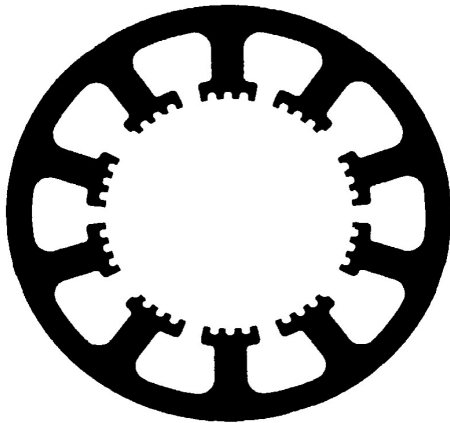


**Preliminary
final version to
download**

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

WinPC-NC

USB



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

Version 3.00

Februar 2017

Legal Notice

Lawful purchase of the **WinPC-NC** control program conveys permission for one person to utilize the **WinPC-NC** control program and its manual. Copying or modifying the **WinPC-NC** control program - including any of its constituent parts - or its manual are forbidden. Furthermore, any unauthorized transmission of the program or its manual - including any extracts - will be legally prosecuted by all means available.

The authors reserve all rights to the **WinPC-NC** control program, its constituent parts, and its manual, including all accepted forms of copyright.

WinPC-NC may not be used in association with medical equipment or food preparation without the explicit, written permission of the author.

Despite extremely thorough testing, it is never possible to completely eliminate all faults and ensure problem free operation. No responsibility can be accepted for damage caused as a result of using our program. We would be grateful to receive feedback, positive and negative, and detailed reports of any problems from lawful **WinPC-NC** users.

Support and reduced cost updates are only available for registered customers. Registration is performed by sending us an email containing the **WinPC-NC** version number, your assigned serial number, and your mailing address. This information may be found on the installation media or visible in the program window when **WinPC-NC** begins execution.

Please register your license !!!!

Burkhard Lewetz
Hardware-Software
Brückenstrasse 7
D-88074 Meckenbeuren
Germany
eMail info@lewetz.de
Homepage www.lewetz.de

May 2017

MS-Windows is a registered trademark of the Microsoft Corporation.
Other products mentioned by name are trademarks or registered trademarks of their corresponding companies

Safety Notices

NC equipment generates magnetic fields which can injury, death, or equipment damage. Refer to your equipment manual for details.

NC equipment can cause injury or death when operated incorrectly. Refer to your equipment manual for details.

NC equipment are an electrical shock hazard. Refer to your equipment manual for details.

NC equipment can cause a fire. Never leave an NC machine alone when it is operating. Always have a fire extinguisher nearby.

NC equipment may use chemicals (e.g. lubricants, coolants, oils) which can cause injury or death. Refer to each chemical's safety and toxicity documents for more details.

NC equipment may cause hearing damage. Ensure one or more forms of hearing protection are in use before nearing the equipment.

NC equipment may can cause eye damage. Ensure one or more forms of eye protection are in use before nearing the equipment.

NC equipment may release airborne particles that can cause lung damage, allergies, immune system disorders, and other serious conditions. Ensure one or more forms of air protection are in use before nearing the equipment.

Table of contents q

The structure of this manual	7
Definitions	7
Use of typography	9
Different versions of WinPC-NC	9
1. What can WinPC-NC do ?	11
2. First Steps	15
2.1. Requirements to the hardware	15
2.2. Installation	15
2.3. Launching the program	17
2.4. First setup and test run	20
2.5. Exiting WinPC-NC	20
3. Operating WinPC-NC	21
3.1. Graphical display of the NC file	21
3.2. Drop-down menus and function keys	31
3.3. The individual menus	31
3.3.1. FILE menu	32
3.3.2. MOVE menu	35
3.3.3. PARAMETER-menu	51
3.3.4. SPECIAL FUNCTIONS menu	54
3.3.5. HELP-menu	63
4. 2D-CAM functions	65
4.1. Overview	65
4.2. Setting options	65
4.3. Example 1	70
4.4. Example 2	74
5. Parameter settings	78
5.1. Basic aspects	78
5.2. Basic settings	79

5.2.1. Basic settings-Interfaces	79
5.2.2. Basic settings Display/Operation	81
5.2.3. Basic settings- X-,Y-,Z-axis.....	84
5.2.4. Basic settings- 4.-axis	87
5.2.5. Basic settings-Accessories	88
5.2.6. Basic settings -Size and dimension	90
5.2.7. Basic settings-Homing	91
5.2.8. Basic settings-JOG move	93
5.2.9. Basic settings-signal wizard	95
5.2.10. Basic settings-Spindle.....	96
5.2.11. Basic settings-Dynamic speed Control	98
5.2.12. Basic settings-monitoring	99
5.3. Coordinates	101
5.4. Tools.....	106
5.4.1. Tools-Activation	106
5.4.2. Tools-Speeds.....	108
5.4.3. Tool-dimension	110
5.4.5. Tool-Changer-Magazine	114
5.5. Miscellaneous parameters.....	116
5.5.1. Miscellaneous-Display/Operation.....	116
5.5.2. Miscellaneous-Job process.....	121
5.6. Technology	125
5.6.1. Technology- Function for mass production	125
5.6.2. Technology-rotary axis.....	126
5.6.3. Technology-Tangential-cutting.....	128
5.6.4. Technologie-3D-print	130
5.6.5. Technology-Laser	132
5.6.6. Technology-Grinding.....	133
5.7. Macros.....	134
5.8. Data format.....	139
6. Initial start-up with the machine	143
6.1. Connecting ncUSB and the machinen.....	143

6.2. Pin assignment LPT ports at ncUSB	145
6.3. Determination of axis resolution	146
6.4. Determination of direction.....	147
6.6. Sequence and direction of reference move	150
6.7. Control of settings	151
6.8. Additional steps	152
7. Signal wizard	153
7.1. Using input and output signals	153
7.2. Assignment of inputs	155
8. Additional Information	161
8.1. Interpreter.....	161
8.2. Error messages	169

The structure of this manual ...

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 and how to adjust the initial specific parameters.
- Chapter 3 :** More detailed descriptions of how to operate the program and the individual functions of **WinPC-NC**.
- Chapter 4 :** 2D-CAM functions for sorting and calculating a tool diameter compensation.
- Chapter 5 :** Explanation of all parameters and the setting options.
- Chapter 6 :** Initial start-up procedure step by step.
- Chapter 7 :** Information concerning definitions and adjustments of input/output signals.
- Chapter 8 :** Further technical information about the program, about the implemented NC format interpreters, error messages and special versions.

Definitions

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

**Job file
or
NC 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
or
Job**

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

Command

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

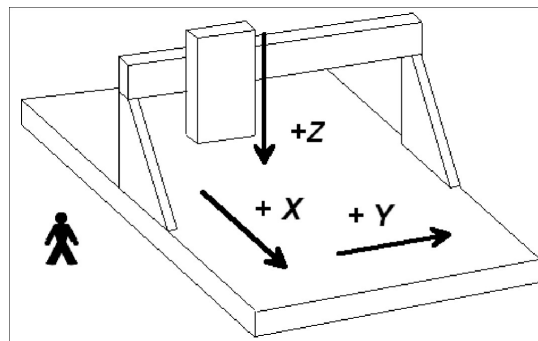
Button

Mouse clickable field to activate a certain function.

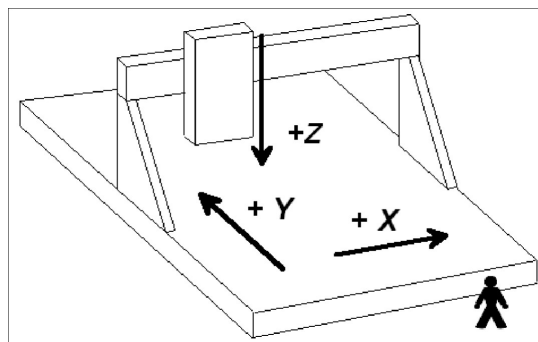
Checkbox

Box for activating or deactivation a certain parameter or function, e. g. signals. An activated checkbox is marked with a cross

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. The assignment of axes depend on your personal position and your viewport to the machine.



Schematic layout of a flat-bed machine with your position aside



Schematic layout of a flat-bed machine with your position in front of

Use of typography

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

Different versions of WinPC-NC

The controlling program **WinPC-NC** is available in four different versions.

Our lowcost program **WinPC-NC Light** offers all necessary functions for beginners, e.g. engraving, milling, drilling and PCB drilling or manufacturing modelling parts. Machine control is made by a PC internal LPT printer port or a small USB LPT plug which is enclosed in delivery.

WinPC-NC Economy. **WinPC-NC Economy** is equipped with additional functions and offers with up to four axes a lot of special functions and provides support of extraordinary mechanical components. This program disposes of much more format interpreters as the version for beginners Machine control is made by one or two pc internal LPT printer ports.

WinPC-NC Light and **Economy** are also available for SMC stepper cards, i. e. motors are not controlled by clocking/direction signals but by SMC signals for well known stepper cards SMC800 and SMC1500.

The functions of **WinPC-NC USB** are equal to those of the Economy version. However, with **WinPC-NC USB** the machine is controlled by a small enclosed USB module and not by an integ-

rated LPT port. The module casing is equipped with 2 connectors which are compatible to the LPT port. Some machines are already equipped with a small board which is required as USB hardware.

WinPC-NC Professional is considered as industrial version and runs only in combination with our external axes controller CPU and is therefore most qualified for all true realtime tasks. The program is fairly independent of windows speed and provides besides utmost stability and reliability additional professional technology functions. Furthermore, we can provide various housing types suitable for switching cabinets according to the individual requirements.

WinPC-NC Starter is a very simple control program which is included to certain OEM machines as an addon software. It cannot be purchased in separate and without machines. The simple functionality enables to create engravings, flat milled parts and drilled PCB boards and leads in easy to learn steps to a save operation of the machine.

For comparing the different versions, please use the document function table. It will provide detailed information concerning capacity and potential operations.

1. What can *WinPC-NC* do ?

*Universal
program*

Das Programm **WinPC-NC** ist eine Software, die aus jedem Standard-Personal-Computer eine universelle NC-Steuerung macht und bis zu 4 Achsen ansteuern kann.

*Required
hardware
accessories
enclosed*

WinPC-N USB works with a small USB module called **ncUSB**. Due to this element CNC machines or stepper motor drives can be accessories directly controlled. The USB module is equipped with 2 LPT enclosed compatible ports of other WinPC-NC versions. Alternatively it is also possible to use another USB hardware which is already installed, e. g. a model provided by our company.

WinPC-NC USB requires a modern PC with at least 2 Ghz clock-pulse rate and 32 bit or 64 bit operating system starting from WindowsXP or higher.



NcUSB module and cable set

By 4 stepper motor axes it is possible to realize any 3D mechanics and to use them for various functions. Standard applications contain:

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

*Exentsive
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 keyboard operation. This makes it easy to learn and master the program.

*Runs on
any modern
PC*

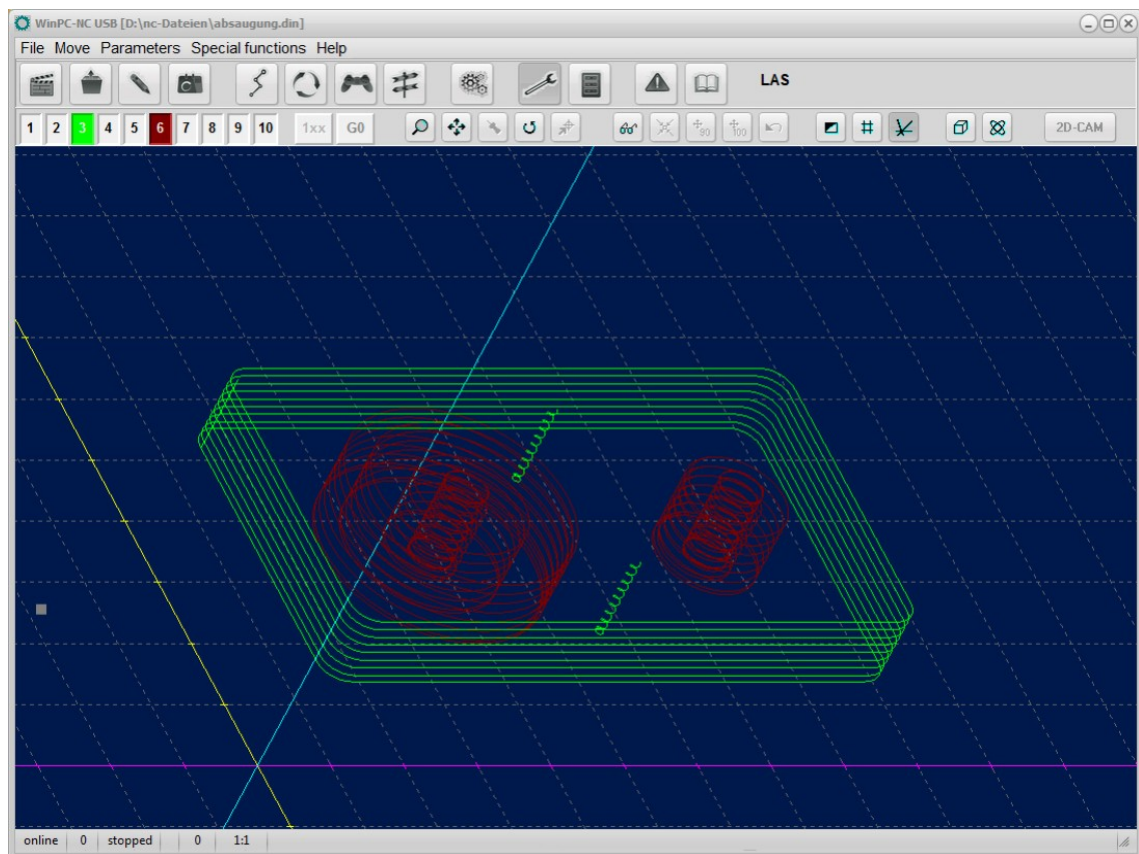
Running **WinPC-NC USB** requires a modern standard PC with any hard disk. USB ports, any kind of graphics card and a 32-bit modern Windows operating system. Notebooks and netbooks with PC corresponding equipment can also be used.



