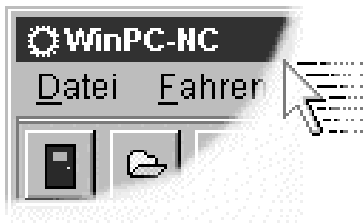


**Milling, plotting, engraving,
drilling, grinding, cutting,
dispensing and much more
besides with...**

WinPC-NC



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

Version 1.70

April 2003

Lawful purchase of the diskette and the manual conveys permission for one person to utilise the **WinPC-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 (P.O.Box) 1221
D-88071 Meckenbeuren
Germany
Tel. ++49-7542-21886
Fax ++49-7542-3889
eMail info@lewetz.de
Homepage www.lewetz.de

April 2003

MS-Windows is a registered trademark 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.
Designer is a registered trademark of Micrografx,
Inc.Borland C++Builder is a registered trademark of Borland International, INC.
Other products mentioned by name are trademarks or registered trademarks of their corresponding companies.

1.700

Table of contents

	The structure of this manual	0-5
	Definitions	0-5
	Use of typography	0-6
	Synonyms	0-6
1.	What can <i>WinPC-NC</i> do ?	1-1
2.	First steps	2-1
2.1.	Installation	2-1
2.2.	Launching the program	2-1
2.3.	Configuring <i>CNC-Control</i>	2-3
2.4.	Exiting <i>WinPC-NC</i>	2-3
3.	Operating <i>WinPC-NC</i>	3-1
3.1.	Graphical display of the NC file	3-1
3.2.	Drop-down menus and function keys	3-5
3.3.	The individual menus	3-5
3.3.1.	FILE menu	3-6
	OPEN	3-6
	OPEN WITHOUT PARAMETERS	3-7
	EDITOR	3-7
	EXIT	3-7
3.3.2.	MOVE menu	3-8
	START	3-8
	JOG	3-9
	JOYSTICK JOG	3-12
	REFERENCE MOVE	3-13
	SELECT TOOL	3-14
3.3.3.	PARAMETERs menu	3-16
	SAVE	3-16
	SAVE AS	3-17
	LOAD	3-17
3.3.4.	SPECIAL FUNCTIONS menu	3-18
	SIGNAL TEST	3-18
	MOTOR TEST	3-19
	CONTROLLER INFO	3-20
	JOYSTICK CALIBRATION	3-21
	CHECK POSITION	3-21
3.3.5.	HELP menu	3-22
	TOPICS	3-22
	DISCLAIMER	3-22
	ABOUT <i>WinPC-NC</i>	3-23
4.	Parameter settings	4-1
4.1.	Tool management	4-1

4.2.	Speeds	4-9
4.3.	Coordinates	4-11
4.4.	Data format and associated parameters	4-15
4.5.	Miscellaneous parameters	4-17
4.6.	Ports	4-23
4.7.	Signals and dwell times	4-23
4.8.	Machine parameters	4-25
4.9.	Macros	4-31
5.	Additional informationen	5-1
5.1.	Interpreters	5-1
5.1.1.	HPGL	5-1
5.1.2.	MultiCAM	5-3
5.1.3.	Drilling	5-3
5.1.4.	DIN/ISO or G-code	5-5
5.2.	Error messages	5-8
5.3.	Special versions of <i>WinPC-NC</i>	5-11

The structure of this manual ...

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

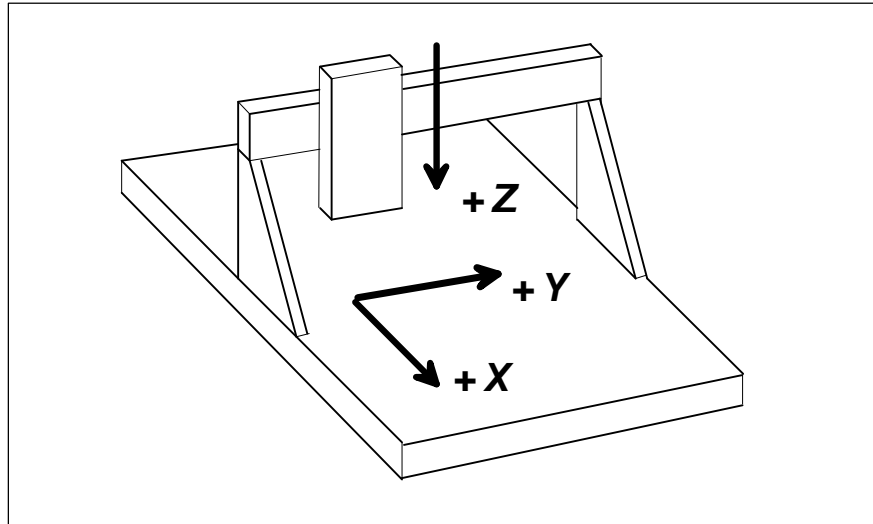
- Chapter 1:** Brief explanation about *WinPC-NC*, the possibilities for using it and the hardware requirements.
- Chapter 2:** Initial start-up procedure, description of how to install the program.
- Chapter 3:** More detailed descriptions of how to operate the program and the individual functions of *WinPC-NC*.
- Chapter 4:** Explanation of all parameters and the setting options.
- Chapter 5:** Further technical information about the program, about the realized NC format interpreters, error 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 *WinPC-NC*. Depending on the application, the file may contain milling, plotting, drilling data 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 *WinPC-NC*.
- Button** Mouse clickable field to activate a certain function
- Checkbox** Box for activating or deactivation a certain parameter or function

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 :



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. <i>'Perform reference movement? Y/N'</i>
Function names	Capital letters, e.g. SIGNAL TEST

Synonyms

DIN/ISO	G code
dispensing	metering
molette	collet chuck
working file	job file
working process	job
reference move	homing
manual move	jog
measuring probe	surface block
feed speed	feed rate
safety clearance	toollift

1. What can *WinPC-NC* do?

universal program

WinPC-NC is a software program which takes any standard personal computer and our CNC-Control axis controller and turns them into a universal NC control system for up to 4 axes.

Axis controller for real-time tasks

The axis controller is required because MS-Windows is a multitasking operating system and either none or only some limited tasks can be run under true real-time conditions.

The 4 stepper motor axes can be used for making any 3D mechanical structures. It is possible to accomplish an extremely broad range of jobs. Standard applications include:

- Drilling
- Milling
- Plotting
- Cutting foils
- Grinding
- Engraving plates
- Dispensing
- Sharpening coining dies

Extensive parameters

The extensive range of options for setting parameters means the program can be adapted to almost all 2 - 4-axis machines.

Clearly structured operator interface

WinPC-NC offers a well thought-out and modern operating concept incorporating drop-down menus and a windows management system with mouse and keypad operation. This makes it easy to learn and master the program.

Runs on any PC

All that is required for operating **WinPC-NC** is a commercially available personal computer with a hard disk, serial port, any graphics card and MS-Windows 95/98/NT/2000 or XP.

In addition, our **CNC-Control** axis controller is required. The Windows-PC with **WinPC-NC** is connected to the axis controller by means of a commercially available null-modem cable, while the digital inputs and outputs of the axis controller are connected to the machine.



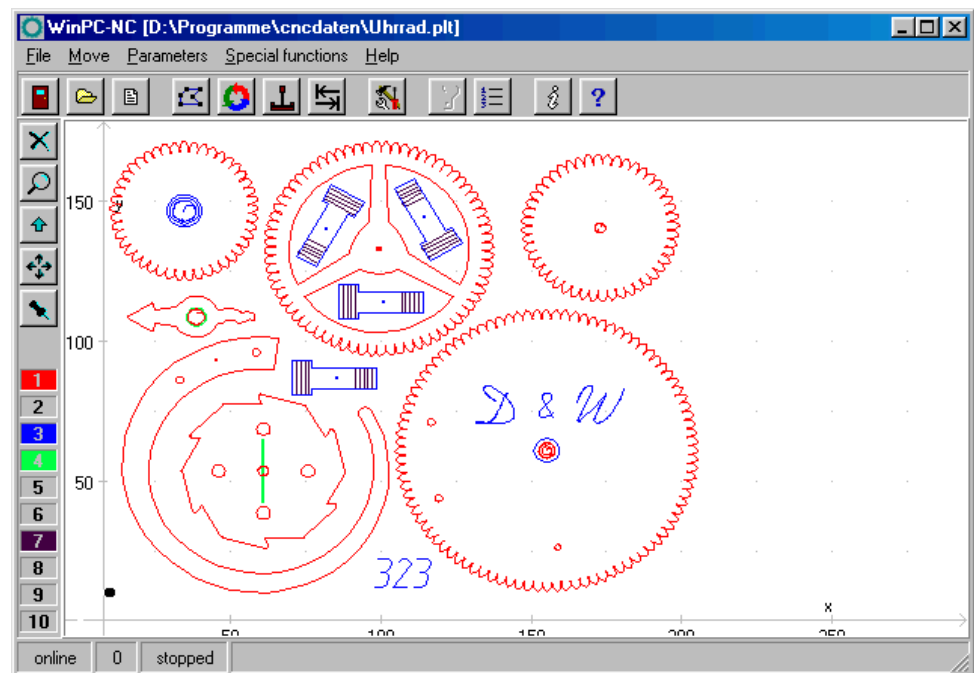
Sample 3-axis milling machine

Special features

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

- 32-bit arithmetic for an almost unlimited range of applications
- Stepping frequencies up to 40 kHz (with our *CNC-Control* axis controller)
- Jog mode in exact steps using the cursor keys or the external keypad
- Graphical display with scale, zoom and shift functions
- Runs under Win95/98/ME/NT/2000 and XP without restrictions
- True Windows multitasking program, i.e. you can switch to your CAD program or surf the web while a process is running on the machine
- Axis resolutions, speeds and ramp length for the X/Y and Z-axis can be set within generous limits. The fourth axis can be programmed as U, V, W, A, B or C.
- Reference and limit switch interrogation

- Recognizes HPGL, G code, MultiCAM and drilling formats
- Extensive tool management function, tools can be selected individually, colors can be set, repetition and feed functions
- Context-sensitive help function and help index
- Speeds can be set between 0.01 and approx. 1000 mm/s
- High-performance integrated editor
- Lots of external signals for synchronizing the procedure, e.g. Start signal, spindle speed reached, machine ready
- Speed control of the drilling/milling spindles
- Units of dimension can be set to mm, inch and mm/s, mm/min and inch/min
- Multilingual, other languages can easily be retrofitted



Main screen of *WinPC-NC*

- Supports various additional signals and sensors
- Supports automatic tool changers (ATC)
- Tool length measurement and compensation
- Flexible macro language for running subroutines at various points in the program
- Special dispensing functions, for example to switch off the dispensing output before the end of the contour is reached

What can *WinPC-NC* do ?

- Supports double X-axes with bridge alignment in reference travel
- Override for feed rate and spindle speed
- automatic measurement of different Z depths by using of a surface probe
- ...there are many other features besides these

2. First steps

2.1. Installation

User-friendly installation

WinPC-NC is installed using a user-friendly setup program. Please insert disk 1 into the drive, select Start, Run and type A:SETUP.EXE. The installation wizard then guides you through the entire procedure.

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 :

WINPCNC.EXE	Control program
WINPCNC.WPI or WTI	Parameter file
WINPCNC.WPW or WTW	Tool file
WINPCNC.HLP	Help texts
WINPCNC.LNG	Messages and texts, multilingual
WINPCNC.KEY	Key assignment for keypad
README	Latest changes to the manual
*.DLL	Some files needed in the Windows folder
*.PLT *.SMM *.DIN	Sample NC files
CNCKONF.*	Several files of the configuration program

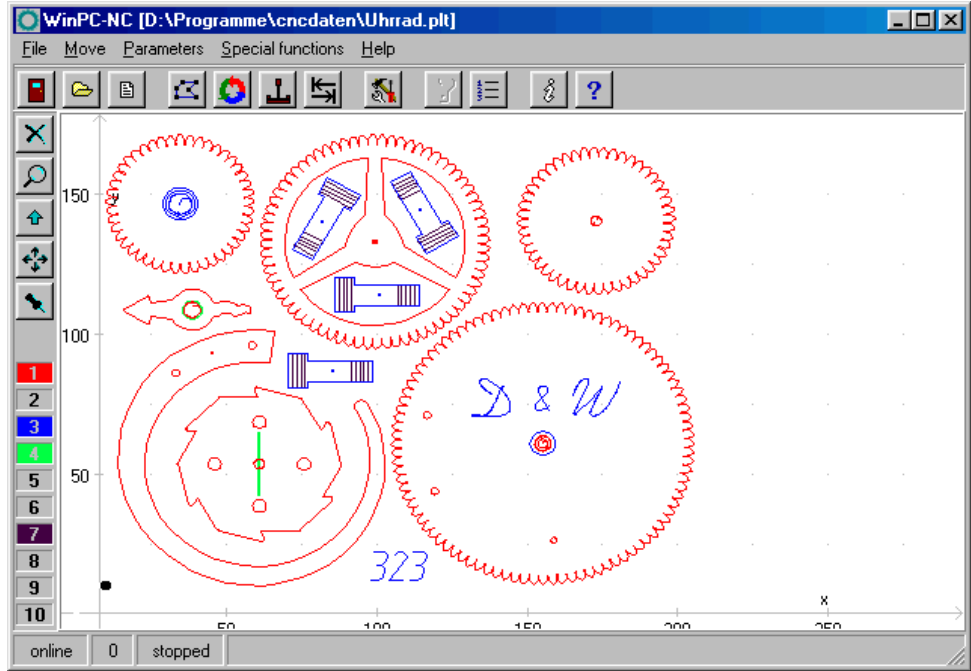
2.2. Launching the program

WinPC-NC is launched simply by clicking the desktop icon or by means of the Start menu.

The **WinPC-NC** working screen appears after a short time.

This screen is divided into several areas :

- Title bar
- Menu bar
- Quick access bar
- Display area
- Toolbar and display buttons
- Status bar



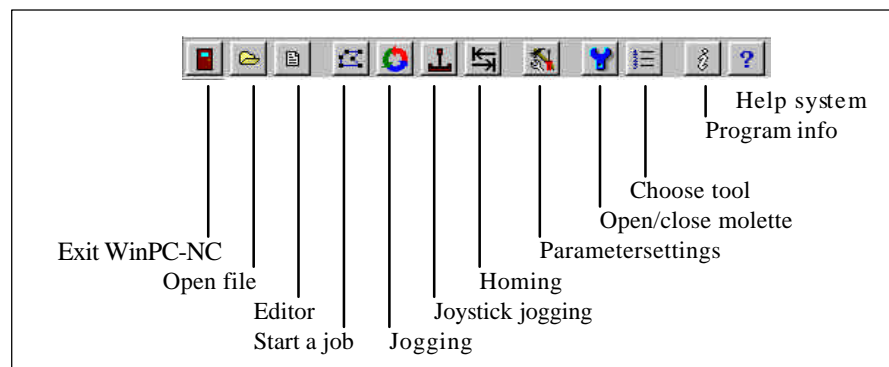
WinPC-NC screen layout

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 quick access bar enables you to select important functions simply by clicking with the mouse.

The function of the individual buttons is as follows :



Button bar in WinPC-NC

The status bar at the bottom displays information about the operating status of the software and the machine, or else a help text about the

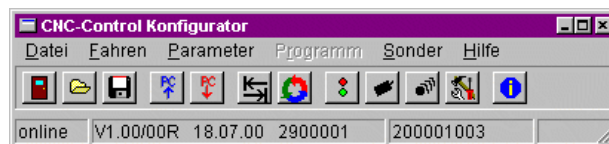
functions which are currently active. Buttons for operating the display and the tools are located along the left-hand edge.

The large area is the working area where *WinPC-NC* displays the currently loaded NC file in graphical format.

2.3. Configuring *CNC-Control*

The first basic settings for the controller should be made using the *CNC-Configurator* utility which is supplied with the controller.

For information about operation and configuration procedures, refer to the manual for *CNC-Control* or *CNC-Configurator*, or alternatively the technical reference for *WinPC-NC*.



CNC-Configurator



When first delivered, axis controllers have default settings which are suitable for most applications. As a result, no further setup work should be required.

2.4. Exiting *WinPC-NC*

You can exit *WinPC-NC* at any time by clicking the cross in the box at the top right of the window, or by selecting EXIT from the FILE menu.

