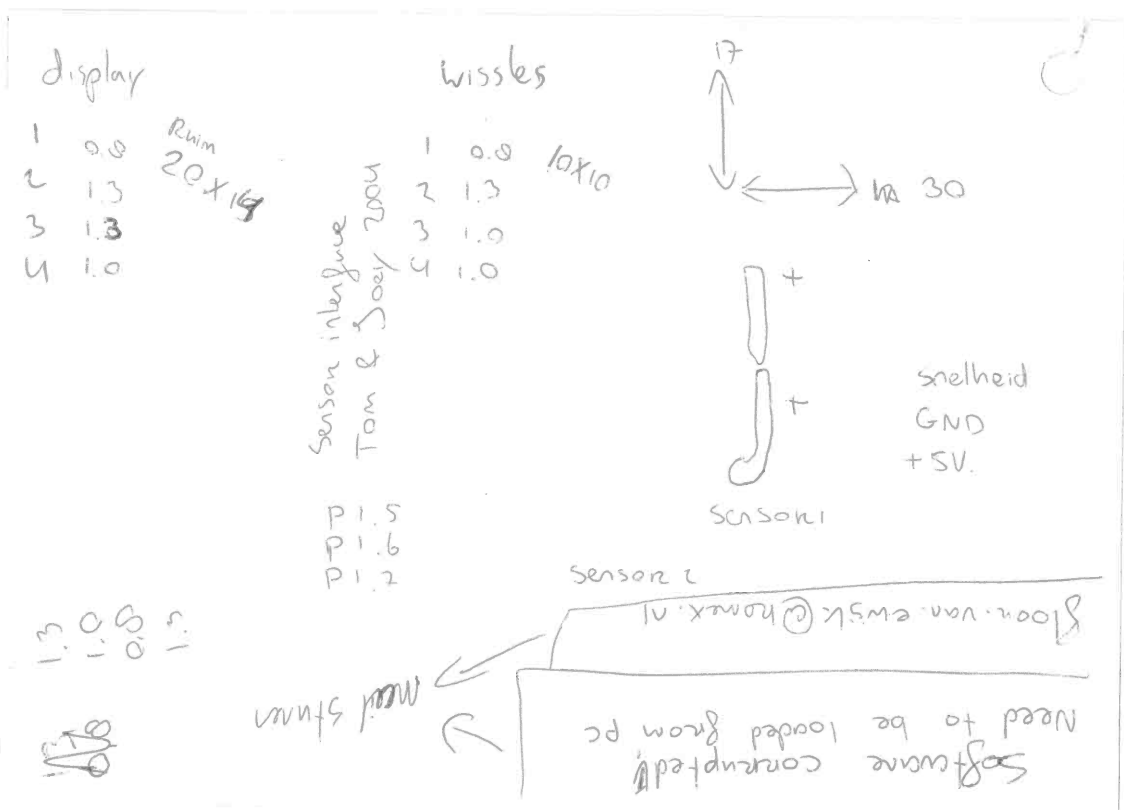


Mape 2005 → MDA elektronica

version 1.0 II

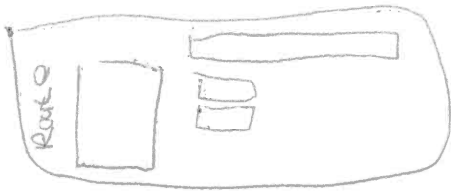


Waar is de help file?



Mape 2005 → MDA elektronica

Version 1.0 II

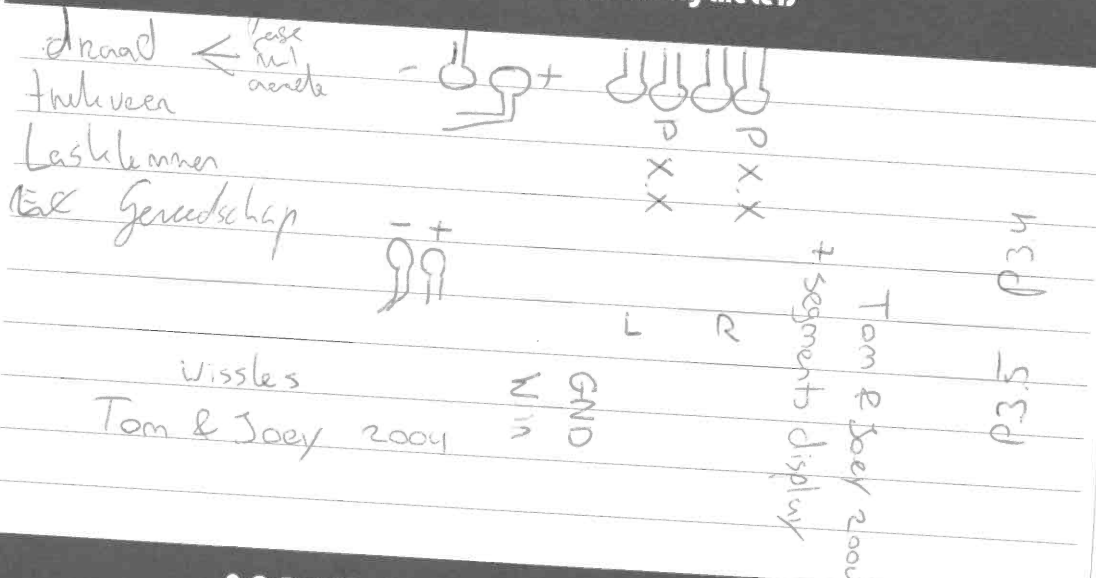


Waar is de help file?