Sample of a 3 axis milling plant

Special functions **Special functions of WinPC-NC USB:**

- Almost unlimited working range
- according to industry standards it controls by clocking/direction signals any commercial stepper motor cards and produces stepping frequencies up to 80 kHz
- Jog mode in exact steps using the cursor keys or the mouse
- Graphical display with scale, zoom, shift, turning and reflecting functions
- Runs from WinXP or higher, 32-bit or 64 bit operating system
- axis resolutions, speeds, backlash compensation, ramp length for the X/Y and Z-axis can be set within generous limits. The 4th axis can be programmed as U, V, W, A, B or C or tangential axis T
- Reference and limit switch interrogation, max. 10 inputs and 8 additional outputs can be set within generous limits
- recognizes HPGL, DIN/ISO, MultiCAM 2D and 3D, drilling formats, ISEL NCP, EPS/AI and DXF (2D)
- extensive tool management function, tools can be selected individually, colors can be set, repetition and feed functions
- Support of automatic tool changers with up to 10 magazine positions
- CAM functions for cleaning up drawings, sorting by tool and position and calculating tool diameter compensations
- Speeds can be set between 0.01 and approx. 1000 mm/s
- High-performance integrated editor or external editor definable
- Lots of external signals for synchronizing the procedure, e.g. start signal, spindle speed reached, machine ready
- Speed control of the drilling/milling spindles, counterclockwise rotation/clockwise rotation
- Units of dimension can be set to mm, inch and mm/s, mm/min and inch/min
- multilingual, other languages can easily be added, 17 languages selectable from menu (date 2/17)



WinPC-NC main screen

- Tool length measurement and compensation
- Flexible macro language for running subroutines at various points in the program
- Override for feed rate and spindle speed
- Comfortable Teachin function
- Repetitions for mass production
- Remote control via external inputs
- Hand wheel support
- Individual adjustments for machine and mechanic parameters and operational and job specific defaults
- Profile administration for easy switching between applications
- ...and there are many other features besides these

2. First Steps

2.1. Requirements to the hardware

WinPC-NC USB is able to perform a direct control of a connected CNC machine and the drives belonging to it by the enclosed USB module **ncUSB**. The required USB driver and the software are to be installed on the computer.

A faultless operation is therefore only guaranteed with personal computers with certain minimum of requirements.

- modern CPU with at least 2GHz clocking frequency
- 32-bit or 64-bit operating system from Windows XP or higher
- a direct USB 2.0 port, no passive USB hub if possible, only original cable set
- standard graphical card, keyboard, mouse, hard disk and other common PC implements
- it is recommended to deactivate any kind of power savings, e. g. screen saver, hard disk, shutdown and reduction of clock-pulse rate. The same applies also to the power saving settings for the USB ports and device manager.

Detailed information and tips for selecting the suitable computer can be learned from further support information documents.

2.2. Installation

userfriendly installation

WinPC-NC is installed using a user-friendly setup program. installation Please insert the disk into the drive and wait for automatic installation start. If the installation is not executed, please start the program SETUP.EXE directly from the CD.

The installation wizard then guides you through the entire procedure.

WinPC-NC USB is delivered together with a separate USB module and cable or a built in USB hardware.

Please connect it to the computer and machine after the software has been completely installed and thus the driver is loaded.

Call up the README file to learn about 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 programm
WINPCNC.WPI oder WTI	Parameter file
WINPCNC.WPW oder WTW	tool file
WINPCNC.WPO	Settings for CAM functions
WINPCNC.MAC	defined macros
WINPCNC.PDF	This manual im PDF format
WINPCNC.LNG	Messages and texts, multilingual
WCNCCON.DLL	Communication modul for <i>ncUSB</i>
README	latest changes to the manual
*.DLL	Some files required in the system folder
*.PLT *.SMM *.DIN	Sample NC files
*.NCP *.EPS *.DXF	

NcUSB connection after software installation

After software installation please plug the supplied module **ncUSB** or machine by the enclosed cable to a vacant USB port. This should result in an immediate hardware identification with the corresponding report concerning the latest installed component.

After a short time the installation of the driver has been completed and you will receive message that the module is ready for use.

2.3. Launching the program

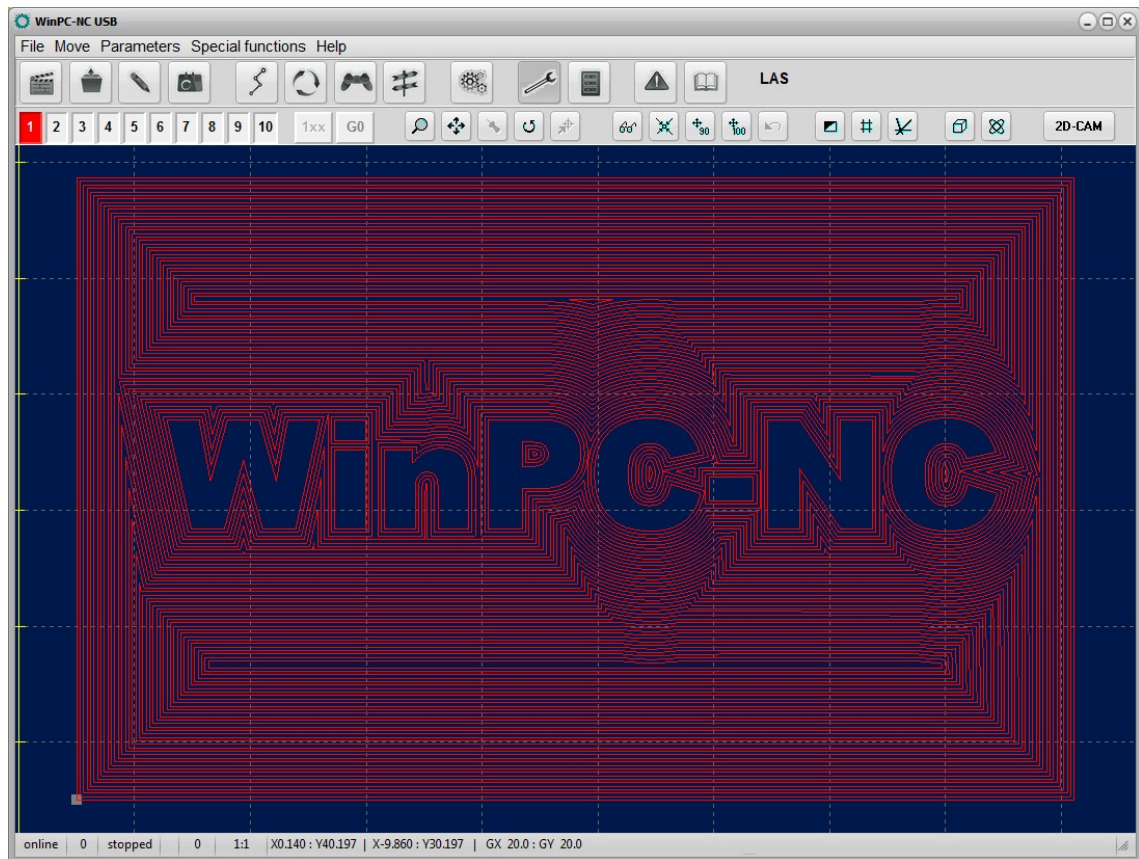
WinPC-NC is launched simply by clicking the desktop icon or by means of the start menu.

A functioning communication to the USB module is signalled by ONLINE in the left bottom corner of the main window. If the module is not plugged or can not be recognized, an error message is displayed and **WinPC-NC** runs solely in the demonstration mode.

In the middle of the screen the **WinPC-NC** working screen appears which is divided into several areas.

- Title bar
- Menu bar
- Bar with speed buttons for easy function call
- Tool bar and display buttons
- Display area
- Status bar

If delivery is made in combination with a machine, probably a menu will be displayed and you can select your machine. Thus any pre-defined settings are automatically carried out and the contact block and resolution of axes should be in conformity with your plant.


















WinPC-NC main screen

*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 speed buttons enable you to select important functions simply by clicking with the mouse.

The function of the individual buttons is as follows:

	End of program		Move Joystick
	open NC file		Reference move
	Load current or new file in the editor		Display of parameter settings
	Call profile management		Handling of molette
	Start Job		Select Tool
	JOG		Info concerning current version
	Move to current zero point		HELP
	Move to current park position	LAS CIR DIS 3DP GRI TAN	Current technology function

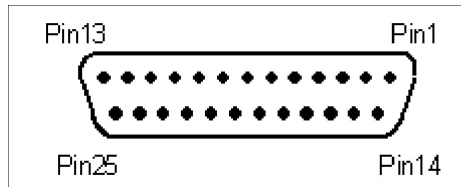
Buttons for using display and tools are placed in the second button bar.

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. The emergency-stop status can be shown by a red marked area at status bar as well. Buttons for operating the display and the tools are located in the second button bar.

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

2.4. First setup and test run

Executing an initial test run with the motors there is nothing else to do than connecting the machine with the LPT1 port of ncUSB and checking or amending the occupation of the controlling signals.



LPT1 port of *ncUSB*, pin layout

WinPC-NC is using the following signals for motor actuation.....

Pin 2	direction motor X	DIR X
Pin 3	clock motor X	CLK X
Pin 4	direction motor Y	DIR Y
Pin 5	clock motor Y	CLK Y
Pin 6	direction motor Z	DIR Z
Pin 7	clock motor Z	CLK Z
Pin 8	direction motor 4	DIR 4
Pin 9	clock motor 4	CLK 4

After this procedure it should be possible to perform a simple test run with function JOG or MOTOR TEST. Please be very careful in performing and be aware of uncontrolled movements of the machine.

Detailed instructions concerning start-up procedure are given in a separate chapter.

2.5. Exiting WinPC-NC

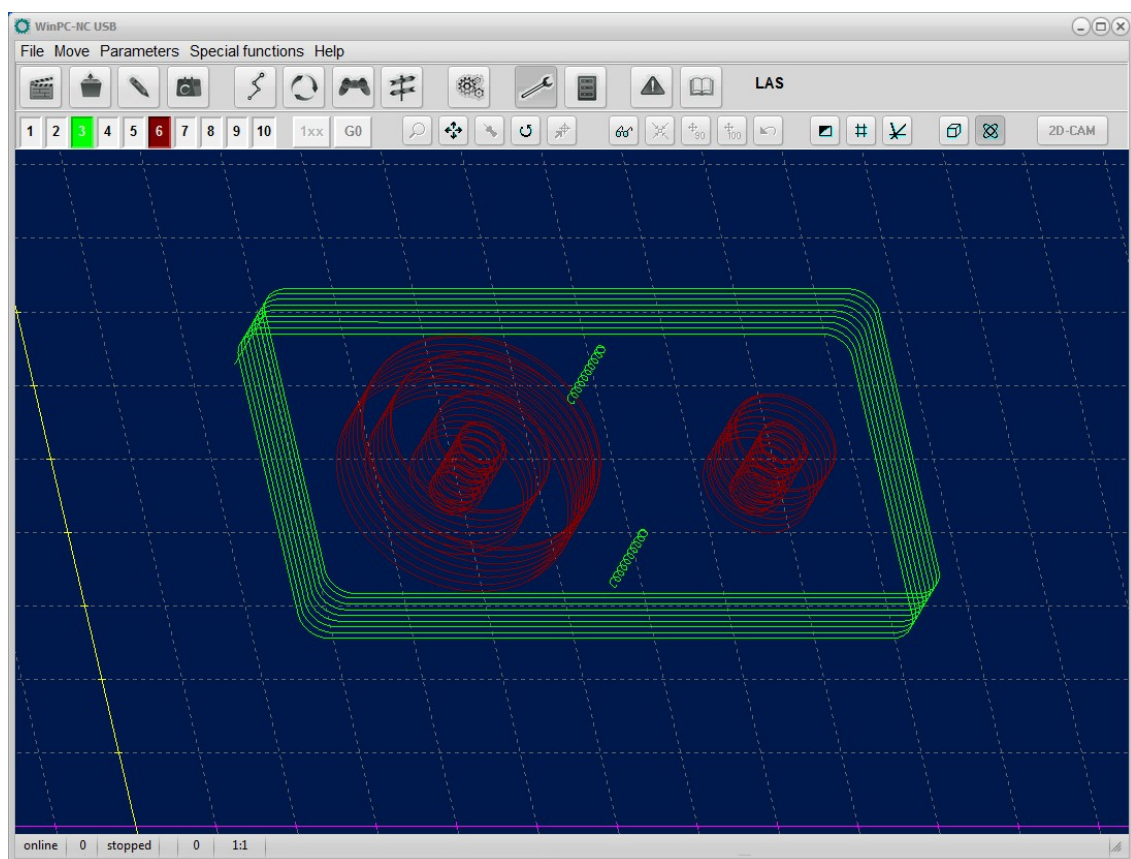
You can exit **WinPC-NC** at any time by clicking the cross in the box at the top right of the window, or by selecting EXIT from the FILE menu.