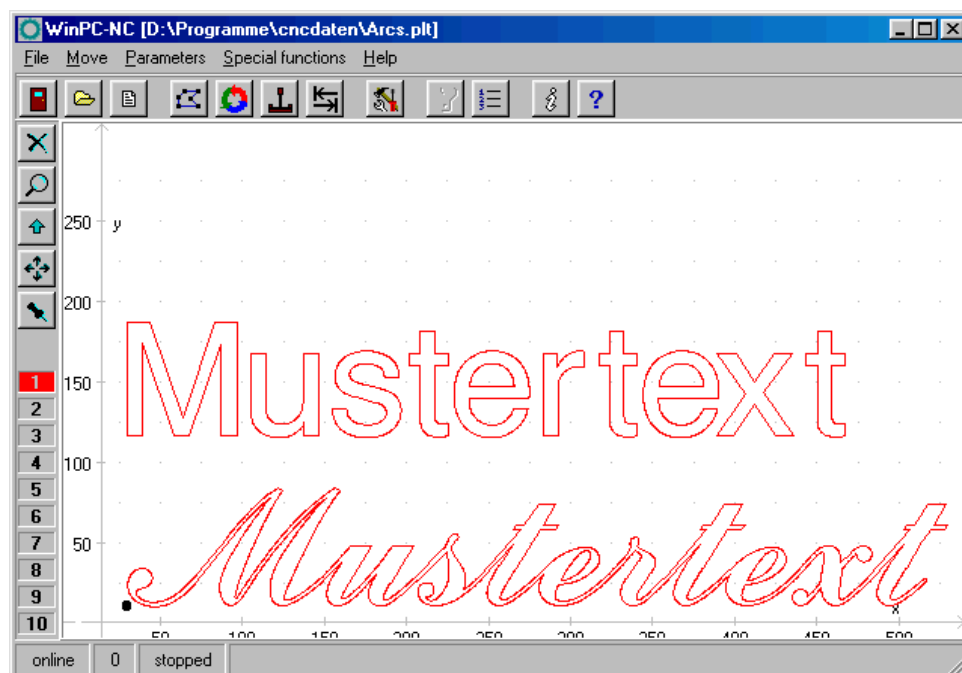
3. Operating *WinPC-NC*

3.1. Graphical display of the NC file

*Graphical
preview of the
NC files*

The graphical preview function in *WinPC-NC* is activated as soon as an NC file is selected. All contours or holes can be seen in the corresponding tool color.

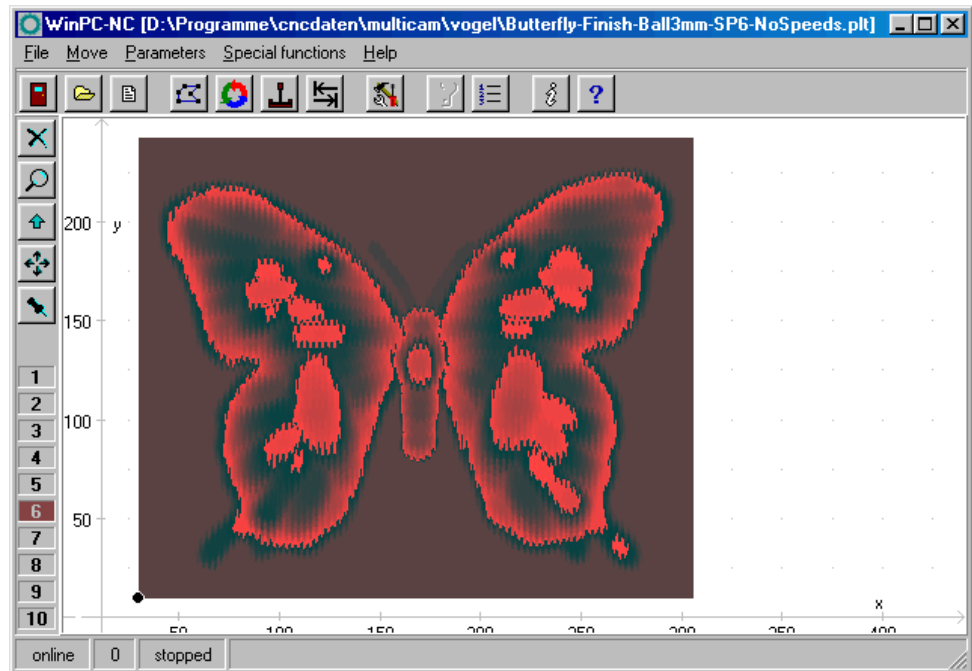
In addition, two rulers can be seen with the current dimensions and positions. The workpiece zero point is identified by a small black dot and the current machine position as a small red dot.



Graphical display of the NC file



3D data in G code programs are only displayed in the X/Y plane, while different Z-axis heights are displayed using colored shading.



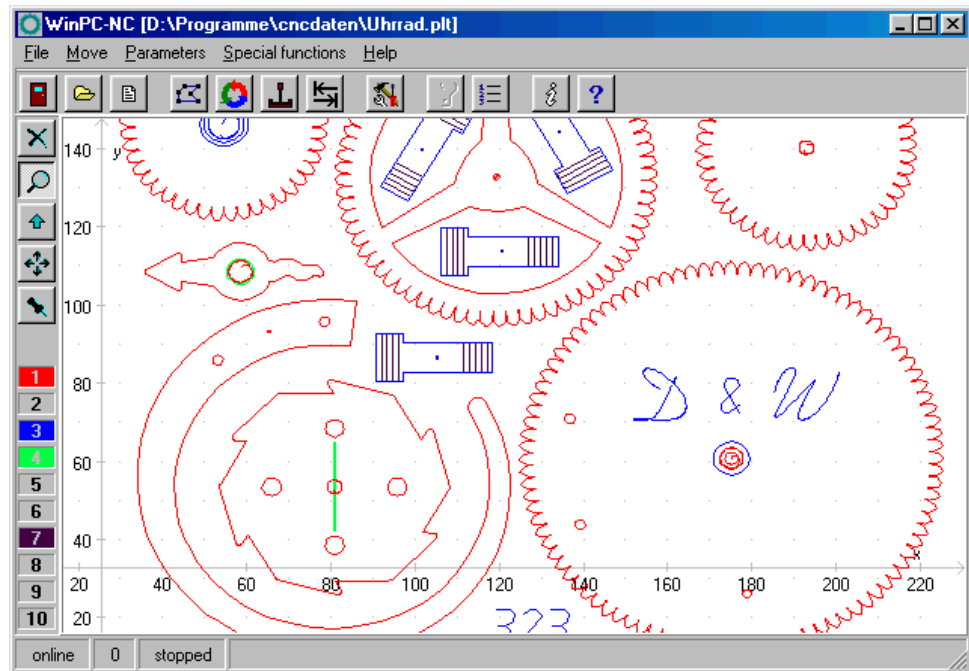
Graphical display of a 3D file with colored shading

The following actions are possible on the graphical display :



Zoom

With the zoom function, it is easy to view certain sections of the screen in a magnified view. After you select the function by clicking the zoom button, the cursor appears as a cross and you can click and drag a rectangle to select the required area.



Zoom in the graphical display of the NC file



Move

The move function makes it possible to use the mouse to move a zoomed image, to enable you to view another part of the NC file in greater detail. After the function has been activated, the cursor appears as a hand. With it, you can click and drag any point to move the image to where you want.

The movement track is displayed as a line while you are dragging. Once you release the mouse button, the graphic appears in the new position to where you have moved it.



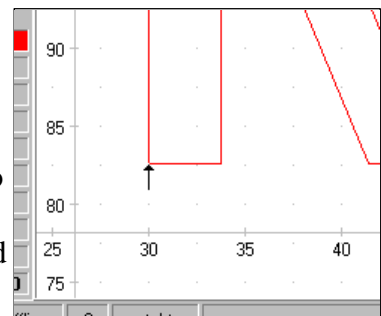
Display original size

Clicking this symbol restores the original size and position. This undoes all zoom and move actions.



Define current machine position

The current machine position is represented by a small red dot in the graphic. Using the function described here, you can position the NC file so the current machine position corresponds exactly to a point or a certain position in the file. The file zero point is simply recalculated internally.



The workpiece and the NC file can be positioned extremely accurately using this simple method.



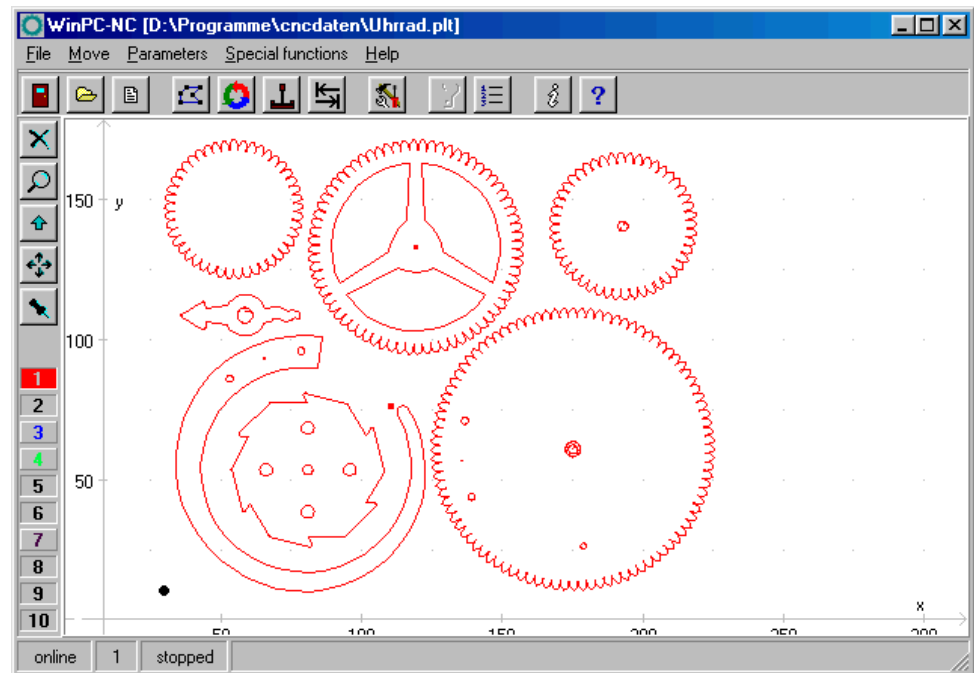
Cancel function

Once a zoom, move or positioning action has been started, it can be cancelled by clicking the cancel button. The cursor reverts to its normal arrow shape.

Set tools active/inactive

All tools which are used in an NC file are displayed along the left-hand edge in the tool frame. They are shown using their current colors. A tool can be deactivated or activated simply by clicking the tool box. Switching in this way has the same effect as activating it in the parameters.

Once tools have been switched, they immediately appear in the graphical display.



Graphical display with inactive tools

3.2. Drop-down menus and function keys

Modern user interface

WinPC-NC has a modern user interface. All functions can be accessed quickly using drop-down menus. In addition, frequently required functions can be activated quickly using the function keys.

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

The menu system is opened or activated by clicking the menu item or pressing one of the shortcut keys for the individual menus.

*Additional
function keys*

Important functions can also be activated using function keys. The function key assignment is set and is indicated in the menu functions.

The most important function keys are :

(F1)	Activate the help system
(F2)	Load new NC file
(shi ft-F2)	Load parameter file
(F3)	Start job process
(F5)	Jog
(shi ft-F5)	Joystick jog
(F7)	Load active or new file in the editor
(F8)	Start homing

3.3. 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 not possible to use the joystick if it has not been defined in the parameters.

3.3.1. FILE menu

The FILE menu combines all functions used for selecting files to process and analyze them. In addition, it is also possible to exit *WinPC-NC* here.

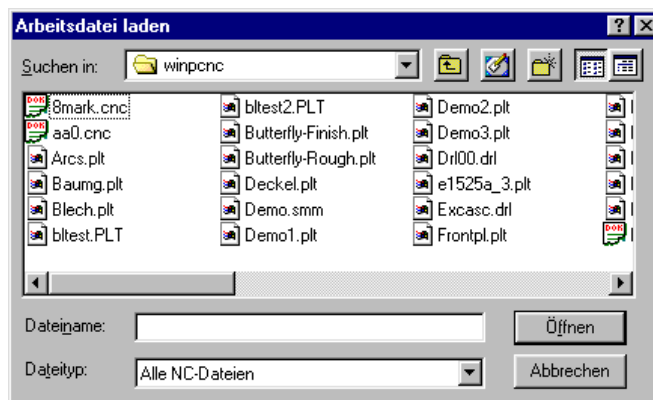
Open...	F2
Open without parameters	
Editor	F7
Exit	

Press the (al t-F) shortcut key to open the File menu.

FILE-OPEN

File selection menu

The FILE-OPEN menu item calls up an interactive function for selecting the working file.



File selection dialog box



In the dialog box, it is possible to change drives and folders, as well as to activate filters for certain file name extensions. The file selection function can also be activated using function key (F2) or by clicking the open button.

Graphical preview of NC files

The graphical preview function in **WinPC-NC** is activated as soon as an NC file is selected. All contours or holes can be seen in the corresponding tool color.

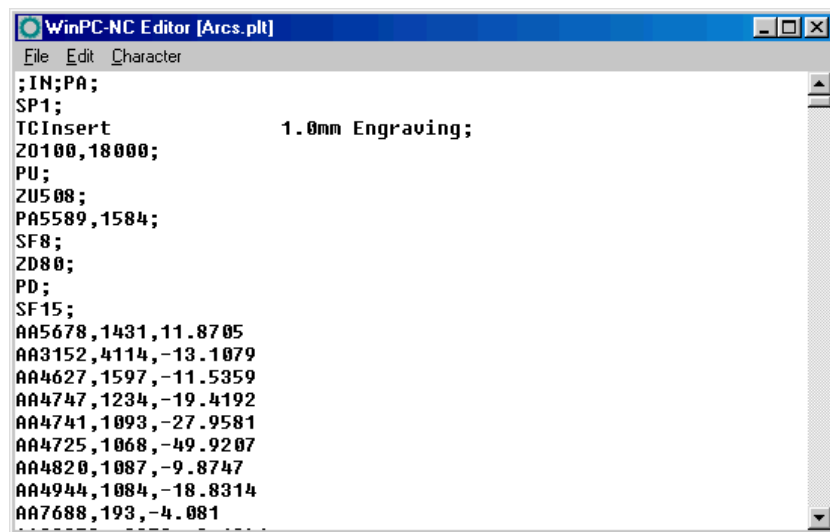
FILE-OPEN WITHOUT PARAMETERS

The second open function only opens the corresponding NC file without taking account of any project parameters. All current parameter settings remain in effect.

FILE-EDITOR

Revising the NC data

The EDITOR menu item activates the editor for creating or revising NC files.



NC file in the integrated editor



The editor can also be activated using function key (F7) or by clicking the editor button.

FILE-EXIT



You can select the FILE-EXIT menu item to exit *WinPC-NC*. Clicking the exit button has the same effect.

3.3.2. MOVE menu

Functions for moving the machine

The MOVE menu groups together all functions which are used for controlling the machine and the tool changer.

Start	F3
Jog	F5
Joystick jog	shift F5
Reference	F8
Select tool	

Press the (Alt-M) shortcut key to open the menu.

Move-START



After a working file has been loaded, the START menu item can be used for processing the job.

Pressing the (F3) key or clicking the Start button has the same effect. **WinPC-NC** controls the X and Y motors during move commands. The Z motor moves up and down during tool movement commands. All three axes can move at the same time in 3D files.

Progress display During machining, a window displays the progress as a percentage.

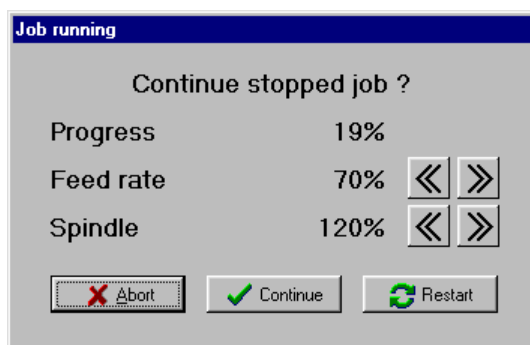


Progress display and speed override

Speed override Clicking the greater than/less than button in the window makes it possible to change the feed speed of the machine as well as the spindle speed in 5 % steps. The valid range is from 10 % to 200 %.

A working process can be cancelled by clicking the Cancel button in the progress display window, or by pressing the (ESC) key. The machine brakes all axes without losing any steps and switches off the spindle and the cooling function.

Resuming an interrupted job When you start again after a cancel, **WinPC-NC** asks whether the interrupted process should be resumed from the point where it was cancelled, or whether the process should be started over again. During an interruption, it is possible to change parameters, clean the tool or even perform jog movement and homing the machine.