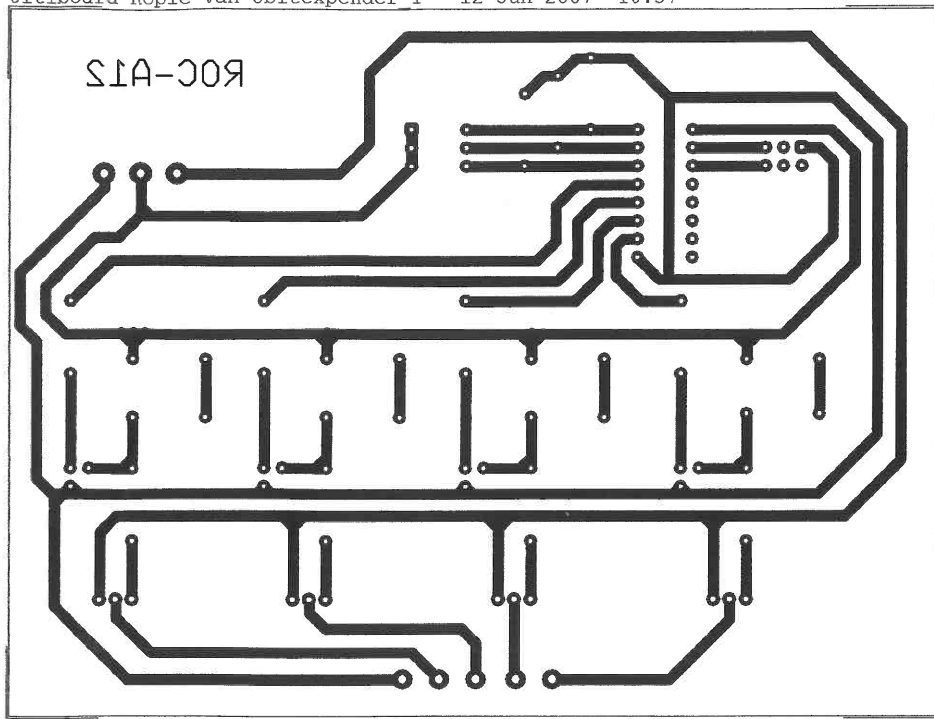
www.dmmetering.com

Dmmetering
Manufacturer of electricity meters

info@dmmetering.com



P.O. Box 92, 2450 AB Leimuiden, The Netherlands
T +31 (0) 172-506815 - F +31 (0) 172-507983



STOFZAK

easy clean papierose

dubbelslag

stoffelte type P10

art. n. 84640010

STOFZUIGER

Industrial Cleaner

n 19154 PI

type AS10

MERK: QHIBLI

Siemens Romex

Reiniger 5

3911 T2 Rheun

#loor van Ewijk.

leverancier

reusmachin

MAPE CMC I

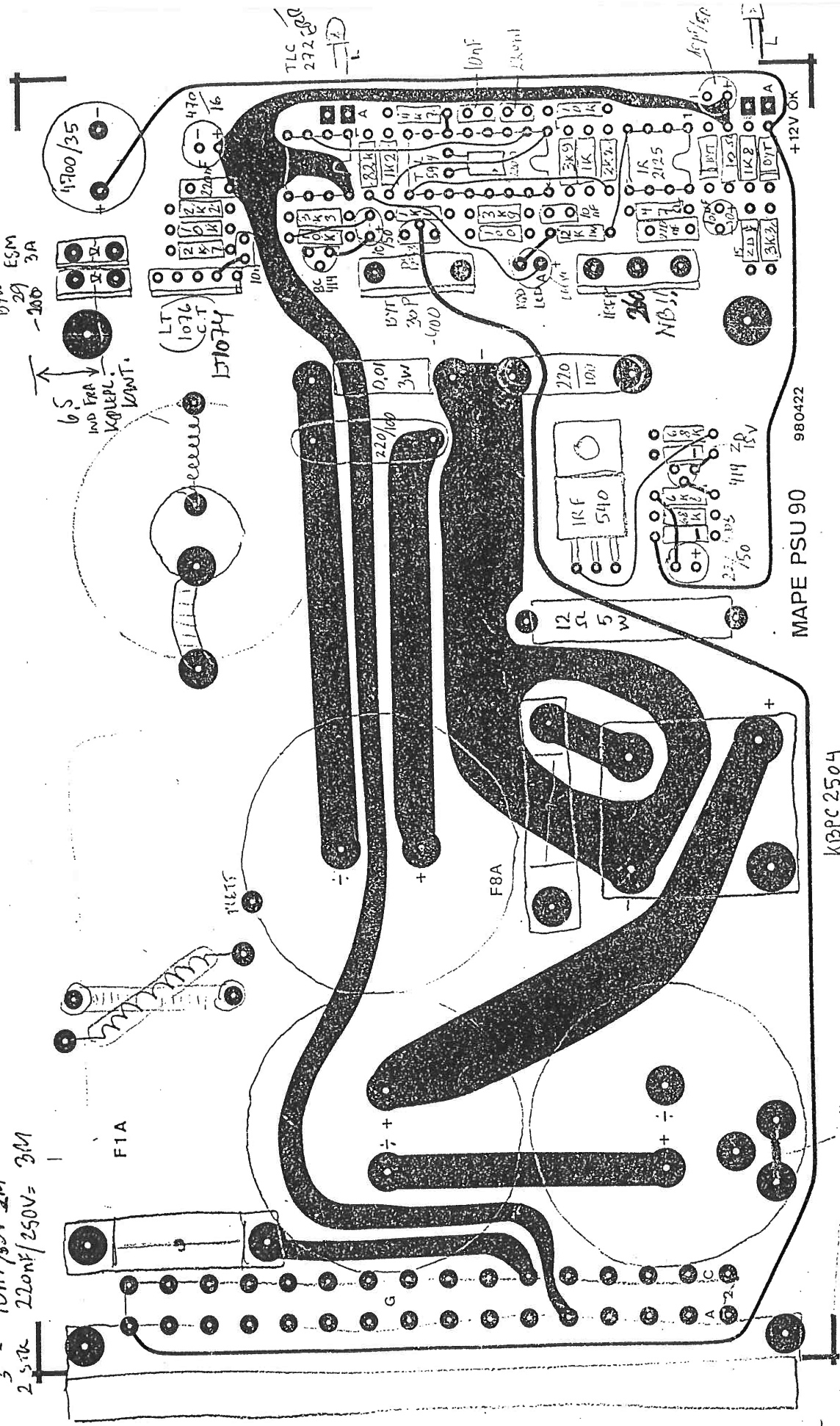
22-4-98.