3. Operating WinPC-NC

3.1. Graphical display of the NC file

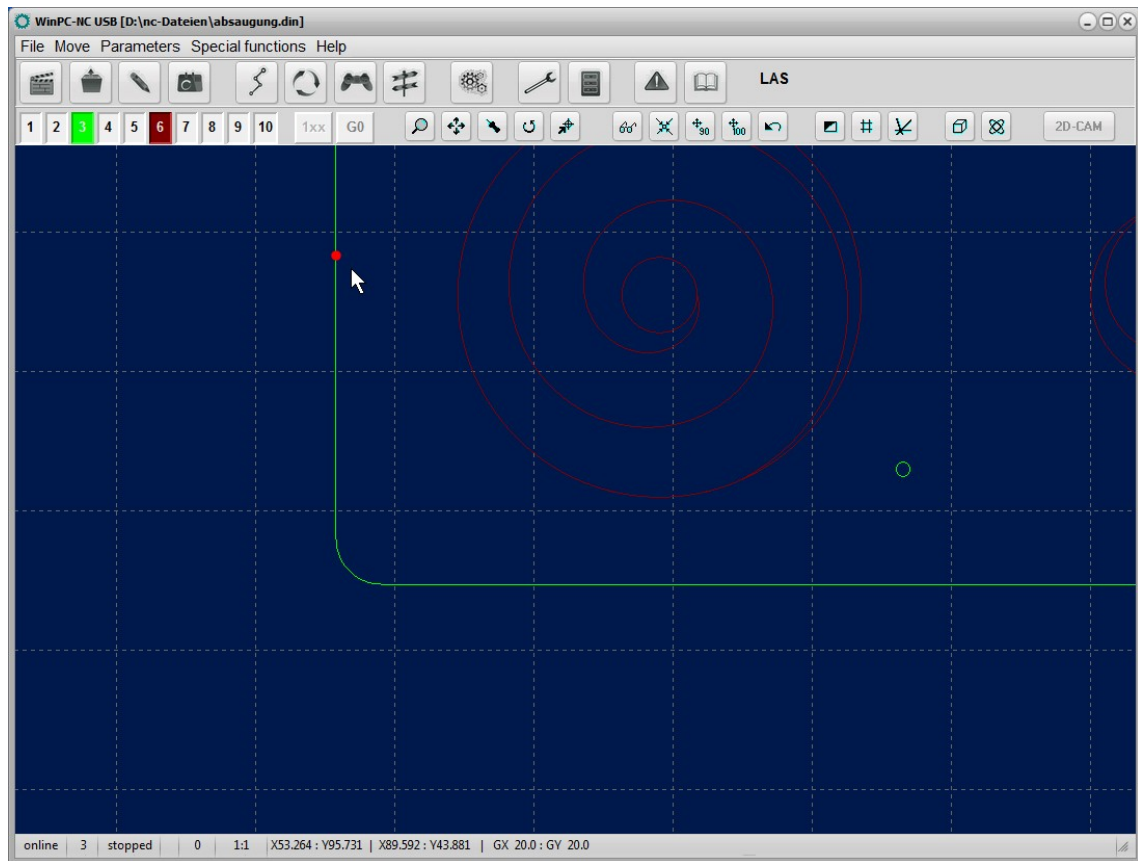
Graphical pre-view of the NC files

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



Graphical display in perspective view after selecting a NC file

Graphical measurements can easily be estimated by visible grid-lines. The distance is displayed below the graphic. The workpiece zero point appears as a small grey quadrat and the current position of the machine is presented as a small red point. Visibility of the actual machine position requires that **WinPC-NC** knows the actual site or a prior reference move.



Zoom in the graphical display of the NC file with zero point and current position

The first view of a loaded file is always the plane view on to the XY level. Several perspectives can be seen by using various display functions and can be even freely zoomed and scrolled.

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.

The zoom function of **WinPC-NC** can be used either by scrolling the mouse wheel or by clicking the zoom button. Then the graphical display has to be clicked by the left mouse pointer. While pressing the left mouse button it is possible to zoom and scroll the graphics by soft movements.

Move

The move function makes it possible to use the mouse to move a zoomed image and enables you to view another part of the NC file in greater detail. For scrolling the view you have to click on to the display window by the right mouse button and keep it pressed. By moving the mouse the view is now scrolled into the individual direction in real time.



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 in the way that the current machine position corresponds exactly to a point or a certain position in the file. The file zero point is simply recalculated internally.

Using this method it is quite simple to position the workingpiece and the NC file with great accuracy.



Move to position

For a speedy move to specific positions within the working area or the graphics please use this function. By a click to the button move to position the cursor appears as a target cross pointer and moves the machine immediately to a clicked position with rapid speed. Using the corresponding zoom factor you are able to move to the desired positions with great accuracy and thus for instance aligning a workpiece prior to clamping.



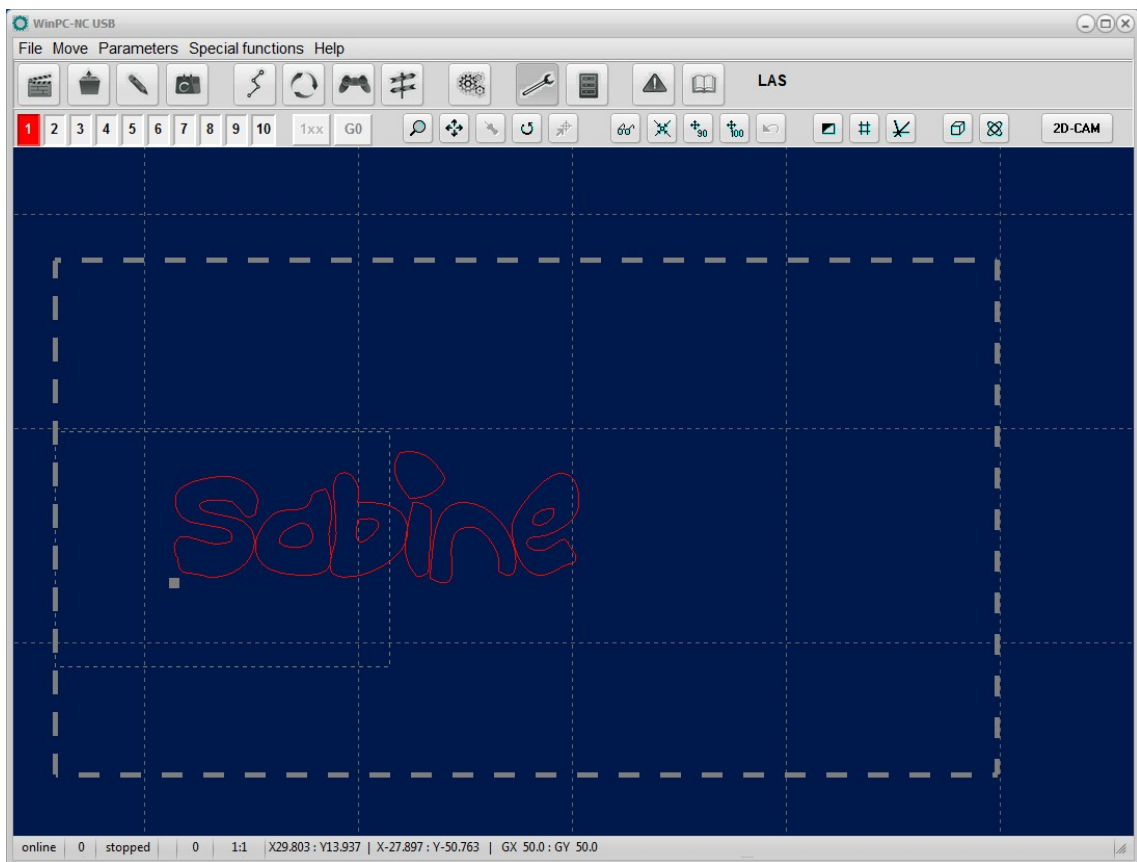
Turning data

By clicking the TURN button it is possible to turn the indicated data by 90°. In this way it is easier to place the data on the material.



Selecting the display area

By means of the button AREA the graphical display can easily be changed. So it is possible to display only the piece to be produced and its corresponding data. There is also the possibility to display the defined workpiece area or the stipulated machine size with the current position. In this way it is easy to recognize the location of the piece and whether it can be worked without any problems within the defined limits.



Display with work piece and working area



Centering within the workpiece area

Concerning engravings and millings it is sometimes advisable to center all data in the pre-set area or on an *empty part*. This is automatically done by the centering function without any size change and the zero point parameters are re-calculated.

Centering area means pre-defined workpiece area. Any defini-

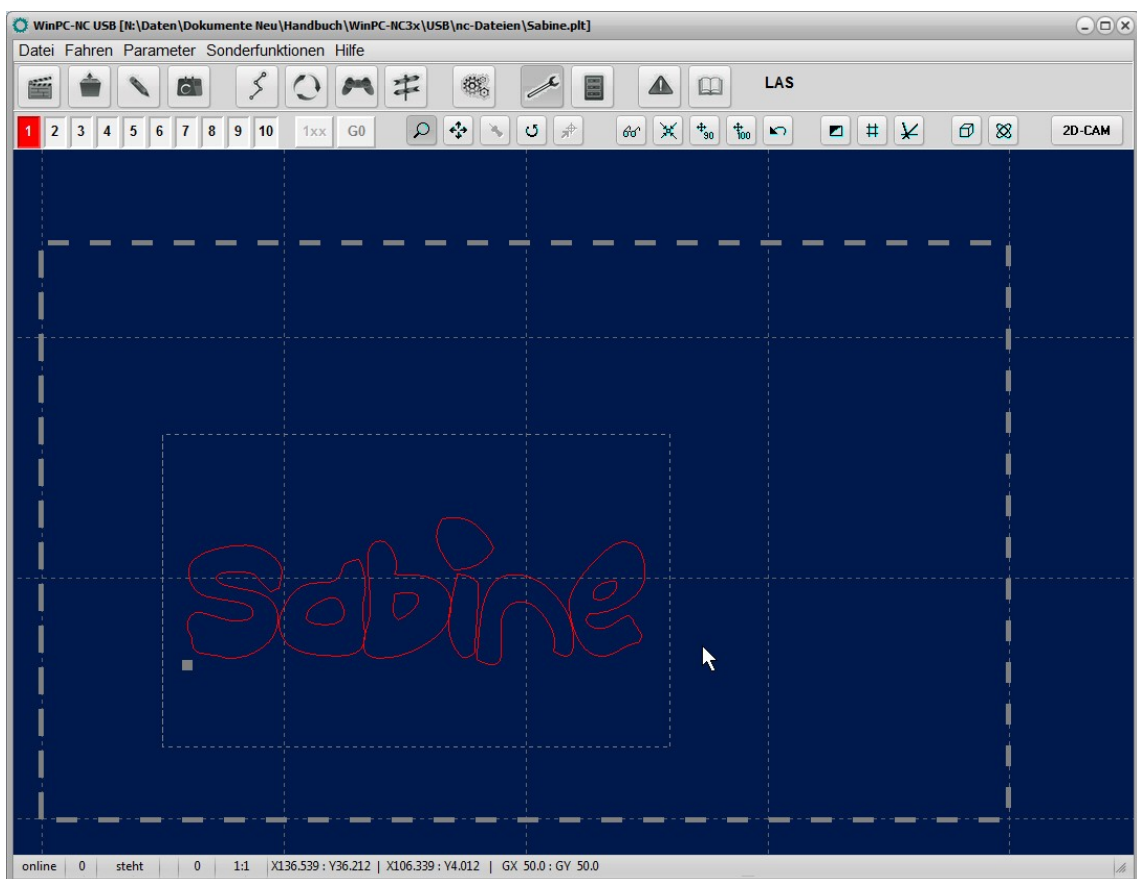
tions and settings can be specified in parameter-coordinates or by manual move to the left bottom or top right corners as well as by the function keys **F5** and **F6**.



Scaling and centering



Besides automatic centering data can additionally be scaled up to 90 % or 100 % of the predetermined workpiece sizes. In this way the engraving is properly centered with correct size and thus ready for being executed.



Data centered within the workpiece are and scaled up to 90%



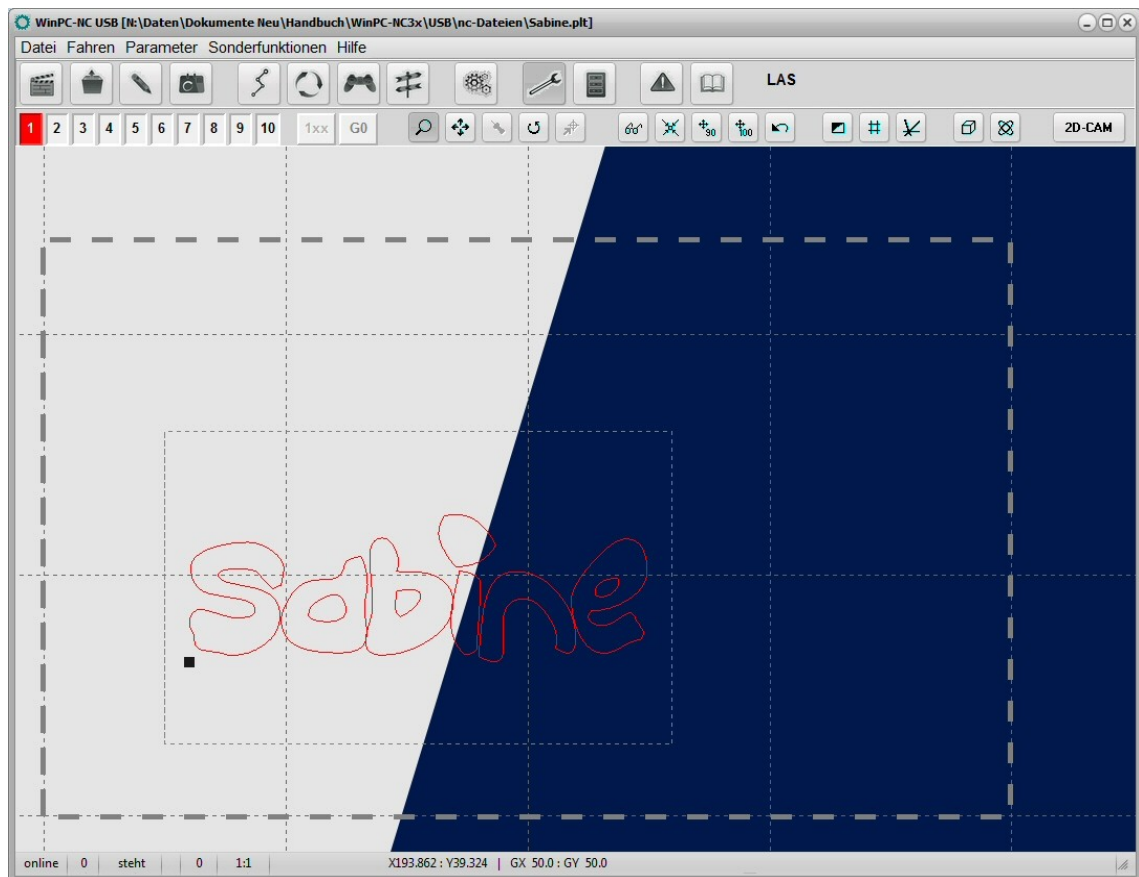
Undo scaling and centering

Automatic scalings and centerings which have been executed last can be cancelled by activating this button and the previous set parameters are restored.