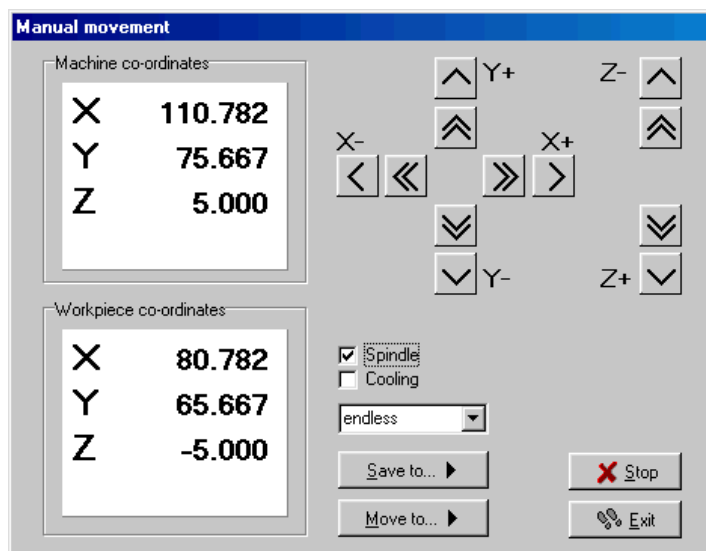


Continuing an interrupted job

MOVE-JOG



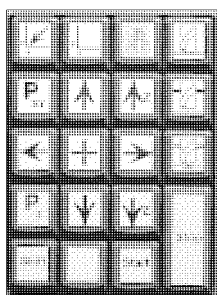
The JOG menu item calls up the manual setup function for the machine. This function can also be accessed by pressing the (F5) function key or with the jog button.



Jog movement

Exact movement with the keyboard or mouse, without loss of steps

In jog mode, it is possible to move all motors step-by-step or continuously with the white arrow keys or with the mouse. Pressing a key briefly or clicking the corresponding direction button causes only one motor step to be performed. Pressing the key for longer or keeping the mouse button pressed causes the motor to move continuously. The changeover time can be defined as a parameter.



Only slow movements are possible using the PC keyboard, whereas with the mouse the double arrow buttons can be used for fast movements and the other buttons for slow movements. The speeds are defined in the parameters.

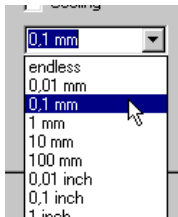
Alternatively, the machine can also be set up and moved using the optionally available external keypad. This handy keypad is directly connected to the axis controller using a 3-5 m long cable, and can easily be taken to the machine.

Step counter display

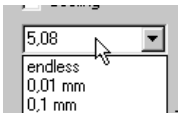
The absolute step counters for each axis are displayed in the top part of the window. Their values relate to the reference point on the reference switches. Below them are located the relative step counters which relate to the zero point of the working piece.



WinPC-NC distinguishes between two systems of coordinates. Firstly the machine coordinates with their origin on the reference switches and which are referred to as the reference point. Then there are the workpiece coordinates with the workpiece zero point, which is usually located in the bottom left-hand corner of the data area.



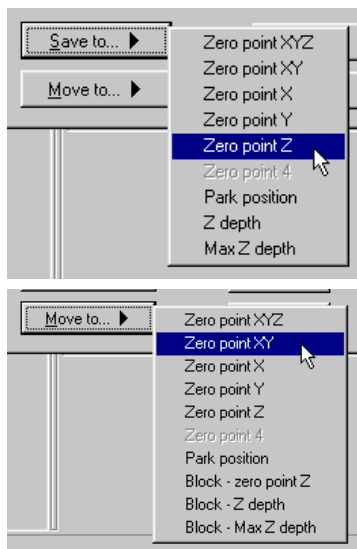
WinPC-NC can move the axes endlessly, i.e. movement continues for as long as a key remains pressed. The axis brakes when the key is released and comes to a stop without any step losses. The second possibility involves moving discrete distances. Distances between 0.01 mm and 100 mm are possible, as well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



In addition to the predefined distances, it is possible to enter any distance in the text box.

Switching additional signals

Clicking the check boxes with the mouse is all that is required to switch both additional signals, Drilling spindle and Coolant pump. This switches the signals on or off. Both signals are switched off when you exit the Jog function.



During manual movement, it is possible to move to and store certain help points. After a position has been reached, it is easy to click the Save button to select required point and save it permanently as a parameter.

Moving to stored help points is just as straightforward. All that is required is to click the Move to button and select the required help point. Then the machine moves to it.

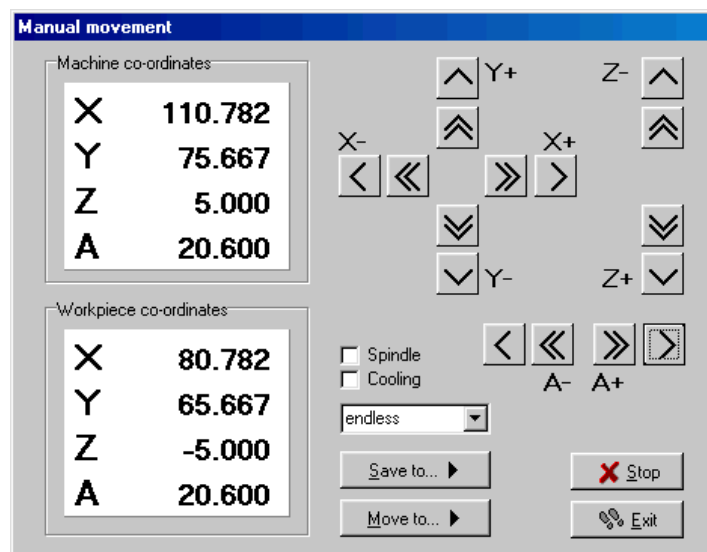
Movement in progress can be interrupted at any time by clicking the Stop button. The JOG function can be exited by clicking the Exit button.

Measuring Z-axis heights automatically

WinPC-NC can automatically determine the various Z-axis heights using a surface probe. The probe should be connected to an input on the axis controller using a cable to allow the probe to move freely. Depending on which Z-axis height you are measuring, place the probe against the clamped workpiece or the table surface to measure the maximum Z-axis depth.

The measuring sequence involves several steps :

1. Move the machine over the point to be measured
2. Bring the measuring probe or surface block into contact
3. Start the measurement. *WinPC-NC* moves the Z-axis downwards slowly until the probe contact trips. Then the axis stops and *WinPC-NC* transfers the measured value and the probe width as parameters. The probe width can be defined as a parameter.



Jogging with the 4th axis

Setting up the 4th axis

If a 4th axis is available, the appearance of the jog dialog box is somewhat different. In this case, it also includes buttons for moving this axis slowly and quickly, as well as boxes for displaying the axis positions.

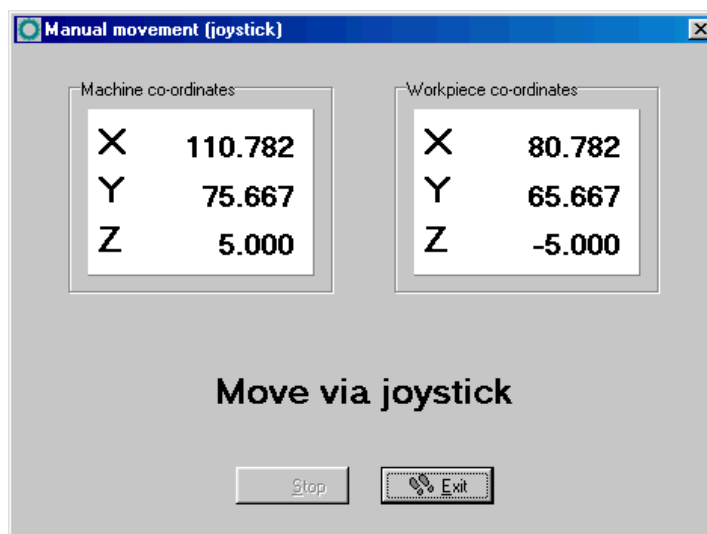
MOVE-JOYSTICK JOGGING



As an alternative to the mouse and keyboard, it is also possible to move the machine manually using a connected joystick. This has the advantage that you can observe the movements directly on the machine and set exact positions. It is much easier to take a joystick to the machine than it would be to move the keyboard and mouse up to the machine.



A connected joystick must be configured in the Windows system prior to use and must be calibrated with a special function.



Jogging with joystick

Jogging with a joystick

When setting up the machine with the joystick, movement in the X and Y-axis directions can be achieved simply by tilting the control handle. Moving the Z-axis requires pressing joystick button 1 as well.

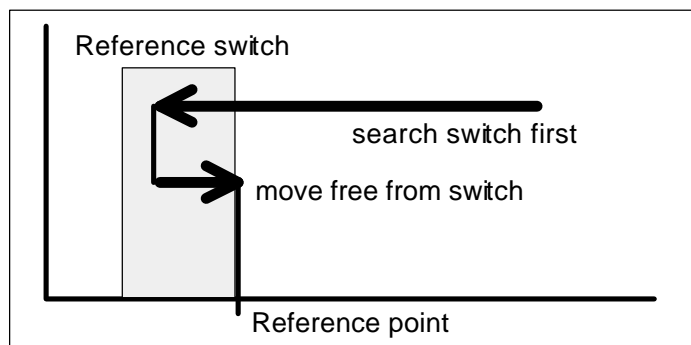
A help point can also be stored directly at the machine using the joystick. The stored position can be selected by repeatedly pressing joystick button 2. To store the position itself, press button 2 at the same time as holding button 1 pressed.

MOVE-REFERENCE

The REFERENCE or homing function moves all axes to the reference switches in a defined sequence.

Reference point of the machine

Every axis starts moving at reference speed 1 and searches for the reference switch. Movement stops when the switch changes its level. Then movement starts at speed 2 in the opposite direction back away from the switch. The edge of the reference switch defines the reference point for this axis.



Sequence of reference movement to the reference switch

WinPC-NC needs to know the current positions of all axes. As a result, reference movement is essential before a working process.



Reference movement is also triggered by pressing the (F8) key, or using the reference button.

MOVE-SELECT TOOL...

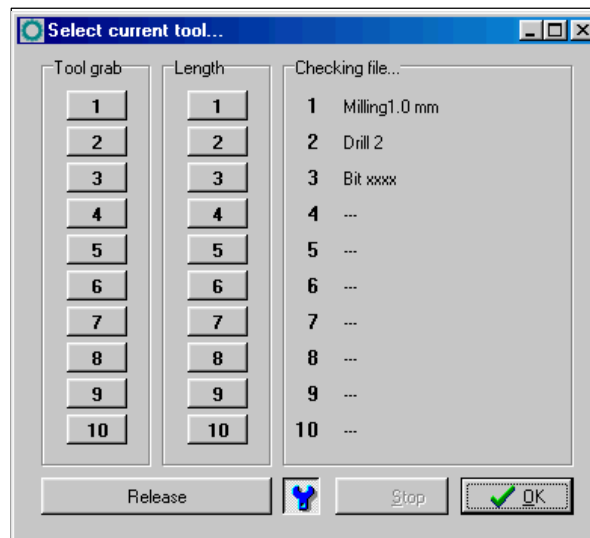
There is a special dialog box for controlling an automatic tool changer or if tool length measurement is activated. It can be activated using the menu function or the tool list button.



The dialog box contains the following possible functions :

- Place an inserted tool in the magazine
- Pick up a new tool
- Measure the length of a tool
- Identify the tool which has just been inserted

As a rule, *WinPC-NC* always remembers which tool was used last and the status of the collet chuck. As a result, this information is retained over more than one session.



Dialog box for selecting, picking up, putting down and measuring tools

*Rapid operation
with buttons*

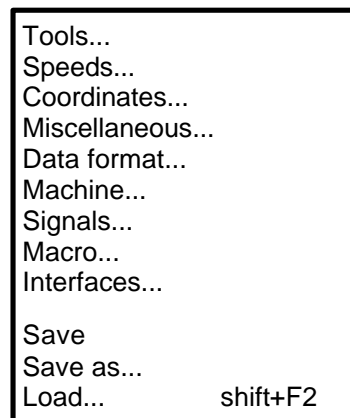
- Release takes the currently clamped tool and puts it down in the magazine
- The collet chuck button closes or opens the collet chuck. There is a security prompt prior to opening the chuck.
- Stop cancels a movement, e.g. to the changer or a measuring movement
- OK closes the dialog box



To inform *WinPC-NC* which tool is currently inserted, all you have to do is click the tool number or name on the right. This may be necessary after the initial startup or following a cancel.

3.3.3. PARAMETERS menu

The PARAMETERS menu contains all the setting options for *WinPC-NC*. It is made up of several dialog boxes which contain the parameters grouped by function.



The shortcut key for activating the PARAMETERS menu is (Alt-P). Clicking the parameters button has the same effect. The individual parameters are explained in detail in a subsequent chapter.

PARAMETERS-SAVE

Saving parameters for a particular project

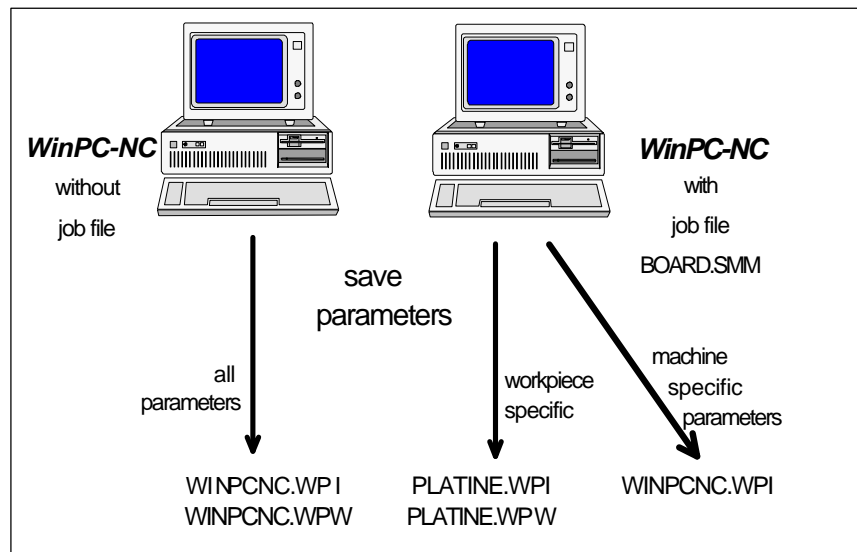
All parameter and tool settings can be stored in files using the SAVE menu item. It is possible to save the settings for a working file or specifically for a project.

The save function operates as follows : When a working file is loaded, **WinPC-NC** stores all workpiece-related settings such as the zero point, scaling, etc. in a parameter and tool file associated with the working file. The files have the same name as the working file, with the endings *.WPI and *.WPW, e.g. SHEET.WPI or ENGRAVE.WPW.

Whenever these working files are loaded in the future, all the settings and tools are once again available without having to be redefined.

The machine-related parameters such as axis resolution, reference directions or the interfaces used are always stored in the standard parameter file WINPCNC.WPI.

If no working file is selected when you save, the save function stores all settings in the default files WINPCNC.WPI and WINPCNC.WPW.



Saving parameters and tool settings

Subdividing between two parameter files offers the advantage that all machine parameters only have to be saved once, and they only have to be saved in this file whenever changes are made.

PARAMETERS-SAVE AS...

The second save function allows you to enter any file name for the project parameters.

PARAMETERS-LOAD...

Loading specific parameters

The LOAD function enables you to load various specific parameter records, e.g. for different materials or working processes.

A window appears with the familiar open dialog box functions. It is possible to select parameter or tool files. The PARAMETERS-LOAD function can also be activated with (Shift-F2).

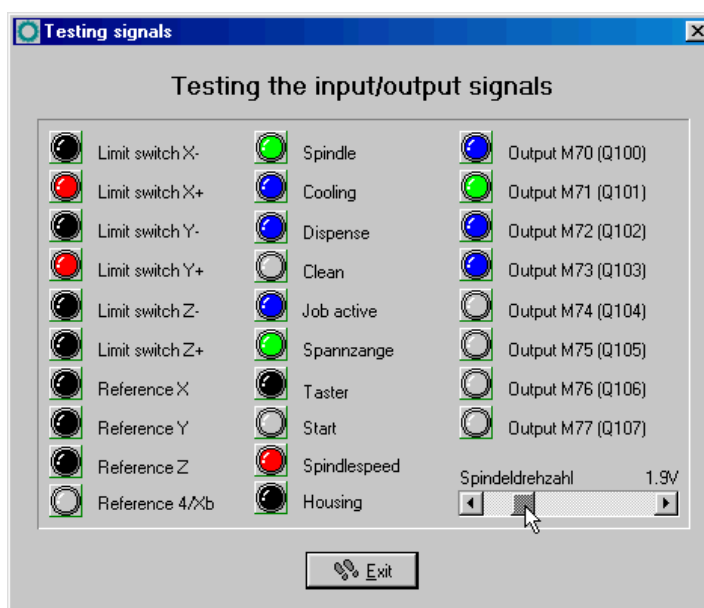
3.4. SPECIAL FUNCTIONS menu

The SPECIAL FUNCTIONS menu gives you access to test functions which you can use to systematically check the mechanism and to ascertain what are the correct or optimum parameter settings. Also, a joystick calibration function and a position check function are integrated here.

