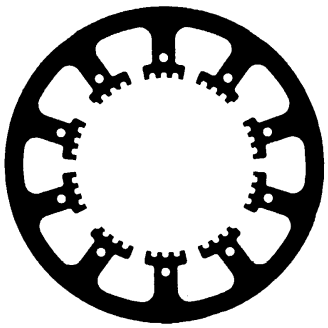
Milling, plotting, engraving,
drilling, grinding, cutting
and much more besides with...



PC-NC

...the software that turns your standard PC into a universal stepper motor NC unit

Version 3.10 and
Version 3.20 (with optional tangential axis)

July 2003

© Copyright Burkhard Lewetz, 1989-2003

Lawful purchase of the diskette and the manual conveys permission for one person to utilise the ***PC-NC*** control program. Copying the diskette and the manual or changing any of the individual files or elements of the manual are forbidden. The authors reserve all rights to the programs and to the manual, in particular the copyright.

This control program has undergone extremely thorough testing. Nevertheless, it is impossible to give a guarantee for completely fault-free operation. Furthermore, no responsibility can be accepted for damage caused as a result of using our program.

Despite the most strenuous efforts, it is never possible to completely eliminate all faults. Consequently, we would be grateful to receive feedback from users.

Burkhard Lewetz
Hardware-Software
Postfach 1221
D-88071 Meckenbeuren
Germany
Tel. (+49) 07542-21886
FAX (+49) 07542-3889
eMail info@lewetz.de
Homepage http://www.lewetz.de July 2003

MS-DOS, MS-Windows, MS-Word are registered trademarks of the Microsoft Corporation.
AutoSketch, AutoCAD, AutoSketch for Windows are registered trademarks of Autodesk AG.
IBM is a registered trademark of the International Business Machines Corporation.
CorelDRAW is a registered trademark of the Corel Corporation.
WordStar is a registered trademark of the MicroPro International Corporation.
Designer is a registered trademark of Micrografx, Inc.
Caddy is a trademark of Ziegler Informatics, Mönchengladbach.
Eagle is a trademark of CadSoft Computer GmbH, Pleiskirchen.
Turbo C is a registered trademark of Borland International, INC.
Other products mentioned by name are trademarks or registered trademarks of their corresponding companies.

3.200

Table of contents

The structure of this manual ...

This manual provides you with all the information needed for using ***PC-NC***. It is divided into individual chapters, the contents of which are summarised below:

Chapter 1: Brief explanation about ***PC-NC***, the possibilities for using it and the hardware requirements.

Chapter 2: Initial start-up procedure, description of how to install the program.

Chapter 3: More detailed descriptions of how to operate the program and the individual functions of ***PC-NC***.

Chapter 4: Batch jobs, explanations of the individual commands and how to use them.

Chapter 5: Explanation of all parameters and the setting options.

Chapter 6: Explanation of the additional program ***HPGL-Opti***.

Chapter 7: Step-by-step procedure for the first setup at the machine and for different standard applications such as drilling, plotting and milling.

Chapter 8: Technical information about the interface configuration, motor actuation and NC data interpreters

Appendix: Fault messages

Definitions

Some of the terms used in this manual may require an explanation:

Job file A file with NC data which is read and processed by ***PC-NC***. Depending on the application, the file may contain milling, plotting, drilling data, data for batch jobs or other types of data.

Job process The process of reading and processing a job file and the resultant actuation of the machine.

Command An individual instruction in the job file which gives rise to actions by the machine or in ***PC-NC***.

Bit Alternatively used for tool.

Descriptions of the machine and the movement directions of the individual axes are made with reference to the following mechanical layout of a flat-bed machine:

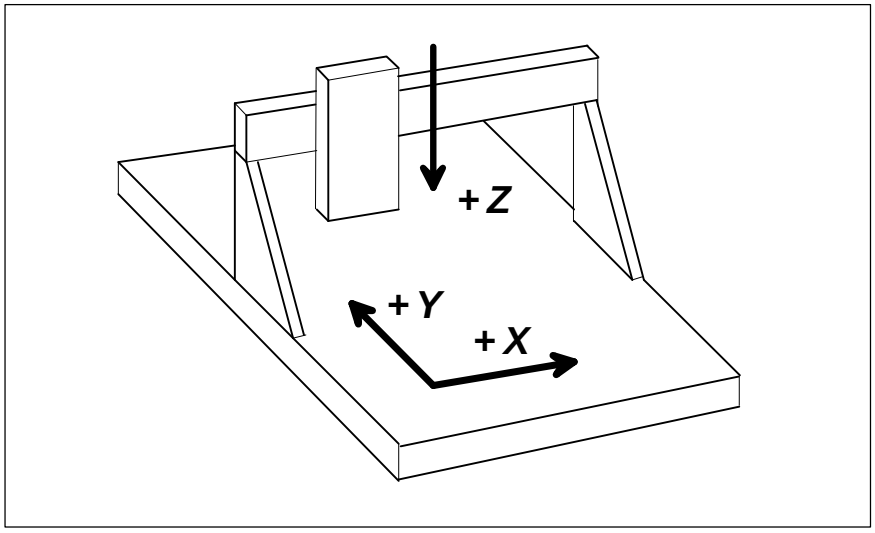


Figure 0-1 Schematic layout of a flat-bed machine

Use of typography

Keyboard entries	Normal script enclosed in a box, e.g. (ENTER)
Cursor keys	Normal script enclosed in a box, e.g. (UP), (LEFT), (PGDN)
Menu functions	Capital letters with menu path, e.g. FILE-DISPLAY
Messages	Italic script, e.g. *´Perform reference movement? Y/N´*
Function names	Capital letters, e.g. SIGNAL TEST

1. What can ***PC-NC*** do?

Universal program

The ***PC-NC*** program is a software package which turns any standard personal computer into a universal NC unit.

No additional hardware

There is no need for either an external processor or other additional intelligent hardware. ***PC-NC*** controls up to 3 stepper motor axes via the parallel printer port, supplying the required clock pulse and direction signals.

The 3 stepper motor axes can be used for making any 2.5-D mechanical structures. It is possible to accomplish an extremely broad range of jobs. Standard applications include:

- Drilling
- Plotting
- Grinding
- Dispensing
- Milling
- Cutting
- Engraving

Extensive parameters

The extensive range of options for setting parameters means the program can be adapted to almost all XYZ machines.

Modern user interface

PC-NC offers a well thought-out and modern operating concept incorporating drop-down menus and a comfortable dialog system. This makes it easy to learn and master the program.

Help function

The context-sensitive help function can provide information and hints for subsequent work at almost any point in the program.

Runs on any PC

A standard personal computer with a hard disc, printer port and any graphics card is required in order to run ***PC-NC***. All types of computer from an 386 through to a Pentium IV can be used.

Figure 1-1 *Cutty* 3-axes machine

Special features

These are some of the special features of *PC-NC*:

- 32-bit arithmetic for an almost unlimited range of applications
- Step frequencies in excess of 20 kHz (measured on a 486/33MHz computer)
- Manual movement controlled by the cursor keys, exact to the step
- Graphic display with a scale on all standard graphic cards
- Process display during processing in graphic form or as a text report
- Shifting and scaling of data using the cursor keys in graphic mode, automatic adaptation to a defined size
- Runs on all PCs from 386 through to Pentium IV
- Speeds, ramp length and ramp profile adjustable as required
- Limit switch polling
- Recognises HP-GL, G code and various drilling formats
- Extensive bit management function

- Additional program ***HPGL-Opti*** included for sorting data and calculating a radius compensation
- Test functions for determining the optimum parameters
- Context-sensitive help function and help index
- Modern user interface with drop-down menus and windows
- Calibration function for reliable workpiece clamping using alternating approach movements from four calibration points
- Directions of all axes can be reversed using the software
- Teach-In function for manual creation of NC data
- Speed selectable in the range from 0.01 to 100 mm/sec.
- Optimisation of NC data, sorting according to bits and minimum unproductive movements
- Definition of batch jobs similar to DOS batch files, allowing the creation of extensive sequences of jobs
- Graphic milling and drilling simulation
- The job process can be started at any point within the NC file, making it easy to resume a job after a cancellation
- Powerful program editor included
- Online Z-axis height correction during processing
- Quick menu for reliable operation in mass production
- External start signal for synchronising the process
- automatic tool length measurement and compensation
- controls an automatic tool changer (ATC)
- output for spindle speed
- foil cutting using a tangential knife with the 4th axis
- available in english, german, italian (more to come)

2. First steps

2.1. Installation

User-friendly installation

Although it is possible to launch ***PC-NC*** from the diskette, we recommend installing the program on your hard disc. The diskette contains an installation program which performs all the required work automatically.

Enter the following command to install the program:

INSTALL (ENTER)

The installation program copies all files to a new directory called C:\PCNC32*.*.

Call up the **README** file to display important changes to the information in the manual. These changes are additional features included after the manual was written.

List of files

The following files are installed to the target directory:

PCNC.EXE	Control program
PCNC.SYS	Parameter file
PCNC.WRK	Bit file
PCNC.HLP	Help texts
PCNC.MSG	Messages and menu texts
README	Latest changes to the manual
HISTORY.TXT	Version history
HPGLOPTI.EXE	external optimisation program
HPGLOPTI.OSY	Parameter file
HPGLOPTI.HLP	Help file
***.PLT *.SMM *.JOB *.DIN**	Sample NC files and batch job

2.2. Launching the program

Launch ***PC-NC*** from the DOS prompt by entering the following command:

PCNC (ENTER)

The ***PC-NC*** working screen appears after a short time. This screen is divided into several areas:

- Title bar
- Menu bar
- Working area
- Status bar
- Help bar
- Function key bar

Figure 2-1 ***PC-NC*** screen layout

Screen layout

The title bar and menu bar are located at the top edge of the screen. The drop-down menus open from the menu bar.

The bottom two lines of the screen contain the function keys which are active for any given program situation. The key allocations change depending on the program status based on the functions which are currently active.

There is a line above the function key lines for explanatory texts. Above this comes the status bar which is for displaying current program states. The large area between this and the menu bar is the working area. ***PC-NC*** uses this area for displaying open windows or messages, depending on the action.

status bar

The following information is shown in the status bar:

- Number of the command currently being processed and the cycle if there are several program repetitions
- Job file (if loaded)
- Current parameter file
- Current bit file
- Status of the drilling spindle and coolant pump signals
- Current bit number and position (bottom/top)
- Fill level of the step buffer
- Machine status (stationary/running)

Additional switches in the launch command line

Additional launch parameters can be used for tasks such as loading a file immediately, using pre-defined parameters or simply adapting the screen display.

The following command structure is used for launching the program together with command line launch switches:

PCNC [file] [parameters] (ENTER)

Explanation:	**file**	Job file which should be loaded as soon as the program is launched
Parameters:	**-m**	Monochrome display
	-h, -?	Parameter display
	-s file	Load a specified parameter file
	-w file	Load a specified bit file

Examples

PCNC abc.plt

Immediately loads the **abc.plt** job file.

PCNC -s others.sys -w others.wrk

Loads special settings from the specified files instead of the standard parameters and bits contained in the PCNC.SYS and PCNC.WRK files.

Restricted operation under MS-Windows

Unfortunately, the full functionality of ***PC-NC*** is not available under MS-Windows. It is impossible to generate step frequencies and control the interface in the required manner under MS-Windows. All the functions for moving the motors are inactive, although it is possible to load and display NC data without any problems.

The additional program ***HPGL-Opti*** can be run under Windows without any restriction.

Please refer to the detailed description in chapter 6 for information about connecting the machine and performing the first steps in operation.

2.3. Exiting *PC-NC*

You can exit ***PC-NC*** at any time using the `(alt-X)` key combination. Alternatively, you can select the EXIT menu item from the FILE drop-down menu.

3. Working with *PC-NC*

3.1. Drop-down menus and function keys

Modern user interface

PC-NC has a modern user interface. All functions can be accessed quickly using drop-down menus. In addition, the function keys can be used for activating functions which are required frequently.

The drop-down menus are divided into several functional groups, e.g. all functions concerned with selecting and processing files are grouped in one menu. All parameters and bits can be set using another menu.

The (F10) key or one of the hotkeys can be used for opening or activating the menu system.

The cursor keys can be used for scrolling through the menus as required. A menu function is selected by pressing the (ENTER) key or by means of the corresponding hotkey.

Additional function keys

Important functions can also be activated using function keys. The assignment of function keys changes depending on the program status and the functions which are active. The keys which are active at any given time are always displayed along the bottom edge of the screen.

3.2. The individual menus

The following text describes all the menus and functions in detail.

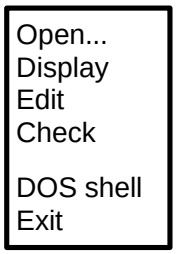
Not all menu items are active all the time. Functions are sometimes unavailable, depending on the program status. For example, it is impossible to display a file in graphic mode if no file has been selected yet.

3.2.1. SYSTEM menu

The SYSTEM menu (the left-most menu in the menu bar) provides information about the ***PC-NC*** program, the version of the program and the licence.

3.2.2. FILE menu

The FILE menu combines all the functions used for selecting and analysing files to be processed. In addition, it is also possible to call up a temporary DOS shell and exit ***PC-NC***.



Press the (F) hotkey to open the File menu.

FILE-OPEN

Using the menu for selecting files

FILE-OPEN is an interactive function for selecting job files.

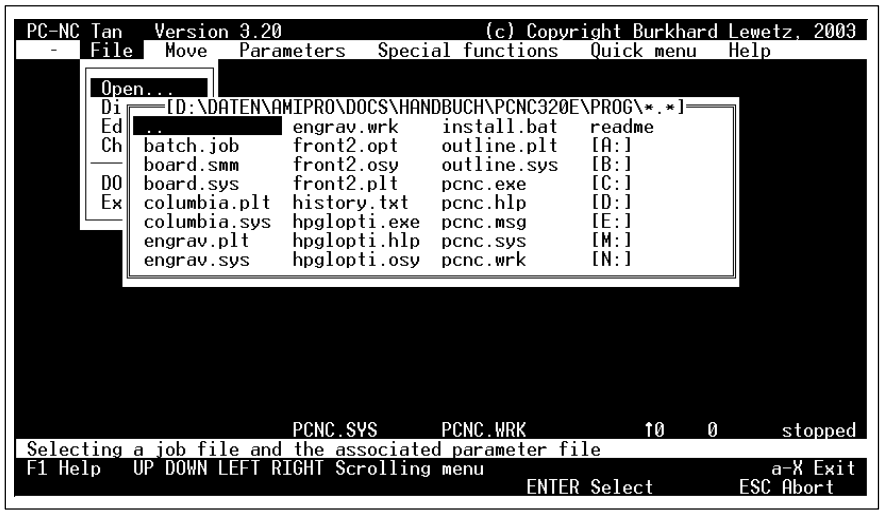


Figure 3-1 File select window

It is possible to change drives and directories in the dialog box. Files can also be selected by pressing the (F2) key.

Loading parameter and bit files subsequently

In addition to loading job files containing NC data, it is also possible to load parameter and bit files with the filename extensions *.SYS and *.WRK. ***PC-NC*** recognises the file type automatically and reads the files accordingly. This function can be used for loading other parameters or bit settings in a targeted manner.

FILE-DISPLAY

Graphic preview with VGA compatible graphics cards

The graphic preview function in ***PC-NC*** enables you to make a quick visual check of the job file which has been selected. The program initially loads all the data in order to determine their extent. The subsequent graphic display is free from distortion and at the maximum possible size.

The graphic preview function works with a standard VGA graphics cards. ***PC-NC*** automatically recognises which type of card is fitted and adapts the graphic output to this card.

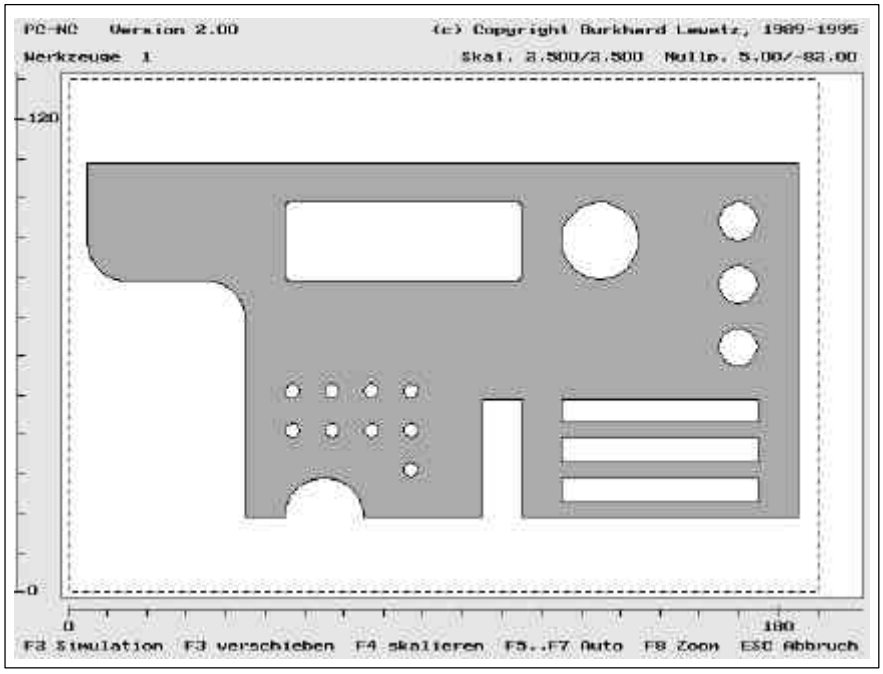


Figure 3-2 Graphic preview function

All the most important data displayed graphically

The following elements appear in the graphic view:

- The information in the job file
- Defined working area (dashed rectangle)

- Zero point (small grey dot at the left-hand bottom corner of the data)
- Current machine position as a red circle (only if in the visible range)
- Current scaling factors for X and Y (based on the parameters)
- Defined zero point for X and Y (based on the parameters)
- Rulers showing the current dimensions
- All required bits and their numbers

G code files can only be displayed two dimensional in the X/Y level.

The following key assignments are active in the graphic display:

(F2)	Starts the milling or drilling simulation
(F3)	Activates the shifting function
(F4)	Activates the scaling function
(F5)	Automatically centres the NC data in the working area
(F6)	Automatically centres the NC data in the working area and scales to 90 % of the size of the working area, side relationships are retained
(F7)	Automatically centres the NC data in the working area and scales to 90 % of the size of the working area, side relationships are not retained
(alt-F5)	Revokes automatic adaptation
(shift-F6)	same as (F6), but scales to 100%
(shift-F7)	same as (F7), but scales to 100%
(F8)	Zoom function. Displays the NC data so they fill the entire format, without the working area.
(+)(-)	Changes the step width
(ESC)	Cancels shift, scaling, simulation or graphic display functions
(Cursor_keys)	Shifts or scales the NC data

The (+) and (-) keys are the grey keys on the right-hand side of the keyboard.

Milling and drilling simulation

The milling and drilling simulation can be used for simulating the entire job process graphically on the screen before the process is actually carried out. This makes it easy to see the sequence in which the individual actions are performed.

PC-NC displays the command number corresponding to each command. The simulation can be cancelled at any point by pressing the (ESC) key.

On-screen adjustment using the cursor keys

Any required adjustments can be made by shifting and scaling functions directly in the graphic display if the dimensions of the working area precisely correspond to the workpiece.

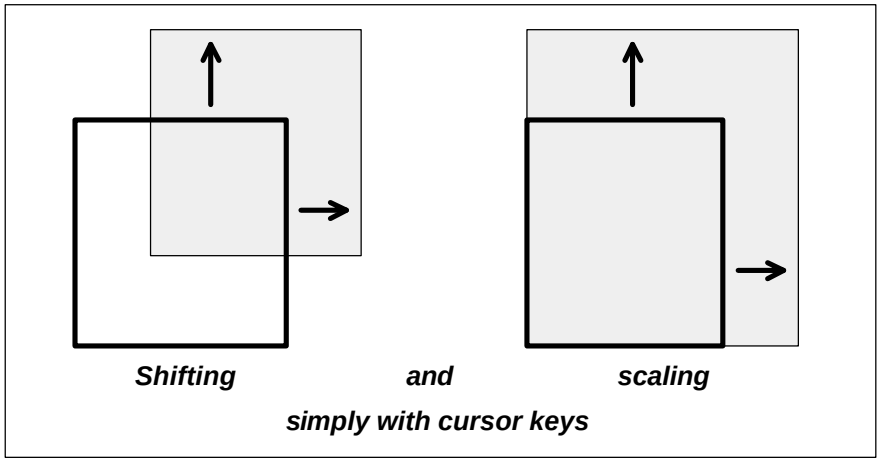


Figure 3-3 Shifting and scaling functions in the graphic display

Shifting and scaling

A light coloured rectangular outline appears when the shifting or scaling function is active. This marks the extent of the job file. This rectangle can be shifted or scaled using the cursor keys, thereby allowing it to be adapted precisely to the defined working area. The step width can be altered by pressing the (+) and (-) keys. The workpiece is displayed again at its new position and at the required size after the function is exited.

Automatic adaptation to the working area

It is also possible to use ***PC-NC*** for scaling and shifting the NC data within the working area automatically. The (F5) function key causes the data to be centred within the working area at their current size.

Maintaining or not maintaining side relationships

The (F6) and (F7) keys can be used in order to centre the data as well as scaling them to 90 % of the size of the working area as well. Using (F6) for this adaptation causes the side relationship of the X and Y expansion to be retained. The X and Y expansions are scaled independently of one another when the (F7) key is used. Pressing (alt-F5) undoes the automatic changes.

The current parameter values are changed by shifting and scaling. It is necessary to save the parameters at the end in order to retain all the new parameters.

The (F6) key also activates the graphic display.

FILE-EDIT

Editing the NC data

The EDIT menu item is used for calling up a build in editor in order to edit or view the NC data. You can define an external editor as well and instruct ***PC-NC*** to activate it instead.

Figure 3-4 NC file in editor window

The editor interface of ***PC-NC*** can be used for alternative programs as well. An example is a signmaking program which can be invoked by ***PC-NC*** after a job automatically.

Please note that there is not the whole memoy available when invoking the external editor or other programs with *PC-NC* running.

Press the (F7) function key to activate the editor.

FILE-CHECK

The CHECK function makes it possible to check the extent within the working area currently set before processing the file.

Checking limits of areas and bits

All the calculated data are listed in a window. The current settings for the working area and the zero point are displayed in the top part of the window. These are followed by the absolute extent of the workpiece data and their extent within the working area, i.e. with the zero point as a reference point.

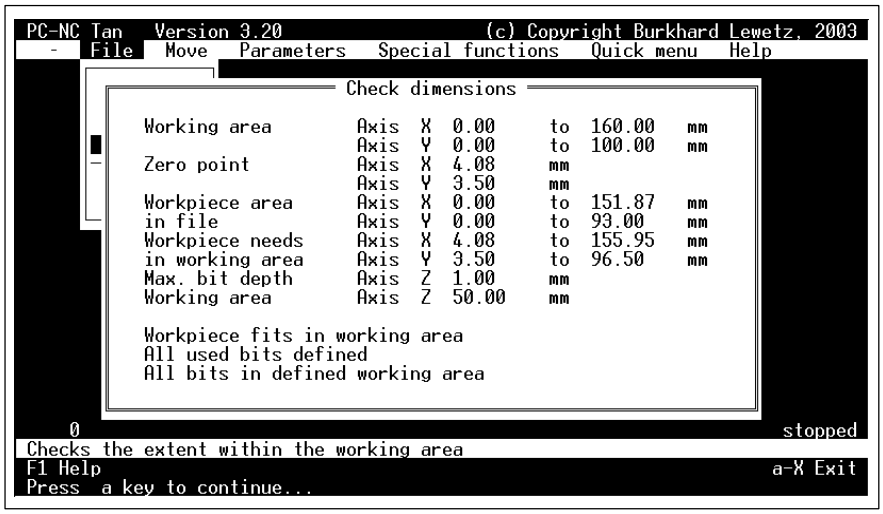


Figure 3-5 Checking the job file

The function also displays whether the workpiece can be processed using the current settings or if the limits of the working area would be exceeded during machining. In addition, the function also checks which bits are required and generates a warning if one or more of the bits have not yet been defined.