3 50K 4700PF/100V
 1 - 4700 / 35V (MONT. TIC SIDST.)
 1 - 470/10V
 3 - 10/50V
 2 - 220NF/50V 2M
 2 - 220NF/50V 1M
 3 - 10NF/50V 1M
 2 50K 220NF/250V = 3M

PSU 90 MONTAGE

- 1 50K 100K
- 1 - 470K
- 1 - 100K
- 1 - 1K
- 1 - 1K1
- 1 - 1K2
- 1 - 1K8
- 2 - 2K2
- 1 - 2K7
- 2 - 3K3
- 2 - 3K9
- 1 - 4K7
- 3 - 6K8
- 1 - 10K
- 1 - 12K/1W
- 2 - 68K
- 1 - 82K
- 1 - 0.015W 3W
- 1 - 12R/5W

- 2 - SIGA 1121D.
- 1 - IAF
- 1 - IAF
- 1 - BRONDEL
- 2 - ZD 15V
- 2 - BYT 01
- 1 - ESM 3A
- 1 - BYW 29
- 1 - BYT 30P
- 1 - 104024
- 1 - IRF 540
- 1 - IREF 260
- 1 - LT-1076CT
- 2 - ROPE LEDS
- 1 - GREEN LED
- 2 - BS 414
- 1 - 32 POL KAWTAKON.
- 1 - PCB 92042E
- 1 - SPOLE ULNECO F-KEANE/10 TRAD
- 1 - SPOLE HEL FOR 12V SW.



MAPE PSU 90 980422

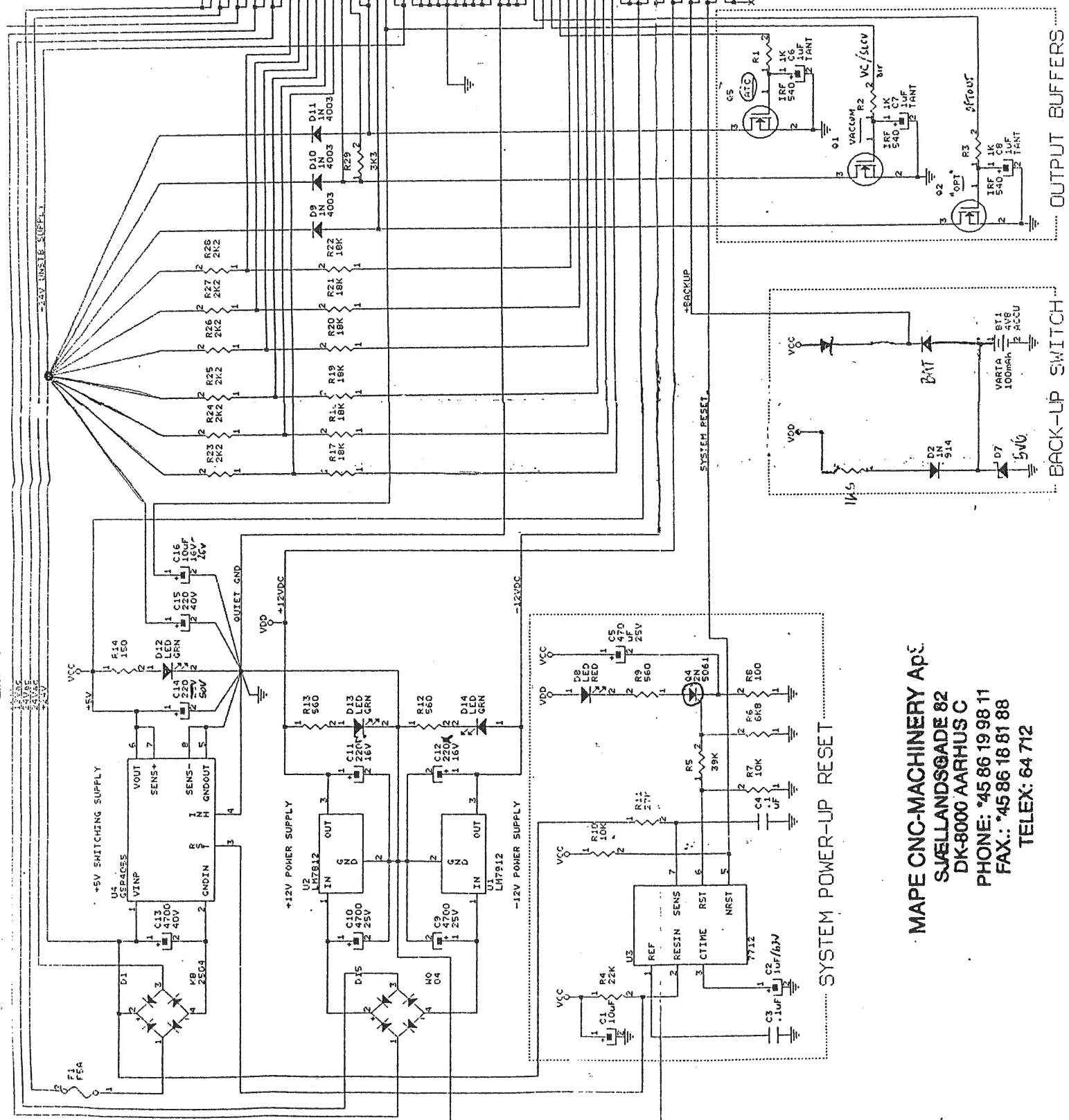
KBPC 2504

13088.

PSU-4

POWERSUPPLY PSU-4.

Copyright Mape Denmark.
The diagrams or parts thereof may in any form be transmitted to 3rd parties. For service use only.



MAPE CNC-MACHINERY ApS
 SJÆLLANDSGADE 82
 DK-8000 AARHUS C
 PHONE: +45 86 19 98 11
 FAX: +45 86 18 81 88
 TELEX: 64 712

- 5M 102R
- 475
- 10052
- 14
- 141
- 142
- 148
- 2K2
- 2K7
- 3K3
- 3K9
- 4K3
- 6K8
- 10K
- 12K/1W
- 68K
- 82K
- 0.015uF 5W
- 12.5/5W