- Signal test
- Motor test
- Controller info
- Joystick calibration
- Service code
- Check positions

SPECIAL FUNCTIONS-SIGNAL TEST

This test function can be used for checking the axis inputs of the controller, i.e. the limit and reference switches, as well as the outputs.



Signal test

Interactive testing of all inputs/outputs

WinPC-NC continuously interrogates the limit switches and reference switches on all supported axes and displays their status. Grayed out switches are not defined in the **CNC-Control** axis controller.

Black indicates the switch is not active, while red displays that the switch is activated.



The switches must be defined and the switching logic set during installation using the *CNC-Configurator* utility. Alternatively, it is possible to use the default settings.

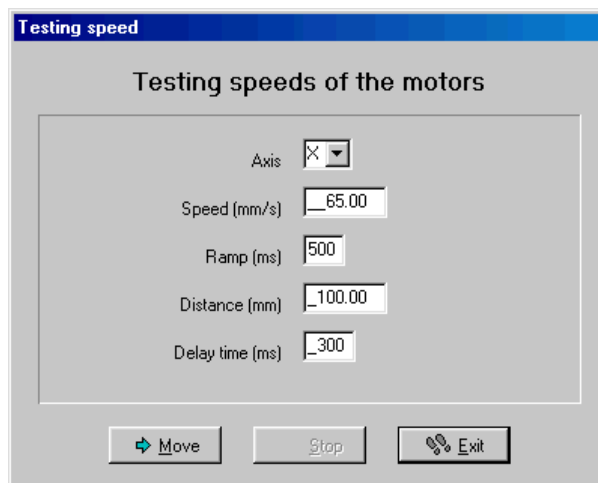
The supported additional signals are tested by simply clicking the switch symbols. This switches the outputs on or off.

Testing the spindle speed

There is a slider in the bottom right-hand corner of the window. This sets the analog output for checking the spindle speed. It can be infinitely varied between 0 V and 10 V.

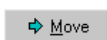
SPECIAL FUNCTIONS-MOTOR TEST

The MOTOR TEST special function is used for ascertaining the optimum speed settings. A window displays all parameters relevant to the step calculation.



Motor test function

The required values can be entered in the parameter boxes, after which a test run can be performed immediately in order to check all parameters.



Clicking the Move button causes *WinPC-NC* to move the selected axis continuously forwards and backwards. By listening to and observing the movement, it is easy to tell whether the parameters are correct for the axis, or whether additional corrections are needed to the speed or ramp length. The test run is cancelled by pressing the (ESC) key or clicking the Stop button.

Optimum parameters

The optimum values for an axis have been achieved if the motor starts up quickly without step losses, and is still able to develop sufficient torque at maximum speed.

Step-by-step setting

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

1. Switch off the ramp length and slowly increase the start/stop speed until the motor stalls. Then reduce the speed value by 30 - 40 %.

2. Test the ramp length with various values. You have achieved a good value if the motor starts up quickly without stalling.
3. Increase the rapid speed in stages. The motor should run quickly while still developing sufficient torque.

Having ascertained the values, you can store them as parameters for the axis in question. All parameters and their functions are explained in a subsequent chapter.

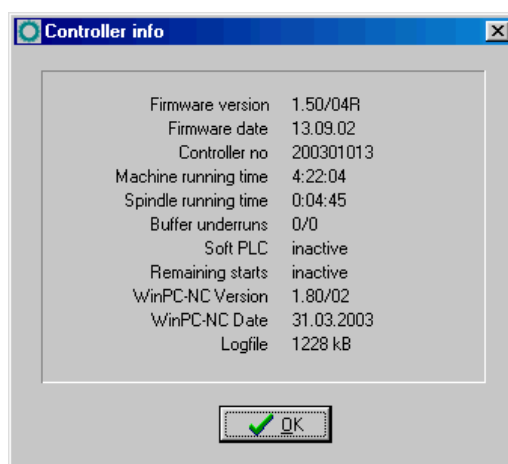


The optimum parameters for a stepper motor axis depend on many factors, e.g. the motor characteristic, the type of drive used (spindle or belt) and the load to be moved.

SPECIAL FUNCTIONS-CONTROLLER INFO

Information about the axis controller

WinPC-NC only works in conjunction with the **CNC-Control** axis controller only. The special CONTROLLER functions reads out and displays the version of the controller.



Display of the controller version

Whenever you have any queries for the manufacturer, always provide the version number of the axis controller as well as the version of **WinPC-NC**.

SPECIAL FUNCTIONS-JOYSTICK CALIBRATION

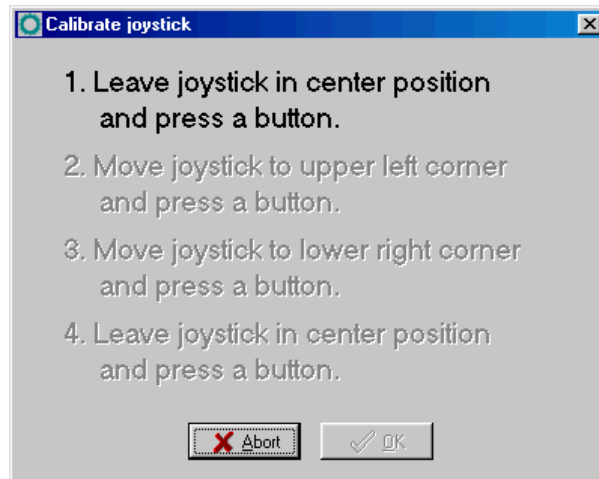
To be able to move the machine with a joystick, **WinPC-NC** needs to know the type and value ranges of the joystick signals. The JOYSTICK CALIBRATION function does this almost automatically.

Determining joystick parameters via menu prompts

Instructions for operating the joystick are displayed in a window. Please follow these instructions exactly. The procedure involves pressing both buttons and moving the lever to certain positions.



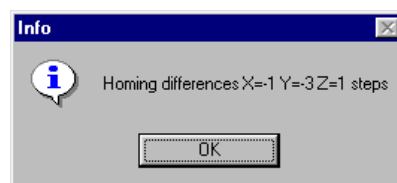
The joystick cannot be used before it has been calibrated.



Joystick calibration

SPECIAL FUNCTIONS-CHECK POSITION

The CHECK POSITION function is another test function in *WinPC-NC*. It can be used for checking the accuracy of the reference switches.



Result of the position check

Check reference position

Relatively small step discrepancies may be due to the switches. Large discrepancies, on the other hand, indicate previous step losses.

It is a good idea to use the CHECK POSITION function if :

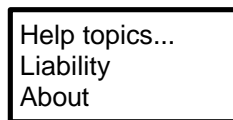
- You presume steps have been lost due to a collision
- You want to determine the maximum machining speed for a tool or a material, which means you want to demonstrate whether or not steps have been lost
- The position was changed during a tool change



It is only possible to check the reference position if reference movement was performed previously and there was no cancel due to a limit switch or a stop signal.

3.4.1. HELP menu

There are three items in the HELP menu.



HELP TOPICS

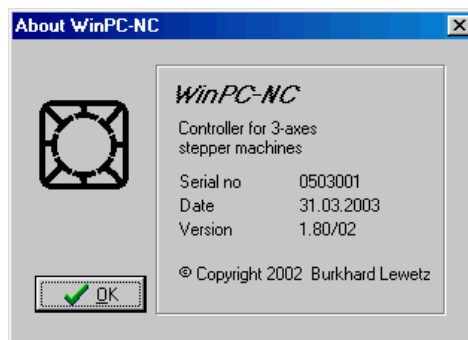
The HELP TOPICS function gives you access to the help system. A dialog box appears containing the main selection.

HELP-LIABILITY

The HELP-LIABILITY function displays a text with the license conditions and a disclaimer. Please read this information carefully before using *WinPC-NC*.

HELP-ABOUT WinPC-NC

Activating this menu function displays information about the current version and revision number.



Information about the current version

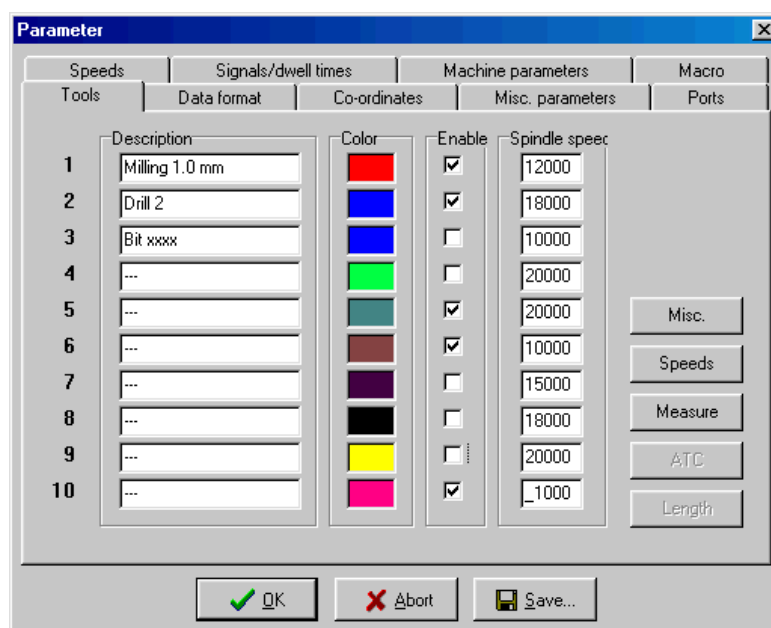
4. Parameter settings

4.1. Tool management

Project-related tool management

WinPC-NC has a user-friendly tool management function. It is possible to define up to 10 tools with specific values for each working project.

The parameters are divided between several dialog boxes. **WinPC-NC** stores all the values defined in these dialog boxes into the tool file using the PARAMETERS-SAVE menu function. Tool files have the *.WPW filename extension.



Tool management (window 1)

Designation

Meaningful tool names

Each tool is identified with a name which is used during tool change prompts.

Color

Colors in the graphical display

The Color parameter for each tool is used in the graphical display of the data. As a result, it is easy to adapt the color display to suit your own requirements or to harmonize with the CAD program you are using.

Clicking the colored box opens a dialog box in which you can select a new color. All colors supported by the current Windows display settings can be selected.

Activation

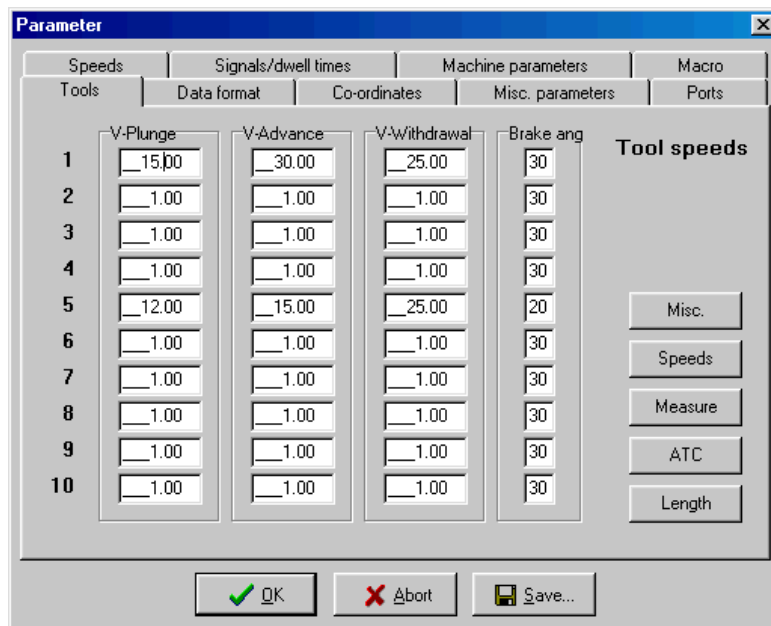
Activating tools individually

Each tool can be individually enabled or blocked. Inactive tools are simply ignored in the graphical display and the commands for them are skipped.

Spindle speed

It is possible to assign a spindle speed to each tool. This is set using a defined analog output when the tool is used.

Tool speeds



Tool management (window 2)

Plunge speed

The plunge speed specifies the speed with which each tool is pushed into the workpiece. It is necessary to consider certain limit values here, depending on the material and the tool.

Feed speed

The feed speed defines the working speed for each tool when the tool is pushed into the workpiece.

This value is irrelevant for straightforward drilling applications. However, if *WinPC-NC* is used for routing, engraving or grinding, then the maximum feed speed depends on the tool used and the material.

Retraction speed

The retraction speed is used for raising or withdrawing the tool from the workpiece.

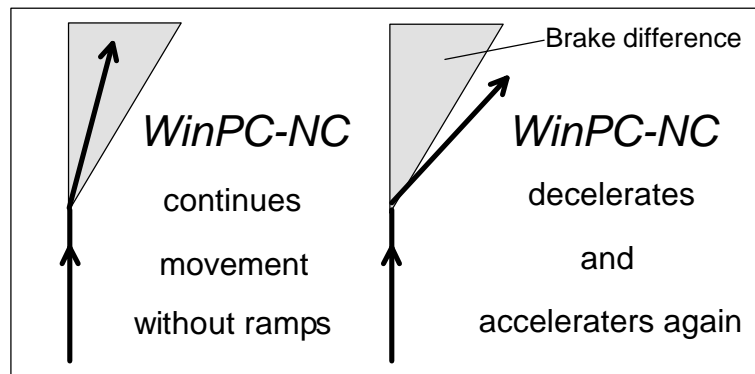
Brake difference

The brake difference specifies the maximum angle differential for subsequent movement stages in which movement takes place at full speed. The value is entered in degrees.

Deterministic speed optimization

At the start and finish of movements, the acceleration and braking function is only activated if the direction of the next movement stage deviates from the previous one by more than an angle value which can be set.

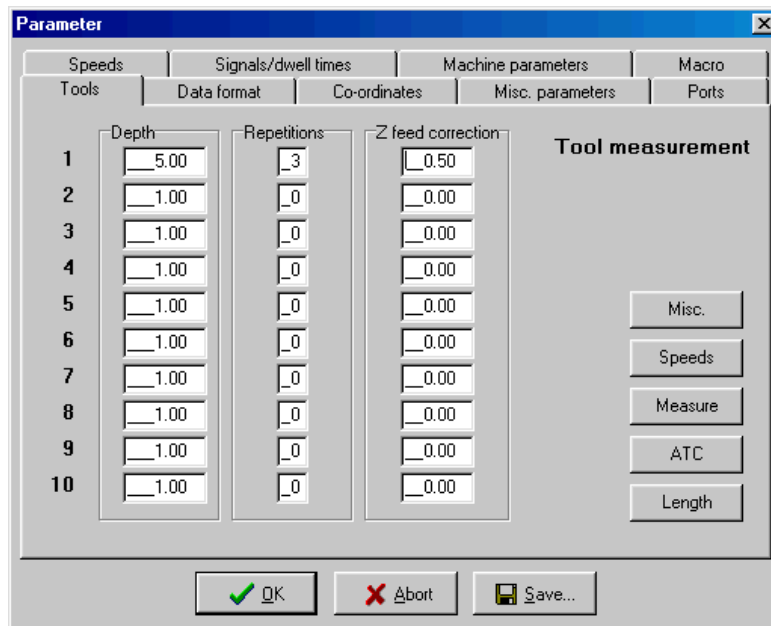
An example of this deterministic speed optimization function can be seen in movement around a circle, which consists of lots of little individual movements. The directions of two successive movement states are only slightly different. As a result, the circular path can be moved in one operation at maximum speed.



Brake difference

Braking and acceleration always take place before and after tool movements. This parameter is irrelevant in straightforward drilling applications and is ignored.

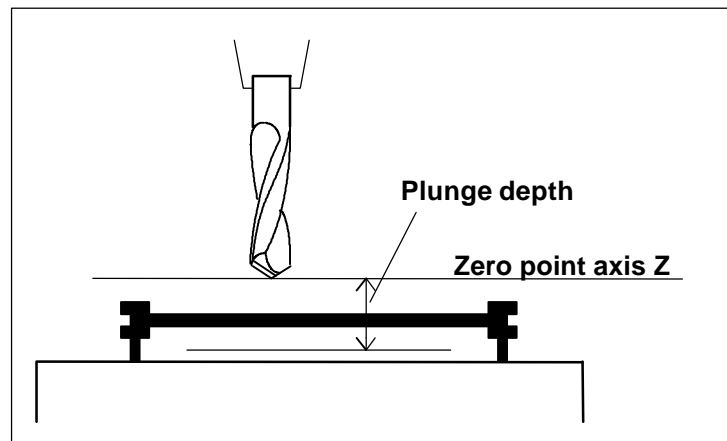
Tool dimensions



Tool management (window 3)

Plunge depth

The plunge depth specifies the distance by which the Z-axis of each tool is moved downwards into the workpiece. The depth is defined in millimeters and is always measured starting from the plane of the zero point.