Change background

This button changes the background color of the graphical screen. A dark blue and a light grey which corresponds more or less to the background color of the previous version. The individual color can be selected depending on your preference.



Background color dark or bright



Activate/Deactivate gridlines

This button activates or deactivates gridlines which serve as a base and are automatically scaled according to the drawing size. In GX and GY in the bottom bar the grid size can be read out.



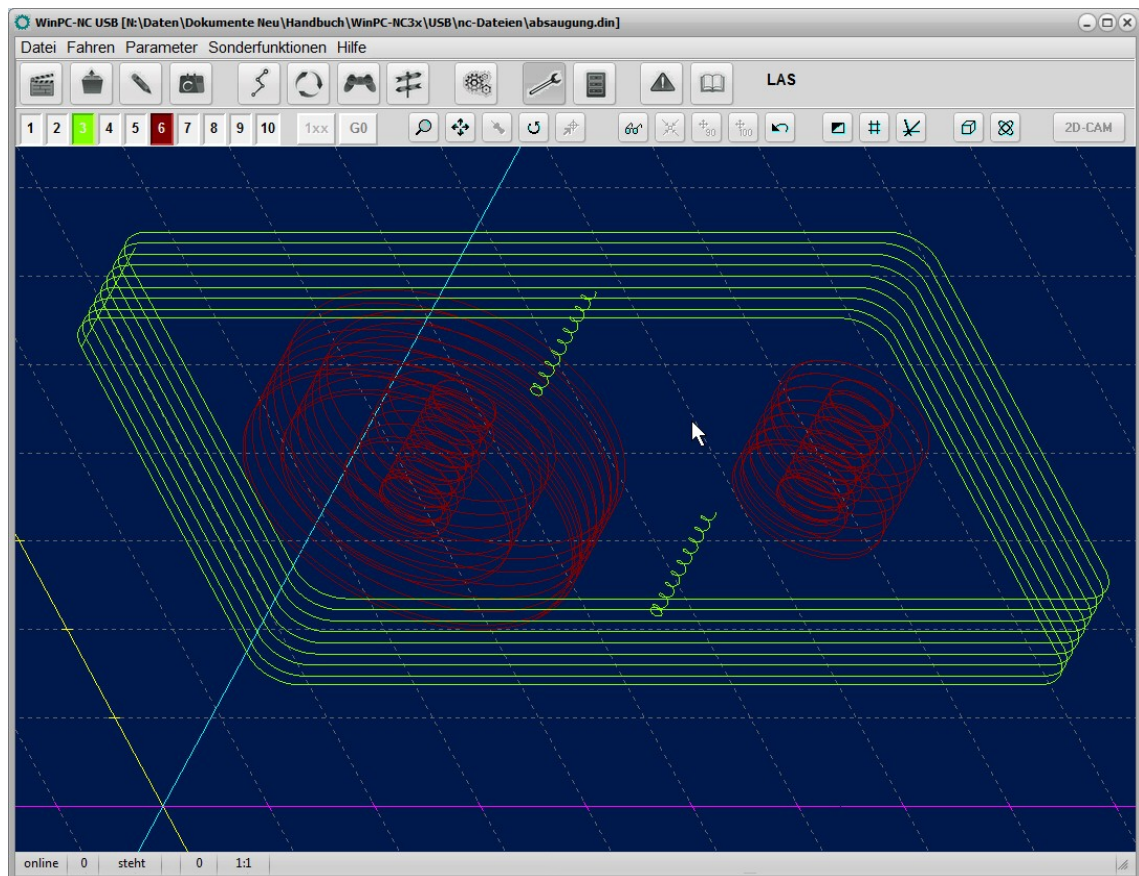
Displaying/Shielding axis of coordinate system

This button switches the display of the axis of the coordinate system. In particular the display and orientation of the Z-axis can be seen as benefit in the perspective display.

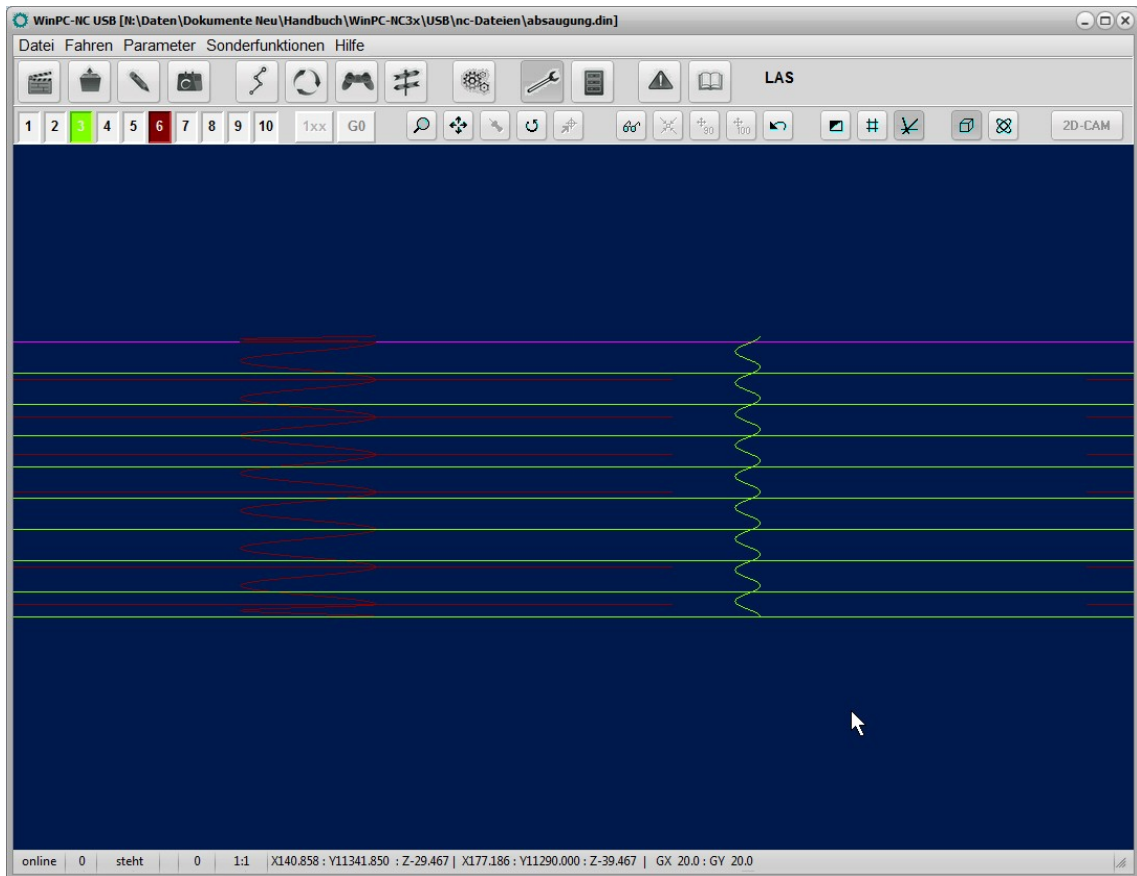


Change perspective /three-dimensional view

Clicking this button means that one of the 4 three-dimensional views jumps to the next. Included as standard the drawing is presented in plan view. With the first click the view changes diagonal forwards, with second click from below and with the third click from the left direction.



Perspective, three-dimensional view with all visible axes



View from the left side on to the Y and Z level with exact position of the mouse



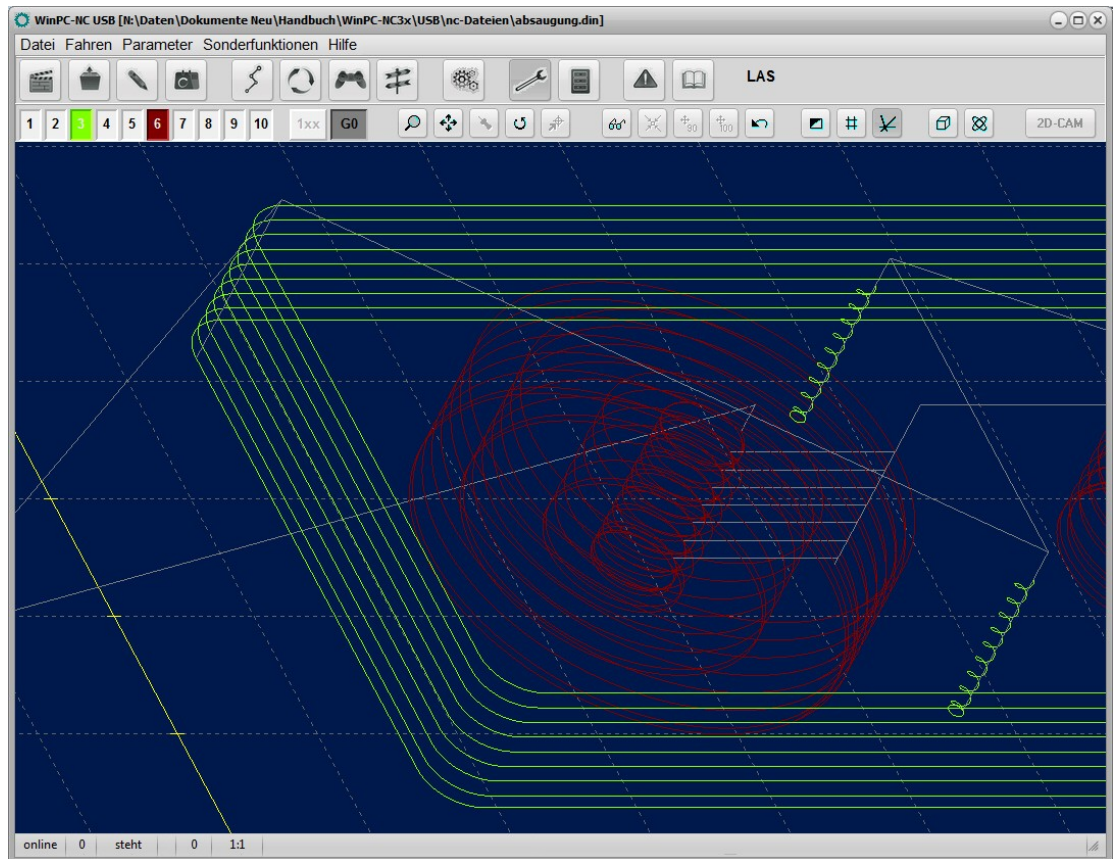
Uninterrupted 3D view, Orbit feature

In this mode the view can freely be turned and rotated. This requires nothing but a click anywhere in the graphics. By moving the mouse it is possible to turn and rotate it in any degrees of freedom and a full 360° view can be realized.



Display of unnecessary trial runs

Clicking this button means presenting or shielding unnecessary trial runs of the drawing.

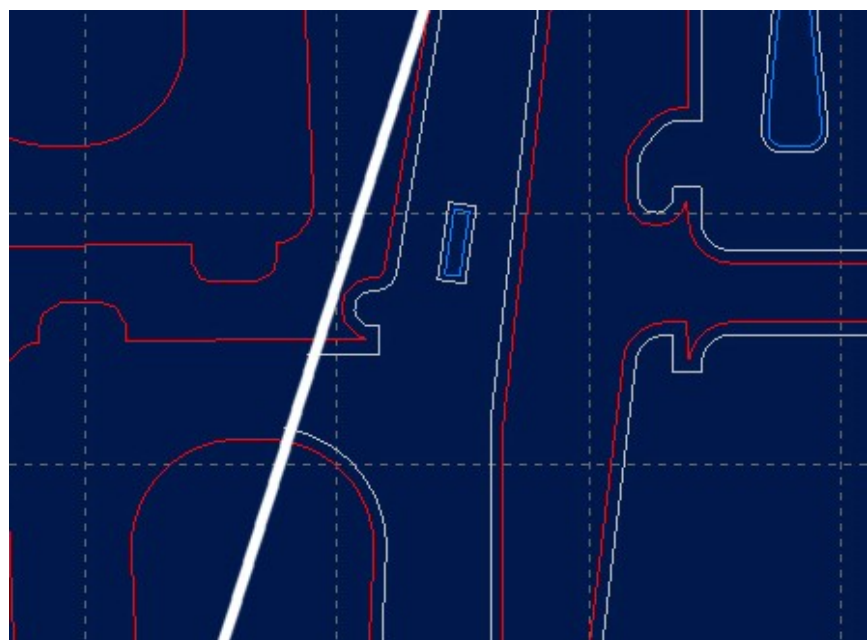


Display of unnecessary trial runs



Displaying or shielding original distances

Subsequently to a radius compensation with the internal 2D-Cam function original directions can be displayed or shielded.



Object after calculation of radius compensation with/without view on the orig. contour



Set tools active/inactive

All tools which are used in an NC file are displayed along the lefthand 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.

As in the example image on the left side, tool no. 1, 2 and 4 are activated. Tool no. 3 is deactivated and tool no. 5 does not appear in the drawing.



CAM functions

WinPC-NC has included simple CAM functions for all 2D data formats like HPGL, EPS/AI, DXF(2D) and drilling data files. This includes a cleanup of data, a new and optimal sorting and the calculation of tool diameter compensations.

A detailed step by step guide can be found in the next chapter.

The main screen of **WinPC-NC** as well as the window of the communication module with the position of axes and the optionally displayed timer can easily be moved and placed side by side. These positions are retained by **WinPC-NC** over more than one session.