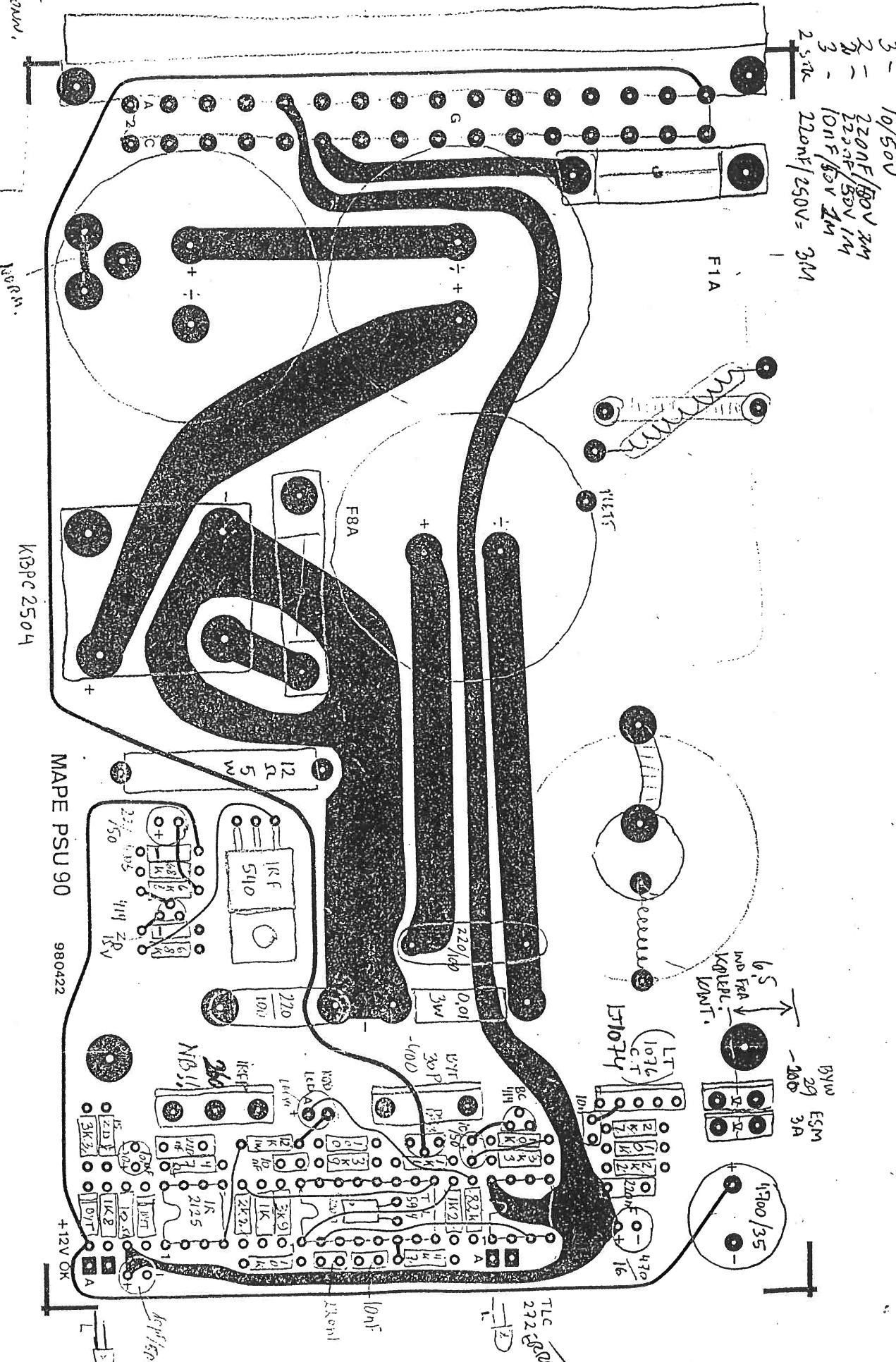
- 3 5M 4700uF/100V
- 1 4700/35V (MONT. TI SIDST.)
- 1 470/10V
- 3 10/50V
- 2 220uF/50V 3M
- 2 220uF/50V 1M
- 3 1015/50V 2M
- 2 220uF/250V = 3M

PSU 90 MONTAGE

22-4-98.

- SIGRETTID.
- 19F
- 79F
- BRONDEL
- ZD 15V
- BYT 01
- ESM 3A
- BYW 29
- BYT 30P
- 144024
- 19F 540
- 19P 360
- LT-1076CT
- RODE LED5
- GRON LED
- BC414

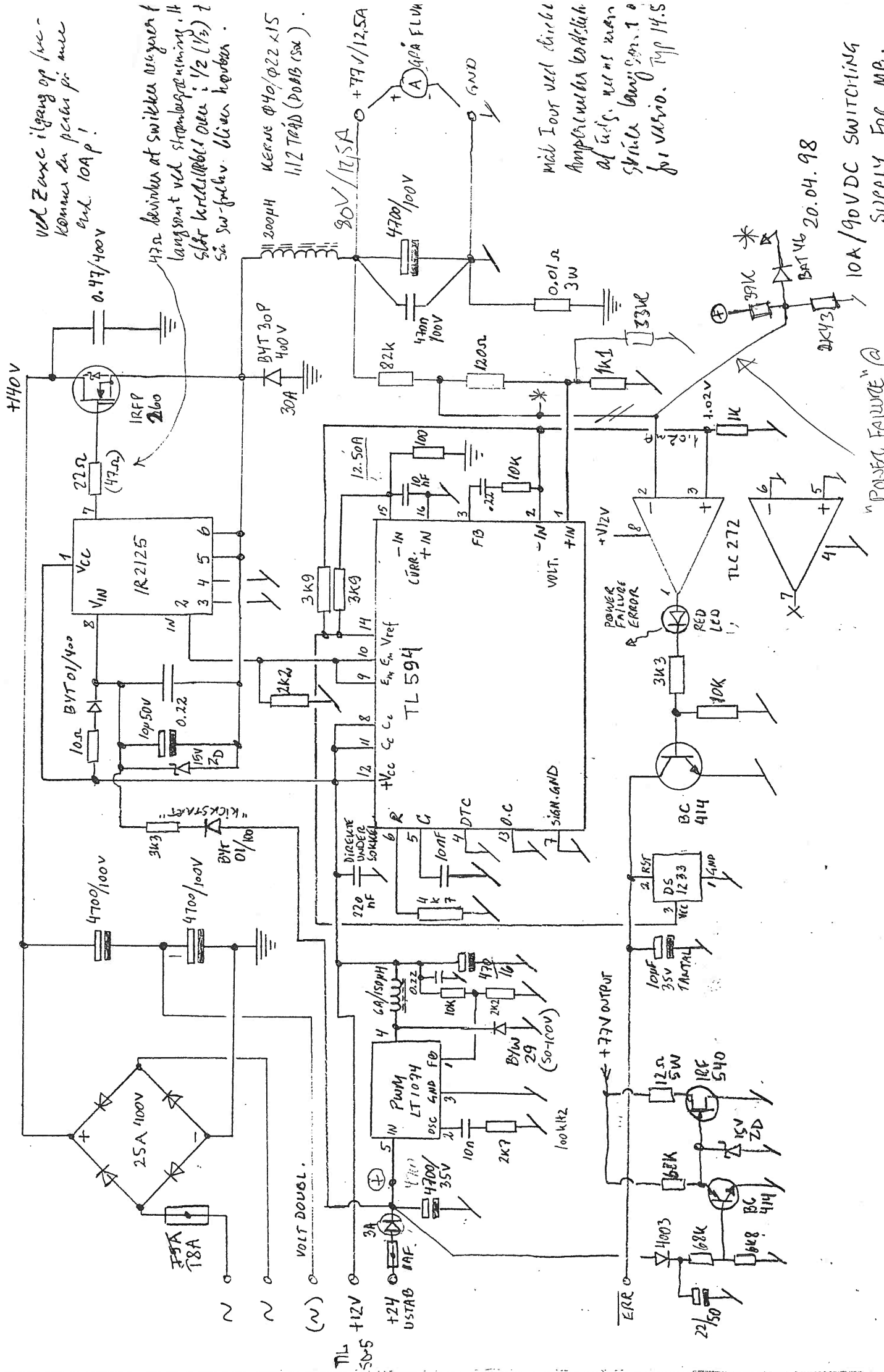
32 POL KONTAKTAR.
 PCB 980422
 SPÅLE UTVÆRD FÆKNEDE/110 70uB
 SPØLE AEL FOR 12V SW.