Plunge depth measured from the zero point of the Z-axis

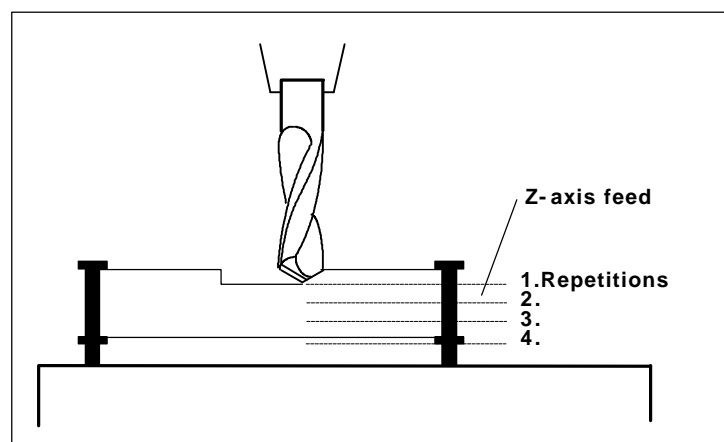
Repetitions

Routing in stages Frequently, several passes are required when routing thick or hard materials. The parameters Repetitions and Z-axis feed mean you do not have to restart a working process several times in succession using different plunge depths.

The repetition always applies to a complete contour line or for a hole, i.e. **WinPC-NC** remembers every insertion point and returns to it after the tool is next withdrawn, in order to start the next pass.

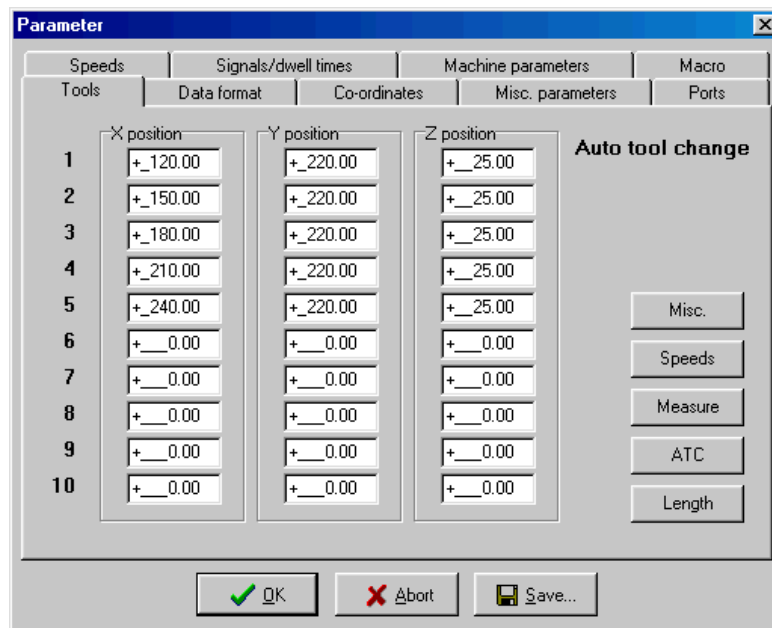
Z-axis feed

During a series of repetitions, the Z-axis feed causes the Z-axis to be moved downwards by the specified value.



Repeats and feed correction

Automatic tool changer (ATC)



Tool management (window 4)

X/Y/Z-axis positions of the tool magazine

Controlling the automatic changer

WinPC-NC can control an automatic tool changer with 10 magazine positions. The exact positions of the magazine locations are defined in this parameter window.

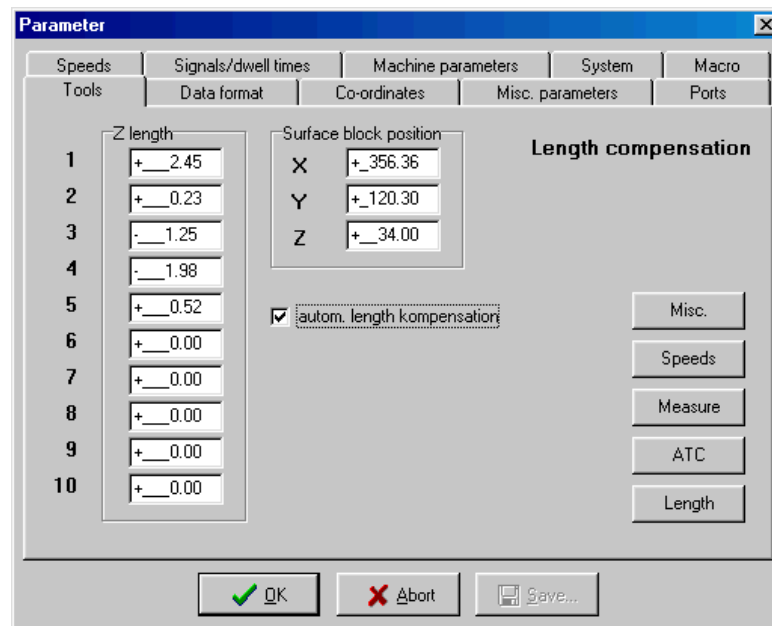
An electrically or pneumatically operated collet chuck is required in order for tools to be changed automatically. This chuck must be switched using a defined output. Macros can be used to define the exact sequence of putting down and picking up as well as movements, waiting times and switching outputs.

The procedure is described in a subsequent chapter.



Parameters for the tool changer are only available if the changer is enabled in the axis controller.

Tool length compensation



Tool management (window 5)

*Compensating
for different
tool lengths*

When you are working with several tools, it is extremely rare that all the tools will be the same length. As a result, determining and maintaining an exact plunge depth is extremely troublesome.

WinPC-NC can automatically measure the different lengths of tools and compensate for differences in length during a working process. This is a very useful function, especially if you are using an automatic changer.



Parameters for tool length measurement are only available if the function is enabled in the axis controller.

Z-axis length

The lengths of the tools used are defined here. Normally, the boxes cannot be edited but instead the values are entered automatically when the tools are measured. The length difference used by the compensation function is calculated on the basis of these parameter values.

Sensor position

*Automatic
length
measurement*

The length sensor must be a switch or a button which the tool can be moved towards from above and is switched when contact is made or pressure is encountered.

These parameters define a position above this sensor in absolute machine coordinates. When measuring, **WinPC-NC** moves to this position and then moves the tool down slowly until the sensor is switched. The distance moved is adopted in the parameters as the tool length.

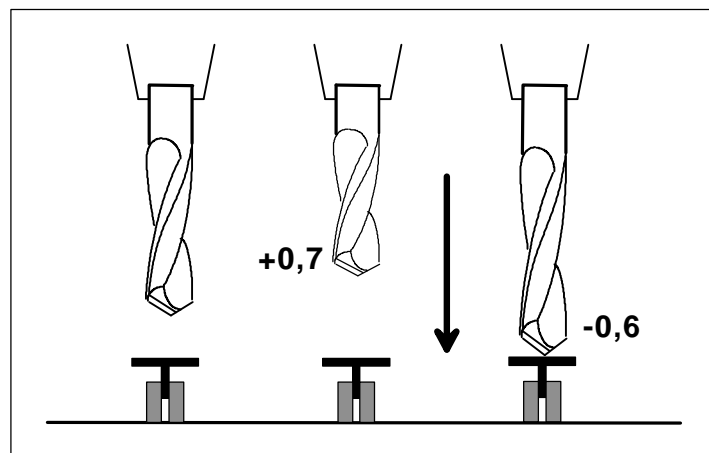


Diagram showing the basic principle of the length sensor and automatic tool measurement

Automatic length correction

This parameter activates automatic tool length compensation. Having a separate parameter to enable this function makes it possible to restrict compensation to projects in which it is really needed.



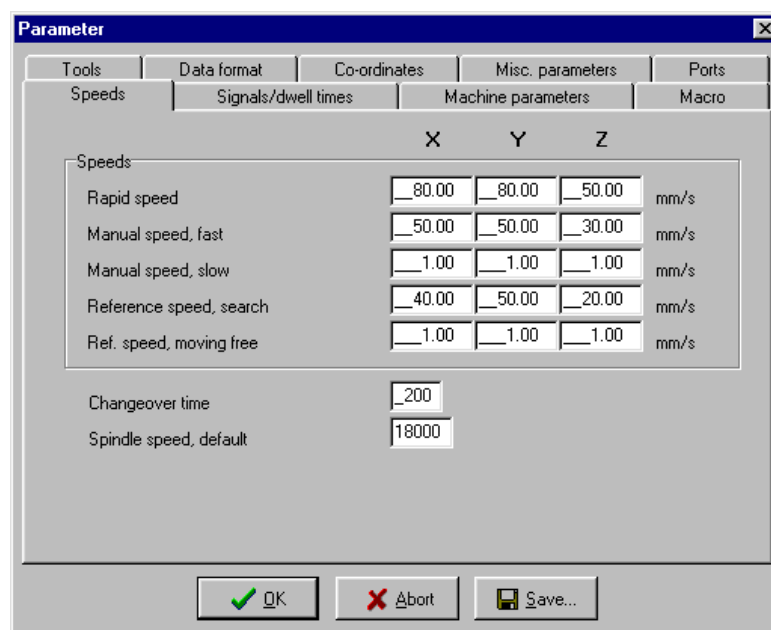
Tools are not automatically measured after being changed. Instead, this measurement must be explicitly selected using the MEASURE TOOL function before each tool is used.

4.2. Speeds

All the parameters which control the speeds of the motors are grouped together in the Speeds dialog box.

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

All parameters must be set separately for X, Y and Z. This means the correct speed can always be calculated and used, even when different axis resolutions are involved.



Speed parameters

Rapid speed

Speed with the tool raised

Rapid speed is used for moving to a new position with the tool lifted out of the workpiece.

These are unproductive movements which *WinPC-NC* always moves at the fastest possible speed.

Jog speeds

These parameters specify the speeds during jog mode. Two speeds can be defined.

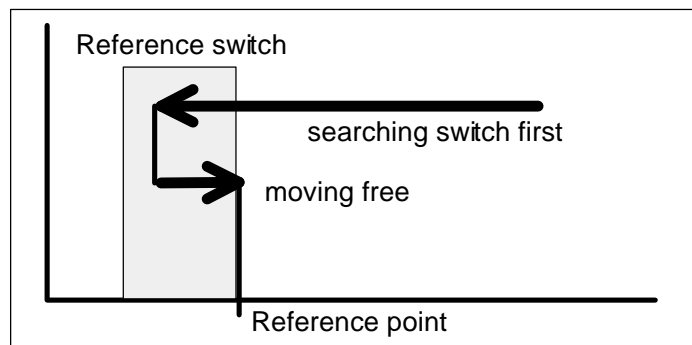
*Individual steps
or continuous
movement*

The motor changes over to continuous movement if the key or the mouse button is pressed for a longer time. It moves at the speeds defined here in this case.

It makes sense to use fast speed to traverse long distances, while exact positioning is facilitated with slow speed or single steps.

Reference speeds 1/2

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



Reference movement of an axis

*Reference point
of the machine*

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



WinPC-NC distinguishes between machine coordinates (with the reference position as their origin) and workpiece coordinates with the workpiece zero point.

Changeover time

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

WinPC-NC outputs individual steps to the motors every time one of the cursor keys is pressed briefly or the mouse button is used to click one of the movement buttons once. If a key or a button is pressed for

longer (precisely for this changeover time) the motor changes over to continuous movement. The movement continues until the key is released.

Standard spindle speed

The standard spindle speed is always used unless another setting is made specifically in the NC program or in the tool parameters. Usually, this applies from switch-on during jog movement or at the start of a working process.

4.3. Coordinates

In the Coordinates dialog box, it is possible to define all parameters relating to dimensions, help points and the unit of measurement.

Co-ordinates			X	Y	Z	
Working piece from			+__0.00	+__0.00	+__0.00	mm
to			+2500.00	+1800.00	+__120.00	mm
Zero point			+__35.67	+__156.80	+__32.60	mm
Park position			+__0.00	+__0.00	+__0.00	mm
Scaling factors			__1.000	__1.000	__1.000	
Tool lift					+__5.00	mm

Unit of measurement in	1/40 mm	
Units	mm + mm/min	
Surface block	+__5.67	mm

Circular axis	
Y axis is circular	<input type="checkbox"/>
Diameter	+__20.00

Setting the working area and the help positions

Machine and workpiece coordinates

It is necessary to distinguish between two types of position definitions. There are absolute machine coordinates with their zero point at the machine reference point. Then there are workpiece coordinates with their origin at the workpiece zero point.

Working area

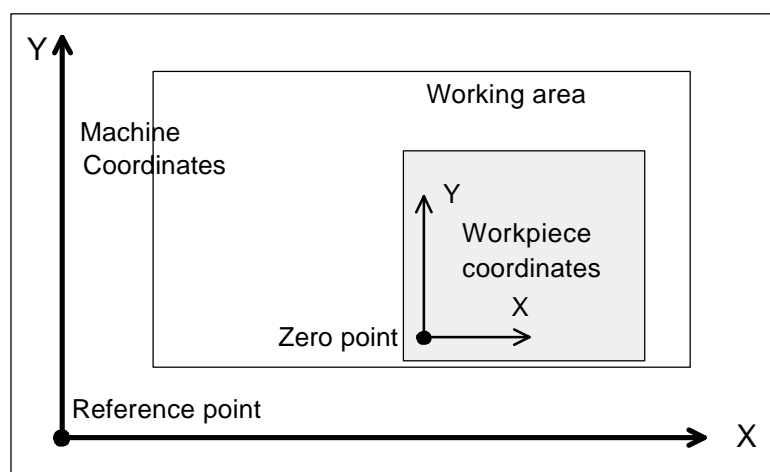
Software area monitoring The working area defines the section of the maximum movement area within which the workpiece is machined. **WinPC-NC** monitors it for the X and Y-axes during the working process. This means the motors are stopped and the process is cancelled if the limits are violated.

Maximum plunge depth for Z-axis The working area for the Z-axis determines the maximum plunge depth to which tools can move without damaging the bed of the machine.

The unit is millimeters and the distances are measured from the machine reference point (machine coordinates).

Zero point

The zero point refers to the reference point of the coordinates in the working file. It is the position with a specific X and Y-axis coordinate within the NC file, e.g. the left-hand bottom corner. All processing distances are measured from here.



Working area and help positions

Zero point as origin of the workpiece coordinates

The zero point can also be defined manually. To do this, it is necessary to move to the required position using jog movement and then save this as the new zero point. The positions of the axes can also be saved individually.

The unit is millimeters and the lengths represent the absolute distance from the zero point to the machine reference point (machine coordinates).

Park position

Defined position for breaks

It is necessary to define a park position if the machine slide is to be moved out of the working area at certain times. This may be necessary for a tool change, for example, or for clamping the workpiece.

It is also possible to make this definition during jog movement by moving into position with the keyboard.

The parked position can be moved to in jog movement or automatically during a tool change and at the end of a working process.

The unit is millimeters and the distances are measured from the machine reference point (machine coordinates).

Scaling factors

It is possible to compensate for calibration differences using the scaling factors. If both the axis resolution and the unit of measurement are set correctly but the machine does not move to the exact length nevertheless, this problem can be corrected using the scaling factors.

The values must be specified to 3 decimal places and are used for multiplying the coordinate values to which the machine is to move.

Safety clearance or toollift

Additional clearance for avoiding collisions

It is possible to define an additional height above the zero point level of the Z-axis as a safety clearance. The tool is lifted above the zero point by this distance during every working process.

On the next plunge movement, **WinPC-NC** first covers the safety clearance down to the zero point at high speed, before pressing into the material with the defined plunge speed.

Unit of measurement

Wide range of predefined units of measurement

The unit of measurement for working data must be defined using this parameter. All coordinate values in the working file are related to a particular dimension.

The possible units are millimeters and inches. HPGL files are usually in units of 1/40 mm or 1 mil, while drilling data are usually in 1/100 mm or also 1 mil.

Units in the program

The UNITS parameter defines the units for dimensions and speeds used in the graphical display and the parameters.

It is possible to select between three options :

Millimeters and millimeters/second	(mm und mm/s)
Millimeters and millimeters/minute	(mm und mm/min)
Inches and inches/minute	(inch und inch/min)

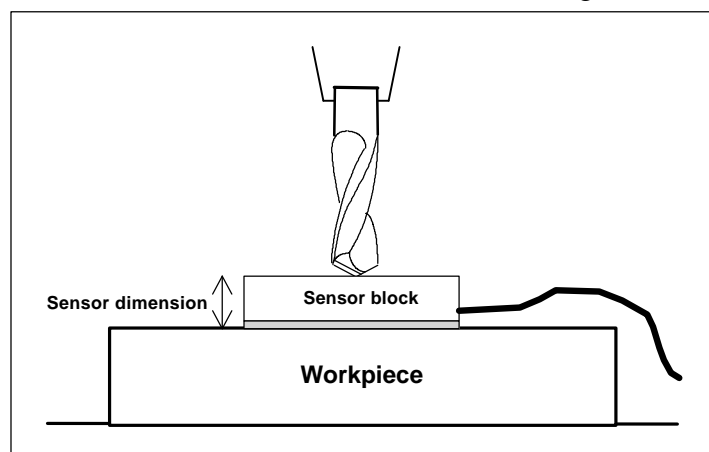
Sensor dimension

*Measuring
Z-axis heights
automatically*

WinPC-NC can automatically measure different Z-axis heights using a sensor block (surface block). The sensor block outputs a signal when it is contacted from above, and the signal is processed as an input to the axis controller.