3.2. Drop-down menus and function keys

modern interface

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

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. Function key assignment is fixed and indicated in the menu functions.

The most important function keys are:

F1	Activate the help system
F2	Load new NC file
shift-F2	Load parameter file
F3	Start job process
shift-F3	Start job process from defined point
F4	Move to XY origin
F5	JOG
shift-F5	Joystick JOG
F7	Load active or new file into the editor
F8	Start reference move
F9	Move to parking position
F10	Open pull down menu

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 analyse them. In addition, it is also possible to exit **WinPC-NC** here.

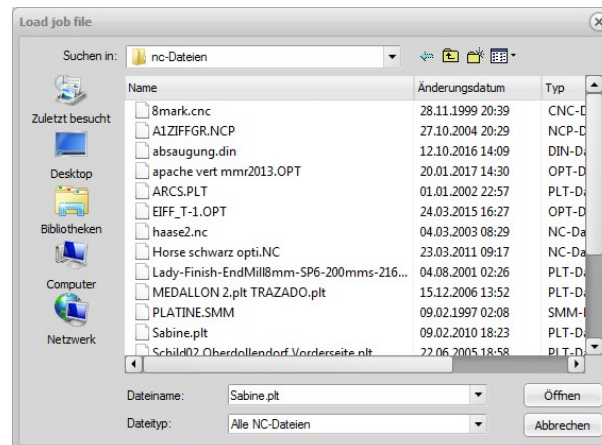
Open...	F2
Open without parameter...	
Last opened	
Editor	F7
Exit	

Press the `alt-D` shortcut key to open the file menu.

FILE OPEN

*File selection
by menu*

The FILE-OPEN menu item calls up an interactive function for selecting a NC 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.

*NC file loading
with previous
parameters*

A preliminary selection can be made via filter. Optionally you can display only NC files with familiar name extensions or all files. Loading formerly used NC files means loading all parameters which have been probably defined or saved for this project previously.

*Graphical
preview of
an NC file*

The graphical preview function in WinPC-NC is activated as soon as it is selected. All contours or vectors can be seen in the corresponding tool color bar.

FILE-OPEN WITHOUT PARAMETERS

The second OPEN function only loads the corresponding NC file without taking account of any project parameters. All current parameter settings remain valid.

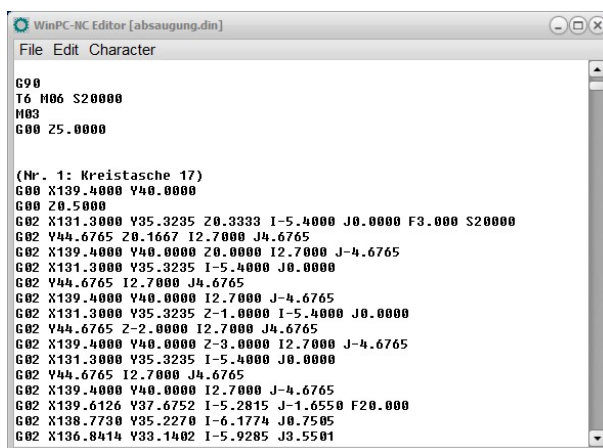
FILE-LAST OPENED

A register displaying last opened files. Any file can be directly opened by a click to the corresponding names.

FILE-EDITOR

*Revising
NC data*

The EDITOR menu item activates the editor for creating or revising the NC files. Optionally you can use the standard editor already integrated in **WinPC-NC** or an external editor which can be determined by parameter and which can be started alternatively with the current file.



NC file with integrated editor



The editor can also be activated by 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 changing.

Start	F3
Start from...	shift-F3
Start single steps	
Zero point XY	F4
Park position	F9
JOG	F5
Joystick jogging	shift-F5
Reference move	F8
Select tool	

The menu is opened by hotkey `alt-F`.

MOVE-START



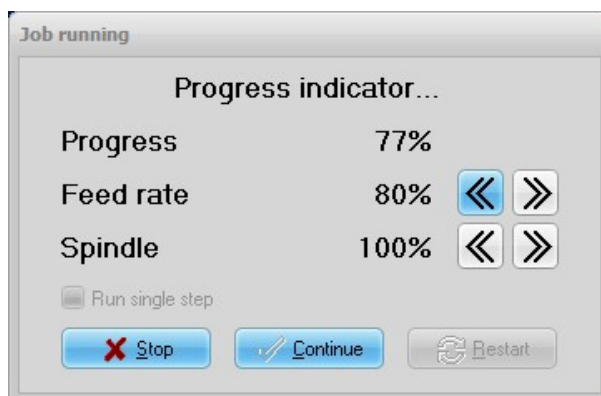
After a working file has been loaded, the START menu item can be used for processing the job. Pressing the (F3) key or clicking the START button has the same effect.

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 or optional even four axes can move at the same time in 3D files.

Progress display

During machining, a window displays the progress as a percentage and the current tool position is marked as a red dot in the graphics in real time.

Additionally available is the display concerning program real time for specific data interpreters, where the actual active command line is marked by a cursor.



Display of progress and speed override

speed override

Clicking the greater than/less than button in the window makes it possible to change the feed rate 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 as 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.

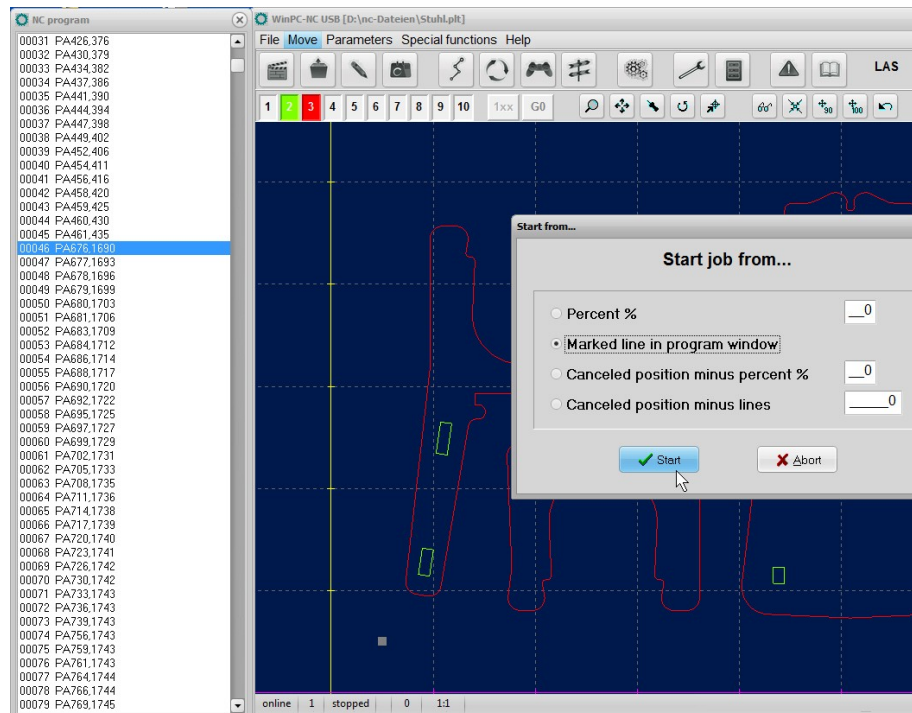
The continuation of an interrupted job has to be enabled in the parameters. The function *Start From* provides an alternative and additional options concerning the continuation process.



Resume of an interrupted job

MOVE-START FROM...

It is possible to start a job not right at the beginning and therefore you can use the function START FROM. There are optionally four different possibilities for selection.



Start from... menu

- Start from specific percent figure
- Start from marked line in the file window
- Start from previous interruption position less a percent figure
- Start from previous interruption position less a number of program lines in the file window

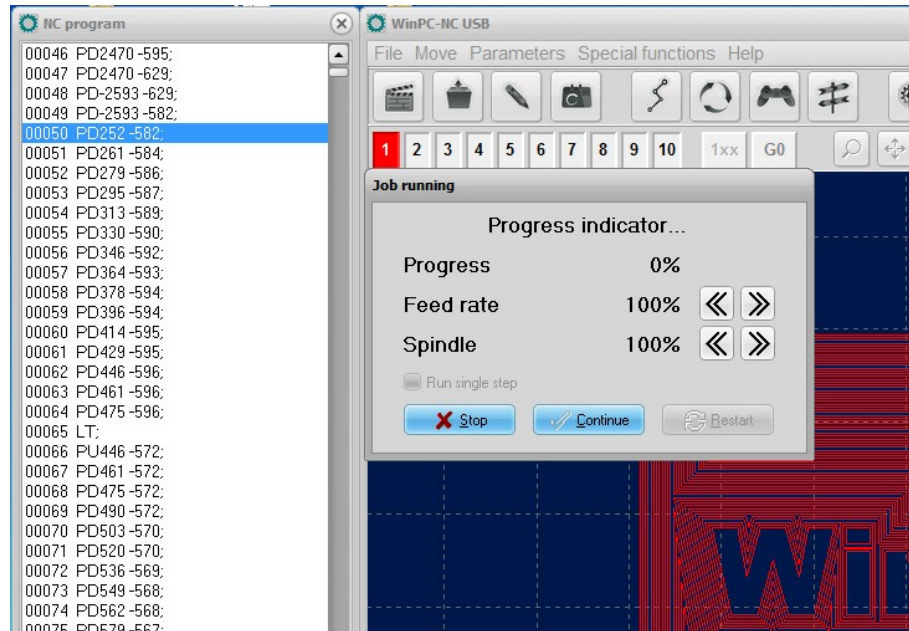
By starting the job **WinPC-NC** recalculates the initial point, moves to it and continues the job.

START-SINGLE STEP

Sometimes it is a good idea to start a job with single moving steps instead of starting it in full speed and rapidly towards the material. By moving in single steps you can easily check the first positions when touching the surface.

Using the function **START-SINGLE STEP WinPC-NC** moves command line for line and you can continue or cancel at each stop.

To continue with next the step just click on *Continue* button. If you want to stop the step by step job uncheck the checkbox and continue in full speed and continuous mode.



Start-Single step

MOVE-ZERO POINT XY and PARK

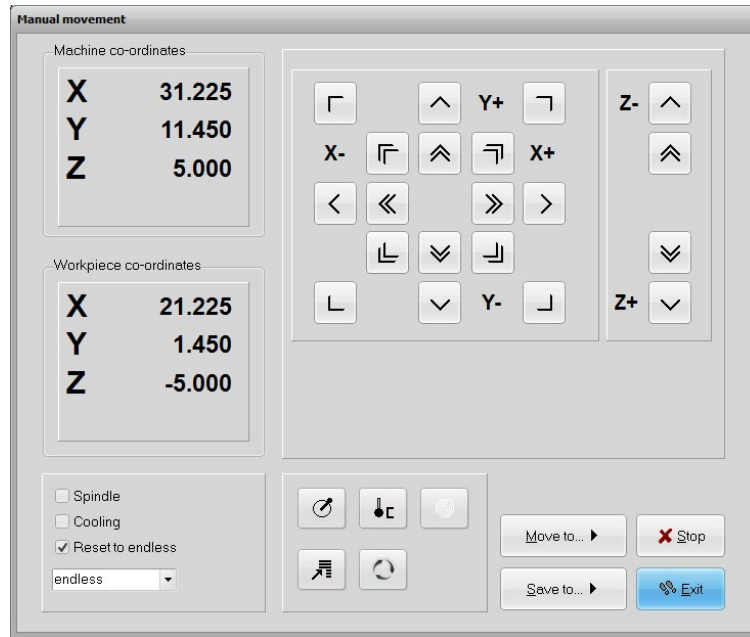


By the functions MOVE ZERO POINT XY and PARK, which can quickly be activated by the buttons **[F4]** and **[F9]** or by speed buttons the two help positions can be reached with ease.

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 function

Exact movement with the keyboard or mouse

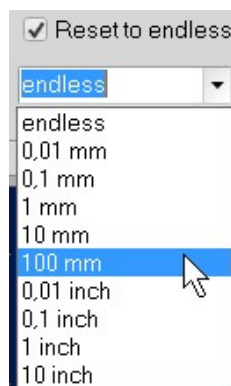
In JOG mode, it is possible to move all motors step-by-step or continuously with the cursor arrow keys in the numeric block or with the mouse. Pressing a key briefly or clicking the corresponding direction button causes only one motor step to be performed. Pressing the key for longer or keeping the mouse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

Via PC keyboard you can execute slow or fast moves. Where the latter is the case, please keep pressed the button **[STRG]**. The righthand arrow keys **[1]** to **[9]** are responsible for individual or diagonal move of the axis X and Y, the keys **[+]** and **[-]** move the Z axes. The speeds are defined in the parameters.

Display of step counters

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.

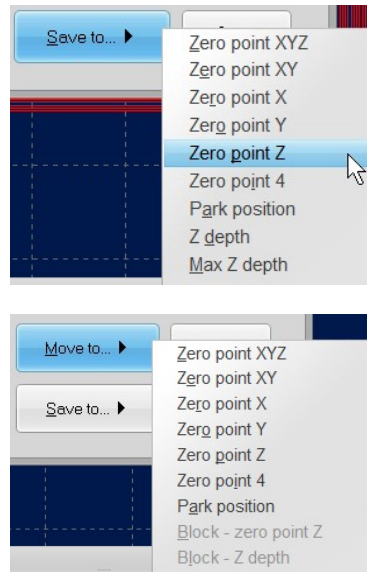
The second possibility involves moving discrete distances between 0.01mm to 100 mm and inches. The current set distance is displayed in the window. **WinPC-NC** moves this distance in any required direction and with both possible speeds, depending -on which button or key is pressed- In addition to the predefined distances, it is possible to enter any distance in the menu line.

Besides the pre-defined distances it is possible to enter any kind of distance in the entry field.

With enabled checkbox **Reset to endless** the individual selected distances are reset immediately in order to avoid new accidental travelling of remote distances.

Switching additional signals

Clicking the check boxes with the mouse all that is required is to switch the 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 and store certain help points. After a position has been reached, it is easy to click the Save button to select the required point and save it permanently as a parameter.