The *Workpiece does not fit in working area* message appears if the working area needs to be enlarged or if the zero point should be shifted.

FILE-DOS SHELL

It is possible to call up a temporary DOS shell using the DOS SHELL function.

Calling up the MS-DOS shell

Not all your computer's main memory is available when you call up the MS-DOS shell. As a result, out of memory errors may occur if you try to launch large programs or carry out complicated commands.

Enter the following command to return to ***PC-NC***:

EXIT (ENTER)

FILE-EXIT

Exit ***PC-NC*** either using the FILE-EXIT menu item or the (alt-X) key combination.

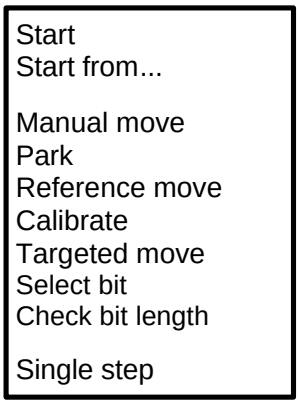
Exiting PC-NC

When you exit ***PC-NC***, the program stops any motors which are still running and closes any files which are still open.

3.2.3. MOVE menu

All functions for moving the machine

The MOVE menu combines all the functions used for activating the machine. It is possible to initiate reference movement, parking movement and manual movements as well as starting or resetting a job process. In addition, a switch can be used for activating single command mode.



Press the (M) hotkey to open the MOVE menu.

MOVE-START

Start processing

After a job file has been loaded, the START menu item can be used for initiating processing. Pressing the (F3) key has the same effect.

Text report or graphic progress indicator

PC-NC opens a report window and displays all the commands in the job file together with their associated interpretations if the currently loaded job file contains NC data. The progress indicator can also appear in graphic mode, however. In this case, ***PC-NC*** displays the distances travelled by the X and Y-axes in the form of lines.

If a batch job is the loaded job file, ***PC-NC*** starts with line-by-line processing. Each command is displayed briefly in a window, after which the corresponding function is performed.

PC-NC activates the X and Y motors in the case of movement commands. The Z motor moves upwards or downwards in the case of commands to move the bits.

If a tangential cutting knife is activated, ***PC-NC*** performs the turn to the moving direction automatically. Small turns are performed online while moving, in case of greater turns, ***PC-NC*** stops moving, turns the knife and starts again.

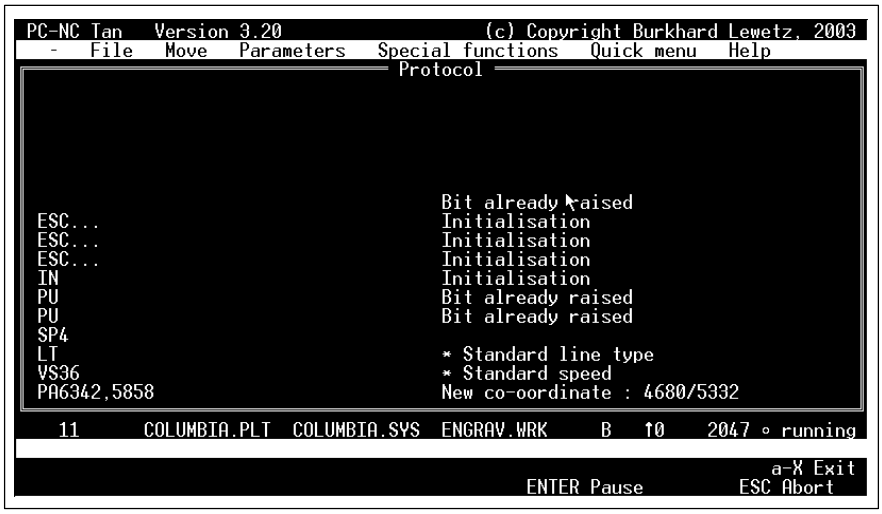


Figure 3-6 Active job process with text report

Pausing at any desired positions

The (ENTER) key can be used for pausing the job process. It takes a few seconds until the command in progress is completed and then the motors are stopped.

It is possible to change certain parameters and move to the parked position during a pause.

The job process can be continued precisely where it left off using the MOVE-START function or by pressing (F3).

Each command in the job file must be confirmed by pressing the (F3) key if single command mode is active.

MOVE-START FROM...

In many cases, it may be sensible to start a job process from a specific NC command instead of at the beginning. This is the purpose of the START FROM... command.

After this function is activated, enter the command number. The job process then starts from this command.

Proceed as follows if you do not know the required command number: use the milling or drilling simulation to simulate the job process from the start and read off the current command number when the simulation reaches the required point.

MOVE-MANUAL MOVE

The MANUAL MOVE menu item is used for the manual movement function. This function can also be accessed using the (F5) function key.

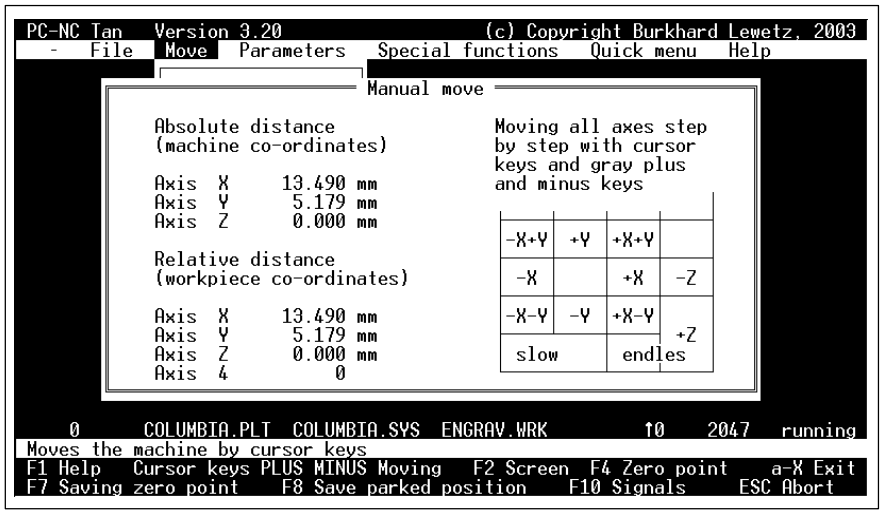


Figure 3-7 Manual movement

Movement precise to the step using the keyboard

When manual movement is active, you can move all motors in a step-by-step or continuous process using the cursor keys. Pressing the key briefly only initiates one motor step whereas pressing and holding the key causes the motor to move continuously. The changeover time can be defined as a parameter.

Display of the step counter

The absolute step counters for each axis in relation to the reference point on the reference switches are displayed in the top part of the window. The relative step counters appear underneath the absolute counters. The relative counters are in relation to the zero point.

_PC-NC_ differentiates between two systems of co-ordinates. On the one hand, there are the machine co-ordinates with their origin on the reference switches, which is also referred to as the reference point. On the other hand, there are the workpiece co-ordinates with the workpiece zero point. This is always located in the left-hand bottom corner of the data area.

PC-NC can move the axes endlessly, i.e. movement continues for as long as a key remains pressed. The axis brakes when the key is released and comes to a stop without any step losses. The second possibility involves moving discrete distances. Distances between 0.01 mm and 100 mm are possible, as well as inch dimensions. The currently set distance is displayed in the window. ***PC-NC*** moves this distance in any required direction, depending on which button or key is pressed.

The following key assignments are active:

(LEFT) (RIGHT)	Moves the X-axis
(UP) (DOWN)	Moves the Y-axis
(HOME) (END)	Moves the X and Y-axes diagonally
(PGUP) (PGDN)	Moves the X and Y-axes diagonally
(PLUS) (MINUS)	Moves the Z-axis
(a) (A)	Moves tangential axis
(F2)	Switches display between large or small digits
(F3)	Moves to the parked position
(F4)	Moves to the zero point of the X and Y-axes
(alt-F4)	Moves to the zero point with the Z-axis
(F5)	Stores the current position in the parameters as the left-hand bottom corner of the working area
(F6)	Stores the current position in the parameters as the right-hand top corner of the working area
(F7)	Stores the current position in the parameters as the zero point of all axes
(ctrl-F7)	Stores the current position in the parameters as the zero point of the X-axis
(shift-F7)	Stores the current position in the parameters as the zero point of the Y-axis
(alt-F7)	Stores the current position in the parameters as the zero point of the Z-axis
(F8)	Stores the current position in the parameters as the parked position
(alt-F9)	Switches to the *Targeted move* function and so makes it possible to move to defined positions or to move specific distances
(F10)	Switches the extra signals *Drilling spindle* and *Coolant pump* on or off
(INS)	Switches between fast and slow movement
(BLANK)	switched between endless move and discrete distances

Displayed units of the tangential axis are steps. There is no need to turn the tangential axis by hand, because it is turned automatically. To check the zero point or offset after a reference move, you have to calibrate the tangential axis once.

The current position is easy to recognise

As an alternative to displaying the absolute and relative step counters, the (F2) key can be pressed to call up the display in large digits. In this case, only the counters relative to the zero point are visible. However, they can be seen from a long way away.

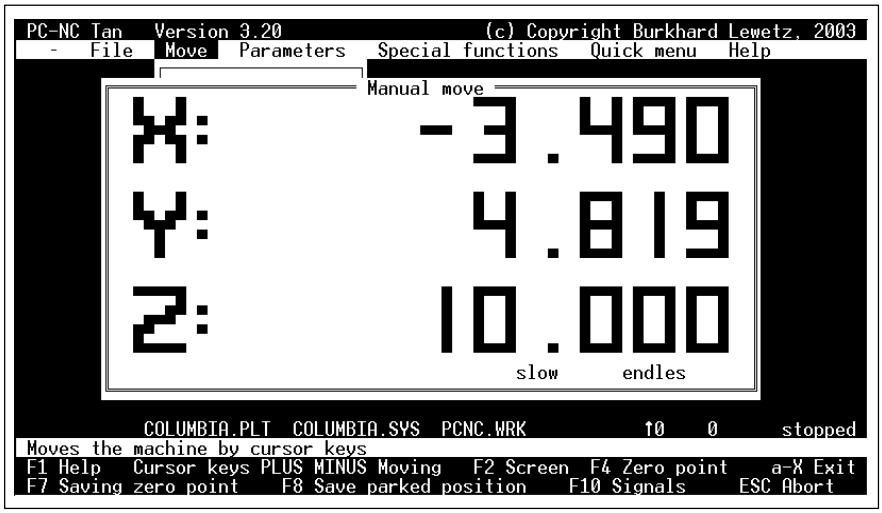


Figure 3-8 Manual movement, step counters in large digits

MOVE-PARK

The PARK function moves all the axes to the defined parked position in a specific sequence. The Z-axis is moved upwards first, after which the X and Y-axes are moved.

Defined parked position

The parked positions for all 3 axes are defined in the parameters.

You can move the axes to the parked position during a pause in the job process (this may be necessary for a bit change, for example). Reactivating the PARK function causes the bit to return to its previous position.

The PARK function can also be activated by pressing the (F9) function key.

MOVE-REFERENCE MOVE

The REFERENCE MOVE function moves all axes to their reference switches in a defined sequence.

Machine reference point

Each axis starts moving at reference speed 1 and searches for the reference switch. The axis stops when the switch changes its level, at which point the axis moves in the opposite direction away from the switch at speed 2. The edge of the reference switch defines the reference point for each axis.

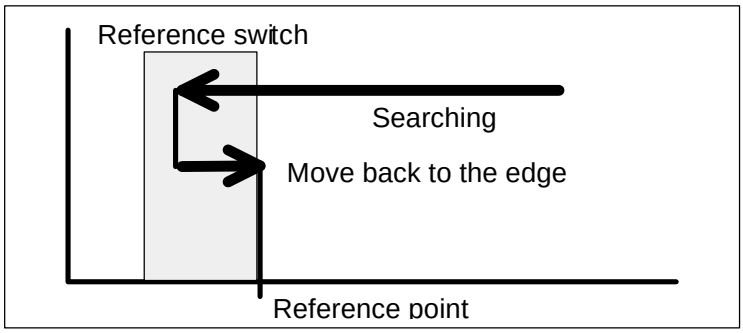


Figure 3-9 Procedure involved in reference movement to the reference switch

PC-NC needs to know the current position of all axes. This is why reference movement is essential before a job process can be started.

Reference movement can also be triggered by pressing the (F8) key.

MOVE-CALIBRATE

Calibrating the workpiece

The calibration function is used for clamping the workpiece correctly. First of all, the job file is loaded, the extent of the data is determined and then all data are displayed in graphic format. Four significant points, usually the calibration points, are marked with a wide ring.

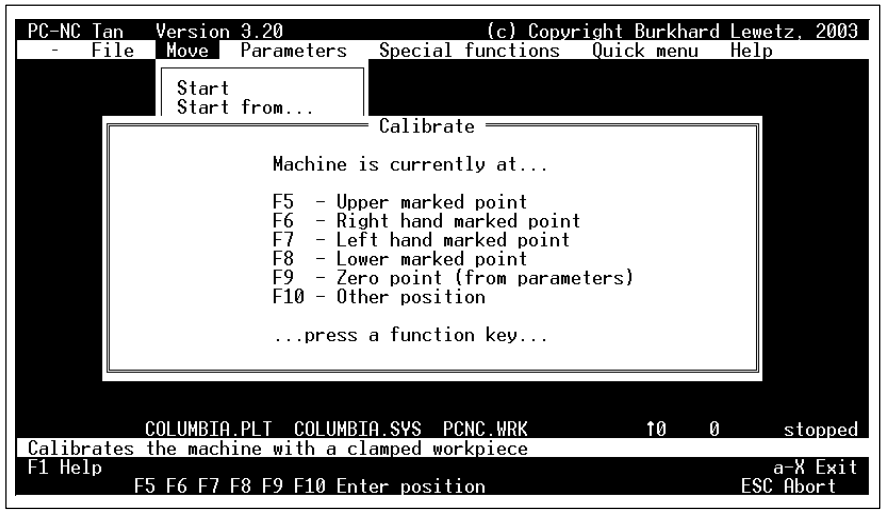


Figure 3-10 Start of the CALIBRATE function

It is possible to specify where the machine is currently located at the start of the calibrate function. If the marking points are known in advance, it is recommend that the machine be moved to one of them before calibration starts.

In this case, ***PC-NC*** calculates the required zero point automatically and updates the co-ordinates in the parameters.

Moving to significant points

The (F5), (F6), (F7) and (F8) function keys can be used to instruct the machine to move to these positions in order to calibrate the workpiece clamping.

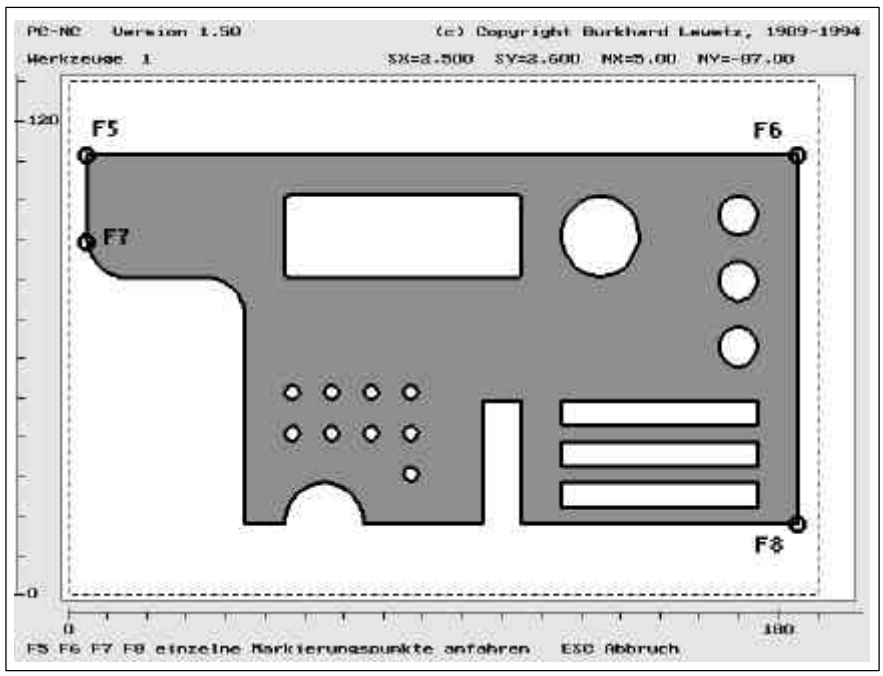


Figure 3-11 Graphic display of the CALIBRATE function with 4 marking points

It is possible to move to and fro between the 4 marking points as often as required. Press the (ESC) key to cancel the procedure.

Typical procedure for calibrating a workpiece:

Procedure for calibrating a workpiece

1. Perform reference movement with the machine.
2. Use the MANUAL MOVE function to move the machine to one of the four marking points.
3. Call up the calibration function and specify where the machine is currently located.
4. Press the (F5), (F6), (F7) and (F8) as often as required in order to move to and fro between the marking points, then align the work piece.
5. Store the parameters with the automatically calculated zero point.

MOVE-TARGETED MOVE

Moving to defined positions

Use the TARGETED MOVE function when you need to move to precisely defined and known positions.

The dialog box can be used for three different target positions. You can select an absolute position in machine co-ordinates (in relation to the reference point) or a relative position in workpiece co-ordinates (in relation to the zero point).

Moving distances

A third option is to define a specific distance from the current position, for instance to move the Z-axis down by 2.5 mm.

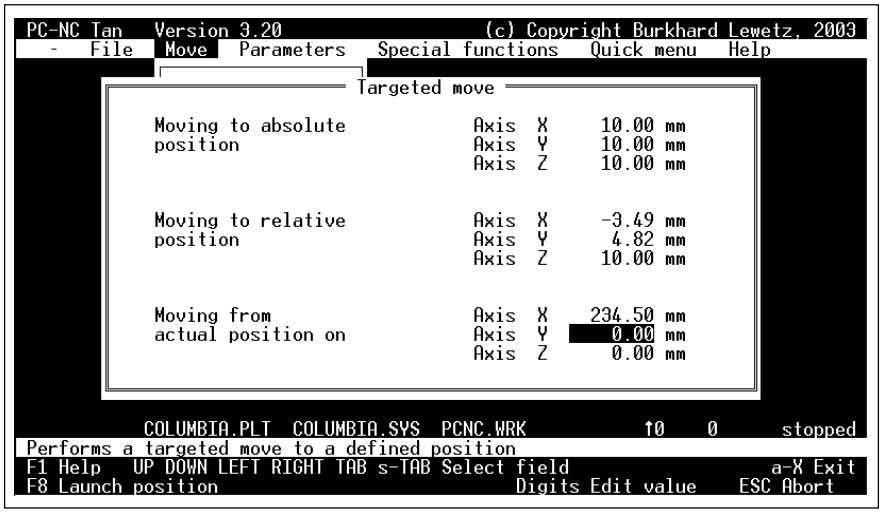


Figure 3-12 Targeted movement

The following key assignments are active:

(UP)(DOWN)	Selects the entry field
(TAB)(sTAB)	Selects a group of entry fields
(ENTER)	Confirms position entries
(F8)	Moves to the target position
(ESC)	Cancels targeted movement

The TARGETED MOVE function can also be activated from the manual movement mode by pressing (alt-F9). It is available in Teach-In.

MOVE-SELECT BIT...

Define first used bit

During normal operation ***PC-NC*** always knows the current mounted bit.

In certain situations it must be necessary to tell ***PC-NC*** which bit to use. This can be...

- after program start if a tool is already mounted
- when using no tool change and you want to use the settings of a certain tool
- if you want to release the tool into the magazine of an automatic tool changer

By choosing the function a menu is opened and you can select the desired bit easily.

Set the molette

In addition you can set the molette of the ATC to open/close state.

To use the automatic tool changer please see the detailed information in chapter 7.

MOVE-CHECK BIT LENGTH

Check length automatically

PC-NC has the functionality to check the length of a bit and to compensate different lengths during a job.

To use this function a length sensor must be mounted and defined.

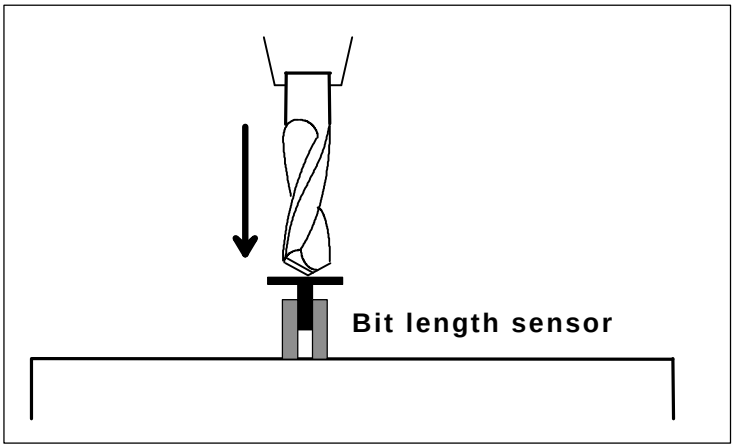


Figure 3-13 Bit length sensor

A detailed explanation of this function can be found in chapter 7.

Length of the reference bit

The function CHECK BIT LENGTH must be used to measure the length of the reference bit. This is the first bit used at the job start. The zero point of the Z axis must be defiend as well. After a tool change in the job, ***PC-NC*** checks the length of the new bit automatically and compensated the differences in length to the first bit.

By using the check length function, ***PC-NC*** calculates the distance from the bit's end to the defined zero point and stores it to the parameters.

MOVE-SINGLE STEP

The SINGLE STEP menu item is a switch which can be active or inactive.

✓ SINGLE STEP

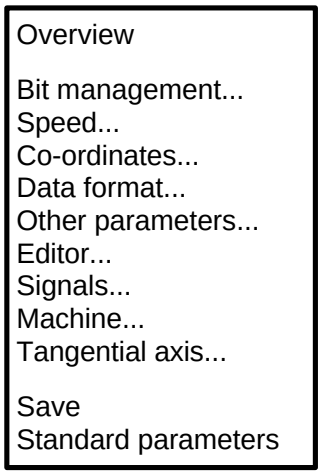
Single command mode

The job process stops after each command when the single step function is active. Implementation only continues after being restarted by pressing the (F3) key.

This function can be used as an easy method of checking individual commands and movement distances. It is possible to change over from single step to continuous operation at any time during a pause.

3.2.4. PARAMETERS menu

The PARAMETERS menu contains all the adjustment options in ***PC-NC***. It is subdivided into several sub-menus in order to split up the parameters into functions.



The hotkey for activating the PARAMETERS menu is (P). The individual parameters are explained in detail in chapter 5.