The measuring procedure involves several steps :

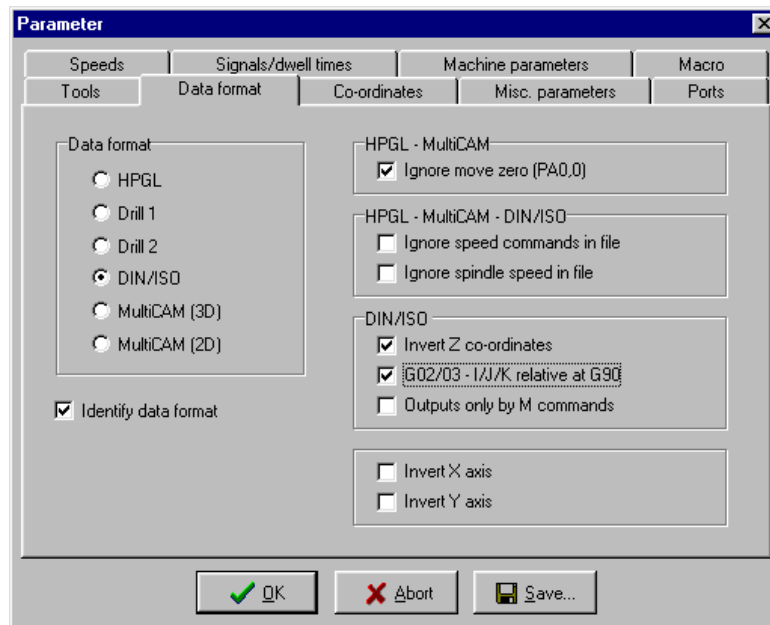
1. Place the sensor block on the workpiece surface for the Z-axis zero point or on the machine bed for the maximum Z-axis depth.
2. Move the tool over the sensor block in jog mode.
3. Start the measurement using the MOVE-JOG menu function.
4. **WinPC-NC** slowly moves the tool down to the sensor and stops when it makes contact. The position is output and added to the defined sensor dimension, the result being stored as a parameter.



Automatic measurement of the Z-axis zero point

4.4. Data format and associated parameters

WinPC-NC can read and process various data formats. The format of the NC file can be selected manually or registered using an automatic recognition function.



Data formats and associated parameters

Data format

WinPC-NC understands various NC data formats, and is able to import these files, display them and perform the movements on the machine. At present, the commonly used plotter format HPGL, MultiCAM, two drilling formats and DIN/ISO or G-codes are possible.

HPGL is from the well known plotters made by Hewlett Packard and is supported by almost every CAD or drawing program. MultiCAM (also referred to as Extended HPGL format) is very common in the USA and offers special 3D expansions for HPGL commands.

Plotting and drilling formats

The drilling formats are based on the following standards: Sieb&Maier1000, Sieb&Maier3000 and Excellon. In other words, it should be possible to process working files in these formats without any problems.

As a rule, machine tools are programmed using the G-codes of the DIN/ISO format.

The appendix contains a list and description of commands in the individual formats. In the event of problems during processing, for example if unknown commands or commands which cannot be interpreted are encountered, then the working file can be analyzed using this description and revised with the editor.



WinPC-NC usually detects the format of the selected file automatically.

Ignore movement to zero (PA0,0)

It is a feature of HPGL and Multicam files that there is usually a PA0,0 command at the end of the program to move to the coordinate zero point. This can be suppressed using this parameter if it is not required.

Ignore feed rate and spindle speeds

HPGL, MultiCAM and DIN formats also contain commands to set the spindle speed and the feed speed. Activate these checkboxes if you want to use the values set in the parameters rather than the commands in the NC file.

Invert Z-axis

Many CAD programs generate negative coordinates for plunge movements of the Z-axis. Activating this checkbox inverts the coordinates of the Z-axis so that these NC files can also be read in and processed correctly.

G02/G03 - I/J/K relative

There are various "dialects" of DIN or G-code programs. In some, the circle parameters I/J/K are specified as absolute values, while in others they are relative distances from the current position. This parameter enables you to distinguish between the types.

Outputs with M commands

In DIN/ISO programs, it is possible to switch almost all outputs such as the spindle, cooling, etc. using M commands. This parameter prevents *WinPC-NC* from operating a signal automatically and forces it to utilize exclusively the M commands which are used.

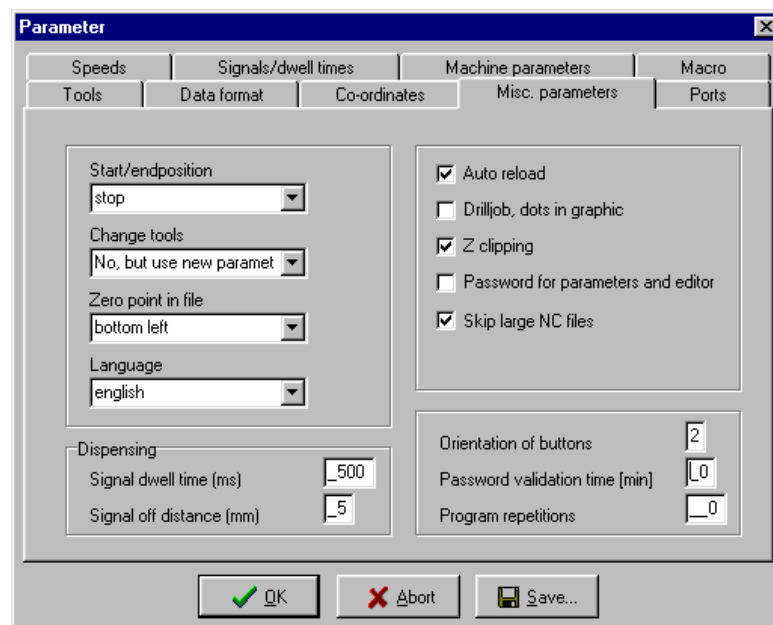
Otherwise, *WinPC-NC* would automatically switch on the spindle at the start of the process and switch the cooling on and off when lifting and lowering the tool.

Mirror axes

The X and Y-axes and their coordinates can be mirrored independently of one another for all formats. A changed parameter is immediately visible in the graphical display.

4.5. Miscellaneous parameters

The miscellaneous parameters include various switches and other settings which are not assigned to any other group.



Miscellaneous parameters

Start and end position

This switch specifies where the start and end point of each working process should be located. The machine also moves to the position after reference movement.

There are 3 possible start and end positions :

stop	<i>WinPC-NC</i> remains at the reference position after reference movement, and at the last coordinate after each working process
Zero point	The machine moves to the defined zero point after reference movement and after each working process
Park position	<i>WinPC-NC</i> moves to the defined parked position after reference movement and after each working process

It is a good idea to move to a certain position after the working process if space is required in order to change the workpiece.

Tool change

The parameter defines how *WinPC-NC* handles the tool change commands during a working process.

There are 5 possibilities for the tool change :

NO	No tool change is performed, the entire working process takes place with the current tool
YES	Performs the tool change and remains stopped in the current position for every tool change
YES to park position	Performs the tool change and moves to the defined parked position for every tool change
NO but new values	No tool change is performed, although the values for plunge depth, plunge speed and feed speed of the new tool are adopted

YES use ATC

Performs the tool change and uses the automatic changer for this purpose

Zero point in file

The workpiece zero point is the point in the NC file which has its position defined in the coordinate parameters. However, it can be located at various points inside or outside the workpiece and these points are defined here.

3 positions are possible :**Bottom left**

The zero point is at the smallest X and Y-axis coordinates in the file.

Coordinate origin

The zero point is at the coordinate origin, i.e. where the CAD program places it for the output. This setting is to be recommended if several files are being used on the same workpiece, e.g. routing and drilling a board.

Centered

The zero point is in the middle of the workpiece, i.e. exactly in the center of the coordinate dimensions in the X and Y-axis directions. This setting is useful for processing round workpieces, e.g. plates.

Language

WinPC-NC is multilingual. The standard version already includes a few languages, and additional languages can easily be bolted on if required. The available languages are listed in a menu.



The language changeover takes place as soon as you select a new language and click Save.

Automatic reload

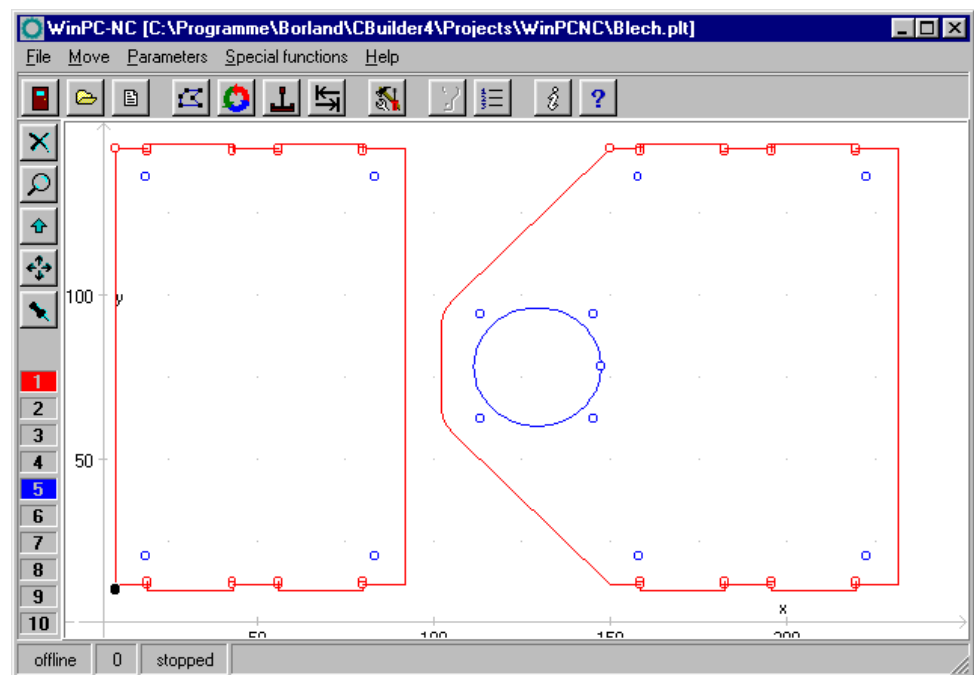
The reload function continuously monitors the date and time of the currently loaded file and reloads the newly modified file if there is a discrepancy.

Using this function, for example, it is possible to edit an NC file in the drawing program, make continuous changes to it and then, when you switch back to **WinPC-NC**, to be able to check all the changes on the screen straight away.

Drill job, dots in graphic

*Marking
insertion points*

The graphical display does not show holes drilled in drilling applications with HPGL, Multicam or DIN/ISO formats. Activating this parameter marks all insertion points with a small circle.



Display as drilling job with marks showing insertion points

Z-axis clipping at range limit

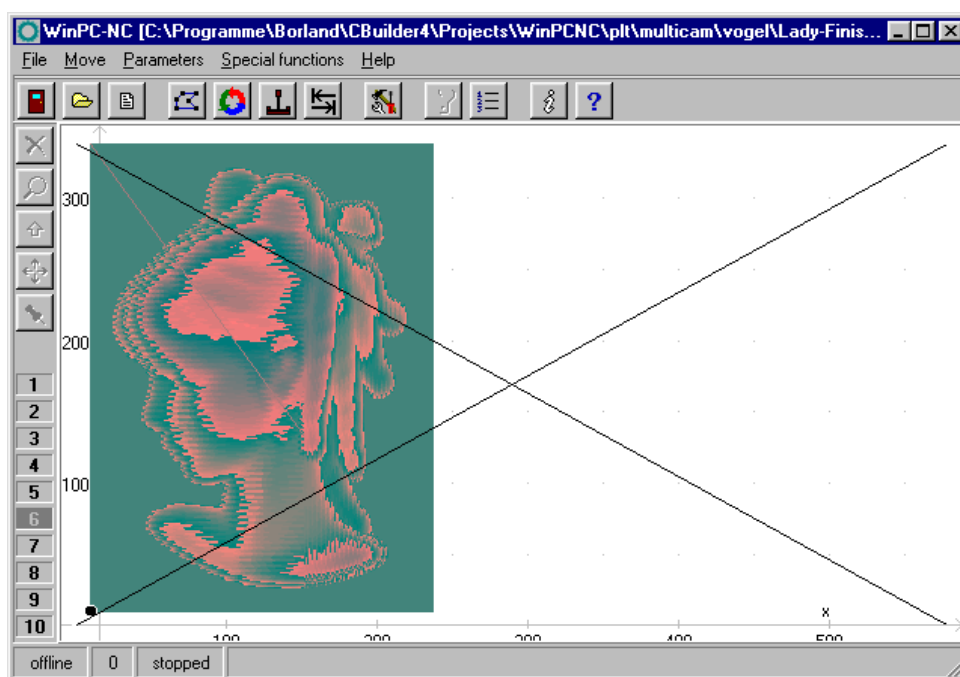
When the Z-axis clipping function is activated, **WinPC-NC** monitors the maximum Z-axis depth and cuts off all deeper movements at this limit.

Skip large NC files

Faster display

The analysis function and graphical display may take an extremely long time with very large NC files. This process can be speeded up by only reading in and displaying the first parts of the file. In such a case, a large cross is superimposed on the graphic display to indicate that not all of the file is visible.

The following functions are not available in this partial view: Zoom, shift and set position.

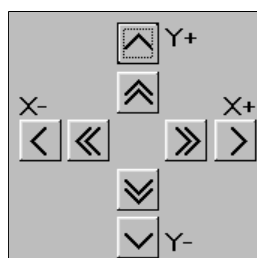


Partial display of a file with marking

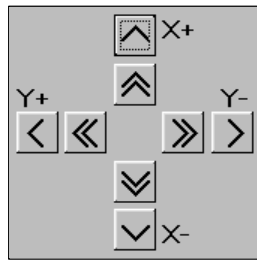
Button orientation

Adapting the movement direction of the buttons

It is possible to use 4 settings to rotate the X and Y-axis directions through 90 degrees at a time in order to adapt the orientation of the movement buttons in jog mode or on the keypad to take account of the machine orientation.



Button orientation position 0



Button orientation position 1

Positions 2 and 3 rotate the X and Y buttons through further 90 degree steps.

Program repetitions

The program repetitions parameter enables you to repeat a working process up to 999 times. This enables series production to be achieved in conjunction with a start signal in DIN/ISO program.

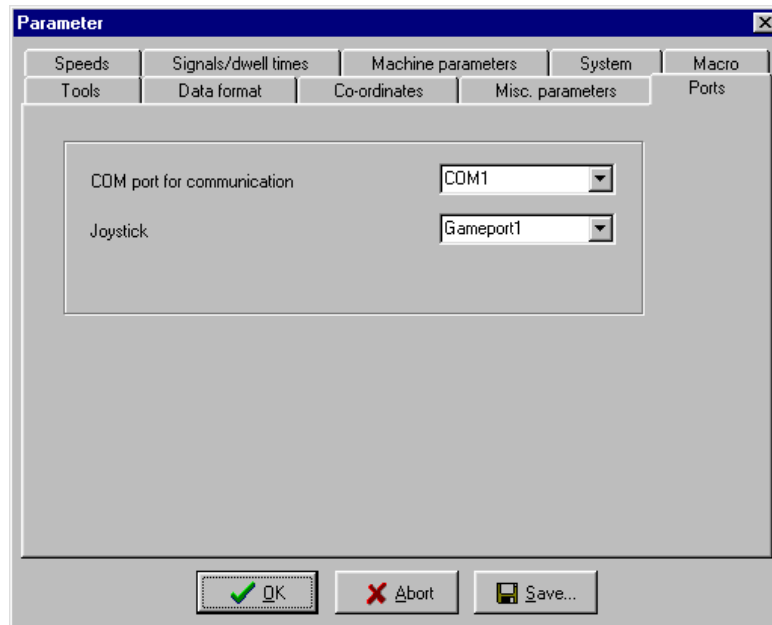
Dispensing parameter

Special dispensing function

WinPC-NC has a technology function for dispensing applications. It is only active if there is a dispensing output available in the axis controller.

The lead time defines a waiting time which is counted down after the metering signal is activated and before the movement starts. The run-on distance defines a distance from the end of the contour at which point the dispensing output is deactivated. This makes it possible to prevent the dispensing material from dripping where it is not required.