Moving to saved 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 the Exit button.

Automatic measurements of Z-heights

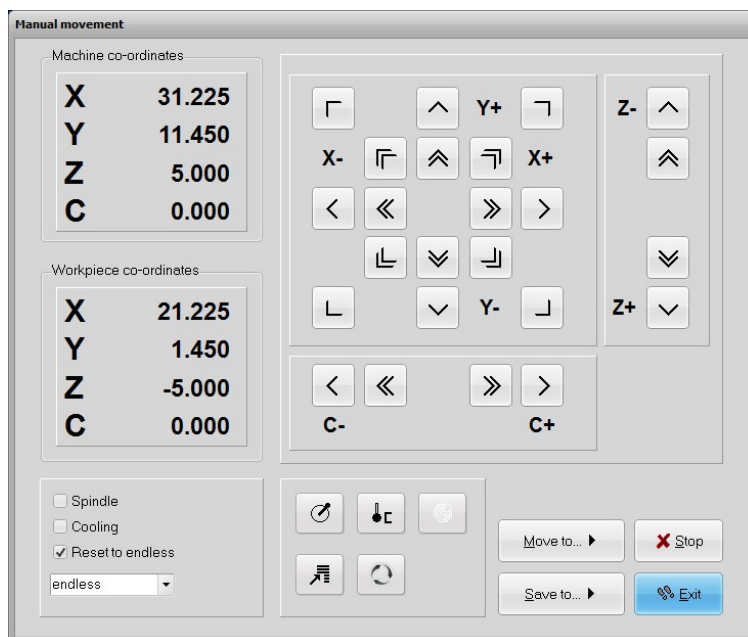
WinPC-NC can automatically determine the various Z-axis heights using a probe or surface block sensor. The sensor should be connected to an input on the LPT port using a cable to allow the block sensor to move freely. Depending on which Z-axis height you are measuring, place the probe against the clamped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

The measuring sequence involves several steps:

- Move the machine over the point to be measured
- Place the probe or surface block
- Start measuring process. **WinPC-NC** moves the z-axis down with slow speed until the probe contacts trips. Then the axis stops and **WinPC-NC** transfers the measured value together and probe width as parameter. The probe width can be defined as parameter.

Setting up the 4th axis

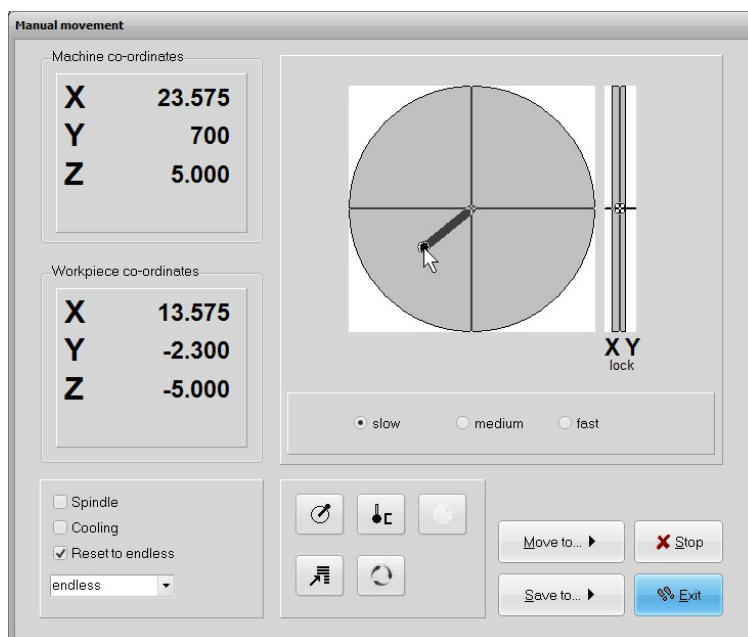
If a 4th axis is available, the appearance of the jog dialog box the 4th axis is some what different. In this case, it also includes buttons, for moving this axis slowly and quickly, as well as boxes for displaying the axis positions. The used axis letter can be defined by parameter.



Jogging with the 4th axis

SOFTSTICK – simulated Joystick

The buttons for movement provide three alternatives for switching the dialog of jog mode. The SOFTSTICK function represents a simulated joystick. The center point can be gripped by the mouse key and thus drawn in any kind of direction while moving the X and Y axes analogously to direction and displacement.

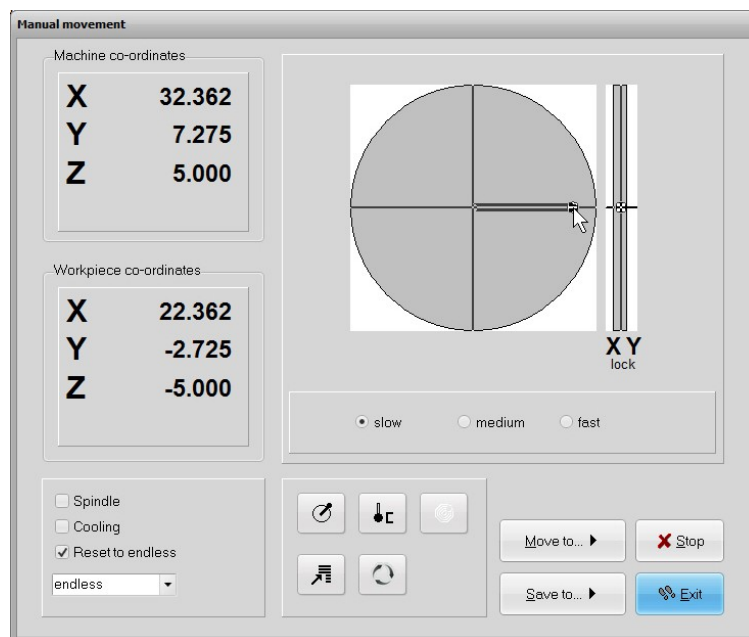


SOFTSTICK function for freely movable axis



Releasing the mouse key results in an immediate deceleration of the moving axis. Optionally there are 3 velocity graduations.

The circle enables moving XY and the right-hand bar allows moving the Z axes. By clicking the lock symbols one of the axis can be fixed and thus prevented from further movements.

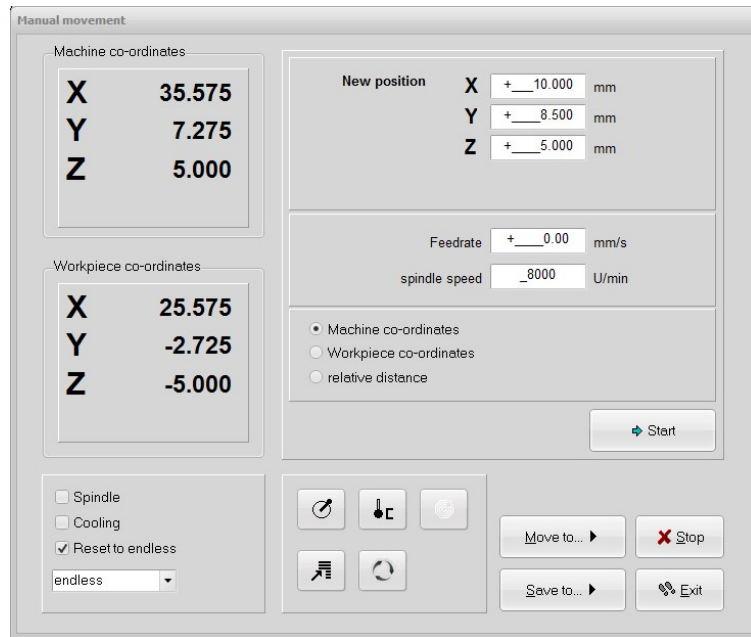


Softstick – move with fixed Y-axes (Lock-Y)

Move to specific positions

Furthermore, you can use the function TARGETED MOVE, for discrete distances in defined velocities and spindle speed. So it is quite simple to carry out any kind of material cut with great accuracy and without a hitch.

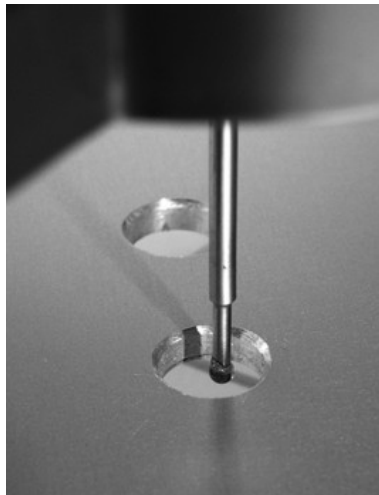




Function Specific Move

Edge probe

The last function to select is the edge probe function and enables you to automatically check and measure an edge of a material or a workpiece. Even the determination of the center of circles is possible.

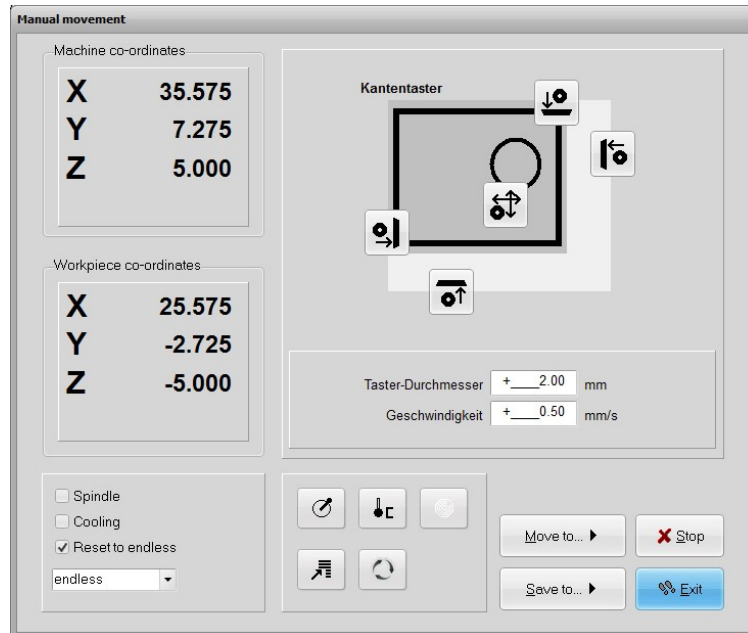


Edge probe for lateral approach

By using the function edge probe **WinPC-NC** can move aside until it touches the edge of a workpiece. The desired moving direction can be selected by pressing the according button. The movement stops automatically when touching the edge and the corresponding input line changes its level. **WinPC-NC** moves the probe back to free it, lifts up to defined tool lift height and moves

again over the workpiece exactly to the measured position while calculating it with the defined probe diameter.

After measurement you can save the new position as zero point of the axis.



Function for edge probe

To measure the center of a circle **WinPC-NC** moves in X and Y direction and touches all in all four points. After all movements it calculates the center of this circle and moves the probe to it exact by the step.

Hand wheel



WinPC-NC supports hand wheels supplied by our company for moving axes and using basic functions as saving help points or executing starting and interrupting a job directly at the workpiece.

Actually we can provide hand wheel **HR-10**, which ensures complete support by **WinPC-NC**. It is equipped with a corresponding USB cable of 5 m length and provides two various modes of movement for all axes of several buttons and thus guarantees fast functioning.



Further information and detailed description concerning buttons can be found in the corresponding documentation for HR-10.

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 is of advantage for observing the movements directly on the machine and setting exact positions. It is easier to take a joystick to the machine than keyboard or mouse.

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



Jogging with joystick

Jogging with 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.

Space-Mouse 3DConnexion

A very popular tool for CAD users is the Space Mouse from-3DConnexion company and **WinPC-NC** can support these parts as well to move the machine in 4 axes and in a very intuitive way.

When using a Space Mouse you first have to define the type of connection under parameters-interface and of course the corresponding drivers must be installed in your Windows system.

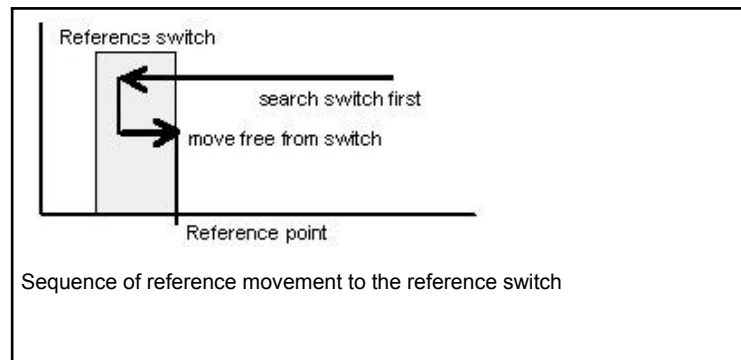


MOVE-REFERENCE

The function REFERENCE MOVE also called homing runs 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 of the machine the reference switch of the machine. 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.



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 the tool change or if tool length measurement is activated. It can be activated using the menu function or the tool list button.

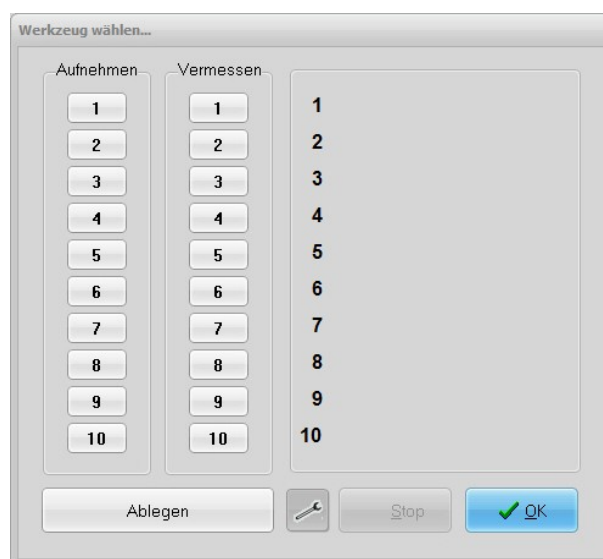
The dialog box contains the following possible functions:

- 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.

In order to be able to use each function of this dialog at least one tool length sensor should be available and set up.