PARAMETERS-OVERVIEW

All parameters in overview

You can use this function to gain a complete overview of all current parameters. ***PC-NC*** displays all the parameters and their associated values in a window.

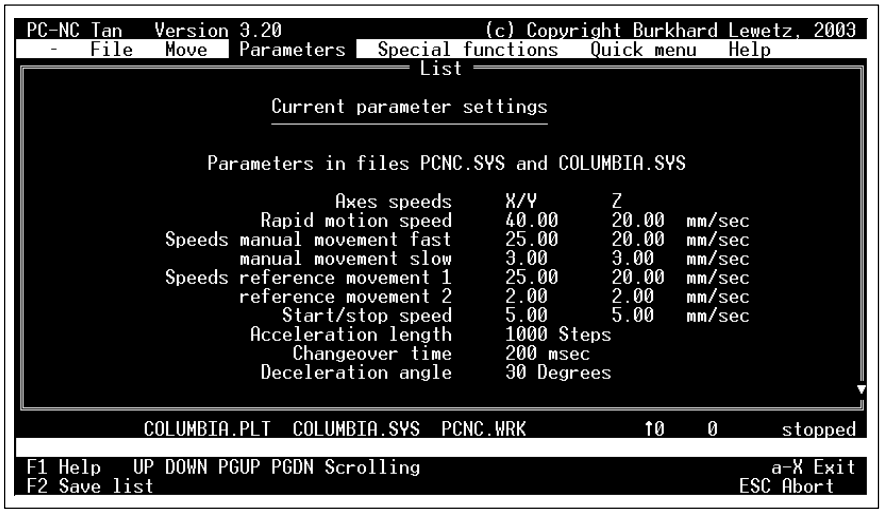


Figure 3-14 Overview of parameters

In addition to the parameters, the overview also shows which parameter files are currently loaded and whether any changes have been made to individual parameters in relation to the stored values.

Permanently saving the overview

Press the (F2) key in order to save the overview display permanently in the PCNC.PAR file. This text file can be used later for archiving purposes, or it can be printed out.

PARAMETERS-SAVE

Saving parameters for a specific project

All parameters and bit settings can be saved in files using the SAVE menu item. It is possible to save the settings as belonging to a job file or a project.

The save function operates like this: ***PC-NC*** saves all workpiece-specific settings such as the zero point, scaling, etc. in a parameter and bit file which belongs to the job file if there is a job file loaded. The files have the same name as the job file and their filename extensions are *.SYS and *.WRK, e.g. COLUMBIA.SYS or NOZZLE.WRK.

Whenever these job files are loaded later, all the settings and bits are restored without them having to be redefined.

The machine-specific parameters such as axis resolution, limit and reference switches or the port used are always saved in the standard parameter file, PCNC.SYS.

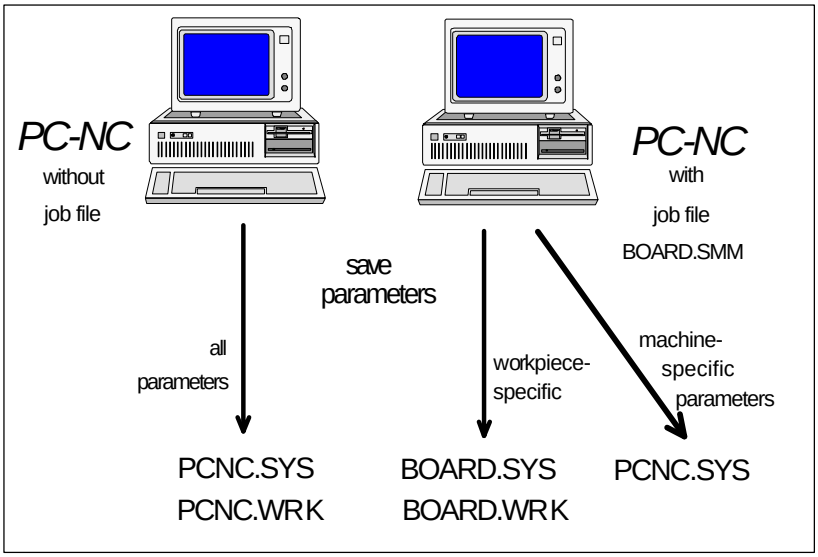


Figure 3-15 Saving parameters and bit settings

The SAVE function saves all settings in the loaded parameter and bit files PCNC.SYS and PCNC.WRK if no job file is specified during saving.

The division into two parameter files offers the advantage that all machine parameters are only saved once and only need to be saved in this file whenever changes are made.

PARAMETERS-STANDARD PARAMETERS

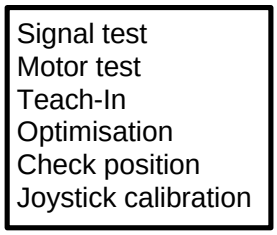
The STANDARD PARAMETERS function is used in ***PC-NC*** for loading the parameters which were active when the program was launched.

This makes it possible to restore the previous status in case your attempts to change the parameters prove unsuccessful.

The program either loads the PCNC.SYS and PCNC.WRK files or the files specified in the launch command line switches -s and -w.

3.2.5. SPECIAL FUNCTIONS menu

The special functions menu provides access to two test functions which can be used to run systematic checks on the mechanical equipment and in order to determine whether the parameter settings are correct and, indeed, at their optimum state. In addition, this menu also contains the following functions: TEACH-IN, OPTIMISATION and CHECK POSITION.



SPECIAL FUNCTIONS-SIGNAL TEST

In addition to the 8 data lines, the parallel printer port also provides 4 extra outputs and 5 inputs for external signals. The clock pulse and direction signals for the stepper motors are connected through the 8 data lines.

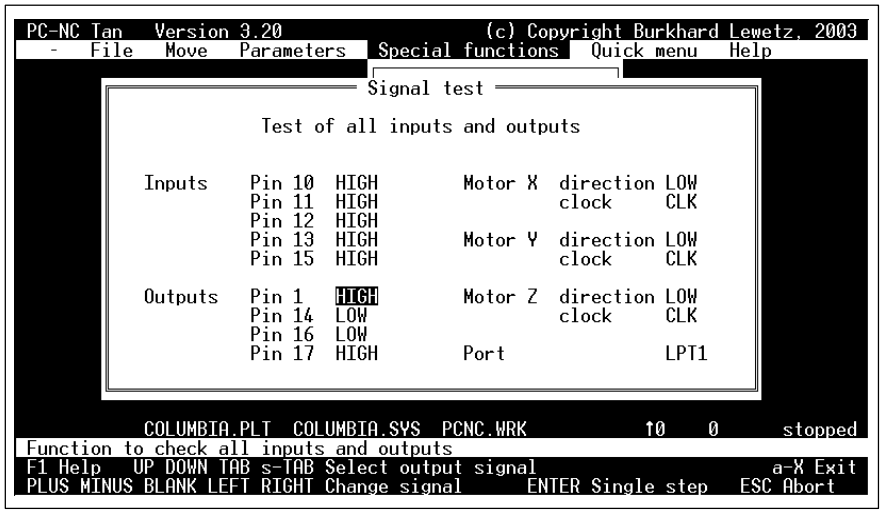


Figure 3-16 Signal test

Checking the port and the signals

PC-NC uses the four extra lines for the following extra signals.

- **Drilling spindle on/off** (pin 1 / relay 1)
- **Coolant on/off** (pin 14 / relay 2)
- **Molette signal** (pin 16 / relay 3)
- **Current reduction** (pin 17)

The SIGNAL TEST function provides a straightforward method of testing the input and output signals from ***PC-NC*** to the machine and for determining the connector assignment.

All inputs and outputs visible

The window displays all outputs and the 5 inputs of the port. The input signals are sampled and displayed continuously which allows the switching of the reference switches to be observed very effectively.

The level of the four extra output signals can be changed easily by pressing the (+), (-) or (SPACE) keys. The machine's reaction can be observed immediately. For example, it is possible to hear the relays switching in the case of pin 1 and pin 14.

The direction signal of each of the three motors can be changed. The (ENTER) key outputs a clock pulse (CLK) each time it is pressed.

It is probable that you have selected the wrong port if the switches on the machine do not produce any reactions in the window and changing the levels of the output signals does not initiate any response either.

SPECIAL FUNCTIONS-MOTOR TEST

The MOTOR TEST special function is used for determining the optimum speed settings. A window shows all the parameters which are relevant for calculating the steps.

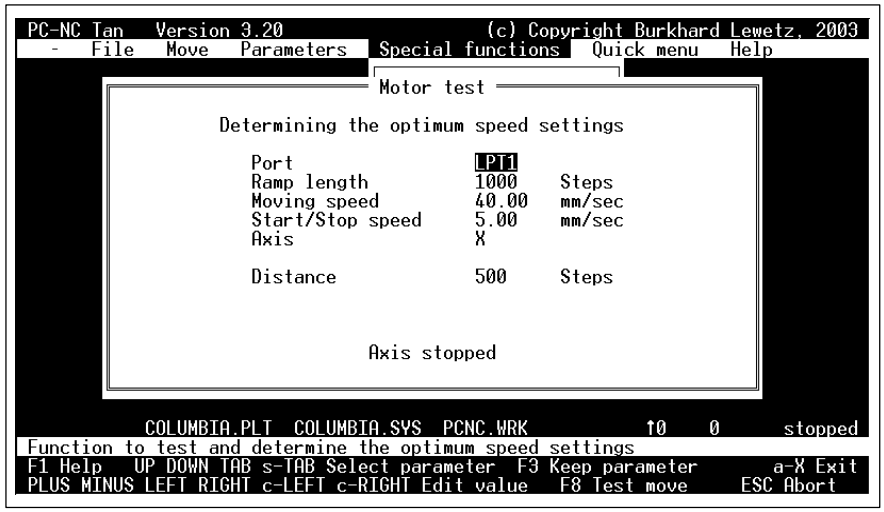


Figure 3-17 Motor test

It is simple to change the values in the parameter fields by pressing the (LEFT) and (RIGHT) cursor keys. The fields themselves can be selected using (UP) and (DOWN).

Testing the motor response

The (F8) key can be used for testing the current settings. In this test run, the speed and ramps for the current motor are tested and movement is performed in alternating positive and negative directions.

The motor response can be tested in a straightforward manner during the test run. It is also easy to check the influence of the individual parameter values on the movements. Pressing the (ESC) key cancels the test run.

Optimum parameters

The values for an axis have attained their optimum status when the motor starts moving quickly without any step losses. It must also generate adequate torque when running at maximum speed.

Step-by-step adjustment

Step-by-step procedure for testing motors X/Y and Z:

1. Switch off the ramp and slowly increase the start/stop speed until the motor reaches its stall point. Then reduce the speed by 30 - 40 %.

2. Test various combinations of ramp length and ramp profile. The values are good if the motor starts quickly and does not stall.

3. Increase the rapid motion speed gradually. The motor should run quickly but nevertheless generate sufficient torque.

You can press the (F3) key to have the parameters determined by this method adopted directly. All the parameters and their functions are explained in chapter 5.

SPECIAL FUNCTIONS-TEACH-IN

Generation of small job files by hand

The TEACH-IN function makes it possible to generate small job files by hand using a very simple method. This means there is no need to use a CAD system for generating the NC data when the workpieces are not very complicated.

The TEACHIN function generates HPGL files and drill files only. To generate G code files please use the editor.

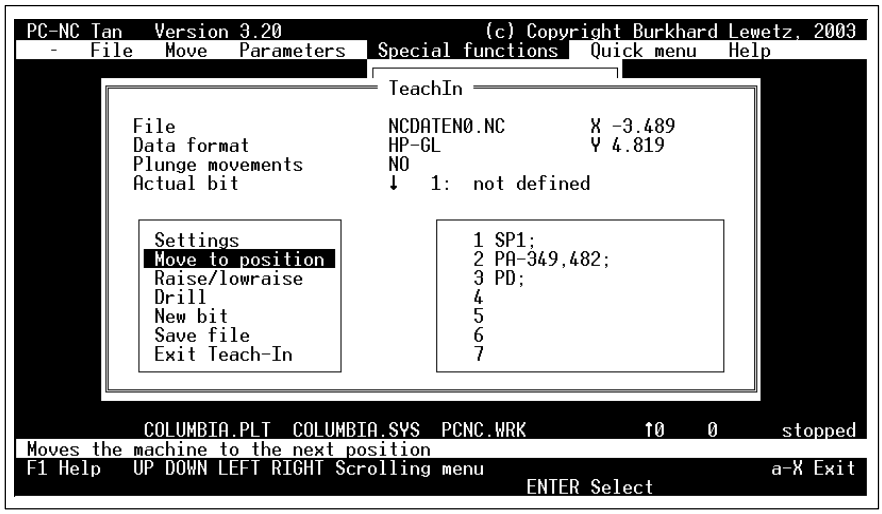


Figure 3-18 TEACH-IN function

The Teach-In window is divided into different areas. The top part contains the current settings. To the right there is a small window which shows the commands which have been saved so far. The box on the left contains a menu for accessing all actions which are part of the TEACH-IN function.

Information about the individual actions:

- **Settings**

This menu item is used for changing settings which are relevant to the TEACH-IN function. For example, it is possible to define the name of

the job file to be generated. In addition, you can specify whether the format of the file should be HP-GL or drilling format. A YES/NO switch is used for defining whether the Z-axis plunge depth should really be moved or only simulated. Finally, it is also possible to specify which is the first bit.

You cannot change the file format once TEACH-IN has started. It is only possible to select undefined bits provided that no real plunge depths have been selected; otherwise, only defined bits can be chosen.

- **Move to position**

PC-NC branches to the MANUAL MOVE function when this menu item is selected. This makes it possible to move to a new target position with the X and Y-axes. The Z-axis cannot be moved with this action. In addition, pressing the (F9) function key makes it possible to repeat the movement of the distance that was just completed. This grid movement is a valuable tool when moving around IC sockets, for example.

- **Raise/lower bit**

This action is used for operating the bit. The bit inserted is raised or lowered, depending on the current position. (This menu item is only active with HP-GL format.)

- **Drill**

This menu item can be used for drilling a hole after a new target position has been reached. The bit is lowered briefly and then raised again immediately. (This menu item is only active with drilling format.)

- **New bit**

A new bit can be selected using a small sub-menu. It is only possible to select undefined bits provided that no real plunge depths have been selected; otherwise, only defined bits can be chosen.

- **Save file**

All actions performed so far, such as move to target position, select and operate bit or drilling, are stored in the required file. They are then available for subsequent automatic operating processes.

- **Exit Teach-In**

This menu item exits the TEACH-IN function. You get the opportunity to save the actions performed so far if you have not already done.

Teach-In procedure

Typical procedure of a TEACH-IN session:

1. Perform reference movement with the machine.
2. Move the Z-axis or all the axes to the zero point.
3. Call up the TEACH-IN function and make the settings for the file name, file format and first bit.
4. Move to the required target position.
5. Drill at this point or move the bit.
   ... Repeat steps 4 and 5 as often as required ...
6. Save the file and exit TEACH-IN.

All current parameters and bits are also saved when the actions are stored in the new job file. A file generated in this manner can subsequently be used and processed like any other job file.

SPECIAL FUNCTIONS-OPTIMISATION

Sorting the NC data

The OPTIMISATION function ensures that the job file is processed efficiently by sorting the NC commands it contains.

The outputs from CAD and layout programs are not always optimised which means the bits have to be inserted several times and the machine frequently has to make unnecessary unproductive movements. In some cases, contours may not be closed but are instead completed in two or more passes.

The optimisation function performs 3 tasks:

1. Sorting the data according to bits so each bit only has to be inserted once.
2. Closing contours so there is no need to start at existing line ends.
3. Ensuring that machining is always resumed at the nearest co-ordinates possible in order to minimise unproductive movements.

Optimising the current job file

The optimisation function can only ever be performed on the job file which is currently loaded. This means it is inactive if no file has been selected yet.

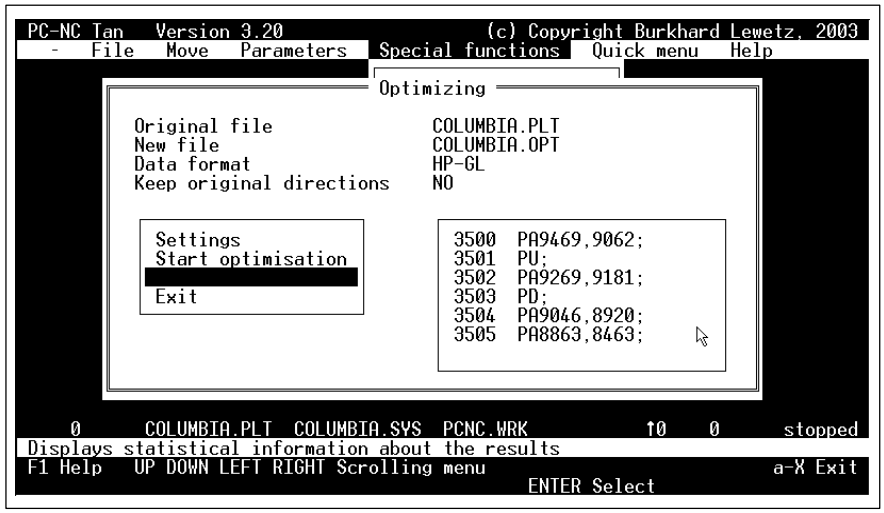


Figure 3-19 Optimising data

The calculation of the truly optimum sequence of movements is a complicated mathematical problem which requires a considerable time to compute. As a result, even ***PC-NC*** does not necessarily produce the optimum solution. However, it does usually give a more effective sequence than the initial file.

The window for the OPTIMISATION function is divided into different areas. The top part contains the current settings. To the right there is a small window which shows the individual commands. The box on the left contains a menu for accessing all actions.

Information about the individual actions:

- **Settings**

The possible settings contain the definition of a file name which should be used for the new file. In addition, it is possible to use a YES/NO switch to specify whether the original movement directions should be retained or if they can also be changed as well for optimisation reasons. The direction of movement should always be retained in engraving and milling data; in contrast this is of no significance for drilling applications.

- **Start optimisation**

Optimisation starts when this menu item is selected. First of all, ***PC-NC*** loads the initial file and determines the dimensions. After this, it sorts the individual commands and finally writes the commands to the target file in the calculated order.

- **Display results**

This menu item provides statistical information about the results when the optimisation process has been completed. The display shows a comparison between the number of bit changes, bit applications and the total length of unproductive movement before and after optimisation.

- **Exit**

This menu item is used for exiting the OPTIMISATION function. The file is saved together with all its parameters and bits. From this point on, it is the active job file.

The optimisation function works with HPGL and drill files only.

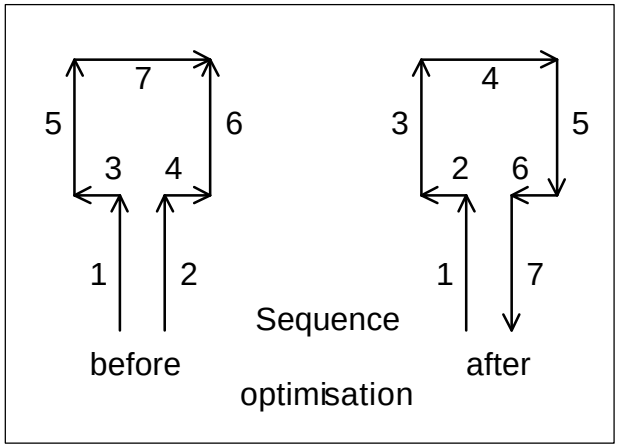


Figure 3-20 Results of the optimisation

Typical procedure

Typical procedure of an optimisation session:

1. Select the job file, i.e. the file to be optimised.
2. Call up the OPTIMISATION function and define the new file name and the movement directions.
3. Start optimisation.
4. Check the results of the optimisation if necessary.
5. Exit the OPTIMISATION function.

External optimisation program

Optimisation requires a great deal of memory. Your computer might not have enough main memory to process large NC files. In such cases, you can perform the optimisation using the external program ***HPGL-Opti***. HPGLOPTI.EXE ships with ***PC-NC*** and it is capable of optimising NC files of any size. Instructions for using it are contained in chapter 6.

SPECIAL FUNCTIONS-CHECK POSITION

The CHECK POSITION function is another of ***PC-NC***'s test functions. You can use it for checking the accuracy of the reference switches or the current position.

Checking the reference position

The process is very similar to the REFERENCE MOVE function. All axes are moved to their reference switches in turn in a targeted manner. Following this, the difference between the actual and expected positions is displayed.

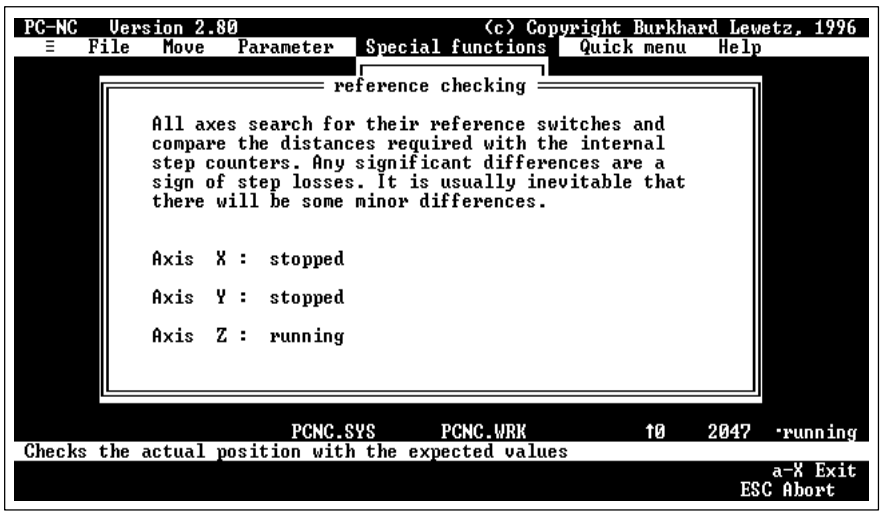


Figure 3-21 Checking the distance to the reference point

Kleinere Schrittdifferenzen können durch die Schalter bedingt sein, große Differenzen deuten dagegen auf vorherige Schrittverluste hin.

Small step differentials may be due to the switch whereas large differentials indicate previous step losses.

It makes sense to use the CHECK POSITION function when:

- You suspect that step losses have led to a collision.
- You want to determine the maximum machining speed for a bit or a piece of material by finding out if there have been any step losses.
- The position was moved during a bit change.

It is only possible to check the reference position providing reference movement has been performed and no positioning movements were interrupted by pressing (ESC).

SPECIAL FUNCTIONS-JOYSTICK CALIBRATION

To use a joystick in manual movements it must be calibrated first. The function leads you thru in several steps and saves the defined angles and values as parameters.