4.6. Ports



Port parameters

COM port for data transfer

This parameter defines the serial port to which our *CNC-Control* axis controller is connected using a null modem cable. *WinPC-NC* signals that the connection is offline in the status bar if the port is set incorrectly, the cable is incorrect or defective or the controller is inoperative (offline).

Joystick

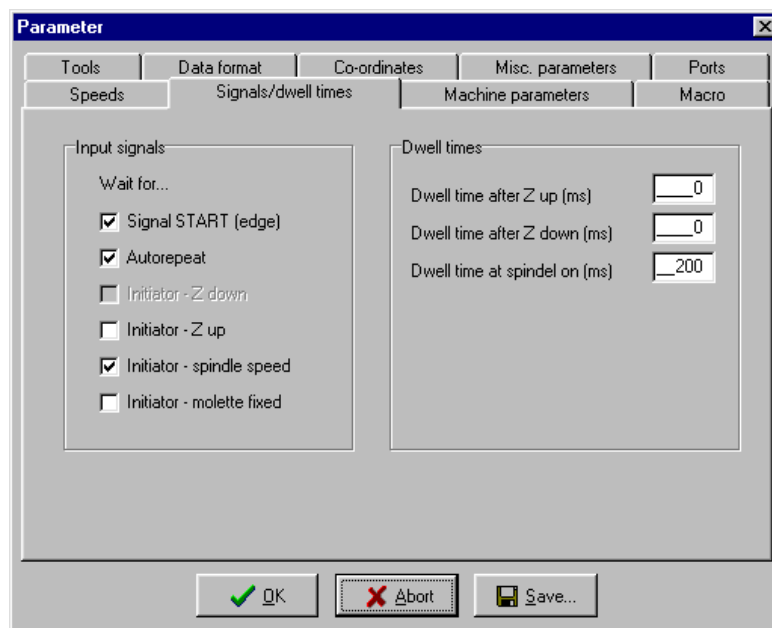
If you want to use a joystick to set up the machine, you must use this parameter to define the game port.



A connected joystick must be configured in the Windows system prior to use and must be calibrated with a special function.

4.7. Signals and dwell times

All input signals and associated parameters are grouped in a separate dialog box.



Signal parameters

Input signals

Synchronization with various sensor signals

WinPC-NC can synchronize the job sequence with various input signals. The option of activating these signals depends whether or not they are present and defined on the **CNC-Control** axis controller. Every time it starts up, **WinPC-NC** reads in the configuration of the controller and only enables the defined signals as parameters.

The various signals and what they mean :

- | | |
|------------------------|---|
| Start signal | Following activation of the working process, WinPC-NC always waits for a positive edge on this signal and does not start processing the NC data until the signal changes from LOW to HIGH. |
| Sensor Z bottom | WinPC-NC can wait for this signal after the Z-axis is lowered. The movements are not continued until the signal is active. This signal is particularly useful with pneumatic Z-axes. |
| Sensor Z top | WinPC-NC can wait for this signal after the Z-axis is raised. Movements to the next insertion point are not continued until the signal is active. |
| Spindle speed | It is possible to use this signal after the drilling spindle starts to indicate when the required speed has been |

reached. *WinPC-NC* waits until this signal goes to HIGH before continuing the process.

Clamping This signal can be used to indicate when the workpiece clamping is activated. The working process cannot be started until this signal is active.

Automatic repeat

Perform the process repeatedly When the start signal is active, it is possible to use this signal to make *WinPC-NC* wait for the next start signal as soon as it finishes a working process. This means no input from the keyboard or mouse is required.

Waiting time after Z up and Z down

In many applications, it is a good idea or indeed a requirement to wait a moment after the tool has been lowered or raised, before starting the machining. This may be the case when working with flexible materials or if you want the tool to freewheel.

The waiting time is defined in milliseconds.

Spindle acceleration time

Acceleration time for drilling spindle The start delay defines a waiting time in milliseconds which always counts down when the additional Drilling spindle signal is activated.

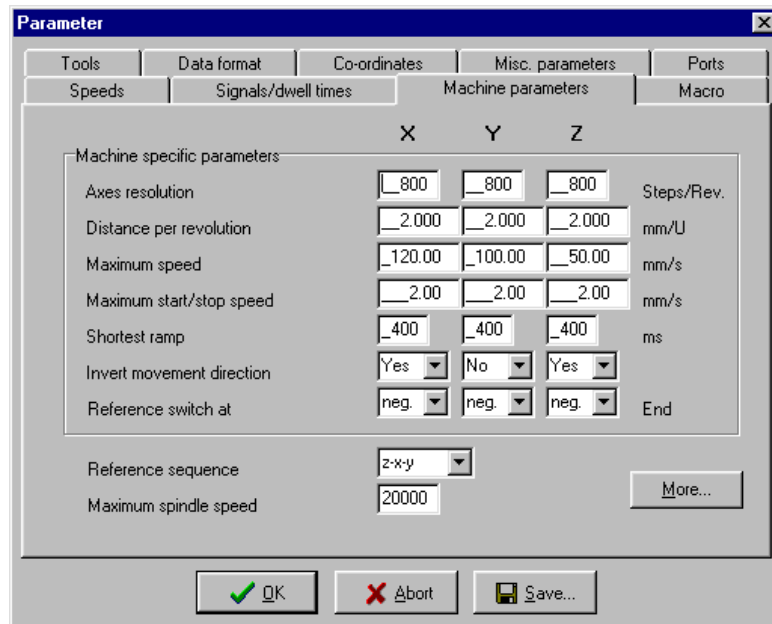
This ensures that the drilling spindle has enough time to accelerate before it is used for the first time.

4.8. Machine parameters

Machine-specific parameters include all settings which relate to the mechanical characteristics of the machine. Only some or even none of these parameters are present in certain OEM versions of *WinPC-NC*. Instead, the parameters will have already been set correctly in advance for the machine in question.



Always take the greatest possible care when setting or modifying any machine-specific parameters, in order to avoid machine defects.



Machine-specific parameters

Axis resolution

The resolution boxes define the number of motor steps per rotation. If you are using gear units or step down/step up ratios, you can enter the calculated values here directly.

The unit is steps.

Distance per rotation

You have to use this parameter to define what distance is moved during one motor rotation. Defining the axis resolution with two parameters offers the advantage that no calculation inaccuracies can arise.

The unit is millimeters with decimal places.

Maximum speed

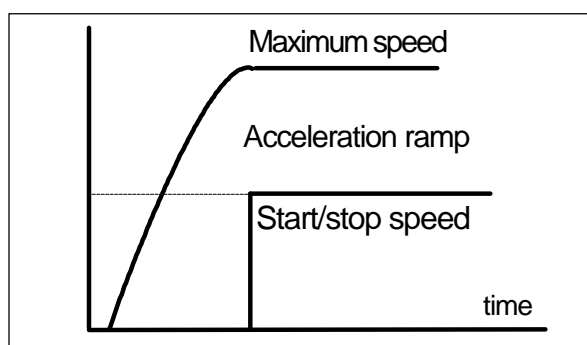
You should use the MOTOR TEST function to ascertain the maximum speed of each axis. This represents the absolute top limit speed with which the axis can be moved.

WinPC-NC monitors the speed entries made in all other parameter windows and always corrects them to the value defined here.

The unit is millimeter/second.

Maximum start/stop speed

The start/stop speed specifies the fastest possible speed with which the stepper motors can start up without ramps. The value is important for calculating path control, because braking at sharp edges or corners does not have to be to a standstill, but only down to this start/stop speed.



Start/stop speed and ramps

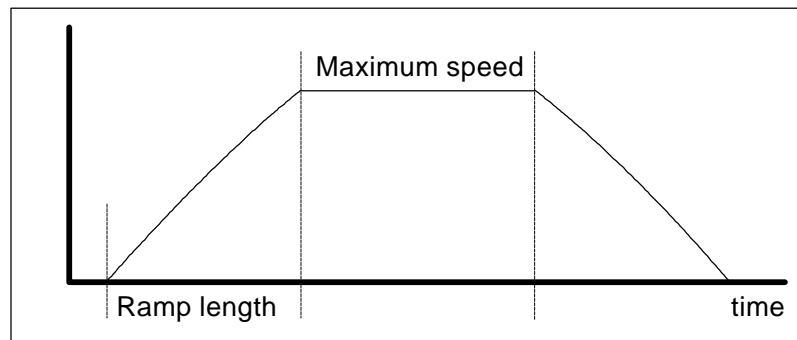
The optimum value must be ascertained by tests, for example with the MOTOR TEST function. To do this, switch off the ramp and increase the speed until step losses occur. Then reduce the resulting value by about 30 % to be on the safe side.

Shortest ramp

The ramp parameter specifies the length of the acceleration ramp until maximum speed is reached, or the deceleration ramp from maximum speed to stationary, in milliseconds.

*Individually
adjustable
ramp length*

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



Speed profile of a motor movement

Accelerating and braking with ramps prevents step losses on the motors and permits faster maximum speeds. The gradient of the ramp is the same in all movements, i.e. the ramp time is shorter for movements at slower speeds.

Invert movement direction

There are two possible remedies if one of the motors is moving in the wrong direction.

Changing over the movement direction

Either swap over the motor winding connections or change the movement direction with this parameter. The direction signal for controlling the motor is then inverted before being output.

Reference switch

This parameter enables you to define in which direction the axis is moved in order to search for the switch. Movement clear from the switch then takes place in the opposite direction.

Reference sequence

Reference movement of the individual axes takes place in a particular sequence. Usually, it is necessary to move the Z-axis upwards first so it is withdrawn from the workpiece. Then the two other axes move to their reference points.

Possible faults with reference movement

After the machine is started up for the first time, there may be faults with reference movement which can be remedied as follows.

- Axis moves in the wrong direction
Remedy : **Define that the reference switch is at the other end of the axis.**
- Axis moves in the correct direction, but stops after reaching the switch and does not move clear again
Remedy: **Define the reference switch at the other end, use the *CNC-Configurator* to change the switch logic.**

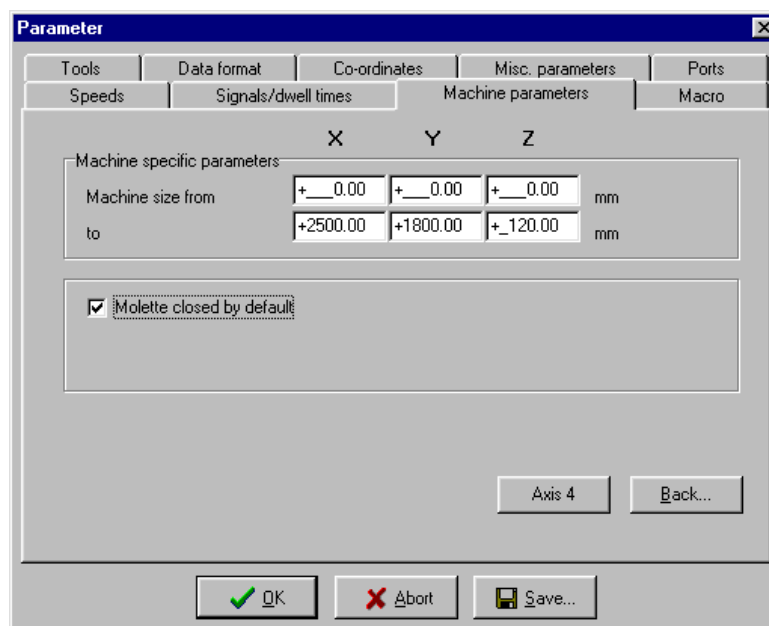
Maximum spindle speed

Speed control in 256 steps

WinPC-NC can control the speed of a drilling or routing spindle. It is possible to define a speed for each tool, or the speed data are taken from the NC files.

The maximum value defines the reference for step 256. All lower values are calculated proportionately between 0 and this value, and are output.

The spindle speed is generated on the *CNC-Control* axis controller, which means the necessary hardware must be available. This involves either an analog output card or up to eight digital outputs.



Second dialog box with machine-specific parameters

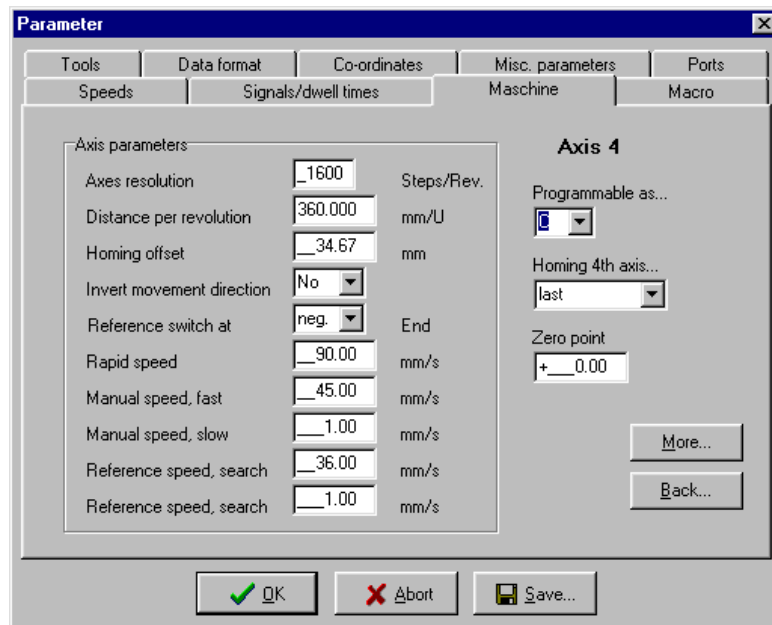
Machine bed dimensions

Machine size The effective movement area of the machine can be defined using these parameters. If area monitoring is activated, **WinPC-NC** checks and monitors these limits during subsequent movements, including jog movement.

Collet chuck after switch-on

WinPC-NC needs to know the switch-on condition of the collet chuck in order to operate the automatic tool changer and the electrical or pneumatic collet chuck correctly. This defines whether the collet chuck has to be opened or closed with an active output.

4th axis settings



4th axis parameters

WinPC-NC and the axis controller can manage and operate a 4th axis. Programming can be done in a DIN/ISO program, for example.



The 4th axis must be enabled in the axis controller before it can be used. Otherwise, the parameter settings are not available.

Axis resolution/Distance per rotation

The resolution parameters operate in the same way as the standard X, Y, Z axes. If you enter 360 or 36 as the distance per rotation, then you can program the 4th axis as a rotational axis in degrees or 1/10 of a degree.

Reference offset

The reference offset defines the angle differential from the zero position after moving clear of the switch. **WinPC-NC** first looks for the switch at search speed and then moves back from the switch in the opposite direction. Following this, the reference offset is moved as a rotation round to the zero position.

Axis 4 programmable as

Letter for programming

The 4th axis can be addressed in DIN/ISO programs using various axis letters. Axes parallel to X, Y, Z are normally designated U, V, W, while rotational axes in the X, Y, Z direction are programmed with A, B or C.

Reference movement 4th axis

The reference movement of the 4th axis can be performed either before or after the other axes.

All other parameters are defined and function in the same way as with the three standard axes.

4.9. Macros

The macros function in **WinPC-NC** offers you a powerful tool for individual adaptations to an extremely wide range of different mechanisms and applications. User-definable macros enable the working sequence to be influenced and configured in many respects.

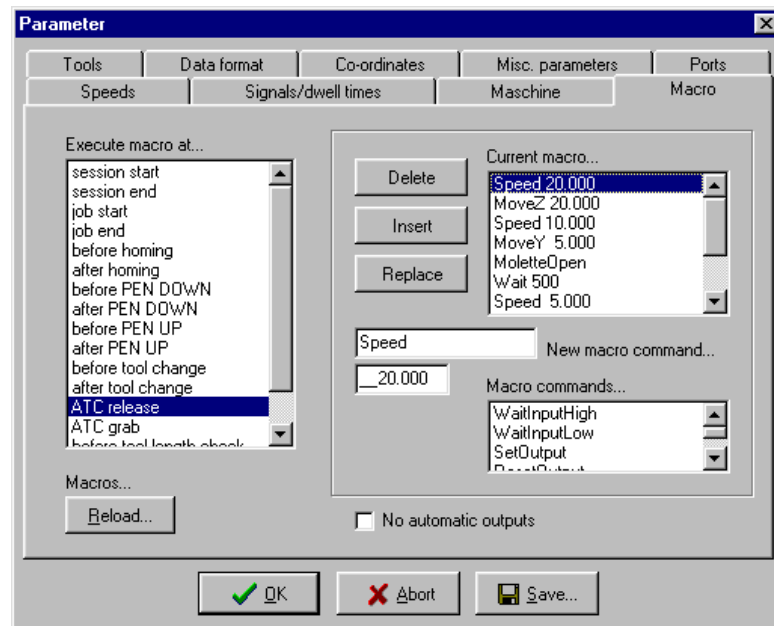


In order for macros to be used in *WinPC-NC*, they must be enabled in the axis controller.