KBPC 2504

MAPE PSU 90 980422

+12V OK



ved Zaxe ilgang op 1µs -
kommer du måske på mere
end 10A p!

47n bevirker at switche reagere
langsomt ved strømregulering. It
skal korrigeres over: 1/2 (1/2) +
så sw-frekv. bliver højere.

200µH KERNEN Ø40/Ø22 x15
1:1,2 TRAD (DORAB (S&C).

80V/12,5A +77V/12,5A

4700/100V
A GFA FLUX

Med 10A ved 77V
Amperetallet er faktisk
af 10A, men som
skulle være 10A
for VARIO. Typ 14.5

BAT 16 20.04.98

10A/90VDC SWITCHING
SUPPLY FOR MB.

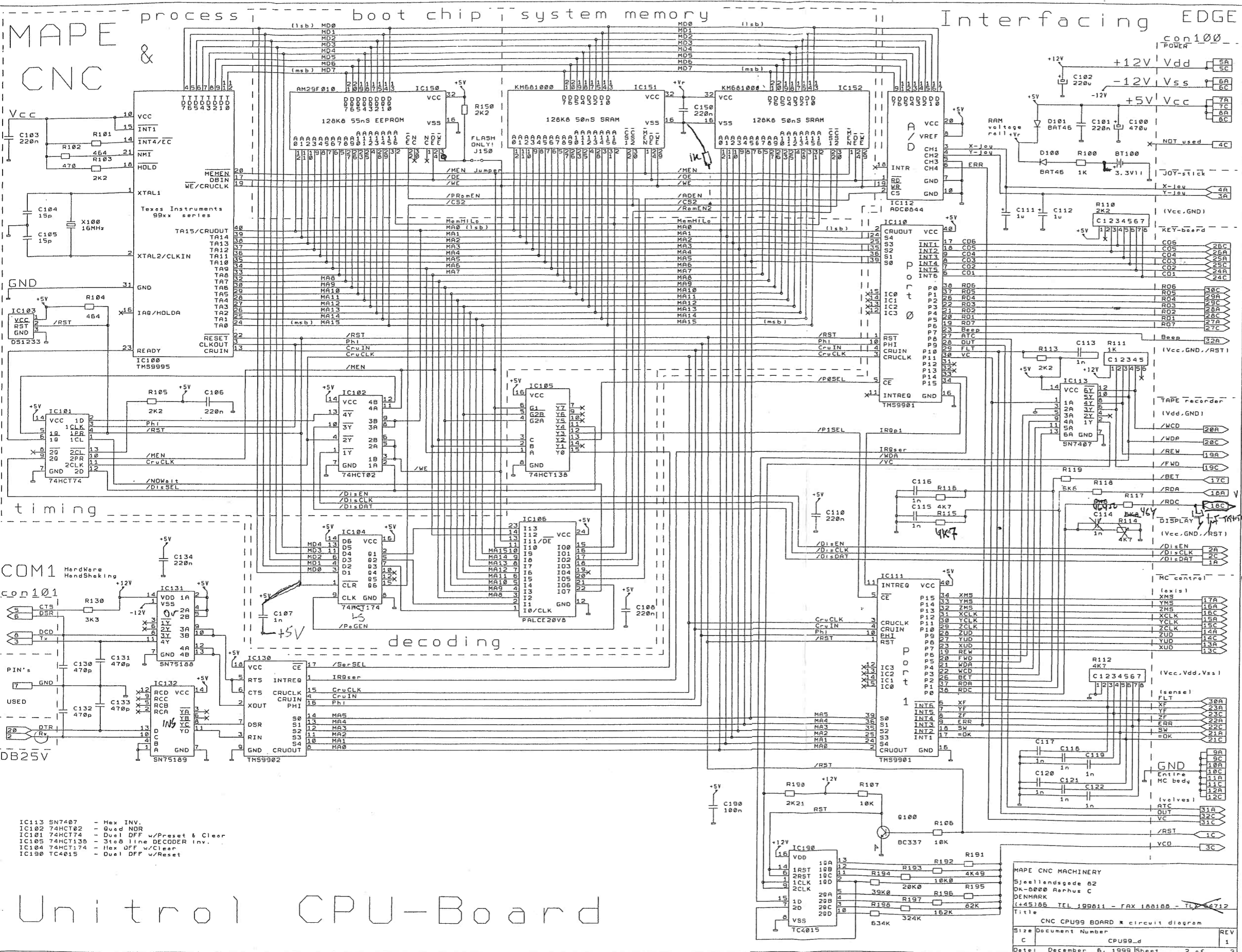
"POWER FAILURE" @

ci: 190-195 Vm NET

CIRCUIT

CIRCUIT

CPU 87.1

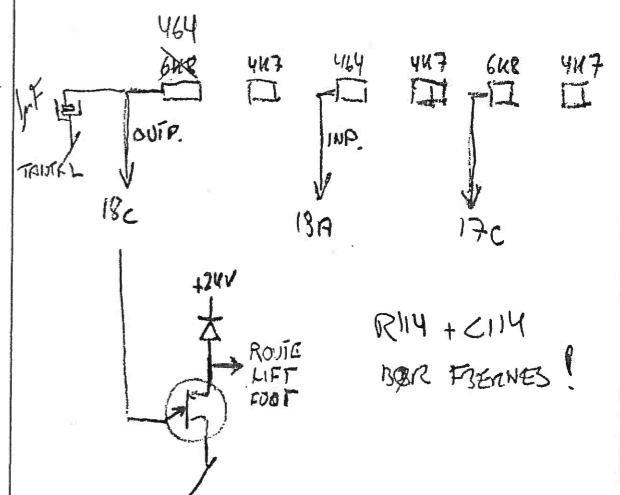


Unitrol CPU-Board

- IC113 SN7407 - Hex INV.
- IC102 74HCT02 - Quad NOR
- IC101 74HCT74 - Dual DFF w/Preset & Clear
- IC105 74HCT138 - 3-to-8 line DECODER Inv.
- IC104 74HCT174 - Hex DFF w/Clear
- IC109 TC4015 - Dual DFF w/Reset