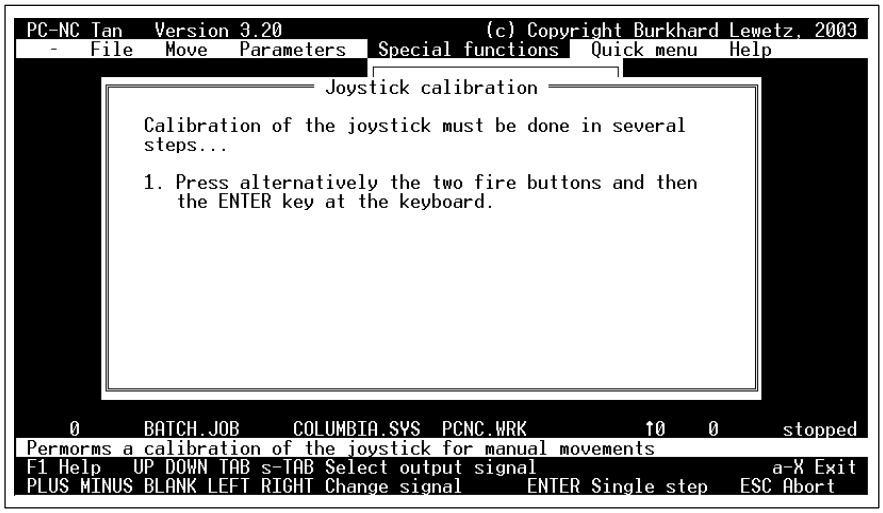


Figure 3-22 Special function JOYSTICK CALIBRATION

The joystick calibration must be done before using it.

3.2.6. QUICK menu

Important functions at your fingertips

The Quick menu gives you quick access to the most important functions required during volume production of workpieces. This menu does not allow the parameters or the machine to be changed any further.

Figure 3-23 Quick menu

The Quick menu appears in the middle of the screen when it is activated. It can be used like all other selection menus.

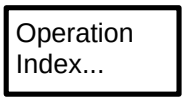
The Quick menu enables the following functions to be activated:

- **OPEN FILE**
- **DISPLAY**
- **START**
- **REFERENCE MOVE**
- **PARK**

You need to exit the quick menu by selecting the last item before you can make any changes to the parameters or to the machine, for example manual movements.

3.2.7. HELP menu

The help menu comprises two items:



HELP-OPERATION

The HELP-OPERATION function provides a detailed explanation about the possible ways of operating ***PC-NC***.

It explains the possibilities of using the menu system as well as how to activate individual functions rapidly using the function keys.

HELP-INDEX

The HELP-INDEX... function gives access to all available texts in the help system. A window opens displaying key words to all the topics.

Figure 3-24 Help index

3.3. Context-sensitive help system

PC-NC ships with a context-sensitive help system. This means it is possible to call up a relevant help text at almost any point in the program.

The help function is not active when motors are running or the program is in graphic display mode.

Help when you need it

Press the (F1) key to activate the help system and press the (ESC) key to leave it again. The explanatory text appears in a window.

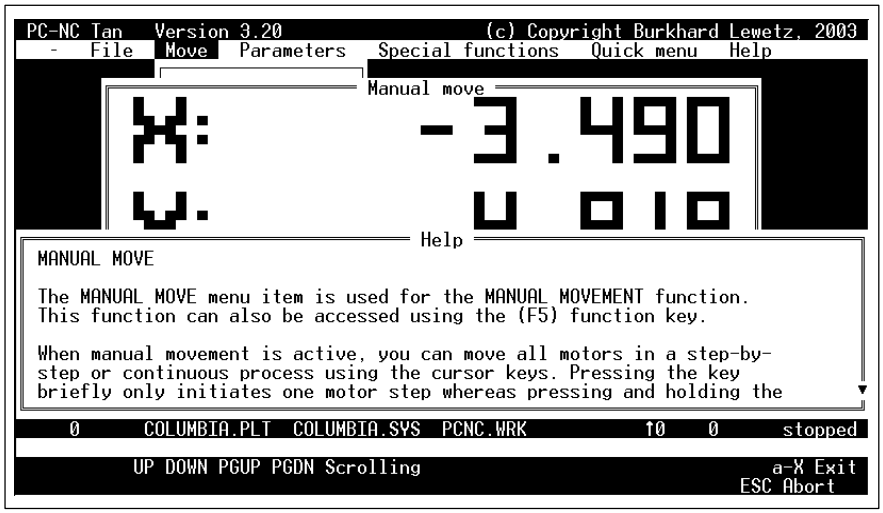


Figure 3-25 Context-sensitive help

You can scroll through the text using the cursor keys. All help texts are also accessible via the HELP-INDEX... function.

4. Batch jobs

4.1. What are batch jobs?

Perform complex processes using batch jobs

PC-NC can process batch jobs which are similar to MS-DOS batch files. A job consists of several individual commands. It allows the operation of ***PC-NC*** to be automated.

Batch jobs are ASCII files and can be created using the editor. The file must have the filename extension *.JOB.

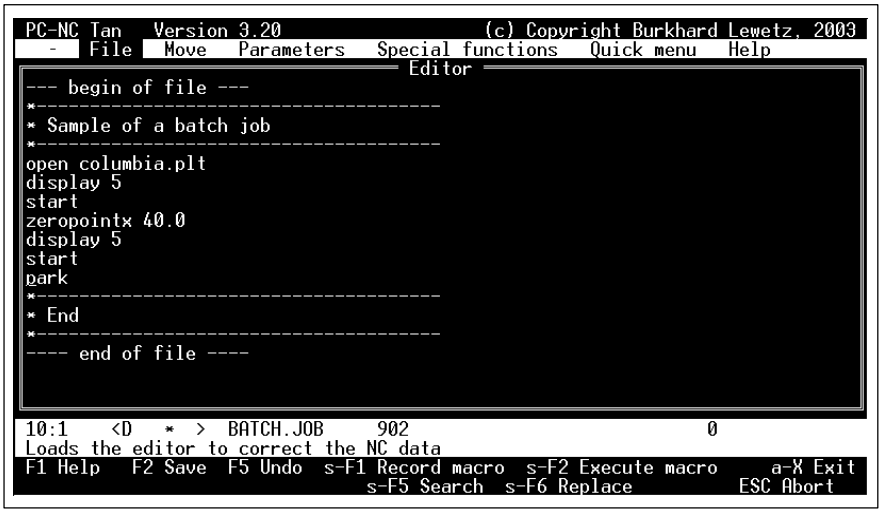


Figure 4-1 Editor with batch job loaded

PC-NC is capable of running batch jobs

The individual batch commands either relate to specific functions of ***PC-NC*** or they cause parameter values to be changed.

Example of a simple batch job:

```
OPEN circle.plt
REFERENCEMOVE
START
```

```
PARK
EXIT
```

4.2. How does *PC-NC* process batch jobs?

PC-NC loads batch jobs just like normal NC files. Batch jobs can be started in this way as well. Processing is started using the MOVE-START function or by pressing the (F3) key.

batch jobs are executed line-by-line

PC-NC reads the job line-by-line, interprets the commands and executes them. Each executable command can be displayed briefly before it is carried out.

Incorrect commands or faults cause a message to be generated. The processing of the batch job is stopped. The fault must be rectified before the batch job can be restarted.

The example above would be processed by ***PC-NC*** as follows: First of all, the CIRCLE.PLT plot file would be opened and then reference movement would be performed. After this, the job process would start using the loaded file. On completion, the machine would be moved to its parked position and ***PC-NC*** would be exited. Each command is displayed for about 2 seconds before it is carried out.

Certain functions cannot be activated when a job is loaded. For example, Teach-In, Optimisation and Data format are unavailable.

4.3. Available batch commands

Each batch command must be in a new line. Remark lines are prefixed by * or ;. Blanks are also allowed.

The possible batch commands are:

AUTOSCALE	Skales the NC data to 90% of the working area, leaves the axes ratio
AUTOSCALE100	Skales the NC data to 100% of the working area, leaves the axes ratio

AUTOSCALEXY	Skales the NC data to 90% of the working area, each axis independent
AUTOSCALE100XY	Skales the NC data to 100% of the working area, each axis independent
BITWITHDRAWALSPEED x	Defines a new withdrawal speed for the bit
CENTER	Sets the zero point to the center of the file
CLEARSTEPCOUNTER	Clears all step counters and defines a new zero point
CLEARSTEPCOUNTERX	Clears the X step counter
CLEARSTEPCOUNTERY	Clears the Y step counter
CLEARSTEPCOUNTERZ	Clears the Z step counter
COOLINGOFF	Switches the coolant output off
COOLINGON	Switches the coolant output on
COOLINGONTIME x	Switches the coolant output on for a defined time
DISPLAY x	Displays the current file in graphic mode for x seconds.
DOS command	Executes a DOS command and then returns to ***PC-NC***. Not all your computer's memory is available.
EXIT	Exits ***PC-NC***.
GOTO x	Continues the batch job from the specified line.
GOTOXTIMES l,x	Continues the batch job from the specified line but only x times
LABEL	Defines a lable for subsequent GOTO commands
MAXX x.xx **MAXY x.xx** **MINX x.xx** **MINY x.xx**	Defines a new working area.

MOVEX x	Moves axis X
MOVEY x	Moves axis Y
MOVEZ x	Moves axis Z
MOVEXY x,y	Moves axis X and Y
MOVEXYZ x,y,z	Moves axis X and Y and Z
OPEN file	Opens the specified NC job file, a parameter or bit file.
PARK	Moves to the parked position.
PARKX x.xx **PARKY x.xx** **PARKZ x.xx**	Defines a new parked position.
PAUSE x	Defines the time during which each executable command is displayed before being carried out; default value 2 seconds.
RAPIDSPEED x	Defines a new rapid speed
REFERENCEMOVE	Performs reference movement.
SCALINGX x.xxx **SCALINGY x.xxx**	Sets new scaling factors.
SPINDLEON	Switches the spindle output on
SPINDLEOFF	Switches the spindle output off
SPINDLEONTIME x	Switches the spindle output on for a defined time
START	Starts a job process with the current NC file.
STARTFROM x	Starts a job process from command x.
STOP	Stops the execution of the batch job and waits until any key is pressed.
WAITPIN x	Waits for an external signal. x defines the pin number 10, 11, 12, 13 or 15 to which the signal is connected. The sign specifies the signal level (+ HI, - LO).

WAITTIME x	Waits for the defined time in seconds
ZEROPOINTX x.xx **ZEROPOINTY x.xx** **ZEROPOINTZ x.xx**	Defines a new zero point.

4.4. Sample batch jobs

Machining a workpiece using several job files

```
; Milling and drilling a board
open board.plt
referencemove
start
open board.smm
start
park
; End of batch job
```

This batch job mills an inserted board and then immediately drills it in one operation. Naturally, the calibration and the parameters must be set and saved prior to starting the batch job.

Machining several workpieces in succession

```
; 3 workpieces offset by 12.5 cm each
open front.plt
zeropointx 30.00
referencemove
start
zeropointx 155.00
start
zeropointx 280.00
start
park
; End of batch job
```

In this case, an X-axis zero point is defined for the first workpiece and then the zero point is shifted through the required distance before each subsequent machining operation is started.

Mass production with external start signal

```
; Clamp workpiece and start by pressing a pedal switch
open part.plt
referencemove
```

```
waitpin -15
start
goto 4
; End of batch job
```

The workpiece is first loaded and reference movement is carried out. ***PC-NC*** then waits for an external start signal from the pedal switch. The signal is applied to pin 15 and must become LOW. Following the machining operation, the execution of the batch job is resumed in line 4, i.e. waiting for the next start signal.

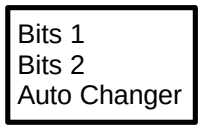
5. Parameter settings

5.1. Bit management

Project-specific bit management

PC-NC comes with a user-friendly bit management function. It is possible to define up to 10 bits with special values for each job project.

The bit parameter settings are divided into 3 dialog boxes.



PC-NC saves all values defined in this dialog box to the bit file using the PARAMETERS-SAVE function. Bit files have the filename extension *.WRK.

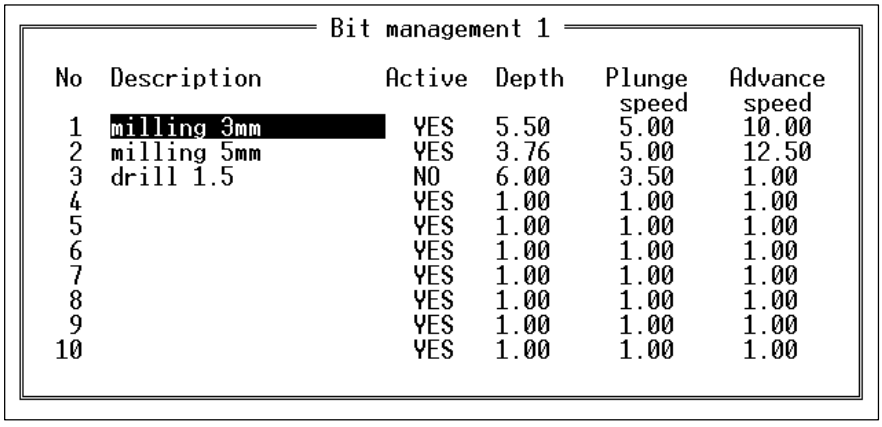


Figure 5-1 Bit management (dialog 1)

All bits used are marked with an asterisk after their number if the loaded file has already been displayed once.

Description

Meaningful bit names

Each bit can be identified by a name up to 20 characters in length. This name is used in prompts to change the bit.

Activation

The plunge depth specifies the Z-axis travel downwards into the workpiece for each bit. The depth is defined in millimetres and always refers to the distance from the plane of the zero point.

Plunge depth

The plunge depth specifies the Z-axis travel downwards into the workpiece for each bit. The depth is defined in millimetres and always refers to the distance from the plane of the zero point.

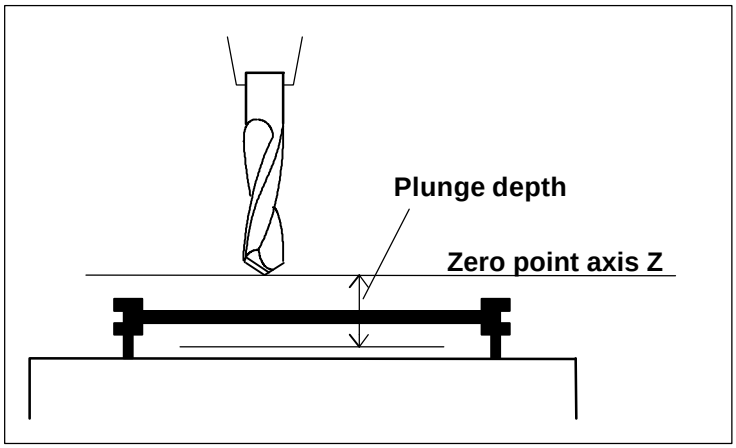


Figure 5-2 Plunge depth measured from the zero point of the Z-axis

Plunge speed

The plunge speed specifies the speed with which each bit is inserted into the workpiece. Certain limiting values must be observed here in accordance with the material and the bit.

The bit withdrawal speed out of the workpiece, i.e. for moving the Z-axis upwards, is defined in the speed parameters.

Advance speed

The advance speed for each bit defines the working speed when the bit is inserted in the workpiece.

This value is not significant for purely drilling applications. However, the maximum advance speed is dependent on the bit used and the material when ***PC-NC*** is used for milling, engraving or grinding.

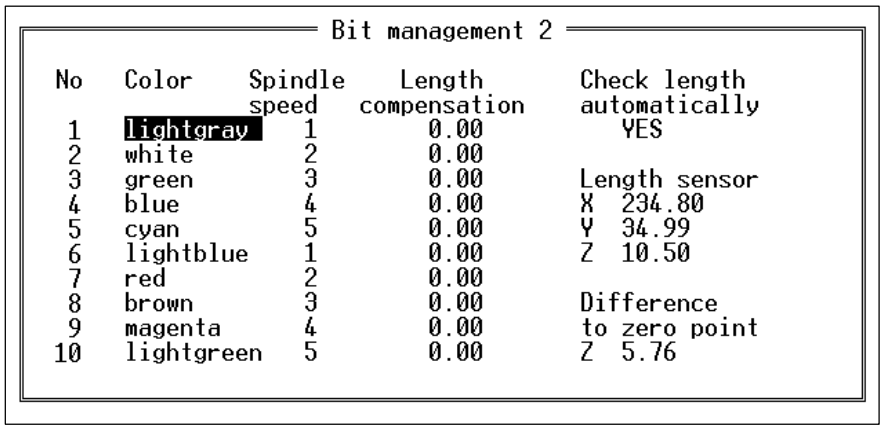


Figure 5-3 Bit management (dialog 2)

Color

Colors in graphical preview

PC-NC can assign colors to each used bit which will be used in graphical display. So it is easy to use the same colors like your CAD program does.

The following colors are available :

blue	**magenta**	**lightblue**	**lightmagenta**
green	**brown**	**lightgreen**	**yellow**
cyan	**lightgray**	**lightcyan**	**white**
red	**gray**	**lightred**	

Spindle speed level

Spindle speed with 4 digital output lines

PC-NC can assign one of 15 levels of spindle speed outputs to 4 output lines. You can use a second or third printer port for generating the output signals.

The 15 levels will be assigned to 4 bits in a defined port address.

Bit length compensation

Compensation of different tool lengths

PC-NC can compensate different bit lengths while executing a job. This means that the different lengths will be moved in addition or too less when moving the Z axis. The defined plunge depth will be reached in every time.

The different lengths can be measured by ***PC-NC*** automatically with the help of a mounted length sensor.

Manual definition or automatic measurement

Of course you can define the differences in bit length by hand as well. More comfortable is the use of the automatic measurement function.

Bit length sensor and height difference

Automatic measurement and compensation

By use of a tool length sensor ***PC-NC*** can automatically check the tool length and compensate it during the job. The sensor must be touchable from top and switch a input line. The connection and definition is the same as for the reference and limit switches.

The parameters define the starting point for the automatic measurement somewhere above the length sensor. From this point ***PC-NC*** moves the bit down until the sensor is reached and the input line switched. The moved distance will be calculated together with the defined difference to the zero point and saved as length parameter.

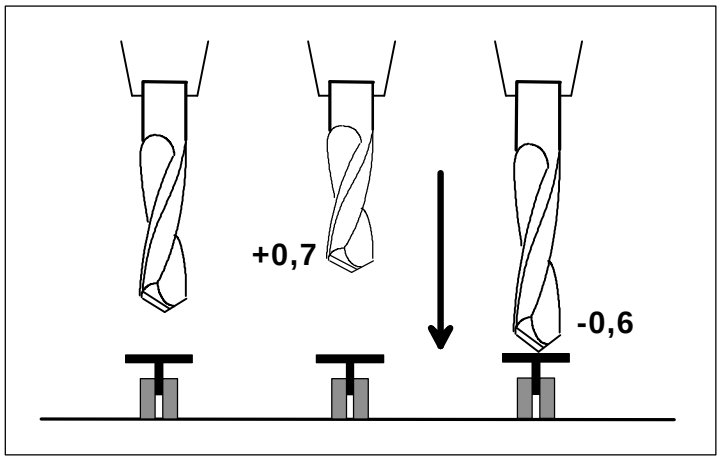


Figure 5-4 different bit lengths

The difference to the zero point defines the distance from the switching point of the sensor to the defined zero point of axis Z of the current job. Normally you use this parameter when measure the first

bit used in the job. By activating the function CHECK BIT LENGTH in the move menu ***PC-NC*** checks this difference automatically.

Check length automatically

If you want ***PC-NC*** to check bit lengths automatically after every tool change command, set this parameter. Otherwise the tool length is not measured, the compensation works anyway.

Automatic bit changer

No	Magazine position Axis X	Axis Y
1	310.00	10.00
2	330.00	10.00
3	350.00	10.00
4	370.00	10.00
5	390.00	10.00
6	410.00	10.00
7	430.00	10.00
8	450.00	10.00
9	470.00	10.00
10	490.00	10.00

Release

	Axis	Distance
1	X	11.00
2	Z	5.60

Grab bit

	Axis	Distance
1	Z	5.60
2	X	11.00

Z lift 23.50

Dwelltime 500

Figure 5-5 Automatic tool changer

Magazine positions

Positions for the automatic tool changer

By using an automatic bit changer, ***PC-NC*** must know about the magazine positions of all bits.

Each time a bit is picked up or released ***PC-NC*** moves to the defined magazine position and executes the commands for releasing or picking up.

The Z co-ordinate will be the same for all magazine positions.

Commands for picking up and releasing

Single move commands

By using the automatic bit changer you can define two individual movements for picking up or releasing a bit.

Example : *Z 2,5*
Y 1,25

After moving to the magazine position ***PC-NC*** lowers the Z axis by 2,5 mm and then moves the Y axis for 1,25 mm. After that the molette will be opened to release the bit.

The moving back to the defined magazine position the movements will be made in the reversed order after the defined dwell time.

Dwell time

The defined dwell time elapses when the magazine position is reached and before ***PC-NC*** moves out again. It is defined in milliseconds.

The use of the automatic tool changer must be enabled in *Other parameters...*

5.2. Speeds

All the parameters which affect the speeds of the three motors are grouped together in the Speed dialog box.

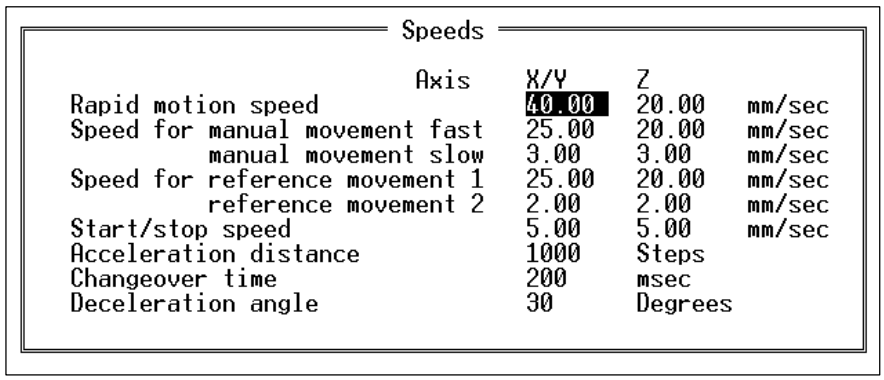


Figure 5-3 Speed parameters

The optimum parameter values for the required speeds can be determined using the MOTOR TEST function.

Rapid motion speed

Speed with bit withdrawn

Rapid motion speed is used in order to move to a new position when the bit is not in the workpiece.

These are unproductive movements which ***PC-NC*** can always perform at the maximum possible speed.

Withdrawal speed for Z axis

The rapid speed for Z axis is the same as the speed while withdrawing the bit from the workpiece.

Speeds for manual movement

These parameters specify the speeds for manual movement. A single step is performed by the current motor each time the cursor keys are pressed.

The motor changes over to continuous movement after the key has been held pressed for a longer period. It is possible to change over from fast to slow movement at any time by pressing the (INS) key.

Rapid movement is advantageous for covering longer distances whereas slow speed makes exact positioning more easy.

Speeds for reference movement 1/2

Each axis starts moving at reference speed 1 and searches for the reference switch. The axis stops when the switch changes its level, at which point the axis moves in the opposite direction away from the switch at speed 2.

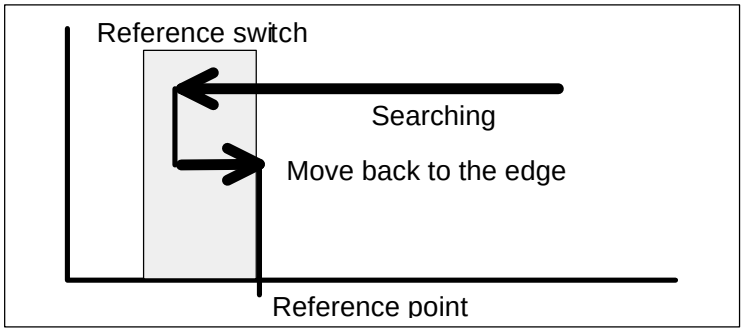


Figure 5-5 Reference movement of an axis

Reference point of the machine

The edge of the reference switch defines the absolute machine zero point or the reference point for each axis. A lower value should be specified for speed 2 so the movement ends with the shortest possible ramp when moving away.