Flexibility with macros

WinPC-NC always stores defined macros in the WINPCNC.MAC file when the parameters are saved. They are always valid and apply to all projects. By clicking the Reload button, it is possible to load the most recently saved macros from the file again.

If you are using an automatic tool changer, it is essential to use macros for the pick and place processes.



Macrodefinition

The dialog box for defining macros is divided into several parts :

- | | |
|------------------------------|---|
| Left-hand list box | Possible points in the program when macros can be run |
| Top right list box | Current macro, a new macro is assembled in this list box using macro commands |
| Bottom right list box | All possible macro commands which can be used to assemble macros |
| Editing boxes | Here, the selected macro commands are combined with the necessary parameter values, e.g. times in wait commands or distances in movement commands |
| Editing buttons | These allow the current macro command in the top right list box to be deleted, replaced or a new command can be added |

Creating macros **Macros are defined in a procedure made up of several steps :**

- Select a program condition by double-clicking in the left-hand list box. This is the program condition for which the macro is to be defined. The top right list box then either displays an existing macro or *No macro defined*.
- Edit the required macro by double-clicking the lines you want to change to transfer them to the editing box. There, you can define different values for the commands. The new command is adopted when you click Replace or Insert.
- In order to create new or partial macros, you have to select the required command from the list box at the bottom right and then complete it in the editing boxes. Following this, the new command can be transferred into the current macro using Replace or Insert.
- To delete macro command, simply move the bar to the corresponding command in the current macro and click the Delete button.

Example 1
Put down tool

Example for putting down a tool in the changer magazine.

Speed 80,00	Set the speed to 80 mm/s
MoveZ 156,34	Move down with the Z-axis
Speed 5,00	Set speed slower
WaitSpindleStop	Wait until spindle has stopped
MoveY 10,00	Move the tool into the chuck holder
ColletChuckOpen	Open the collet chuck
Wait 500	Wait 500 ms for pneumatic system
Speed 2,00	Set very slow speed
FahreZ -5,00	Move 5 mm up from tool
Speed 80,00	Set speed back to fast
MoveZ -151,34	Move Z-axis all the way up
MoveY -10,00	Move Y-axis back to starting position

Example 2
Insertion when flame cutting

Example of a macro defining an insertion procedure when flame cutting. It is always performed before *PenDown* commands.

MoveZ 50,00	Move down with the Z-axis
Wait 10000	Wait 10 seconds at pre-heating position 1
MoveZ 20,00	Move down again by 20 mm
Wait 2000	Wait 2 seconds and pre-heat
SetOutputHigh 100	witch on oxygen with output 100
Wait 500	Wait another 500 ms before movement

Do not operate signals automatically

*Outputs
automatic or
with macros*

Normally, **WinPC-NC** operates a few output signals automatically. For example, the *Job in progress* output is set at the start of a working process and reset at the end of after a cancel. Equally, the following signals are also set automatically: Drilling spindle, Coolant pump, Dispensing, etc.

If it is better for the signals to be operated with the macros or if you want to change the times when they are operated, then you must switch off automatic operation with this parameter.

Following this, all signals can only be operated using macros.

5. Additional information

5.1. Interpreters

At present, interpreters are implemented in WinPC-NC for several NC data formats.

- HPGL
- Drilling data
- Multicam
- DIN/ISO or G-codes

5.1.1. HPGL interpreter

Plotter data with the HPGL interpreter

The HPGL interpreter recognizes the most important commands in HPGL plotter language. Successful tests have been carried out with various CAD and drawing programs.

Drawings from...

- AutoCAD®
- Eagle®
- CADDy®
- AutoSketch®
- HCAM®
- Tommysoft CAD®

Windows HPGL drivers

...and other CAD systems can be processed without any problems. Furthermore, the interpreter can also work with HPGL data from the standard Window driver HP-Plotter, which programs such as...

- CorelDRAW®
- Designer®

...and other Windows drawing programs use for preparing their data and storing data in a file. Alternatively, it is also possible to use a driver for the HP 7475 or HP Color Pro plotter.

HPGL syntax

The following commands produce a response from 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]];	Lift and move tool
PD [x,y[,x,y]];	Lower and move tool
SP x;	Select tool
IP x,y[,x,y];	Define scaling points
SC x,y,x,y;	Define scaling factors
CI r[,a];	Draw complete circle
AA x,y,a[,c];	Absolute arc
AR x,y,a[,c];	Relative arc

In addition, the interpreter recognizes the following commands although no response is produced on the machine:

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

The following non-standard expansions can also be used:

PP ;	Drill hole, lower/lift tool
FC t;	Wait t milliseconds
FC100,t ;	Wait t milliseconds
FC101,i ;	Wait for HIGH level at input i
FC102,i ;	Wait for LOW level at input i
FC200,a ;	Set output a to HIGH
FC201,a ;	Set output a to LOW

HPGL language prescribes the following syntactical rules, amongst others, which must be observed when creating or revising the data.

- There must always be a pair of coordinates x,y
- All commands with coordinates must have no, one or more coordinate pairs
- Every command must be followed by a semicolon (;) or a new command
- With PA and PR, the subsequent commands are also set to absolute or relative positioning
- The last command should lift the tool (PU;)

<i>HPGL example</i>	PU; PA1000,1000; SP1; PD; PA1000,2000; PA2000,2000; PA2000,1000; PA1000,1000; PU;	<i>Start by lifting tool</i> <i>Then move to 1000/1000</i> <i>Select tool 1</i> <i>Lower tool</i> <i>Outline square</i> <i>Lift tool</i>
---------------------	---	---

5.1.2. MultiCAM-Interpreter

HPGL expanded with 3D commands The MultiCAM format is a 3D expansion of the HPGL standard. It includes all previous HPGL commands and some additional ones by means of which the Z-axis can be moved synchronously with the other axes.

A differentiation is made between 2D and 3D modes. There must always be 3 coordinates in 3D mode, i.e. one for the Z-axis as well.

ZZ0/1;	Activates 2D or 3D mode
ZUz;	Toollift height of the Z-axis over the zero point (overwrites the safety clearance parameter)
ZDz;	Plunge depth (overwrites the depth values in the parameters)
SFv;	Feed rate
ZOd;	Spindle speed

The syntactical rules of HPGL language apply analogously to MultiCAM.

5.1.3. Drilling data interpreter

Various drilling data formats The drilling data interpreter recognizes data in the following standards: Excellon, Sieb&Maier1000 and Sieb&Maier3000.

A drilling file always starts with one or two percent signs and ends with the command M30. In between, there can be any number of lines with X and Y coordinates and the tool number.

Drilling data syntax

The syntax is as follows:

```
% oder %%  
[Xnumber]/[Ynumber][T x]  
.....  
M30
```

There are two different formats. In format 1, the coordinate values are always 5-digit numbers, although it is possible to leave off the trailing zeros (e.g. X021 corresponds to 2100).

In format 2, leading zeros are discarded and the coordinate values therefore comprise different numbers of digits.

The drilling formats prescribe the following syntactical rules, amongst others, which must be observed when creating or revising the data.

- Every command must be in its own line
- If coordinates and a tool command are contained in the same line, then the tool is changed first before drilling takes place at the position
- The coordinates can also include a decimal point, e.g. X123.456
- All lines before the percent sign(s) are regarded as remarks and are skipped

Sample of drilling data in format 2

Short drilling program in format 2	<i>Remark lines</i>
%	<i>Start of program</i>
T1	<i>Select tool 1</i>
X1000Y1000	<i>Drill hole at 1000/1000</i>
X1200Y2340	<i>Drill hole at 1200/2340</i>
X2700Y2950T2	<i>Tool 2 drill hole at 2700/2950</i>
Y1000	<i>Drill hole at 2700/1000</i>
M30	<i>End of program</i>

And here is the same example in format 1:
(always 5-digit values, with trailing zeros discarded)

<i>Sample of drilling data in format 1</i>	Short drilling program in format 1	<i>Remark lines</i>
	%	<i>Start of program</i>
	T1	<i>Select tool 1</i>
	X01Y01	<i>Drill hole at 1000/1000</i>
	X012Y0234	<i>Drill hole at 1200/2340</i>
	X027Y0295T2	<i>Tool 2 drill hole at 2700/2950</i>
	Y01	<i>Drill hole at 2700/1000</i>
	M30	<i>End of program</i>

5.1.4. DIN/ISO or G-code interpreter

WinPC-NC also contains a small G-code command interpreter. The G-code language is used in professional and industrial applications for programming 2 and 3-axis machine tools.

The following commands are supported:

G commands	G00	Linear interpolation with rapid speed	
	G01	Machining, linear interpolation	
	G02	Clockwise circular movement	
	G03	Counterclockwise circular movement	
	G04	Dwell time, time with F command	
	G17	Plane XY	
	G18	Plane XZ	
	G19	Plane YZ	
	G70	Dimensions in inch	
	G71	Dimensions in mm	
	G90	Absolute position information	
	G91	Relative position information	
	G98	Define subroutine	
	M functions	M00	Program halt
		M02	End of program
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	
M66		Change tool, same as M06	
M70	Set output 100 to HIGH		

Additional information

	M71	Set output 101 to HIGH
	M72	Set output 102 to HIGH
	M73	Set output 103 to HIGH
	M74	Set output 104 to HIGH
	M75	Set output 105 to HIGH
	M76	Set output 106 to HIGH
	M77	Set output 107 to HIGH
	M-70	Set output 100 to LOW
	M-71	Set output 101 to LOW
	M-72	Set output 102 to LOW
	M-73	Set output 103 to LOW
	M-74	Set output 104 to LOW
	M-75	Set output 105 to LOW
	M-76	Set output 106 to LOW
	M-77	Set output 107 to LOW
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 or inch/min)
	I,J,K	Circle parameters
	T	Select tool
	X,Y,Z	Coordinates
	U,V,W	Coordinates for 4th axis, if activated
	A,B,C	Coordinates for 4th axis, if activated
	R	Radius of arcs
	L	Call subroutines

*Example:
Square with
rounded
corners*

```
%prog2
N001 G90
N002 G71 T1 M6
N003 G00 X110 Y100 Z10
N004 G01 Z11
N005 G01 X190
N006 G03 X200 Y110 J10
N007 G01 Y190
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
N015 M30
```

*Start of program
Absolute coordinates
Dimensions in mm, tool 1 rapid
speed to 1st position
Plunge movement with Z
Feed movement in straight line
Arc about center point
etc.*

*Rapid speed to zero point
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 speed cannot be influenced by commands; instead, rapid traverse is used for G00 and tool feed speed for other commands
- The command number can be defined using N commands
- For arcs, you can either use the I, J, K commands to define the center point or R to program a radius. Positive radii produce an arc less than 180° and negative radii an arc greater than 180°.
- Subroutines are defined at the end of the main program after M30, using G98 Lx. The definition ends with G98 L0. The subroutine call can be positioned anywhere with Lx.

<i>Example:</i>	%prog2	<i>Start of program</i>
<i>Drilling cycle as</i>	N001 G90	<i>Absolute coordinates</i>
<i>subroutine</i>	N002 G00 X110 Y100	<i>Rapid traverse to 1st position</i>
	N003 L1	<i>Call drilling cycle</i>
	N004 G00 Y110	<i>Next position</i>
	N005 L1	<i>etc.</i>
	N006 G00 Y120	
	N007 L1	
	N008 G00 X0Y0	
	N009 M30	<i>End of program</i>
	N020 G98 L1	<i>Define drilling cycle</i>
	N021 G01 Z20	
	N022 G00 Z0	
	N023 G01 Z25	
	N024 G00 Z0	
	N025 G01 Z30	
	N026 G00 Z0	
	N027 G98 L0	<i>End of subroutine</i>

5.2. Error messages

Error messages occurring when **WinPC-NC** is running appear in a small window in the middle of the screen. Messages in a gray box do not indicate there is a fault, but only provide information.

The following error messages may occur :

- **Limit switch reached**

The machine has reached one of the limit switches. It is necessary to move back from the switch and perform reference travel again before continuing work.

- **Machine not switched on**

The connected controller or machine is not ready or not switched on.
WinPC-NC works in demo mode.

- **HPGL syntax error**

There is an invalid HPGL command in the plotter file. It does not correspond to the syntactical rules and therefore cannot be processed by **WinPC-NC**.

- **Connection canceled**

The connection to the controller is interrupted. This can happen with long cables or interference on the line. Please use the shortest possible null modem cable for connecting the controller and do not route it close to power supply units, high-frequency components and other sources of interference.

- **Status stack full/empty**

An internal error has occurred in *WinPC-NC*. Please report this error to the program authors, providing as many details as possible.

- **Incorrect parameter**

A parameter in the WPI or WPW file is incorrect or the parameter file has been corrupted.

- **Stop signal detected**

The stop signal has been detected. Please rectify the fault.

- **No reference movement performed yet**

WinPC-NC and the controller always need to know what their current position is. Please always perform reference travel after restarting the controller.

- **Error xx from controller**

An internal error has occurred in *WinPC-NC*. The causes may be as follows:

- The connection is disrupted because the serial cable is routed past live parts / is defective / is not plugged in correctly
- The axis controller is running with a very old firmware version Please report this error to the program authors, providing as many details as possible.

- **Data cannot be displayed**

The NC data for the current file cannot be displayed graphically. Either they do not match the selected format or they do not contain any tool actions.

- **Tool not defined yet**

A tool is needed which has not yet been programmed in the tool management system. The appropriate parameters must always be defined prior to using tools.

- **No tool file found**

No tool file can be found. *WinPC-NC* therefore continues to work with default values.

- **Machine is at limit switch**

When both limit switches on an axis are defined, *WinPC-NC* is unable to tell which switch has been reached. Please move the axes in question back from the limit switch manually and then restart reference travel.

- **Service code required**

The axis controller can no longer be controlled and requires a service code before continuing work. Please contact the supplier.

- **Command rejected by controller**

An internal error has occurred in *WinPC-NC*. The causes may be as follows:

- The connection is disrupted because the serial cable is routed past live parts / is defective / is not plugged in correctly
- The axis controller is running with a very old firmware version Please report this error to the program authors, providing as many details as possible.

- **Machine not ready**

The machine has not been switched on or there is a fault. The READY input on the axis controller signals this condition.

- **WinPC-NC does not work with this axis controller**

The axis controller used cannot be controlled with *WinPC-NC*. Please contact the manufacturer.

- **Keypad fault**

A fault has occurred in the connected keypad or there is a discontinuity in the cable. Please check the connection. Please contact the manufacturer or the supplier if the fault reoccurs.

- **Communication failed / Timeout**

There is a fault in the connection between *WinPC-NC* and the axis controller. The causes may be as follows:

- The connection is disrupted because the serial cable is routed past live parts / is defective / is not plugged in correctly
- The axis controller is running with a very old firmware version Please report this error to the program authors, providing as many details as possible.

- **Protective hood not closed**

The protective hood or other protective device has not been closed and therefore no process can be started. The machine can still be set up.

- **Command rejected during machine movement**

There is a fault in the connection between *WinPC-NC* and the axis controller. The causes may be as follows:

- The connection is disrupted because the serial cable is routed past live parts / is defective / is not plugged in correctly
- The axis controller is running with a very old firmware version Please report this error to the program authors, providing as many details as possible.

- **File with invalid format**

The selected file does not match the set data format. Change the parameter or the data format to HPGL or drilling format.

- **Help file not found**

The file with the help texts cannot be found. It is called WINPCNC.HLP and must be located in the current folder or in the *WinPC-NC* folder. No help texts can be displayed.

- **Workpiece exceeds working area**

The set working area was exceeded during machining of the current workpiece. It may be possible to rectify this condition by performing reference travel. If not, either change the unit of measurement, move the zero point or increase the working area.

- **Next position outside working area**

The current working process has been cancelled because the next position to which movement should take place is outside the set working area. Correct the settings and restart the working process.

- **Unknown command**

The NC file contains an unknown command and the working process has therefore been stopped. It can now be cancelled or continued with the next command.

- **Cannot create file**

WinPC-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.

- **No park position defined**

You can only move to the park position if you have defined the corresponding coordinates, i.e. they are set to values other than zero.

- **Required tools not defined**

The working file uses tool numbers which have not yet been defined. The plunge depth and plunge speed must be defined for every tool, and HPGL files also need the feed speed to be defined.

- **Tool moves in too far**

The plunge depth of a tool or the depth and the defined feed corrections are deeper than the maximum plunge depth or the limit of the Z-axis working area.

- **G code syntax error**

There is a syntactical error in the G code file being processed. Please use the editor to correct the line in question and restart the working process.

5.3. Special versions of *WinPC-NC*

Special applications on request

The authors are always grateful for opinions and suggestions. If required, hardware and software solutions can be worked out for your special application, or our standard products can be adapted to optimize their functions for you. Please ask us for a quote without obligation.