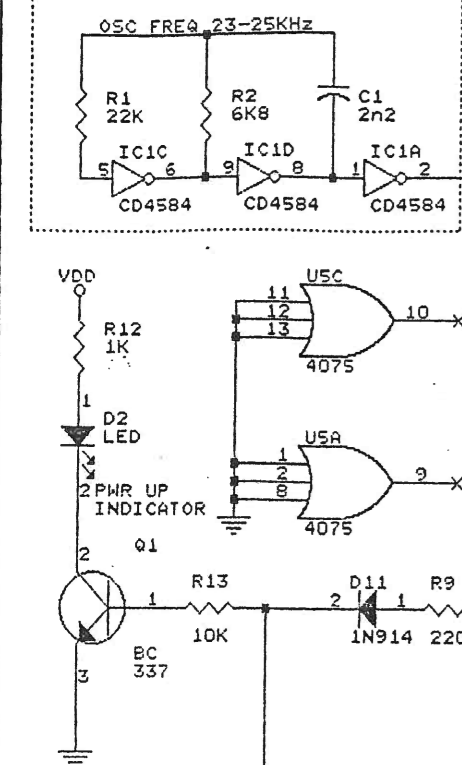
MAPE CNC MACHINERY
 Sjeallandsgade 82
 DK-8000 Aarhus C
 DENMARK
 (+45) 86 19 96 11 - FAX 186188 - TEL 66712

Title: CNC CPU99 BOARD * circuit diagram
 Size: Document Number: CPU99_d
 Date: December 6, 1999 Sheet 2 of 2