_PC-NC_ differentiates between machine co-ordinates with the reference point as their origin and workpiece co-ordinates with the workpiece zero point.

Start/stop speed

The start/stop speed specifies the maximum speed in mm/sec. with which the ramps can start their movement. The acceleration ramps do not have to start with speed 0, but instead can set off at a specific starting speed in order to optimise the positioning times.

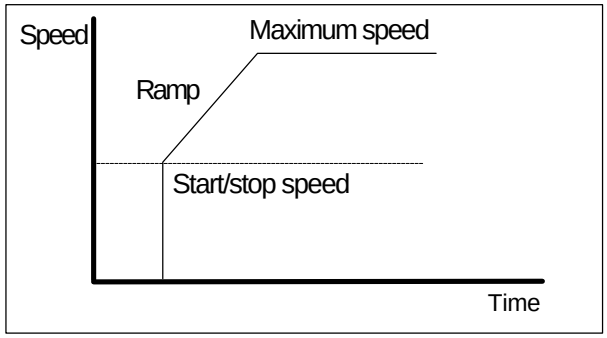


Figure 5-6 Start/stop speed and ramps

The optimum value can be determined by testing, e.g. using the MOTOR TEST function. To do this, switch off the ramp and increase the speed until step losses occur. The value determined by this method should be reduced by about 30 % to provide a safety margin.

Acceleration/deceleration distance

The acceleration/deceleration distance parameter specifies the length of the acceleration and deceleration ramps in motor steps.

Ramp lengths individually adjustable

During each movement, the speed is increased until the maximum speed is reached. At the end of each motor movement, the speed is reduced slowly until the motor comes to a stop.

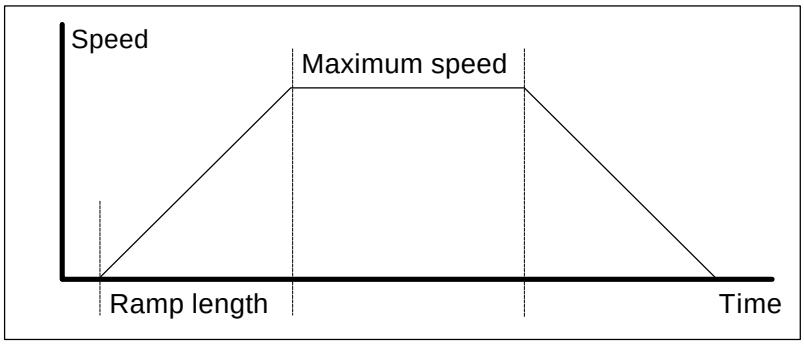


Figure 5-7 Speed profile for a motor movement

Acceleration and deceleration using ramps prevents step losses on the motors and permits faster maximum speeds.

Optimum ramp profile

The calculated default ramp profil runs well at most machines and mechanics. By use of unusual or strange mechanical components some reloadable ramp profiles can lead to much better axes movements. Additional ramp profiles are available on request.

Changeover time

The changeover time defines the time delay in the transition from single step to continuous movement for manual setup.

PC-NC sends single step signals to the motors each time the cursor keys are pressed briefly. If a key is held pressed for a longer period, namely precisely this changeover time, the motor changes over to continuous movement. Movement continues until the key is released.

Deceleration angle

The deceleration angle specifies the maximum differential angle for subsequent movements where the bit continues its movement at maximum speed. The value is specified in degrees.

Look ahead speed optimisation

The acceleration and braking functions are only active at the start and finish of movements if the direction of the subsequent movement is greater than a limiting angle. This angle can be adjusted.

A circular movement can be viewed as an example of this progressive speed optimisation function. A curve comprises many short individual movements. The directions of two successive movements are only marginally different from one another. Consequently, the circle can be moved in one operation at maximum speed.

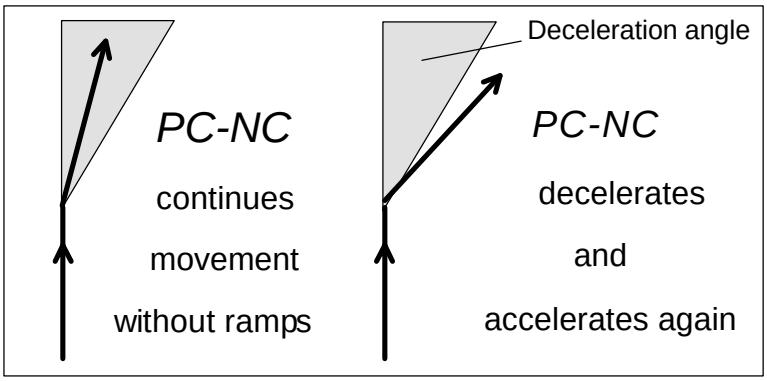


Figure 5-8 Deceleration angle

Decelerated and acceleration always take place prior to and subsequent to bit movements. This parameter is not significant for purely drilling applications and it is not evaluated.

5.3. Co-ordinates

The Co-ordinates dialog box can be used for defining all parameters relating to dimensions, utility points and the unit of measurement.

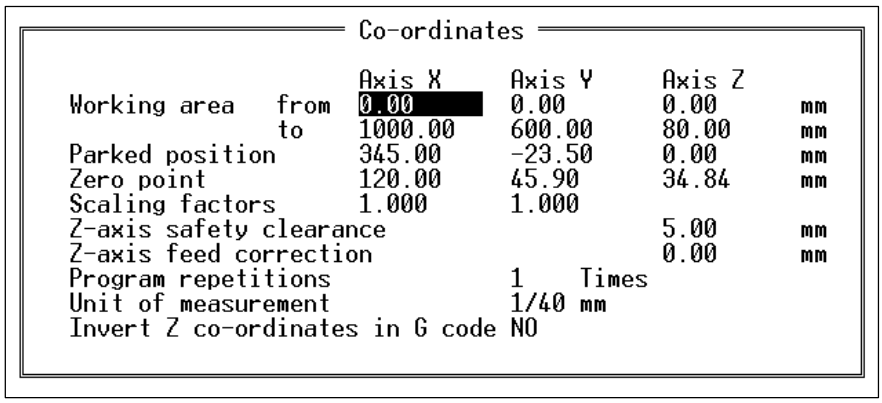


Figure 5-9 Setting the working area and the utility positions

Machine and workpiece co-ordinates

Every time a position is defined, it is necessary to differentiate between two types of definition. There are the absolute machine co-ordinates whose zero point is the reference point of the machine. In contrast to these, there are the workpiece co-ordinates whose origin is the workpiece zero point at the bottom left.

Working area

Software area monitoring

The working area defines the extent of the maximum movement area within which the workpiece can be machined. ***PC-NC*** monitors this area during the job process, i.e. the motors are stopped and the procedure cancelled if the limits are violated.

The working area is not monitored during manual movement and reference movement.

Maximum Z plunge depth

The working area for the Z axis defines the maximum plunge depth which bits can be moved without touching the machine bed. The area is checked at each job start.

The unit of measurement is the millimetre and the distances are in relation to the machine reference point (machine co-ordinates).

Parked position

It is necessary to define a parked position if the machine carriage has to be moved out of the working area from time to time. This may be necessary, for example, in order to change a bit or for clamping the workpiece.

It is also possible to define the parked position by moving to it manually using the keyboard.

The bit can be moved to the parked position either using the MOVE-PARK function, by manual movement or automatically during a bit change and at the end of a job process.

The unit of measurement is the millimetre and the distances are in relation to the machine reference point (machine co-ordinates).

Zero point

The zero point refers to the point of reference of the co-ordinates in the job file. It is the position defined by the smallest X and Y-co-ordinates, i.e. it is always located at the bottom left-hand corner of the data area. All distances for machining are calculated from this point.

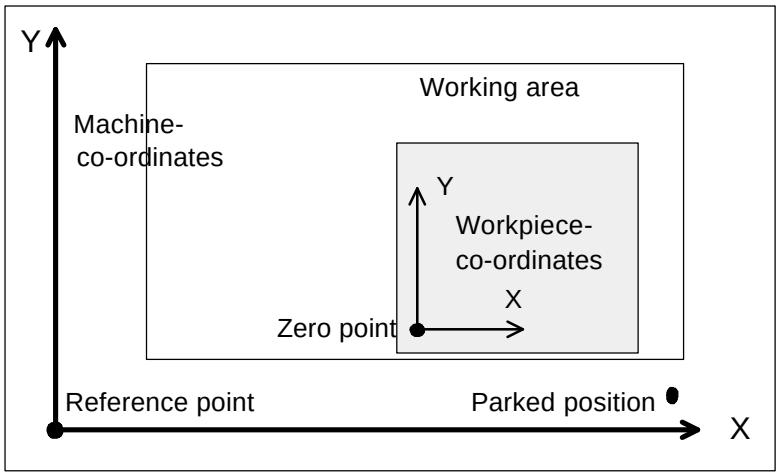


Figure 5-10 Working area and utility positions

Zero point as the origin of the workpiece co-ordinates

The zero point can also be defined manually. Do this by moving to the required position manually and then pressing the (F7) key to save this as the new zero point.

The unit of measurement is the millimetre and the lengths represent the absolute distance from the zero point to the machine reference point (machine co-ordinates).

Scaling factors

The scaling factors can be used for compensating for calibration differentials. If the exact length is not moved although the axis resolution and the unit of measurement are both set correctly, the problem can be corrected using the scaling factors.

The values must be specified to three decimal places. The values are used for multiplying the co-ordinates to which the bit moves.

Using 3D G code programs, the X scaling factor is used for Z as well.

Z-axis safety clearance

It is possible to define an additional height above the zero point plane of the Z-axis as a safety clearance. The bit is raised beyond the zero point by this distance during each job process.

During the next plunge operation, ***PC-NC*** first moves in rapid motion through the safety clearance until reaching the zero point. The bit then penetrates the material with the defined plunge speed.

Z-axis feed correction

The feed correction for the Z-axis causes the zero point of the Z-axis to be shifted downwards by the specified value each time a program is repeated.

This means the bit penetrates the material more deeply each time the program is repeated. The extra penetration is the correction value.

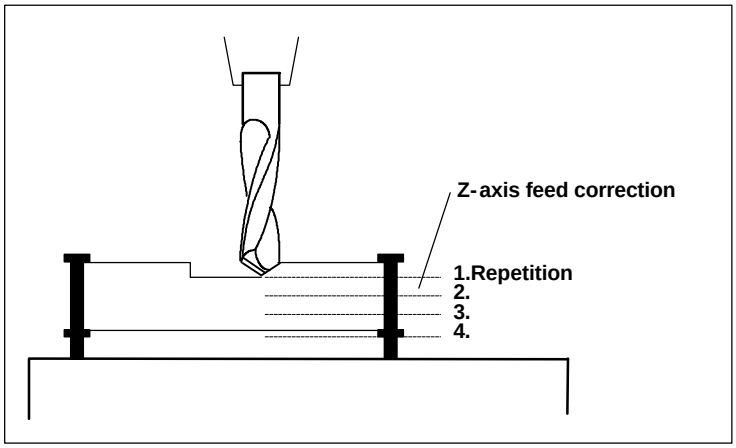


Figure 5-11 Program repetitions and feed correction

Program repetitions

Milling out step-by-step

Several passes are often required when milling thick or hard materials. The two parameters *Program repetitions* and *Feed correction* are used in order to avoid having to start a job process several times with different plunge depths.

The number of program repetitions causes the job file to be processed several times, i.e. the job file is started from the beginning *Repetitions-1* times after it reaches the end.

Unit of measurement

Wide range of pre-defined units of measurement

The unit of measurement for the job data must be defined using this parameter. All co-ordinate values in the job file are given in relation to a specific dimension.

Several millimetre and inch units can be used. HP-GL files are usually in units of 1/40 mm whereas drilling data are in 1/100 mm or 1/1000 inch.

Invert Z co-ordinates

G code programs often move the Z axis down with negative distances which is not so in ***PC-NC***. To use these programs without problems, please invert all Z co-ordinates.

5.4. Data format

The data format of the job files is defined in a sub-menu. At present, the commonly used HP-GL plotting format, G codes and two drilling formats are supported.

Plotting and drilling formats

The drilling formats are based on the following standards: Sieb&Maier 1000, Sieb&Maier 3000 and Excellon. In other words, it ought to be possible to process job data in any of these formats.

Professional milling machines are usually programmed with G codes.

Chapter 8 contains a list of commands and command descriptions for the individual formats. You can use this description to investigate the job file in case you encounter problems during processing, for instance unrecognised commands or commands which cannot be interpreted. In certain circumstances, the job file can be edited accordingly using an editor.

_PC-NC_ recogizes the current data formate in most files automatically.

5.5. Other parameters

Other parameters groups together various switches and other settings.

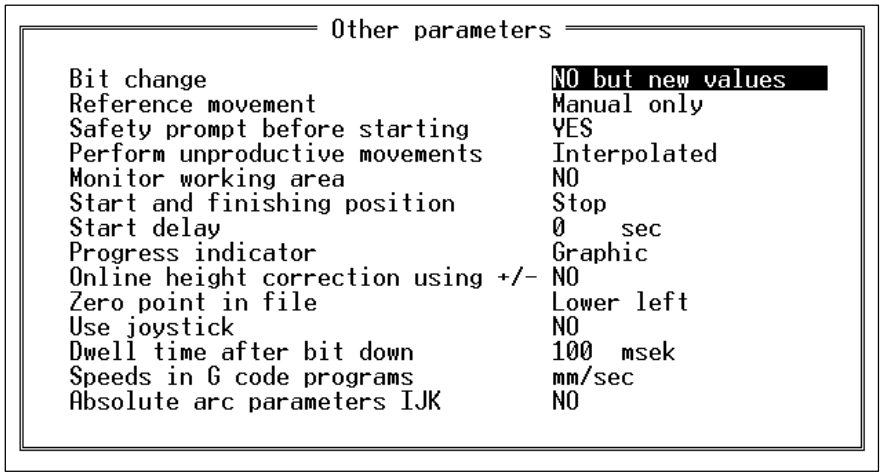


Figure 5-12 Other parameters

Bit change

This parameter defines the method by which ***PC-NC*** treats the bit change commands during a job process.

There are 5 possibilities for a bit change:

NO	No bit change is performed and the entire job process is performed with the current bit.
YES	The bit change is performed and the carriage always remains in its current position for the change.
YES at Parked position	The bit change is performed and the carriage always moves to the defined parked position.
NO but new values	No bit change is performed although the program subsequently takes into account the values for plunge depth, plunge and advance speed of the new bit.
YES use ATC	The bit change is performed automatically using the automatic tool changer. All necessary ATC parameters must be defined.

Reference movement

PC-NC needs to know the current position of the individual motor axes exactly. The position is not known when the machine is switched on, in case of faults or after a job process has been cancelled. For this reason, reference movements have to be performed occasionally.

Automatic reference movement

All axes move to their reference switches in a specific sequence during reference movement. The positions at these switches represent the machine reference point which is used for calculating all absolute distances.

There are three options:

Manual only	Reference movement is only triggered by the user.
At program launch	Automatically whenever ***PC-NC*** is launched.
Before job process	Automatically before each job process.

PC-NC always prompts the user to carry out reference movement when it does not know the absolute positions. Responding to this prompt with **NO** has the effect that ***PC-NC*** assumes the current position is to be the zero point.

Safety prompt before starting movement

This switch activates a safety function which causes the program always to ask if everything is ready before the machine starts moving. This function has been implemented for reasons of safety and can be switched off at any time.

Perform unproductive movements

Undesirable resonance problems may arise at high speeds and certain step frequencies on the motors. This either leads to step losses on an axis or an incorrect movement direction.

Positioning movements can be performed in three different ways in order to prevent this happening during unproductive movements at rapid motion speed, i.e. when the bit is raised from the workpiece.

The following settings are possible:

Interpolated Moves to the target position in a straight line; this is the standard setting and the fastest option.

Diagonal Moves part of the way diagonally i.e. at a 45° angle then follows the X or Y-axis for the remainder of the distance.

X/Y separate Movements along the X and Y-axis are performed independently.

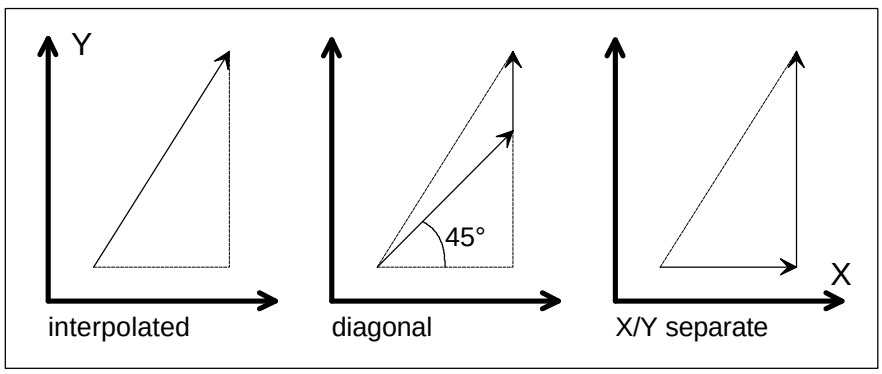


Figure 5-13 Unproductive movements

Interpolated unproductive movement is the standard setting. It should not be changed unless you experience resonance problems on the machine.

Monitor working area

This switch can be used for switching on and off the function of monitoring the size of the working area.

Monitoring always ought to be active under normal conditions in order to prevent accidental collisions with the edges of the machine or clamping devices.

The working area is monitored during job processes when the switch is activated.

It may be sensible to switch off the monitoring function for test purposes.

Start and finish position

This switch specifies where the start and finish point should be for each job process. The bit is also moved to this position after the reference movement.

There are 3 possible locations for the start and finish position:

Stop	***PC-NC*** stops at the reference point after completing reference movement and stops at the co-ordinates last occupied after each job process.
Zero point	The bit is moved to the defined zero point after reference movement and following each job process.
Parked position	***PC-NC*** moves to the defined parked position after reference movement and following each job process.

It is sensible to move to a specific position after the job process if you need space to change the bit.

***PC-NC* asks whether it really should move to the zero point if the zero point is outside the working area and working area monitoring is active.**

Start delay

Acceleration time for the drilling spindle

The start delay defines a waiting time in seconds. This time must always elapse when the extra signal *Drilling spindle* is activated.

This method ensures that the drilling spindle has enough time to accelerate before being used for the first time. The *Spindle accelerating* message appears during the waiting time.

Progress indicator

PC-NC continuously delivers information about the progress of machining during a job process.

Two types of progress indicator are possible:

Text ***PC-NC*** displays all commands and their interpretations in a report window, e.g.

PD	Lowering bit
PA123,345	New co-ordinates 123,345

Graphic ***PC-NC*** switches to graphic mode and shows each movement covered as a coloured line. The colours correspond to the individual bits. A percentage display shows the progress of machining.

Online height correction using +/-

Correction of plunge depth whilst machining is in progress

Online height correction of the Z-axis has to be enabled using the parameters. Once enabled, the Z-axis can be corrected by 1/100 mm or at least one step each time the (+) and (-) keys are pressed. This correction can be performed whilst the job process is taking place.

The Z-axis immediately moves downwards by this distance every time the (+) key is pressed. Pressing the (-) key moves the axis upwards. Corrections to distances made in this fashion are retained in

subsequent actions, i.e. the distances do not revert to their original values automatically.

The (+) and (-) keys are the grey keys on the right-hand side of the keyboard.

Zero point in file

The workpiece zero point can be defined to four different positions.

origin the zero point is exacty at the position where the CAD or drawing program defines it. Normally this is the origin of the co-ordinates sytems.

lower left the zero point is the most left and lowest co-ordinate in the file.

upper left the zero point is the most left and largest co-ordinate in the file.

center the zero pint is exactly the middle of the working piece.

Use joystick

To use a joystick for manual movements activate this parameter.

Dwell time after bit down

This parameter defines a dwell time after moving the bit down. It is defined in milliseconds.

Speeds in G code programs

Speed definitions can be handled different by ***PC-NC***. They can be ignored to use the parameter settings or they can be interpreted as mm/sec or as mm/min.

Absolute arc parameters I/J/K

Arc parameters I/J/K in G code programs are relative by default. Some postprocessors generate them as absolut positions. To handle this set the parameter to YES.

5.6. Editor

It is necessary to define an external editor or viewer program in order to use the FILE-EDIT menu item for editing and viewing the job and parameter files, or indeed any other files. By default the build in editor is used.

Figure 5-14 Parameter settings for the editor

Use external editor

This parameter instructs ***PC-NC*** to use the build in editor or the external one, defined in the next line. You can even use programs for generating data instead of an external editor, like sign making programs .

Editor command line

External editor The full name of the editor program and, if necessary, its path should be entered. The current filename is added to the editor name when the program is launched from within ***PC-NC***. Any other settings required when launching the editor can be accommodated by writing a batch file containing all the necessary information and entering the batch file's name in this box.

Automatic launch

PC-NC can invoke an external editor or similar program automatically after a job process. For instance you can generate a serial number to work with next automatically.

5.7. Signals

All digital signals and according parameters are grouped into this dialog box.

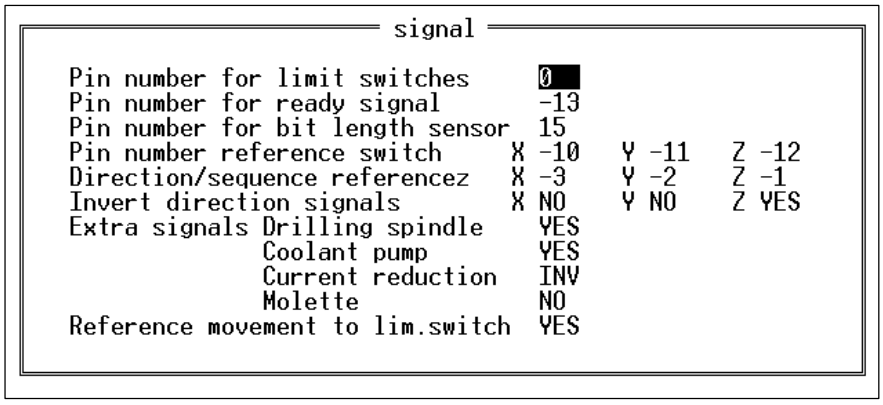


Figure 5-15 Signal parameters

Pin numbers for switches

Assignment of switches to lines of the LPT port

The limit and reference switches must each be connected to one of the 5 input lines or the parallel port being used. The lines on pin numbers 10, 11, 12, 13 and 15 are available. Each switch used must be assigned to one of these pins.

NO or NC contacts

An optional sign can be used in order to define the switch as an NO or NC contact. The SIGNAL TEST function can be used for determining the precise pin numbers and switch types or the logic level.

It is also necessary to apply a ready signal to one of the input lines if ***PC-NC*** should also recognise the machine's status. ***PC-NC*** uses this signal to determine whether the machine is switched on or off.

0 must always be entered as the pin number for unassigned signals.

Example

Reference X	**pin 10**	NO contact on pin 10
Ready signal	**pin -15**	NC contact on 15

When several switches have to be applied to one pin, they must be grouped into a common line and connected as a logical OR in the case of NO contacts or a logical AND in the case of NC contacts.