Using an automatic tool changer requires the corresponding activation in the parameter functions and equipment of an automatic or pneumatic collet chuck.



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

*Rapid
operation with
buttons*

Function of buttons :

- Release takes the currently clamped tool and puts it down in the magazine
- The molette button closes or opens the molette. There is a security prompt prior to opening it
- 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..

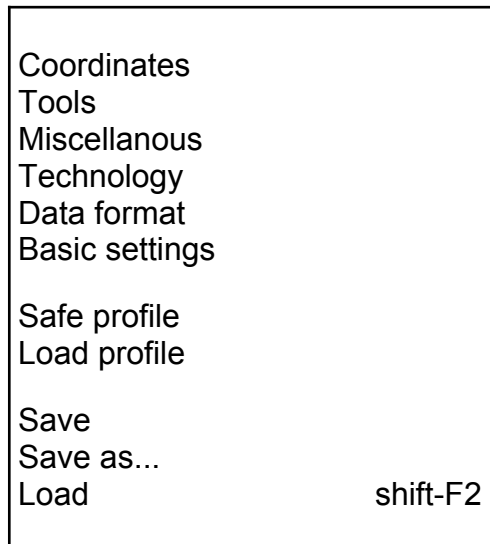
*Mode of
pplication*

Concerning length measuring and length compensation please proceed as follows...

1. Insert a tool – at best the one you want to use first.
2. Load the required job if is not already activated and visible in the graphics.
3. Information to **WinPC-NC** concerning the current active tool via the tool change dialog and ensuing manual measurement. For this please simply click the measurement button for the corresponding tool. The new tool numer is immediately displayed in the status bar and **WinPC-NC** moves the tool to the length sensor for automatic length measuring. The information concerning which tool and the corresponding length is now available.
4. The next step is loading the job and defining the zero point by inserted tool, especially defining the Z zero point. This can be effected as usual, e. g. by scratching the surface
5. Finally please start the job. **WinPC-NC** asks for a change with each new tool and then meaures the tool length automatically. The exact length differences to the reference tool are calculated and respected with subsequent tools concerning the Z height.

3.3.3. PARAMETER-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.

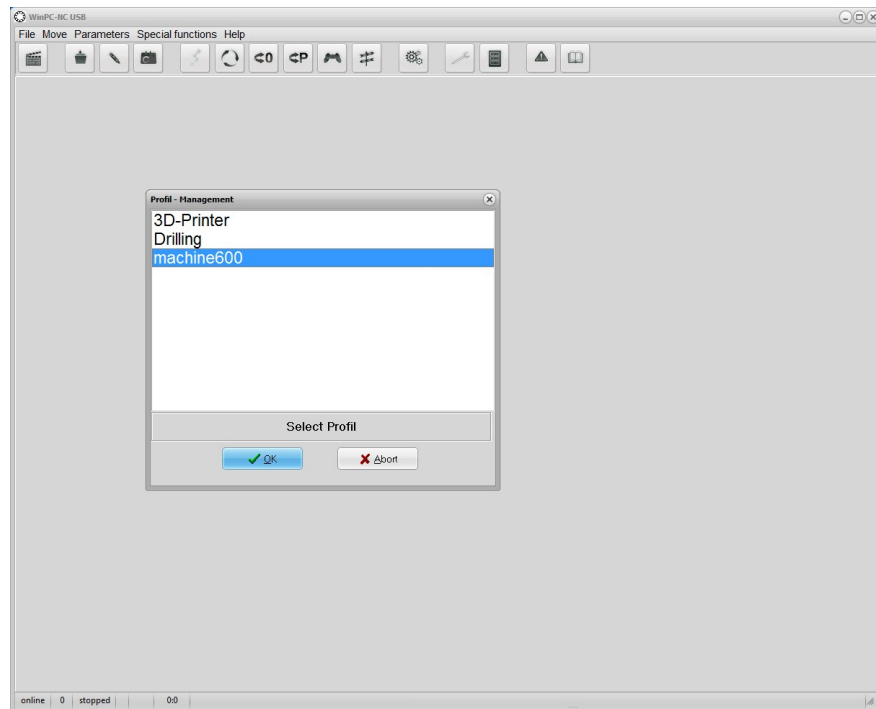
Parameters-SAVE/LOAD

Saving a complete profile

WinPC-NC offers a function to save and reload complete profile settings for different applications like milling, tangential cutting or 3D printing. The saved setup file includes all parameter settings of **WinPC-NC** itself and of the machine as well as defined macros and more settings.

Next to a system changeover all applications can easily be activated by selecting the corresponding setup file and the software can be properly configured.

Files with machine-set-up are marked with *.WMS and they comprise any settings concerning the machine, **WinPC-NC**, tool settings, macros, messages and additional characteristics.



PARAMETER-SAVE

*Saving
parameters
specifically
for a 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 for a project.

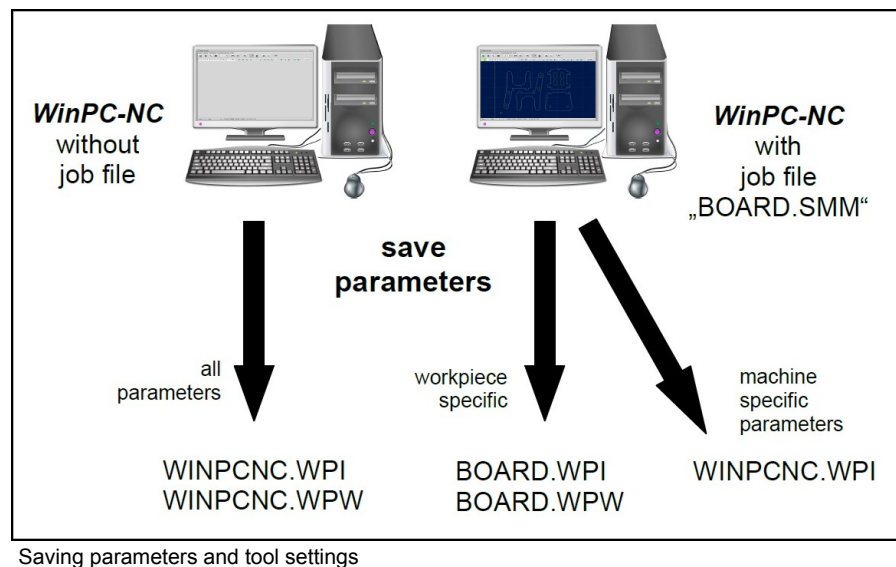
The SAVE function operates as follows : When a working file is loaded, **WinPC-NC** stores all workpiece-related settings such as the zero point, scaling, etc. in a parameter and tool file associated with the working file. The files have the same name as the working file, with the endings*.WPI and *.WPW,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

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

Display and selecting profile management

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.



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. In this way it is possible to produce parameter settings for various operating procedures or materials.

Please avoid an overwriting of the standard parameter file WINPCNC.WPI during saving the parameters by the function SAVE AS....

PARAMETERS-LOAD...

*Loading
specific
parameters
records*

The LOAD function enables you to load various specific parameters, 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.

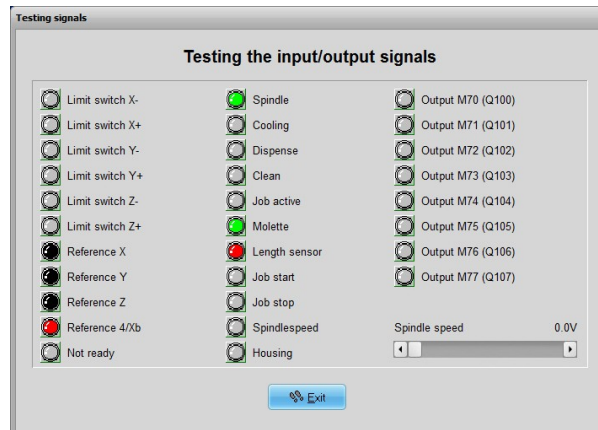
3.3.4. SPECIAL FUNCTIONS menu

The SPECIAL FUNCTIONS menu gives you access to two test functions which you can use to systematically check the mechanics 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
Status information
Joystick calibration
Check position
Factory settings
Teachin

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 have not already been defined or no input/output pins have been allocated.

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

Definiton of the switches and setup of the switching logic must be executed during installation by the signal wizzard under parameter-basic settings-signal wizzard.

The supported additional signals are tested by simply clicking the LED symbols. This switches the outputs on or off and display changes from blue to green.

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 0V and 10V and outputs this value binary coded at LPT2 port of **ncUSB** or as a PWM signal.

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

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 X/Y and Z-axis :

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.

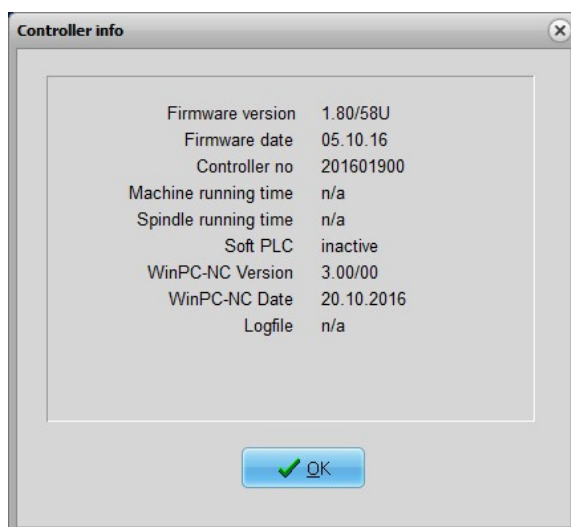
It is absolutely impossible to draw any conclusions from the ascertained values for one axes regarding the max. speed of the total plant system or regarding the possibly reachable speeds during the job performance.

SPECIAL FUNCTIONS STATUS INFORMATION

Information about the communication module

WinPC-NC USB can solely be used in conjunction with the external accessory module ncUSB which must be connected by the delivered cable. The different versions of the communication module and the WinPC-NC main program are displayed in the corresponding window.

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



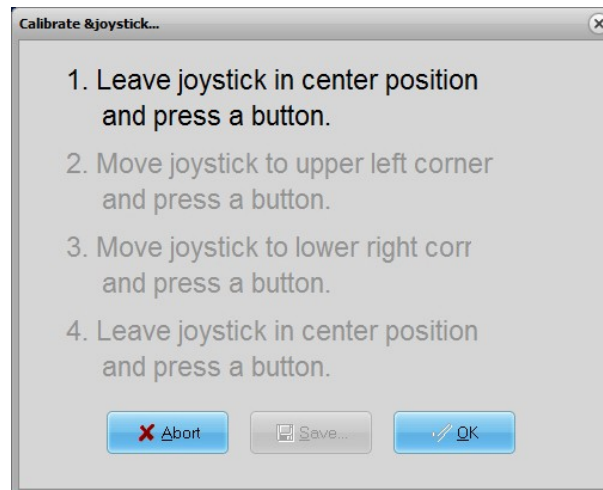
SPECIAL FUNCTIONS-JOYSTICK CALIBRATION

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.

Determination joystick parameters menu

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.

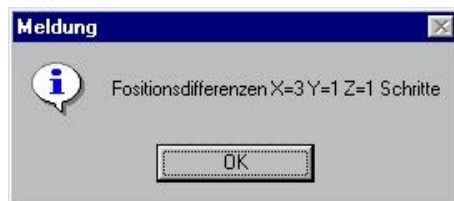
The joystick cannot be used before it has been calibrated.



Joystick calibration

SPECIAL FUNCTION CHECK POSITION

The CHECK POSITION function is another test function in **WinPCNC**. 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.

WinPC-NC can check and calculate exact positions and step differences only if the touched reference switch can be moved over to break down and decelerate in a controlled move. It is not possible if the axis sticks to the block or end of axis while braking.

Please make sure your reference switches have sufficient distance to the end block of each axis.

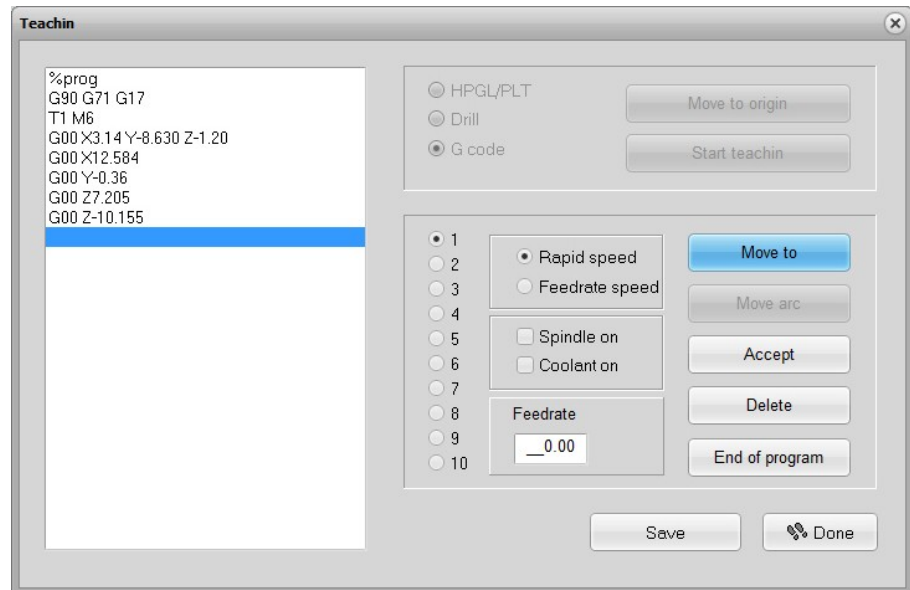
SPECIAL FUNCTIONS FACTORY SETTINGS

When getting **WinPC-NC** in combination with a machine it will have predefined settings according to your machine or you will get a selection menu at first startup to choose your machine type.

In case of a mistake in settings or selection you can restore all definitions to the state of delivery and reselect your type or begin to setup your personal machine and mechanics. It will be active with next program start.