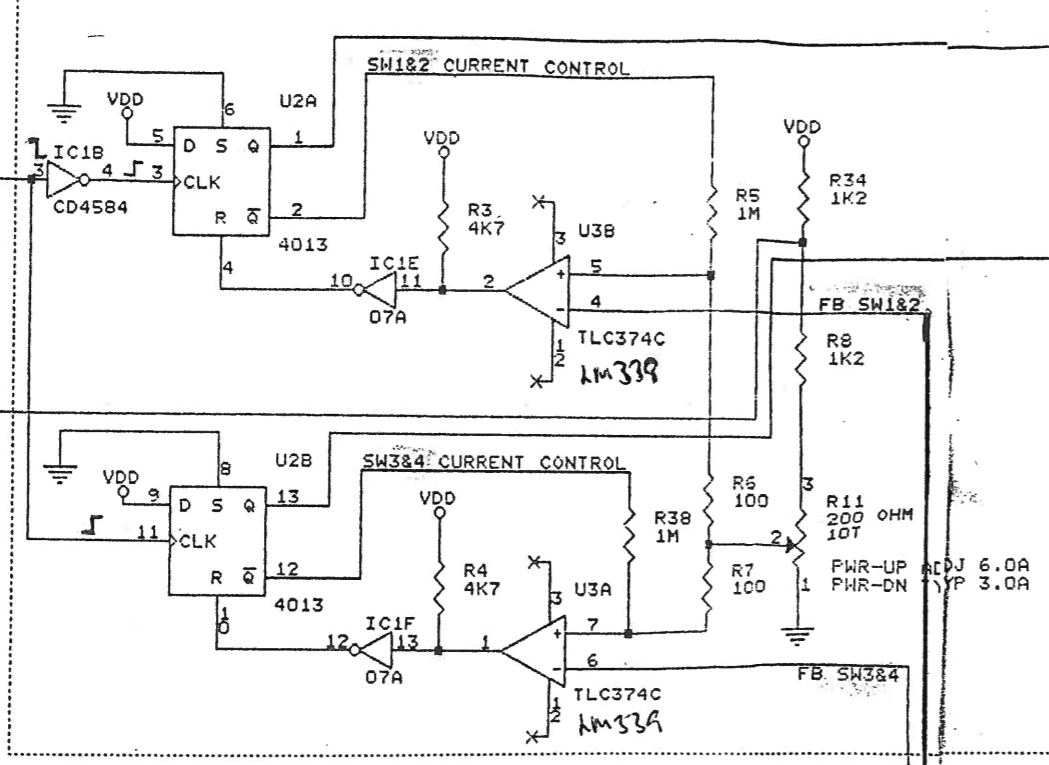


SD-5

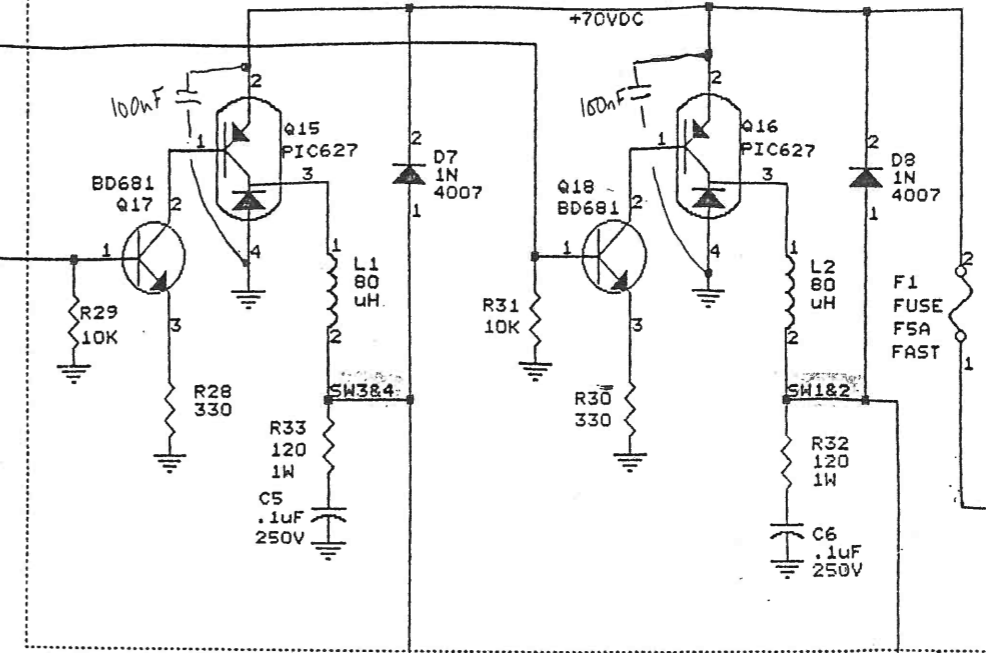
SW OSC



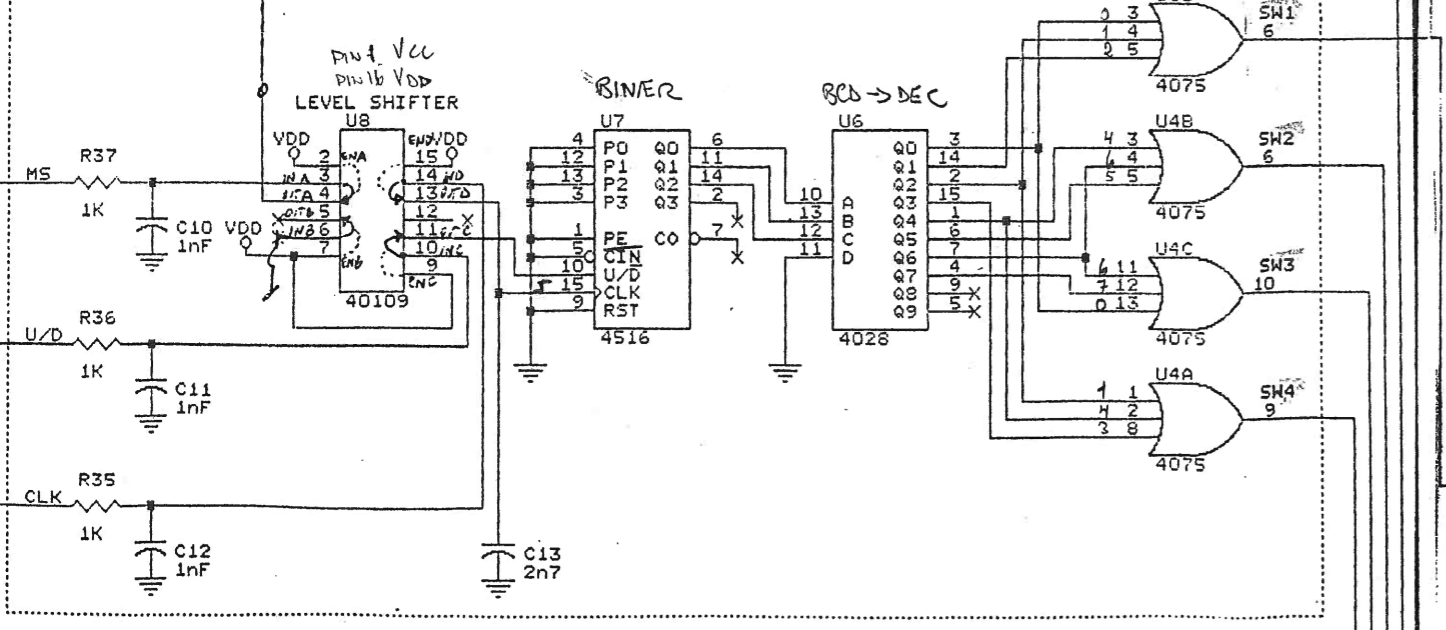
CURRENT CONTROL



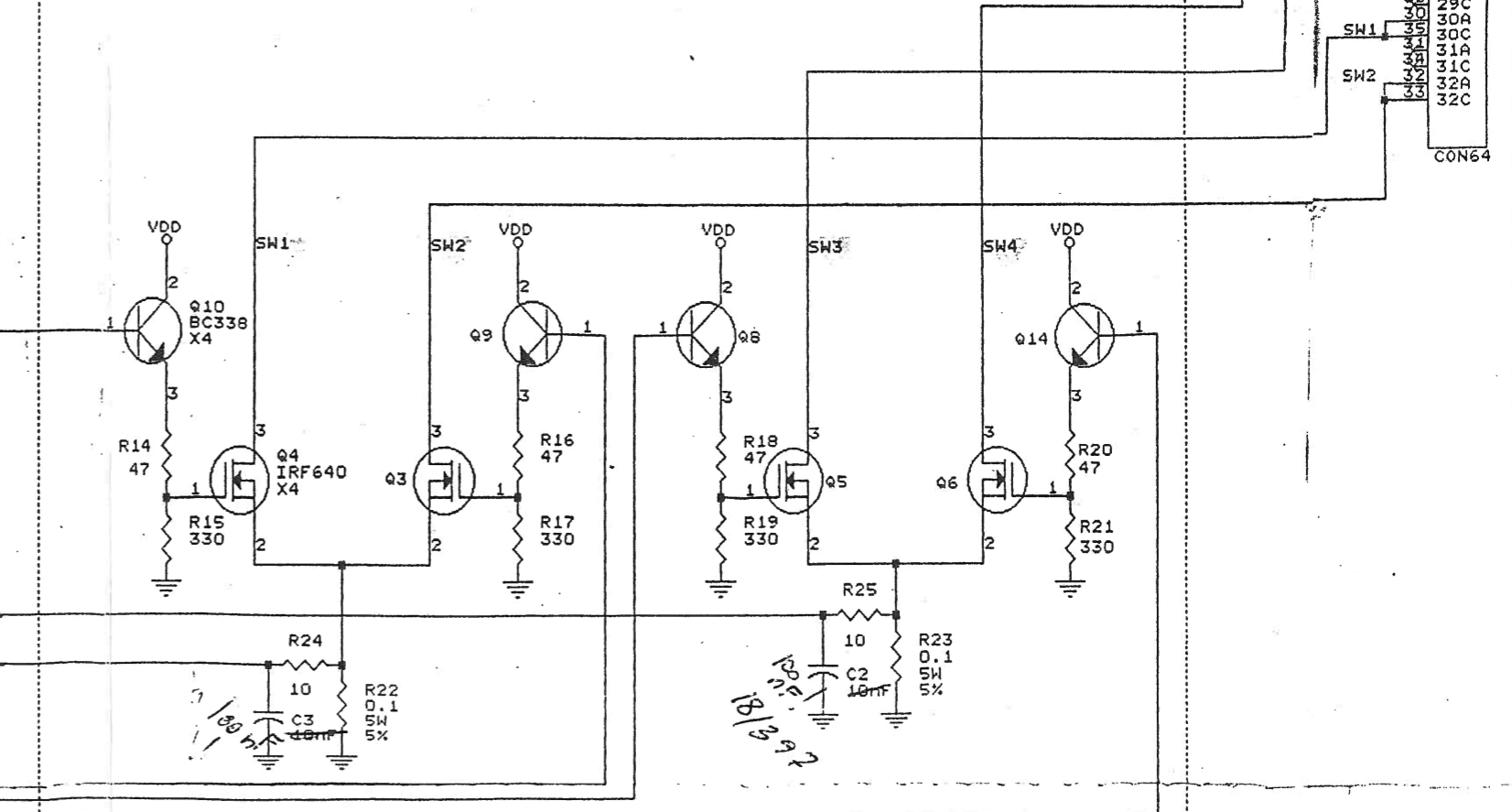
OUTPUT CURR GENERATORS



INPUT CONDITIONER & DECODER



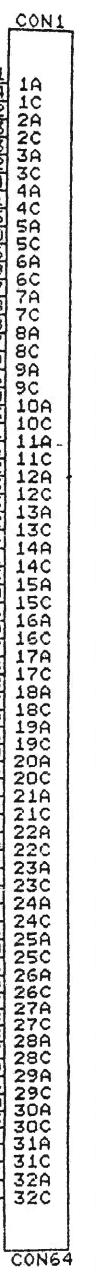
OUTPUT STAGE SW1-4



NOTE:
R8=1K2 AND R9=220
GIVES 6.5A MAX, AND P_{down}=2.9-3.0A

MAPE CNC-MACHINERY ApS
Sjællandsgade 82
DK-8000 AARHUS C
PHONE: *45 86 19 98 11
FAX: *45 86 18 81 88
TELEX: 64 712

STEPMOTOR DRIVE SD5.
PROPERTY OF MAPE MACHINERY APS DENMARK.
COPYRIGHT MAPE MACHINERY APS, DENMARK.
THE DIAGRAMS OF PARTS THEREOF MAY NOT
IN ANY FORM BE TRANSMITTED TO 3rd PARTIES.
FOR SERVICE USE ONLY:



Values for status-registers:

Date: 5/10-01

revision
0307.00

File: STATUSNY2000.XLS RETTET 03.07.2000, KM computer/MS-WORD/DOKUMENTER/MAPE & WORKS.

Serial No: 0110282 Customer: Siemens-Rowex Model: CNC-1

(SARONIX BV)

	X	Y	Description:	
0	9880	9830	Correction XY leadscrews (PPM/10.000)	
1	4800	-14250	X toolpod, distance (from ref.) +/-.	Y toolpod, distance (from ref.) +/-.
2	0	1500	Delta X toolpods (+/-).	Delta Y toolpods +/-.
3	0	0	X-position Laser Z1.	Y-position Laser Z1.
4	0	0	X-position Laser Z2.	Y-position Laser Z2.
5	0	0	Laser beam-height vertical.	Laser resolution: Steps/step-increment.
6	25	16	A06 Default (x1000 rpm.)	A16 Default (mm/S)
7	16	300	A26 Default (mm/S)	A46 Default (mm/S)
8	320	2	A36 Default (NB:In steps)	A50 Default (0-3)
9	0	0	Ref Axis Configuration (0-3)	Autoparameter load 0=On, 1=Off
10	100	30	Ref. Speed (Towards endstop; mm/S)	Ref. Speed (away from endstop; mm / S.)
11	0	1		HF-Gen. Type 0=Auto, 1=Mape, 2=S & M, +4=4 polet