There are two methods for polling the limit and reference switches. The switches have to be applied differently depending on the method used.

1. **No** independent reference switches and reference movement is performed to the limit switches:

   Pin ref. X - Both X-axis limit switches (OR/AND)
   Pin ref. Y - Both Y-axis limit switches (OR/AND)
   Pin ref. Z - Both Z-axis limit switches (OR/AND)
   Pin limit switch - 0

2. Reference switches **and** limit switches available:

   Pin ref. X - X-axis reference switch
   Pin ref. Y - Y-axis reference switch
   Pin ref. Z - Z-axis reference switch
   Pin limit switch - All limit switches as common (OR/AND)

PC-NC can differentiate between which or the two types is being used purely from the *Reference movement to limit switches* parameter.

Please refer to the detailed description in chapter 7 for information about connecting the switches and defining the parameters for them.

Direction/sequence for reference movement

Reference movement of the individual axes is performed in a specific sequence. It is usually necessary to move the Z-axis upwards first, out of the workpiece. The other two axes then move to their reference points.

The sequence in which the axes are moved is defined in these fields. An optional sign can be added to define the direction in which the switch is sought. Movement away from the switch is then made in the opposite direction.

Example

X-axis	**- 3**	**First, Z-axis in the positive direction,**
Y-axis	**2**	**then Y-axis in the positive direction,**
Z-axis	**1**	**finally X-axis in the negative direction.**

Possible faults during reference movement

There may be some faults during reference movement after the machine is taken into operation for the first time. If so, the faults can be rectified as follows:

- Axis moves in the wrong direction
  Remedy: Change over the sign in the *Direction/sequence* parameter.

- Axis moves in the right direction but stops after a short time after which the *Fault at reference switch* fault message appears. Alternatively, the axis moves up to the switch but does not move away again.
  Remedy: Change over the sign in the *Direction/sequence* parameter and change over the sign in the *Pin number for reference switch* parameter.

- Axis moves in the wrong direction and stops after a short time after which the *Fault at reference switch* fault message appears.
  Remedy: Change over the sign in the *Pin number for reference switch* parameter.

Please refer to a detailed description in chapter 7.

Invert direction signals

There are two ways to change the direction of rotation of a motor if one is rotating in the wrong direction.

Mirroring or rotating the data

You can either swap over the connections on the motor windings or use this parameter in order to change the direction of rotation. The direction signal for controlling the motor is then inverted.

The direction signal for the X and Y motors can be swapped over. It is also possible to mirror or rotate the NC data in this manner.

Extra signals

The three switches in these entry fields enable or disable the activation of extra signals.

Switching the drilling spindle and coolant on and off

These switches must be used to enable any of the following signals: *Drilling spindle, Coolant pump, Current reduction* or *Molette* if they are needed at the machine.

There are three possible states for each signal:

YES	Signal switches to HIGH when it is active
NO	Signal is not used
INVERTED	Signal switches to LOW when it is active

Reference movement to limit switches

This parameter should be activated if the machine should perform reference movements up to the limit switches of the axis end stop instead of the actual reference switches.

Reference switches used as limits as well

The reference movement is performed in the same way as for separate reference switches, i.e. the system first looks for the switch and then moves away from it in the opposite direction.

5.8. Machine-specific parameters

Machine-specific parameters include all the settings related to the mechanical characteristics of the machine and its connection to the personal computer via a port.

Be very careful when setting or altering the machine-specific parameters since they can lead to faults on the machine.

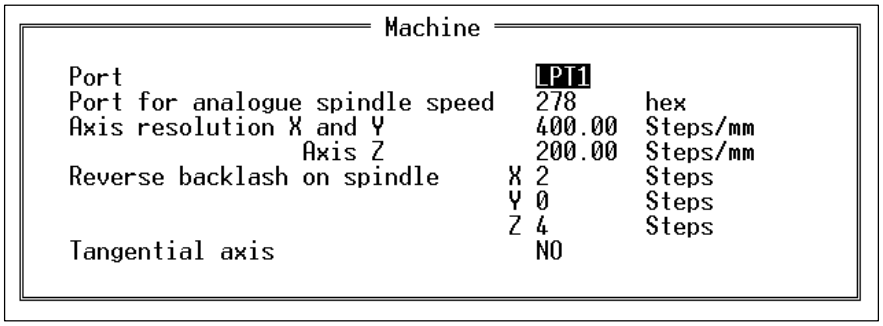


Figure 5-16 Machine specific parameters

Port

This is for defining the parallel printer port via which the machine is connected to the personal computer.

PC-NC automatically recognises the number and addresses of the ports fitted and presents these for selection.

Axis resolution

The two resolution fields define the number of motor steps per millimetre of travel. It is of decisive importance for these values to be specified accurately so that distances and speeds can be calculated.

The values are calculated using the following formula:

$$Resolution = \frac{No.of\ steps\ per\ rotation}{Spindle\ pitch\ in\ mm\ per\ rotation}$$

Reversing backlash on spindle drives

Compensation for reversing backlash

Some drives cannot be set to completely eliminate all backlash when there is a change of direction. This always produces small differences which can accumulates during a job process. This parameter makes it possible to compensate for this reversing backlash.

Parameters are available for all axes in order to define the number of unproductive steps. This number of motor steps is then output in addition each time that axes change direction.

The default value is 0 and should not be changed if it is possible to set the drives to zero backlash.

Port for analogue spindle speed

Spindle speed in 15 levels

PC-NC can assign one of 15 levels of spindle speed to 4 output bits. You can use a second or third printer port for generating the output signals.

Tangential axis

Enables the tangential axis as 4th axis. All parameters for tangential cutting are avalable if enabled only.

5.9. Tangential axis

Settings for a tangentional cutting knife

The optional tangential axis for controlling a cutting knife must be enabled in the machine specific parameters.

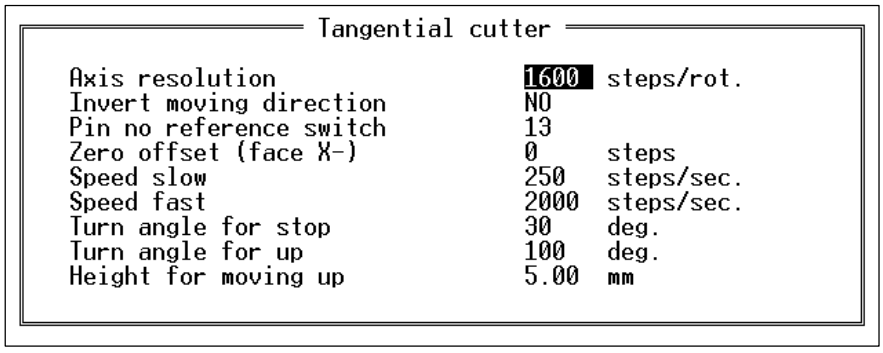


Figure 5-17 Parameters of the tangential axis

Axis resolution

Defines the number of steps per full rotation of the tangential knife.

Invert direction signal

There are two ways to change the direction of rotation of the tangential knife if it is rotating in the wrong direction.

You can either swap over the connections on the motor windings or use this parameter in order to change the direction of rotation. The direction signal for controlling the motor is then inverted.

Speeds tangential axis

These two parameters define the rapid speed and the slow speed of the tangential axis. They are used in a job movement, during reference and jog movements.

The units are steps/sec.

Pin number and offset

The reference switches of the tangential axis must be connected to one of the 5 input lines or the parallel port being used. The lines on pin numbers 10, 11, 12, 13 and 15 are available.

An optional sign can be used in order to define the switch as an NO or NC contact. The SIGNAL TEST function can be used for determining the precise pin numbers and switch types or the logic level.

The offset parameter defines the number of steps which must be moved after the reference movement to face the knife to the X- direction. The negative X direction means the zero point of the tangential axis and must be set absolute precise.

To define the optimum setting do the following steps...

1. define the offset to 0
2. perform a reference movement
3. go to jog mode and turn the knife exactly to the X- direction
4. define the displayed number of steps as the offset parameter
5. save settings

To get good results the offset of the cutting knife after a reference movement must be defined very exactly.

Turning angle and heigth

Performing the turns of the knife

The two turning angle settings define limits for different cutting functions. If two following lines differ in direction less than the stop angle, ***PC-NC*** moves on without brake and turns the cutting knife while moving.

If the difference in direction is more the the stop angle but less the angle for moving up, ***PC-NC*** stops movement, turns the knife and continues.

If the difference in direction is more the the angle for moving up, ***PC-NC*** stops movement, moves Z axis up to the defined height, turns the knife, moves down again and continues cutting.

6. Additional program ***HPGL-Opti***

6.1. Overview

Manipulating HPGL files

HPGL-Opti is a convertion tool to modify HPGL files. The vectors created by common drawing or CAD programs are sorted in order to optimise the following work at a CNC machine.

The most important functions of *HPGL-Opti* are :

- user friendly operation with pull down menus and context sensitive help
- sorting all vectors for minimum unproductive movements
- definable sequence of used bits
- free assignment of bits to pens in the file
- creating closed contours with catching grid
- bit radius compensation
- moving inside or outside a contour
- moving clockwise or counterclockwise
- splitting the output to different files
- graphical display of the data

6.2. Operation

Same operation as with PC-NC

All screen layouts, parameter dialogs and the operation is similar to ***PC-NC***. The function keys for equal functions are identical. Learning and working with is very easy.

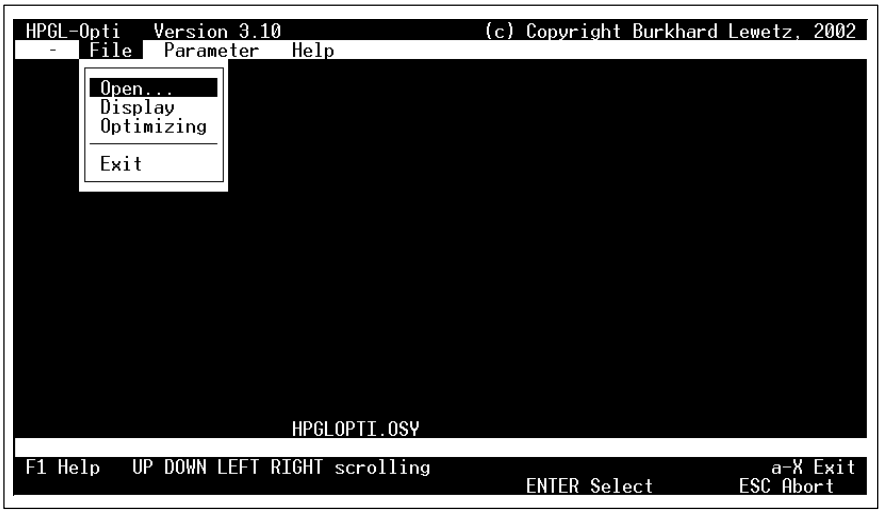


Figure 6-1 ***HPGL-Opti*** screen layout

There are four menus INFO, FILE, PARAMETERS and HELP. The most needed functions can be reached with function keys as well.

Menu structure

The menus in summary :

INFO — The system menu shows information about the current version and the liability.

FILE-OPEN — The FILE-OPEN menu item affords access to an easy-to-use function for selecting existing program files. Any stored parameters for a NC program are also opened. All files belonging to a project have the same file names with different file name suffixes.

It is possible to change drives and directories in the dialog box. Programs can also be selected by pressing the (F2) function key.

FILE-DISPLAY — The graphic preview function in ***HPGL-Opti*** enables you to make a quick visual check of the job file. The program initially loads all the data in order to determine their extent.

The subsequent graphic display is free from distortion and at the maximum possible size.

The graphic preview function works with a standard VGA graphics card.

The following elements appear in the graphic view...

- The information in the job file
- Current scaling factors for X and Y (based on the parameters)
- Defined zero point for X and Y (based on the parameters)
- Rulers showing the current dimensions
- All required bits and their numbers

The following key assignments are active in the graphic display...

(F2) Starts the milling simulation
(ESC) Cancels simulation or graphic display function

The milling simulation can be used for simulating the entire job process graphically on the screen before the process is actually carried out. This makes it easy to see the sequence in which the individual actions are performed.

There is no online help during graphic display.

The (F6) key also activates the graphic display.

FILE-OPTIMISING

The optimisation function ensures that the job file is processed efficiently by sorting the NC commands it contains.

The outputs from CAD and layout programs are not always optimised which means the bits have to be inserted several times and the machine frequently has to make unnecessary unproductive movements. In some cases, contours may not be closed but are instead completed in two or more passes.

The optimisation Function performs several tasks:

1. Sorting the data according to bits, so each bit only has to be inserted once.
2. Closing contours so there is no need to start at existing line ends.

3. Ensuring that machining is always resumed at the nearest co-ordinates possible in order to minimise unproductive movements.
4. Sorting bits in the defined sequence.
5. Calculating a radius compensation and shifting the contour inside or outside
6. Generating movement directions as defined

Typical procedure of an optimisation session:

1. Select the job file, i.e. the file to be optimised.
2. Define all necessary parameters, join the new file to the old one
3. Start optimisation.
4. Check the results of the optimisation if necessary.
5. Repeat the procedure and do not join the files
6. Save the results and exit ***HPGL-Opti***

Optimisation requires a great deal of memory. Your computer might not have enough main memory to process large NC files.

The (F3) key also activates the optimisation function.

FILE-EXIT

Exits ***HPGL-Opti***

PARAMETERS-BIT MANAGEMENT

HPGL-Opti comes with a user-friendly bit management function. It is possible to define up to 10 bits with special values for each project.

The following settings are available...

- assignment of bits to pens
- sequence of bits in output file
- grid to close contours and to combine subsequent vectors
- radius for compensation
- moving direction, clockwise or counterclockwise
- shifting of new contour, inside or outside the original line

PARAMETER-OTHER	With other parameters you can define different settings not belonging to single bits. You can define a shift of the co-ordinate zero point, scaling factors, to join the old and new file for testing purposes and you can select the typ of algorithm for the radius compensation. Joining the old and new file is very helpful for a visual control of calculated radius compensations.
PARAMETER-SAVE	All parameters and bit settings can be saved in files using the save menu item. It is possible to save the settings as belonging to a job file or a project. The save function operates like this. **HPGL-Opti** saves all workpiece-specific settings such as the zero point, scaling, etc. in a parameter file which belongs to the job file if there is a job file loaded. The file has the same name as the job file and it's filename extension is *.OSY. Whenever these job files are loaded later, all the settings and bits are restored without them having to be redefined.
HELP-WORKING WITH	Gives a short explanation of the typical operation tasks with ***HPGL-Opti.***
HELP-INDEX	Summary of all help topics.

Key assignment

The following keys are available :

(F1)	activates the online help
(F2)	opens the file selection dialog
(F3)	starts the optimisation
(F4)	opens the dialog of the bit parameters
(F6)	shows NC data in grphical display
(F10)	opens the drop down menues
(ESC)	cancels a running function
(alt-X)	exits from ***HPGL-Opti***
(F2)	starts the simulation during graphical display

The created files get an new file name extension. The first file will be named *.OPT, the second *.OP1 and so on.

6.3. Parameters

Parameter settings are grouped in two dialog boxes.

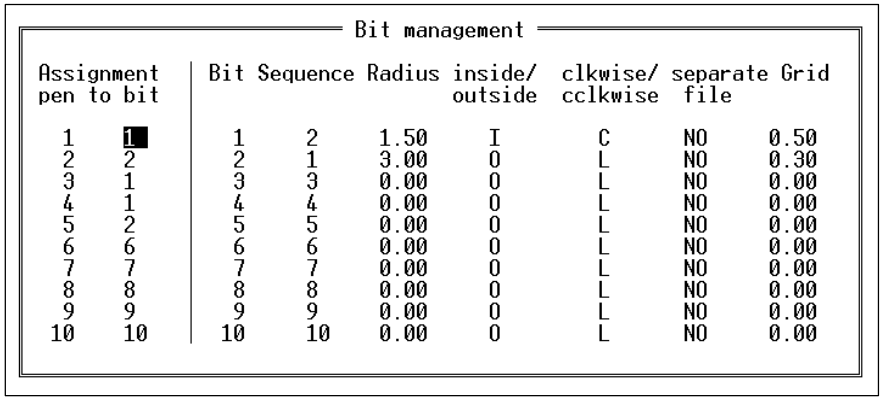


Figure 6-2 Bit management settings

Bit assignment

Assigning pens to bits

You can assign different bits to the used pens in the original file. For instance it is possible to assign the same bit to more than one pen in the old file.

Sequence of bits

These parameters define the sequence of appearance of the bits in the new created file.

Radius

Radius compensation

To calculate the radius compensation you have to define an exact radius for each bit you want the compensation done. The new toolpath is shifted inside or outside the original line.

Radius compensation is only possible at closed contours and not for engraving purposes.

Inside/outside

This parameter defines the direction the new line is shifted while compensatation of the bit radius. You can select inside or outside the original line.

Radius compensation is only possible at closed contours and not for engraving purposes.

Moving direction

The moving direction of the new contour depends of the type of bit. Some bits may used as left cutter others as right cutter.

Create separate file

Split to multiple output files

The new toolpaths of each bit can be separated to single files. This may be useful when you want to work with this files in different orders or sequences. If you define a single bit to have it's own file all other bits will be written to the standard output file.

File names of serarate output files will be *.Oxx, with xx indicating the bit number.

Grid

One of the most important functions of ***HPGL-Opti*** is to check the contours and to combine single vectors to get a closed contour. Sometimes contours seem to be closed but have little gaps or crossed vectors.

This gaps and crosses can be combined to closed contours by using the grid. It works like a window in which the ending point of one vector and the bginning point of the next vector must be.

Units of grids are mm.

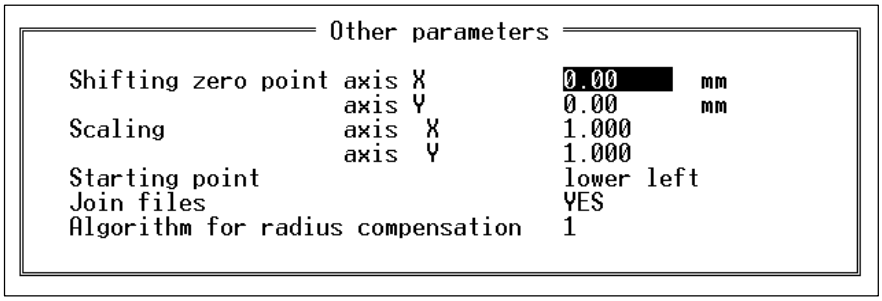


Figure 6-3 Other parameters at ***HPGL-Opti***

Shifting zero point

These parameters can shift the zero point in the new created file by the defined values. Default is shifting by 0.

Scaling factors

With the scaling factors you can modify the size of the new contour by X or Y parameters.

Valid parameters are between 0.001 and 99.999.

Starting point

In the selection menu you can define one of eigth possible starting points. At the lower left starting point HPGL-Opti first searches for the less Y co-ordinate and after for the less X co-ordinate. The left lower starting point works in the opposite order.

Eight positions are available :

lower left	**left lower**
lower right	**left upper**
upper left	**right lower**
upper right	**right upper**

Join files

Old and new created file joined together

This switch set to YES causes ***HPGL-Opti*** to combine the old file and the new created one. To display this in graphical is very helpful to check the calculated toolpath after the radius compensation.

To get a single file with the new toolpath only just repeat the job with the original file reloaded and set the join parameter to NO.

Algorithm

HPGL-Opti contains two different algorithms to calculate the radius compensation. Unfortunately it is not possible to recogize the better result automatically. Please check it by yourself and choose the algorithm that fits your needs best.

6.4. Performing the optimisation

Step by step

The optimisation must be done in subsequent steps...

1. Select the job file, i.e. the file to be optimised.
2. Define the pen to bit assignment of all used pens in the file
3. Check the bits for the radius compensation and define the radius, the direction of movement and the inside/outside parameter.
4. Define the join files parameter with YES
5. Start optimisation by pressing (F3).
6. Check the results in the graphical display
7. Repeat the procedure by reloading the original file and set the jo in files parameter to NO
8. Save the results and exit ***HPGL-Opti***

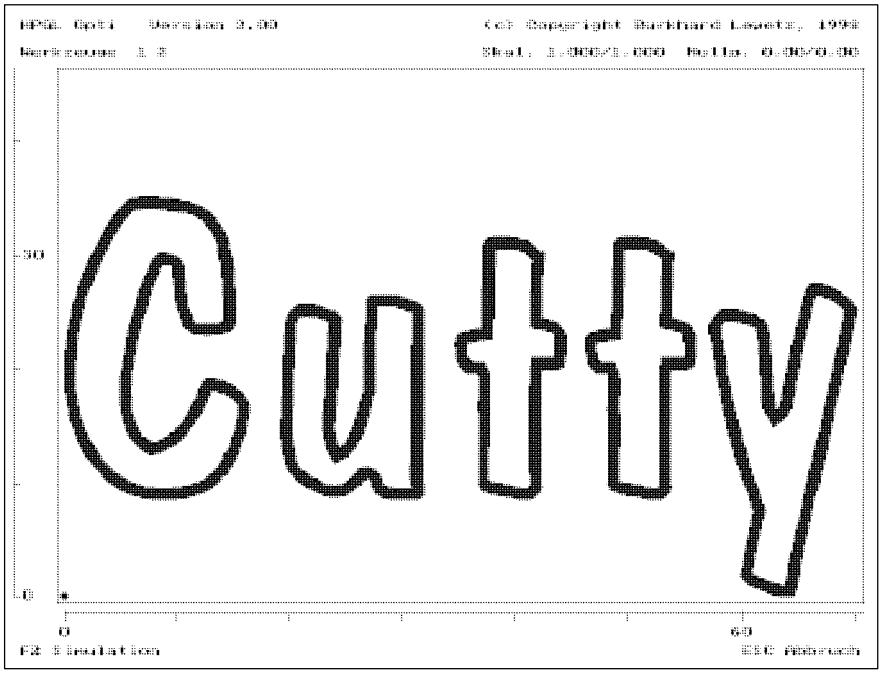


Figure 6-2 Engraving after radius compensation

7. How can I....

This chapter gives detailed informations about specific tasks using ***PC-NC*** and about typical applications.

How can I...

...start ***PC-NC*** for the first time ?
...use the automatic tool length compensation ?
...use the automatic tool changer (ATC) ?
...use ***PC-NC*** for plotting ?
...use ***PC-NC*** for drilling ?
...use ***PC-NC*** for milling ?
...use ***PC-NC*** for milling boards ?
...use ***PC-NC*** for engraving and grinding ?
...pause a job process ?
...use the quick menu ?
...adjust my NC data ?
...use the tangential axis for cutting ?

7.1. ...start *PC-NC* for the first time ?

Carry out the following steps or checks in order to configure ***PC-NC*** after installing it on your machine.

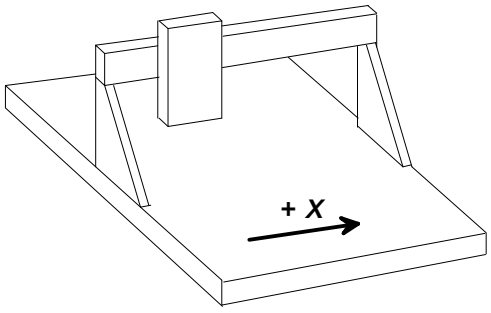
Define the port

You have to define which interface you are using to connect the machine. Use the [P] and [M] keys to open the machine parameters dialog box and set the appropriate port (**LPT1** or **LPT2**). Press [ESC] to close the dialog box again. Please note that you always have to save any changes you make to the parameters. Do this using the PARAMETERS-SAVE function.

Checking and setting movement directions

Call the MOVE-MANUAL MOVE function by pressing the [M] and [M] keys. Now move all axes using the cursor keys and check the movement directions on the machine.

Each key is assigned to a specific axis and direction of movement. Start with the X-axis.



Pressing the [RIGHT] cursor key produces the following movement on the machine.

Pressing the [LEFT] key should move the X-axis precisely the opposite direction.

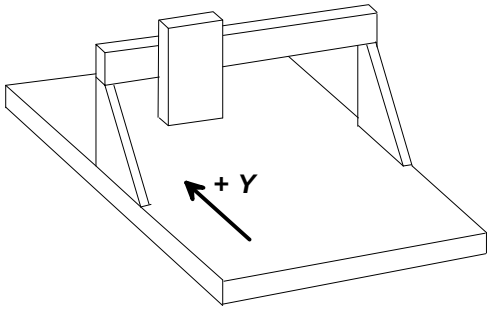
One of the following faults may apply if this is not the case:

Fault	Possible cause	Remedy
Machine does not move at all	Wrong port	Set the right port.
	Wrong port assignment	Check the assignment (see the appendix in the manual).
	Machine switched off	Check the motors are being powered.
Wrong axis moves	Wrong port assignment	Check the interface assignment (see the appendix in the manual) and correct it.
X-axis always moves in the opposite direction, i.e. to the right when the [LEFT] key is pressed and to the left when the [RIGHT] key is pressed	Direction signal operating with incorrect logic	Change over the *Invert direction signal* parameter in the machine parameters.

Check the other two axes after the X-axis and correct the movement direction using the parameters if required.

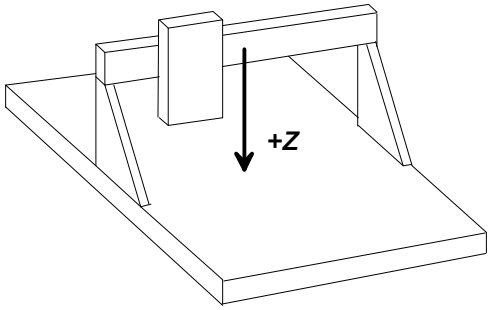
£

Pressing the [UP] key must cause the machine to move backwards.



+

Pressing the grey [+] key causes the Z-axis to move downwards; it moves upwards when the [-] key is pressed.



The movement directions can be changed in the software using the machine-specific parameters. Call up the dialog box from the main menu by pressing the [P] and [M] keys. You only need to change over the *Invert direction signal* parameter for the axis in question.

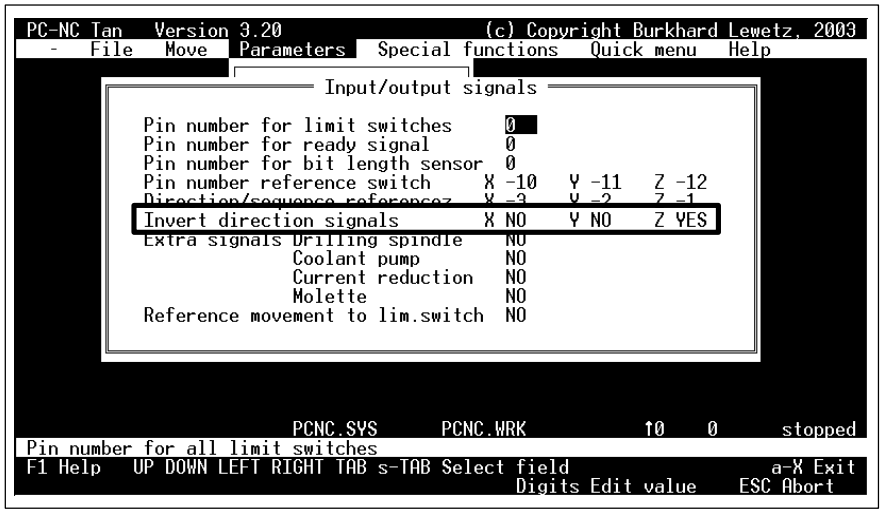


Figure 7-1 Setting the moving directions

Setting reference switches

You can use the SIGNAL TEST special function as a very easy method of checking the function of the limit and reference switches. Press the [S] and [S] keys to activate the function. Alternatively, you can select the function from the drop-down menus.

Now operate the switches on your machine by hand and observe the reaction on the screen. Each switching operation must be clearly displayed. The corresponding signal levels are marked by **LOW** and **HIGH**.

It is easy to determine what type of switches are used.

Signal is LOW when the switch is not operated and to HIGH when the switch is operated

↓

Switch is an NO contact

Signal is HIGH when the is not operated and changes changes to LOW when the switch is operated

↓

Switch is an NC contact

Make a note of the switch type (NO or NC contact) of each reference switch on each axis. Also note down the pin number of the port to which the switch is connected.

The switch settings are made in the field in the machine-specific parameters dialog box which you call up by pressing the [P] and [M] keys. Enter the pin number determined for each axis in the *Reference switch pin number* parameters. If an NC contact is being used as the switch, you also have to enter a negative sign.

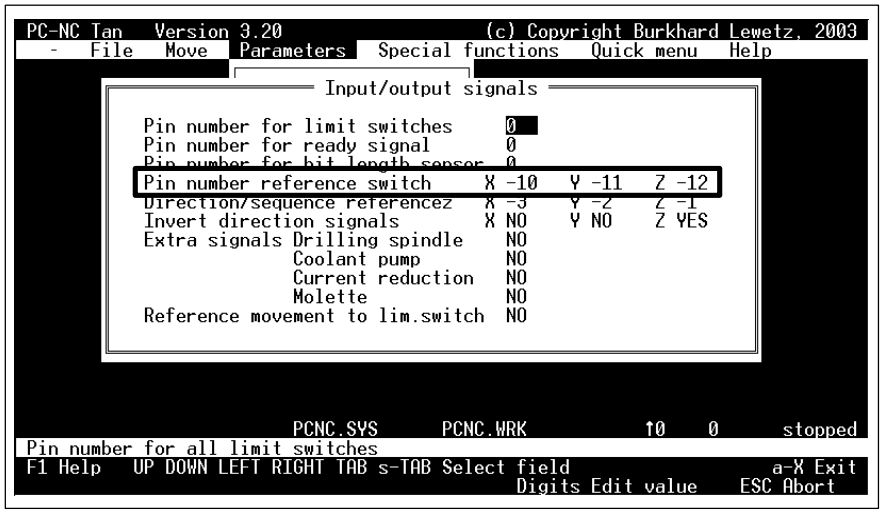


Figure 7-2 Setting pin numbers

Example

The display for pin 11 changes continuously in the SIGNAL TEST function when the X-axis reference switch is operated. It indicates a **HIGH** level when it is not operated and the display changes to **LOW** when the switch is operated by hand.

The switch is an NC contact and the right value for the *Reference switch pin number* for the X-axis is -11

Setting sequence/direction of reference movement

The *Direction and sequence of reference movement* parameters define in which sequence the axes should move to their switches during reference movement, and at which end of the axis the reference switch is attached. Normally speaking, the Z-axis should be moved first since this allows the bit to be withdrawn upwards out of the workpiece in case of a cancellation. The other axes come after the Z-axis. Consequently, you should enter the sequence 3 2 1 in the parameter fields for the X, Y and Z-axes.

After this, find out on which end of each axis the reference switch is attached (on the positive or the negative end). You do not need to make any changes if the switch is at the positive end, i.e. if the axis has to move in the positive direction to reach the reference switch. Put a negative sign in front of the *Direction/sequence of reference movement* parameter if the switch is on the negative end. Make these settings for each axis.

Example

X-axis	- 3	**First, Z-axis in the positive direction,**
Y-axis	2	**then Y-axis in the positive direction,**
Z-axis	1	**finally X-axis in the negative direction.**

Checking the settings

Your machine should now perform reference movements correctly. After completing all the parameter settings and saving them, press the [F8] key or select the MOVE-REFERENCE MOVE menu item to perform reference movement.

During reference movement, all axes move up to their switches in the selected sequence. After this, they move back away from the switches at a slower speed. All axes stop exactly at the edge of the switch and mark the reference point there.

One or more of the settings is incorrect if this process does not take place automatically or if it is cancelled and a fault message is displayed. The table below gives information about possible faults and how to solve them.

Fault	**Remedy**
Axis moves in the wrong direction	Change over the sign in the *Direction/sequence* parameter.
Axis moves in the right direction but stops after a short time after which the *Fault at reference switch* fault message appears. Alternatively, the axis moves up to the switch but does not move away again.	Change over the sign in the *Direction/sequence* parameter **and** change over the sign in the *Reference switch pin number* parameter.
Axis moves in the wrong direction and stops after a short time after which the *Fault at seference switch* fault message appears	Change over the sign in the *Reference switch pin number* parameter.

7.2. ...use the automatic tool length compensation

Tool length measured automatically

PC-NC has an comfortable function to measure tool lengths automatically and to compensate differences in the following job. This guarantees the same moving depth of tools every time.

Requirements

The measurement of the tool length is performed by a length sensor. It must be mounted somewhere at the edge of the machine bed and can be touched from the top. The sensor has to be assigned to a input pin like all other switches.

The exact position can be measured manually or moved to with the jog function. The co-ordinates must be defined within the parameters.

The difference in length is always according to the reference or master tool, which normally is the current one mounted.

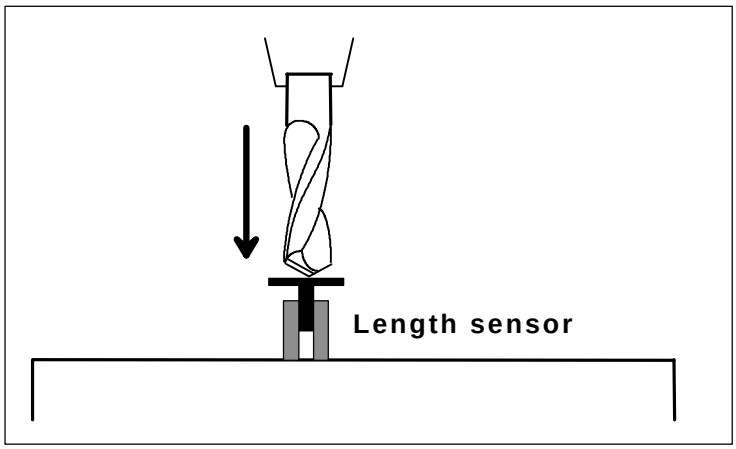


Figure 7-3 Moving to the length sensor

Compensation of different lengths

Each used tool can be defined with a difference in length, which will be added to the plunge depth every time it is used. A tool 2mm shorter than the master tool will be lowered for 2mm more.

Operation step by step

1. define all parameters of the sensor
2. mount the first tool, which will be the master tool

3. define the workpiece's zero point in the usual manner
4. enter the function MOVE-CHECK TOOL LENGTH. ***PC-NC*** moves to the length sensor and measures the master tool. This leads to a parameter *difference to zero point* which will be set automatically.
5. start the job, change tools, let them measured automatically and compensated by length.

7.3. ...use the automatic tool changer (ATC)

The controlling of an automatic tool changer is one of the professional functions of ***PC-NC***. In combination with the automatic length measurement and length compensation you can use ***PC-NC*** for mass production.

Requirements

You need several mechanical components for the ATC. There must be a magazine at one edge of the machine bed, where up to 10 tools are located. Second is a electric or pneumatic molette to pick up and release the tools during the job. The molette is controlled open or closed via the digital output signal at pin 16 (version 3.20) or pin 9 (version 3.10) of the printer port.

Necessary definitions

For each magazine position the exact X and Y co-ordinates must be defined in the parameters. The Z height is the same for all positions. ***PC-NC*** first moves to this height and lowers the Z axis after to perform a release or pick up service.

Moves during release and pick up

For each release and pick up service you can define two certain movements with one of the possible axes. Each movement selects an axis and a distance to move.

Example :	Release	Z	3,6
		Y	2,0
	Pick up	Y	-2,0
		Z	-2,5

A tool change according to these definitions is performed like this. ***PC-NC*** moves first over the magazine position of the current tool.

Then it lowers Z for 3.6 mm and moves Y for 2 mm. Now the molette opens to release the tool and the dwell time elapses. After that ***PC-NC*** moves Y back for 2 mm and Z up for 2.5 mm.

To pick up the new tool ***PC-NC*** performs the same tasks but moves to the new magazine position first and closes the molette at the magazine position.

Operation step by step

1. mount a magazin and the automated molette
2. define the magazine positions and the heigth to move first
3. define the movements for releasing and picking up
4. set the parameter *tool change* to use the automatic tool changer
5. if a tool is mounted already, please select it by using the menu item MOVE-SELECT TOOL
6. start the job process and let ***PC-NC*** change the tools automatically

7.4. ...use *PC-NC for plotting ?*

Bear the following points in mind and take the appropriate action if you want to use ***PC-NC*** for plotting:

1. Connect the retaining magnet for a drawing pen or the cutter knife to the **Coolant pump** signal on pin 14 of the port. This signal becomes HIGH for **Pen down** commands and LOW for **Pen up**.

2. The definitions for bit management and the parameters for the Z-axis are not required. However, in order to obtain a brief waiting time when operating the pen, you should enter a small travel distance for the imaginary Z-axis. This distance leads to a delay as the pen is raised and lowered, thus preventing smudges on the paper.

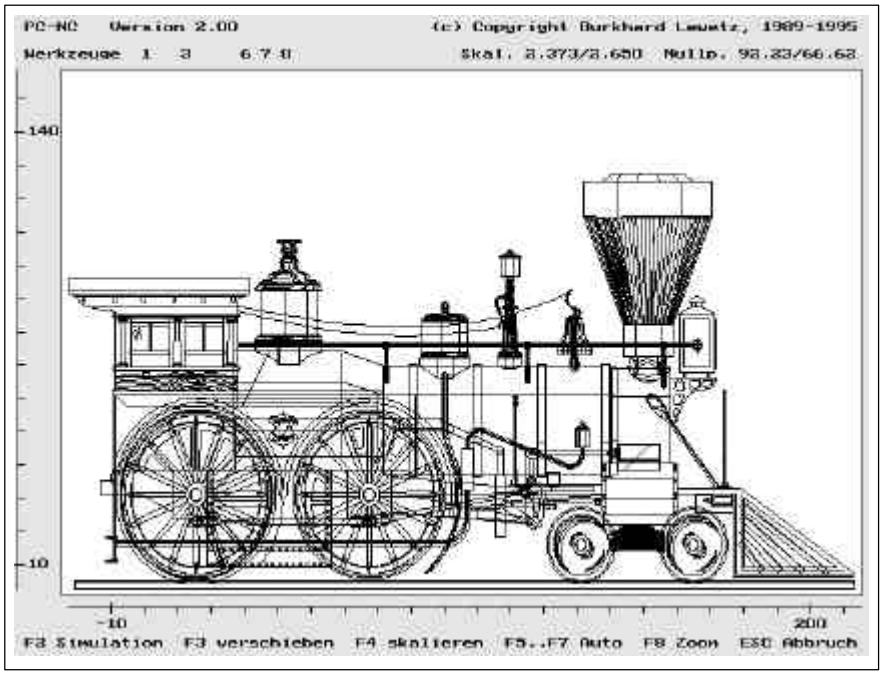


Figure 7-4 Plotting with ***PC-NC***

3. Fix the paper or the material being used firmly onto the working surface and make sure it is smoothed out.

4. Only use plot files in HP-GL format.

7.5. ...use *PC-NC* for drilling ?

Bear the following points in mind when using ***PC-NC*** for drilling:

1. Define bit management data for all the drill bits you will need.

2. Connect the coolant pump and the drilling spindle to the appropriate signals.

3. Use a soft carrier material (plastic or wood) to cushion the workpiece, e.g. a board, when clamping it and use the calibrate function to align it if necessary.

4. Set the drilling depths so as to take account of different drill bit lengths.

5. Do not set the speed of movement of the drilling axis (Z-axis) so fast that there will not be sufficient torque to drill a hole in the material.

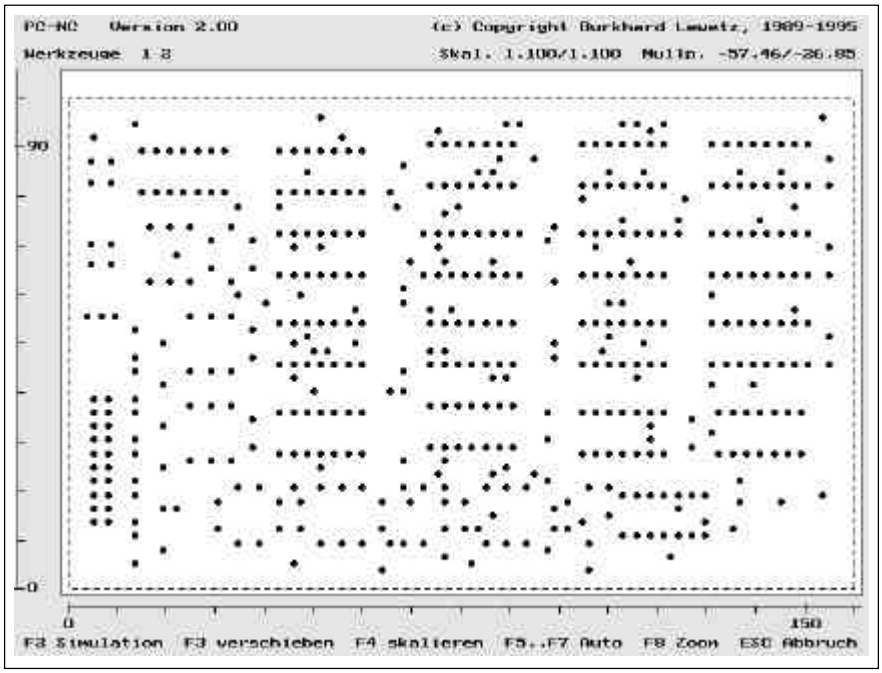


Figure 7-5 Drilling with ***PC-NC***

6. Use drilling files in Sieb&Maier or Excellon formats. Other formats can easily be adapted using the build in editor.

7.6. ...use ***PC-NC*** for milling ?

Bear the following points in mind when using ***PC-NC*** for milling:

1. Define all the milling cutters you will need in the bit management function.

2. Do not select an excessively fast advance speed for the milling cutters. The value set must be suitable for the work piece and milling cutter materials as well as the torque capacity of the X and Y-motors.

3. Connect the coolant pump and the milling cutter spindle to the appropriate signals.

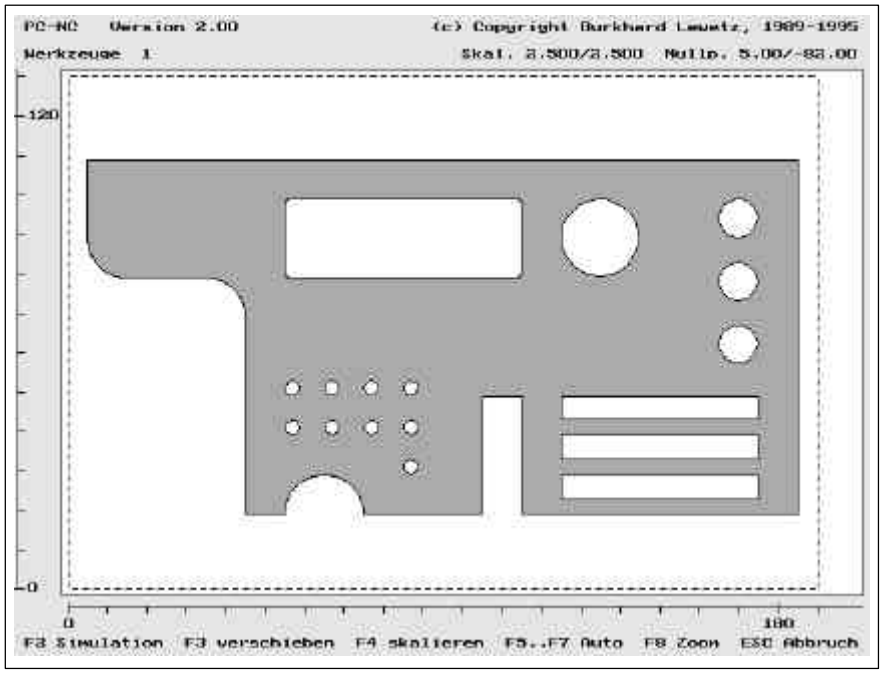


Figure 7-6 Milling with ***PC-NC***

4. Clamp the workpiece so it hangs clear and use the calibration function if required.

5. Set the working area so collisions with the clamping devices are impossible.

6. Use job files in HP-GL format and take the milling cutter radii into account when preparing the files.

7.7. ...use *PC-NC* for milling boards ?

Milling the insulation material off boards is a popular method of making prototypes in particular. What is more, it produces faster results than the normal etching method.

You can use the following method if the layout program has not already generated the data required for the insulation channels:

1. Generate the layout as a bitmap file (e.g. TIFF) using layout programs such as Eagle® or CADdy®.

2. Trace the bitmap file automatically using a drawing program such as Designer® or CorelTrace® (part of the CorelDRAW® package). This converts the bitmap data into vector format.

3. Save the result as an HP-GL file.

4. Use ***PC-NC*** to mill the HP-GL data onto copper-coated board material. The conductor pathways and pads should be retained and only the insulation channels are milled out.

5. Brush off the milled board well to remove any protruding burrs.

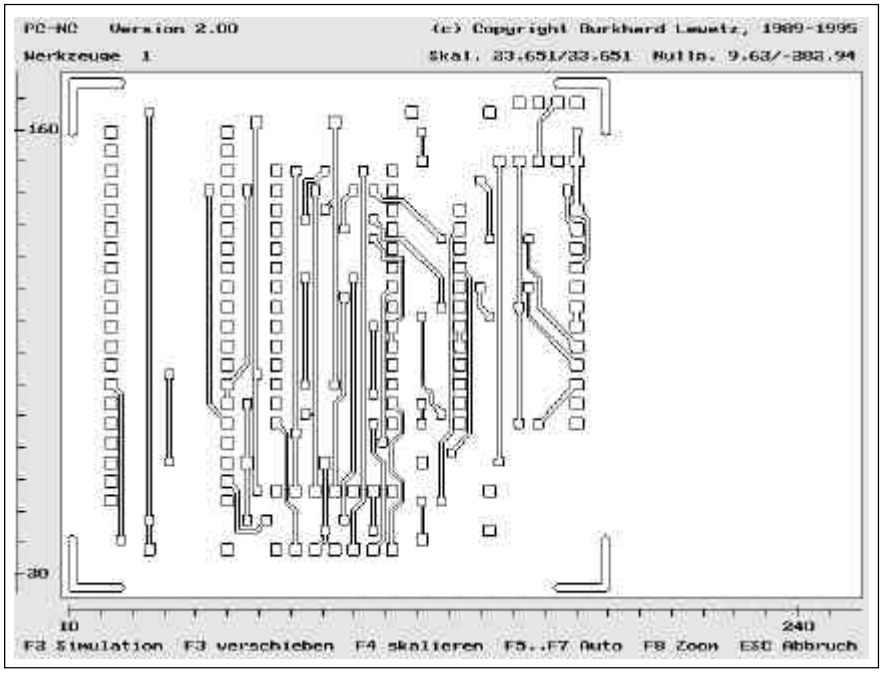


Figure 7-7 A board for outline milling shown in graphic mode

7.8. ...use ***PC-NC*** for engraving and grinding ?