12	4	4	XY-Error Limit (Step)	Z-Error Limit (Step)
13	300	300	XY-High Power Speed	Z-High Power Speed
14	180	12500	Max Speed (mm/S)	XY Axis resolution (uM / 1000 steps). Starting value addition (%)
15	100	50	Acc-Time (mS)	
16	100	50	Decc-Time (mS).	
17	300	25000	Max Speed (mm/S)	Z-Up Axis resolution (uM / 1000 steps). Starting value addition (%)
18	36	33	Acc-Time (mS)	
19	10	33	Decc-Time (mS)	
20	150	100	Max Speed (mm/S)	Z-Down Max Stroke Speed (mm/S) Starting value addition (%)
21	10	33	Acc. Speed (mS)	
22	10	33	Decc. Speed (mS)	
23	0	0	Tool-diam. max undertolerance (0.01mm)	Tool-dia. max overtolerance (0.01mm)
24	0	0	Z1 X-offset.	Z1 Y-offset.
25	0	0	Z2 X-offset.	Z2 Y-offset.
26	1	1	Data-Out specification:0=9600 1=19200.	Data-In specification: 0=9600 1=19200.
27	0	0	Z1 Reference +addition (steps).	Z 2 Reference + addition (steps).
28	60000	30000	X-Step @ ref-point.	Y-Step @ ref-point.
29	5	50	Axis-configuration.	Tool-depth (steps).
30	60	1696	Max. Spindle-RPM.	Z-Axis height (steps). NB:+28step TEST!
31	3635	18091	Ref. point distance (in steps).	
32	13000	500	Ref. point co-ordinater (externally).	
33	20	800	X=Number of tool-pods	Y=Air valve open/shutting timer (mS).
34	11000	11650	Min. pos. limit (steps), @ positioning.	
35	60250	48250	Max. pos. limit (step), positioning.	
36	0	10	Rpm OK timer mS	
37	1000	282	Model No. CNC-	Serial Production No.
38	0	975	N/A (Knækpos. - Timer mS).	Mains voltage adjustment.
39	0	0	1 = Cancel automatic Z-UP function.	Code A33:Y x 10
40	1	51039	Unit: 0=1/2000", 1=1/100mm, 1/1000".	Checksum

Spindle type: Jäger 42-25 Left No: Right No: 266/2

Laser-signal: VDC (typ 1.6-1.8V).

Distance adj. tool to workplane (@20,32mm ringsetting): 1,0 mm.