Bear the following points in mind when using ***PC-NC*** for engraving.

1. Define all the bits you will need in the bit management function.

2. Set the advance speed of the bits appropriately for the depths of grinding. If necessary, run tests with the grinding bits and remnants of the material.

3. Connect the grinding spindle to the appropriate signals.

4. Take the lengths of the bits into account when setting the plunge depths. It is best to clamp the bits with a wood block as a stop.

4. Clamp the workpiece so it is level and cannot slip out.

5. Use job files in HP-GL format. Programs such as Corel DRAW® can be used for making attractive graphics and lettering. The data output using the CorelDRAW® export function can be processed by ***PC-NC*** without any problems

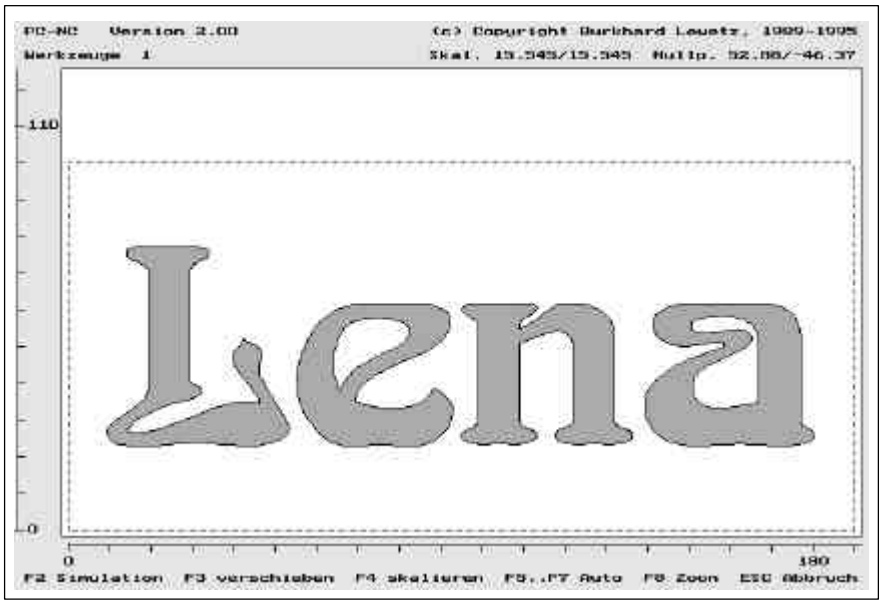


Figure 7-8 The original for an engraving

7.9. ...pause a job ?

The (ENTER) key can be used for pausing a current job process. It takes a few seconds until the command in progress is completed and then the motors are stopped.

During a pause you can do the following...

- check and modify certain parameters (speeds, tool parameters, display...)
- move to parked position to clean or change the tool
- activate the online help
- cancel the job process by pressing (ESC)

The job process can be continued precisely where it left off using the MOVE START function or by pressing (F3).

7.10. ...use the quick menu ?

The QUICK menu gives you quick access to the most important functions required during mass production of workpieces. This menu does not allow the parameters or the machine to be changed any further.

The following functions are available :

- FILE OPEN
- FILE DISPLAY
- START a job process
- REFERENCE MOVEMENT
- PARK

Quit the quick menu by using the last menu item EXIT.

7.11. ...adjust my NC data ?

PC-NC has the functionality to adjust NC data by shifting and scaling manually or automatically to a predefined size or to the defined working area. This all is possible in the graphical prewiew by simply pressing different function keys.

The following keys can be used :

(F3)	shift NC data somewhere in the whole working area.
(F4)	scale NC data in X or Y direction
(F5)	center NC data in current size within the defined working area.
(F6)	scale and center NC data to fit into the working area by 90% and retain side relationships.
(F7)	scale and center NC data to fit into the working area by 90% and do not retain side relationships, i.e. scale X and Y independent.
(shift-F6)	scale and center NC data to fit into the working area by 100% and retain side relationships.
(shift-F7)	scale and center NC data to fit into the working area by 100% and do not retain side relationships, i.e. scale X and Y independent.
(alt-F5)	undoes any automatic shifting and scaling

By shifting and scaling ***PC-NC*** recalculates the parameters of the zero point and the scaling factors. To keep them please choose PARAMETER-SAVE after finishing adjustment.

7.12. ...use the tangential axis for cutting ? (version 3.20 only)

PC-NC Tan can control a 4th axis for tangential cutting. Tangential movement means, that ***PC-NC*** always turns the blade of the cutting knife to face the moving direction. This function is necessary for cutting foils in very high quality.

Requirements are the mechanical parts of the cutting knife and the 4th stepper motor to turn the knife axial to the Z axis. The parameters and the function itself must be enabled with one parameter in the machine specific parameter dialog.

For using the tangential axis for the first time please do the following steps :

1. connect a reference switch at the tangential axis, test it by use of the SIGNALTEST function and define the corresponding pin number as parameter
2. define the axis resolution
3. check the slow and fast speeds by manual tests and define them as parameters
4. perform a reference movement including the tangential axis
5. activate the MANUAL MOVE function and turn the tangential axis by pressing the keys (a) and (A) to a zero position. The blade of the knife must face to the negative X direction. You easily can perform a few cuts for testing simply by moving the X axis. The offset must be defined as parameter.
6. perform another reference movement and check the correct direction of the 4th axis (X-).

After defining all parameter settings test the cutting function with a simply NC data file. Sometimes it may be necessary to optimise the turning speeds once again.

8. Further information

8.1. Pin assignment

Port assignment

PC-NC controls the machine via one of the parallel printer ports. This is a 25-pin SUB-D socket with the following assignment:

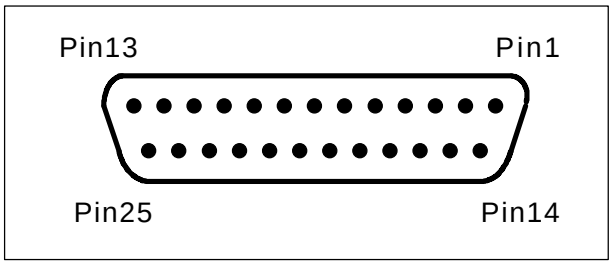


Figure 8-1 Printer port socket

Pin 2	Motor X direction
Pin 3	Motor X clock pulse
Pin 4	Motor Y direction
Pin 5	Motor Y clock pulse
Pin 6	Motor Z direction
Pin 7	Motor Z clock pulse
Pin 1	Drilling spindle on/off
Pin 14	Coolant pump on/off
Pin 17	Current reduction when motor is not running
Pins 10/11/ 12/13/15	Free for signals from the limit and reference switches as well as for a ready signal
Pins 18-25	Signal earth (0V GND)

Version 3.10

Pin 8	always HIGH (5V TTL)
Pin 9	Molette
Pin 16	Boost signal, HIGH during ramp movements

Version 3.20 with tangential axis

Pin 8	Motor 4 direction (tangential axis)
Pin 9	Motor 4 clock (tangential axis)
Pin 16	Molette

The signals on the parallel printer port can be used for controlling all stepper motor output stages which operate with a +5 V TTL level at their inputs.

Always turn off the machine and the PC before connecting them up. For reasons of safety, you should always make provision for an emergency off switch which deenergises the motors.

Extra signals

***PC-NC* controls the extra output signals as follows:**

Boost (3.10 only) HIGH during ramp movement, can be used to increase the current at the output stages temporarily.

Current reduction HIGH when no motor is running or inverted, depending on the parameter.

Drilling spindle HIGH during the entire job process except during a bit change. (Can be activated or inverted using the parameter.)

Coolant pump HIGH when the bit is in the workpiece and is moving in the workpiece. LOW before the bit is withdrawn from the workpiece. (Can be activated or inverted using the parameter.)

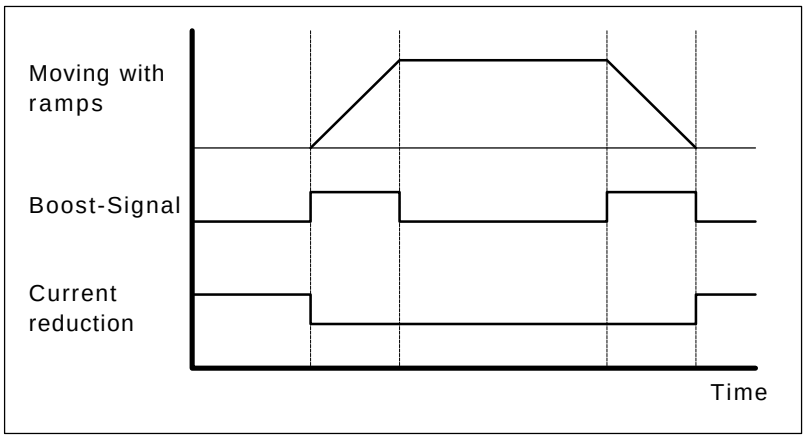


Figure 8-2 Signal profile for boost and current reduction

Take great care when connecting up the equipment in order to avoid damaging the machine and the PC. Ideally, the signals to and from the machine should be electrically isolated.

spindle speed output

PC-NC can control spindle speed with a 4 bit signal at LPT2 or LPT3. 15 different speed levels are possible and binary coded according to the following table.

Signals at pins 2 to 5 :

Example :

Speed level	5 4 3 2	Speed level	5 4 3 2
1	0 0 0 1	9	1 0 0 1
2	0 0 1 0	10	1 0 1 0
3	0 0 1 1	11	1 0 1 1
4	0 1 0 0	12	1 1 0 0
5	0 1 0 1	13	1 1 0 1
6	0 1 1 0	14	1 1 1 0
7	0 1 1 1	15	1 1 1 1
8	1 0 0 0		

8.2. Interpreters

At present, interpreters for three NC data formats have been implemented ***PC-NC***.

HP-GL interpreter

Plotting data with the HP-GL interpreter

The HP-GL interpreter recognises the most important commands in HP-GL plotter language. Successful tests have been performed with various CAD and drawing programs.

Drawings generated by the following CAD systems, amongst others, can be processed without any problems:

- AutoCAD®
- AutoSketch®
- Eagle®
- CADdy®

Windows HP-GL driver

As well as this, the interpreter can also understand the default Windows driver *HP-Plotter* which is used by the following programs, amongst others, for preparing their data and storing the data in a file:

- CorelDRAW®
- Designer®

Alternatively, a driver for the HP 7475 or HP Color Pro plotters can also be used.

HP-GL syntax

The following commands result in actions by the machine:

PA[*x,y*[*,x,y*]];	Move to absolute position(s)
PR[*x,y*[*,x,y*]];	Move to relative position(s)
PU[*x,y*[*,x,y*]];	Raise and move bit
PD[*x,y*[*,x,y*]];	Lower and move bit
SP*x*;	Select bit
IP*x,y*,[*,x,y*];	Defining scaling points
SC*x,y,x,y;*	Defining scaling factors
CI*r*[*,a*];	Move a full circle
AA*x,y,a*[*,c*];	Move an absolute arc
AR*x,y,a*[*,c*];	Move a relative arc
PP[x,y[,x,y]];	Perform a drill at the position, PD and PU

The interpreter also recognises the following commands, although these do not produce any response from the machine:

DT...;	**LT...;**
LB...;	**VS...;**
IN...;	**CA...;**

The list below presents some of the rules of syntax in the HP-GL language. These rules must be observed when writing or editing the data:

- Co-ordinates must always appear in pairs (x,y).
- All commands with co-ordinates must have either none, one or several pairs of co-ordinates.
- Each command is terminated by a semicolon (;) or a new command.
- PA and PR also set absolute or relative positioning for the subsequent commands.
- The final command should raise the bit (PU;)

HP-GL example

PU;	*Start with bit raised*
PA1000,1000;	*Then move to 1000/1000*
SP1;	*Select bit 1*
PD;	*Lower bit*
PA1000,2000;	*Describe a square*
PA2000,2000;	
PA2000,1000;	
PA1000,1000;	
PU;	*Raise bit*

Some NC files contain complex commands which cannot be processed or displayed by *PC-NC*. In such cases, you can use the PrintGL shareware program to convert them into simple vector commands. The data produced by PrintGL are fully compatible with *PC-NC*.

Drilling data interpreter

Various drilling data formats

The drilling data interpreter recognises data in the following formats: Excellon, Sieb&Maier 1000 and Sieb&Maier 3000.

A drilling file always starts with one or two percentage signs and finishes with the command M30. Between there limits there may be any number of lines with X and Y co-ordinates and the bit number T.

Drilling data syntax

The syntax is as follows:

% or %%
[Xnumber][Ynumber][T*x*]
.....
M30

There are two different formats. In format 1, the co-ordinates are always entered as five-digit numbers (the trailing zeros can be omitted, e.g. X201 corresponds to 2100).

The leading zeros are dropped in format 2 and the co-ordinate data therefore have differing numbers of digits.

The list below presents some of the rules of syntax of the drilling formats. These rules must be observed when writing or editing the data:

- Each command must start in a new line.
- In a line containing co-ordinates and a bit command, the bit is changed before the position is drilled.
- Co-ordinates can also have decimal points, e.g. X123.456
- All lines prior to the percentage sign are assumed to be remarks and are not evaluated.

Example of format 2 drilling data

Short drilling program written in format 2	*Remark lines*
%	*Start of program*
T1	*Select bit 1*
X1000Y1000	*Drill hole at 1000/1000*
X1200Y2340	*Drill hole at 1200/2340*
X2700Y2950T2	*Bit 2, drill hole at 2700/2950*
Y1000	*Drill hole at 2700/1000*
M30	*End of program*

Here is the same example in format 1:
(Always 5 digits, trailing zeros deleted.)

Example of format 1 drilling data

Short drilling program written in format 1	*Remark lines*
%	*Start of program*
T1	*Select bit 1*
X01Y01	*Drill hole at 1000/1000*
X012Y0234	*Drill hole at 1200/2340*
X027Y0295T2	*Bit 2, drill hole at 2700/2950*
Y01	*Drill hole at 2700/1000*
M30	*End of program*

G code interpreter

PC-NC has a G code interpreter included. G code programs are used with professional milling and turning machines with 2 or 3 axes.

The following commands are available :

G commands	G00	go with rapid speed
	G01	linear interpolation with feed speed
	G02	clockwise circular interpolation, feed speed
	G03	counterclockwise circular interpolation
	G04	dwell time (time in ms with F command)
	G17	plane XY
	G18	plane XZ
	G19	plane YZ
	G27	move to parked position
	G70	units in inch
	G71	units in mm
	G74	performs a reference move

	G90	absolute position information
	G91	relative position information
M functions	M00	Program halt
	M03	Spindle on
	M04	Spindle on
	M05	Spindle off
	M06	Change tool
	M07	Coolant on
	M08	Coolant on
	M09	Coolant off
	M16	Wait for input, input number in F
	M30	End of program
Other	N	Record number commands
	S	Spindle speed
	F	Pin number (with M16 command)
	F	Dwell time (with G04 command)
	F	Feed (without M16 and G04 command) Speed in set unit of measurement (mm/s or mm/min)
	I,J,K	Circle parameters
	T	Select tool
	X,Y,Z	Coordinates

Example: Square with rounded corners

```
%prog2                          Start of program
N001 G90                        Absolute coordinates
N002 G71 T1 M6                  Dimensions in mm, tool 1 rapid
N003 G00 X110 Y100 Z10          speed to 1st position
N004 G01 Z11                    Plunge movement with Z
N005 G01 X190                   Feed movement in straight line
N006 G03 X200 Y110 J10          Arc about center point
N007 G01 Y190                   etc.
N008 G03 X190 Y200 I-10
N009 G01 X110
N010 G03 X100 Y190 J-10
N011 G01 Y110
N012 G03 X110 Y100 I10
N013 G01 Z10
N014 G00 X0 Y0 Z0               Rapid speed to zero point
N015 M30                        End of program
```

Bear in mind the following points when writing G code programs:

- The programs must be written using an editor or an external program.

- The command number can be defined using N commands
- For arcs, you can optional use a third coordinate for moving a helical contour
- comments in round brackets () will be ignored

8.3. Restrictions under MS-Windows

To control stepper motors directly with clock/direction signals you must have exclusive access to all PC components. ***PC-NC*** is an absolute hard realtime application. Popular multitasking operating systems like MS-Windows cannot support a realtime application with the necessary services.

Therefore *PC-NC* cannot be run under Windows.

9. Appendix

9.1. Error messages

Error messages occurring when ***PC-NC*** is running appear in a small red window in the middle of the screen. Messages in a grey box do not signal a fault state, they are used for information purposes.

The following fault messages may arise:

- **Out of memory**

  The main memory is not large enough to run ***PC-NC*** or a specific function. Close TSR programs or take other measures to make more memory available. ***PC-NC*** requires about 300 kB when running.

- **Parameter file not found**

  No parameter file can be found. Neither the file specified in the command line launch switch nor the default parameter file PCNC.SYS have been loaded. ***PC-NC*** continues to run using the default values.

- **Drive not ready**

  The selected drive is not ready. Make the drive ready and try again.

- **No port**

  The printer port specified in the parameter file does not exist on the computer. ***PC-NC*** runs using the LPT1 port.

- **File not found**

  The specified or selected file cannot be found. Check the drive and the path details.

- **Cannot display data**

  The NC data in the current file cannot be displayed in graphic mode. They are either not in the selected format or the data does not contain any bit action instructions.

- **Bit not defined**

  The data calls for a bit which has not been defined in the bit management function. The appropriate parameters must be defined before bits can be used.

- **HP-GL syntax error**

  The plotting file contains an invalid HP-GL command. It does not follow the rules of syntax and consequently cannot be processed by ***PC-NC***.

- **Maximum speed is xx.xx**

  The maximum speed specified is faster than the maximum speed permitted with this system clock pulse setting. ***PC-NC*** corrects the value automatically.

- **Execution cancelled**

  Movement command was interrupted when the (ESC) key was pressed. Perform reference movement before initiating any further actions.

- **Bit file not found**

  No bit file can be found. Neither the file specified in the command line launch switch nor the default parameter file PCNC.WRK have been loaded. Consequently, ***PC-NC*** cannot start any job process.

- **Invalid file format**

  The selected file is not in the selected file format. Change the parameter for the data format to HP-GL or drilling format.

- **Graphic system fault**

  Your computer is equipped with a graphics card which ***PC-NC*** cannot identify unambiguously. Unfortunately, NC data cannot be displayed in graphic mode.

- **No external editor defined**

  The FILE-EDIT menu function can be used for calling up an external editor in order to edit NC data. However, no editor has been defined. ***PC-NC*** therefore merely displays the file in a window, in which case it cannot be edited.

- **Help file not found**

  The file containing the help texts cannot be found. Its filename is PCNC.HLP and it must be located in the active directory or the ***PC-NC*** directory. It is impossible to display help texts.

- **Working area too small**

  The working area is too small for the selected workpiece. Remedy this problem either by changing the unit of measurement, moving the zero point or, of course, by increasing the size of the working area.

- **Editor not found**

  The external editor cannot be found. Check the launch path and the name of the editor in the parameters.

- **Workpiece extends beyond working area**

  The working area was exceeded when the current workpiece was being machined. This situation might be remedied by performing a reference movement. If not, either change the unit of measurement, move the zero point or increase the size of the working area.

- **Next position outside working area**

  The current job process was cancelled because the next position to be moved to is located outside the working area. Correct the settings and restart the job process.

- **Unknown command**

  The job process was halted on encountering an unknown command in the NC file. You can now cancel the process or continue, skipping the unknown command.

- **Incorrect launch switch disregarded**

  A command line launch switch is invalid or its format is incorrect. It was disregarded. The correct command line launch switch format is:

  PCNC [file] [parameters] (ENTER)

Explanation:	**file**	Job file which should be loaded as soon as the program is launched
Parameters:	**-m**	Monochrome display
	-h, -?	Parameter display
	-s file	Load a specified parameter file
	-w file	Load a specified bit file

- **PCNC.EXE fault - cancel**

  The PCNC.EXE file is damaged and should be re-installed from the original diskette. There may be a virus in your system.

- **Not all PCNC functions available under Windows**

  Unfortunately, the full functionality of ***PC-NC*** is not available under MS-Windows. All the functions for moving the motors are inactive, although it is possible to load and display NC data and define parameters.

- **Limit switch reached**

  One of the limit switches has been triggered. The machine carriage seems to have moved against a lateral limit. Switch to the MANUAL MOVE function and use the keyboard to move the machine carriage back from the limit switch. You must perform a reference movement before initiating any further actions.

- **Cannot create file**

  PC-NC tried unsuccessfully to create a file on the current drive. Either the drive is not ready or it is full, write protected or you do not have any access rights. Correct the problem or change to another drive.

- **Cannot create temporary file**

  PC-NC tried unsuccessfully to create a temporary file on the current drive. Temporary files are used for displaying help texts or for storing data temporarily. Either the drive is not ready or it is full, write protected or you do not have any access rights. Correct the problem or change to another drive.

- **No parked position defined**

  You cannot move to the parked position before defining the appropriate co-ordinates, i.e. the co-ordinate values are not equal to zero.

- **Required bits not defined**

  The job file uses bit numbers which have not been defined. The plunge depth and plunge speed must be defined for each bit. Also, for HP-GL files, the advance speed must be defined for each bit.

- **Error in batch command**

  The command of a batch job currently being processed does not comply with the rules of syntax. Compare the command with the description in the manual or the online help and correct the problem.

- **No value specified**

  The batch command currently being processed requires a value which is not defined in the batch job.

- **Value in batch command out of range**

  The value in the batch command currently being processed is not within the specified range. Compare the value with the valid ranges in the overview of parameters.

- **No file specified**

  The OPEN batch command requires the filename of the NC file you want.

- **Position not yet known**

  The CHECK POSITION function cannot be implemented until the current reference position is known. Either you have not yet performed a reference movement or an ongoing positioning movement was cancelled by pressing (ESC).

- **No pin number specified**

  The WAITPIN executable command requires the pin number where the external signal is connected to be specified. The signal level is defined using the sign. Pin numbers 13, 15 or -13, -15 are possible.

- **Incorrect pin number**

  The WAITPIN executable command requires the pin number where the external signal is connected to be specified. The signal level is defined using the sign. Pin numbers 13, 15 or -13, -15 are possible.

- **GOTO out of range**

  The GOTO executable command causes processing to be continued from the specified line. However, the batch job is shorter than the GOTO target used.

- **Cannot nest batch job**

  No further batch jobs can be opened and processed when one is already running.

- **No DOS command specified**

  The DOS executable command requires a runnable DOS command as its operand. However, no such command has been defined.

- **Bit depth too much**

  The defined bit depth is more than the defined working depth of the Z axis.

- **Length sensor not defined**

  The automatic length check function can be used with a defined length sensor only. Please define a pin number like the reference switches.

- **Text not found**

  The defined text cannot be found within the file loaded in the editor.

- **Maximimum length reached**

  The maximum length of a line is reached in the text editor. Please break the line into two new ones.

- **G code systax error**

  There was a syntax or command error found in the loaded G code program. Please check all lines according to the interpreter description in the last chapter.

- **Error at joystick**

  The keys or positions of the joystick cannot be recognized at the calibration function. Please check the port and the joystick itself.