SPECIAL FUNCTIONS TEACHIN

By means of the function TEACHIN simple programs in various formats can be created by moving and following the contours.



Special function Teachin

Creating programs interactive.

The new program is displayed in the teachin window on the left side of the dialog. In the top part actions can be activated prior to the teachin process.

The large window on the right-hand displays all available actions, e.g. selecting the tool, speed settings, operating the spindle and cooling and provides the possibility to produce various contour shapes.

Executing commands depends on the selected data format. Commands as switching the spindle or cooling are useless with the HPGL format.

Teachin step by step **Gradual procedure with teachin:**

1. Execute reference move and call up teachin function
2. Stipulate the desired data format and determine the zero point by moving
3. Start teachin process. Now the program head will be automatically set up and displayed in the program window
4. In the following steps you can create as many contour elements as necessary for the desired program either by moving the new line ends or by definition of circle points. Tool chan-

ges and unproductive movements to new start points and plunge positions are possible

5. Click on exit button and the program is automatically finished
6. The new created program should be stored before leaving the function.

Possible actions during the teachin process are as follows :

<i>Move line</i>	Moving to a new position. This moving step is either an operation in open curcuit with high speed or immersed with feed rate. Several lines can be teachd successively and the function is left if no other movement has taken place.
<i>Circle arc</i>	Teachin of a circle or arc is taking place with always three points. The first point is also the current position. First any point on the circle arc is being moved to and finally to the final point. By these three points a circle or arc command can be cleary created.
<i>Accept</i>	This function inserts the actual position of the cursor bar into the program.
<i>Delete</i>	Deletes the line of the cursor bar. With this function it is also possible to delete and correct previous commands.
<i>Exit</i>	Inserts the necessary commands for exiting the program and finishes the new created NC program.

New commands are always inserted at the actual cursor position and enables the user to make up for overlooked actions.

3.3.5. HELP-menu

There are three items in the help menu:

Help topics
Liability
About **WinPC-NC**

HELP-TOPICS

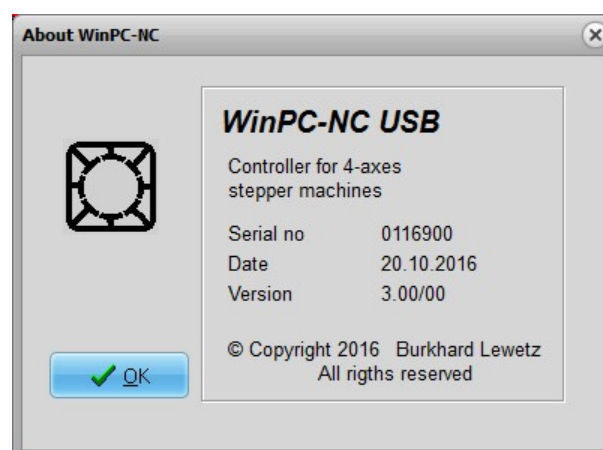
The HELP TOPICS function gives you access to the help system. A dialog box appears containing the main selection.

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

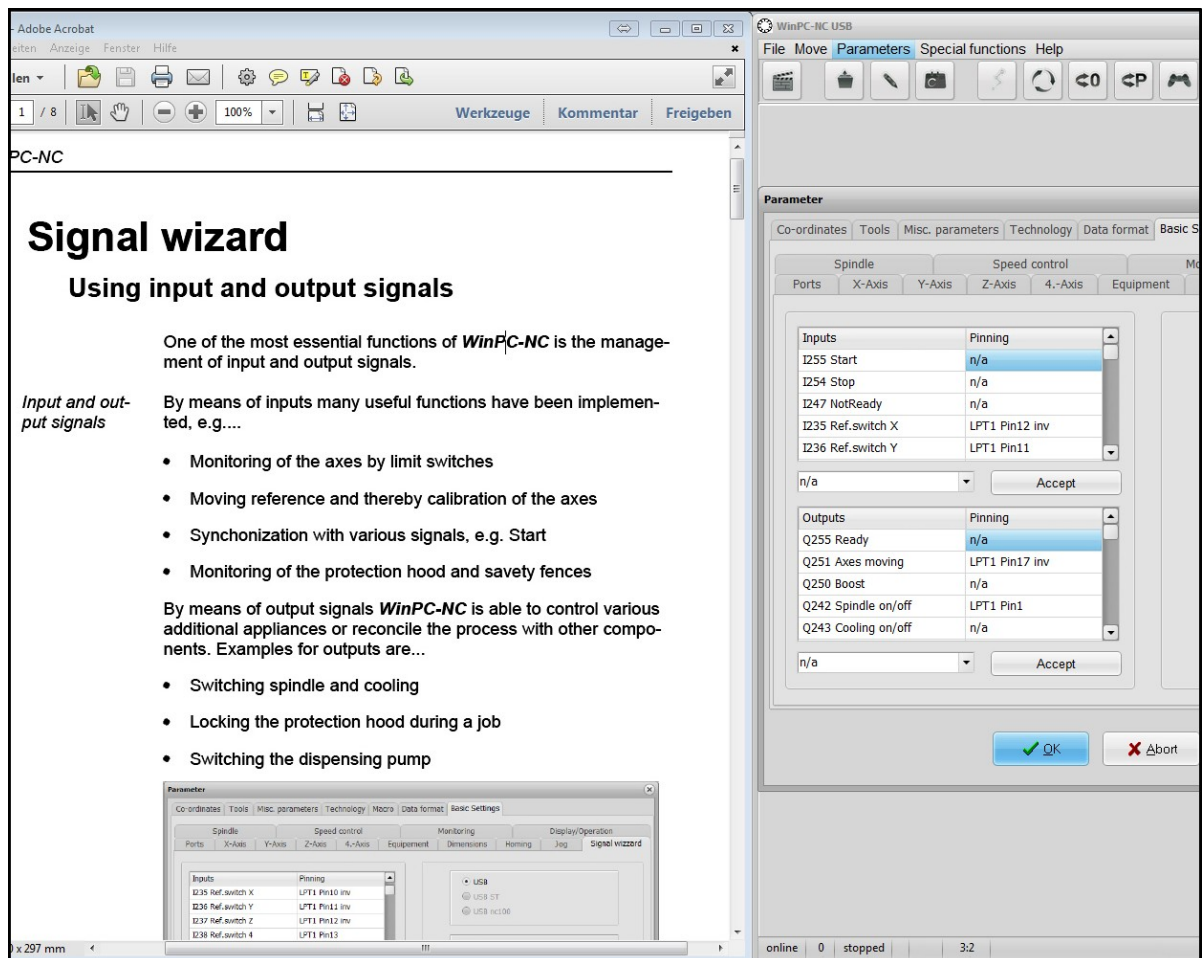
Activating this menu function displays information about the current version and revision number.



Information about the current version

Independently of this there is always the possibility to call up a context-related help by function key **[F1]**, which gives information concerning the actual opened parameters or the possible applicable actions in the present situation.

The ONLINE-Help is provided in pdf format and thus requires the installation of a pdf reader. We recommend to use the freeware Acrobat-Reader or other similar programs as Foxit-Reader or Sumatra-Reader.



Open ONLINE-Help for parameters

4. 2D-CAM functions

4.1. Overview

Concerning 2D data **WinPC-NC** provides special functions for sorting and preparing data for subsequent processing. Following functions are selectable and adjustable in a separate input dialog area:

- Assigning tools according to use
- Cleaning up data, deleting zero vector and double lines
- Setting output sequence according to tool number
- Optimizing empty moves
- Tool diameter offset compensation in contours

*Relates to any
2D data*

The CAM functions are applicable to any imported 2D data, - i. e. to all files in the formats HPGL, EPS/AI, DXF (2D) and drilling formats.



The CAM functions are activated by a click to the 2D-CAM button after loading the job and the dialog box is opened without delay and provides all available setting options and functions

4.2. Setting options

Available functions and settings:

Cleaning up data

All zero length vectors and double lines are deleted and removed from the drawing. Usually these modifications are not visible. However, they may result in failures concerning the subsequent calculations.

Scaling data

All data and lines are scaled in size prior to the subsequent calculation. This is an essential feature for using the CAM functions as a subsequent consideration of the tool radius for the required target size must always be observed. If scaling is made afterwards via **WinPC-NC** parameter settings, the tool radius offset is also scaled.

Different settings can be made for the X and Y axes.

Searching contours/lines

Activating this function means that **WinPC-NC** tries to create closed contours or continuous extended lines out of many individual lines and therefore the individual vectors are resorted and linked.

During the drawing process it may happen that consecutive lines do not exactly match to each other and as a result there are small spacings or contour crossovers. This kind of inexactness can be eliminated by the definition of a catching grid. The sorting function always tries to consider at first all the lines showing equal initial coordinates and end coordinates. Please note, that the fuzzy search via catching grid a tolerance is only made if no exact follow-on line is found.

By using an additional button the CAM function is induced to find and mark enclosed lines and contours, i. e. elements which are completely surrounded by other contours. This is an essential feature for a radius correction later on.

Calculating tool diameter offsets

For compensating diameters or radii of used tools in path contours it is possible to compute a radius correction for closed contours and thus the actual paths can be offset by a special set distance to the inside or outside.

The radii values are set in the next dialog box. By using an additional button it is possible to assume the grey-coloured original lines and tool numbers exceeding 100 for the new graphics data output. Thus the visual control is simplified and former lines are not considered while job processing.

Optimizing empty moves

Prior to the output of the calculated data in a new NC file, **WinPCNC** tries to optimize or minimize the empty moves between lines and contours. This saves processing time and helps to increase the plant's efficiency.

First of all the surrounded elements are considered and thereafter the corresponding contours. Thus it can be avoided that workpie-

ces are machined that have previously already been completely milled.

	New tool no	Radius	Inside	Climb mill	Sequence
1	_1	_1.500	<input type="checkbox"/>	<input type="checkbox"/>	_1
2	_2	_1.000	<input checked="" type="checkbox"/>	<input type="checkbox"/>	_2
3	_3	_2.000	<input type="checkbox"/>	<input type="checkbox"/>	_3
4	_4	_0.000	<input type="checkbox"/>	<input type="checkbox"/>	_4
5	_5	_0.000	<input type="checkbox"/>	<input type="checkbox"/>	_5
6	_6	_0.000	<input type="checkbox"/>	<input type="checkbox"/>	_6
7	_7	_0.000	<input type="checkbox"/>	<input type="checkbox"/>	_7
8	_8	_0.000	<input type="checkbox"/>	<input type="checkbox"/>	_8
9	_9	_0.000	<input type="checkbox"/>	<input type="checkbox"/>	_9
10	10	_0.000	<input type="checkbox"/>	<input type="checkbox"/>	10

Second dialog of CAM functions

New tool number

These input dialog boxes are responsible for resorting or summarizing existing tool or pen tool settings prior to any other functions. Sometimes it is advisable to summarize several drawing components for a tool which is to be machined for more effective processing later on.

Radius

The radius of all the used tools is defined for the radius compensation. The path which is to be recalculated is accordingly placed with contours to the inside or outside.

Inside

This button determines the direction of the relocated new path. The line offset is made to the inside if you have clicked to Inside, otherwise the line offset is made to the outside.

Climp milling

Travel direction of the milling tool along the contour is defined by climb milling and up-cut milling.

Sequence of operatione

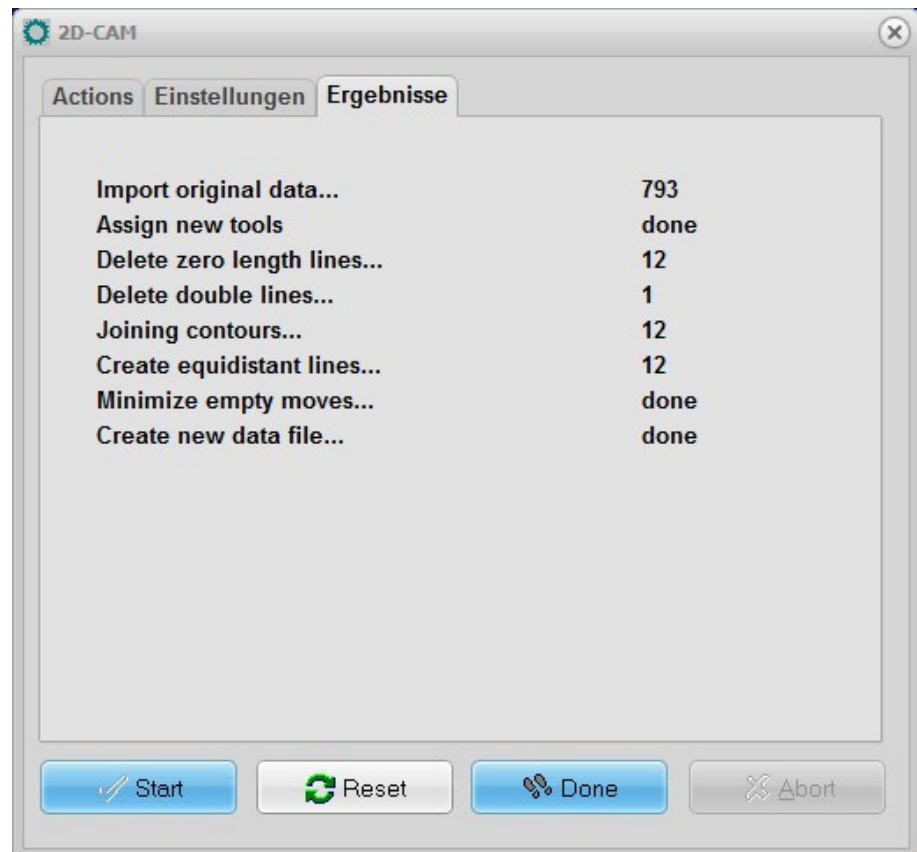
Finally the tool sequence is determined for creating the new file. All elements of this tool are always jointed and possibly an empty move optimization is carried out.

*Newly created
file
is immediately
loaded*

After completion of all required calculations and resortings **WinPCNC** creates a new 2D file in an intern used format and gives the preceding name of the project and the ending *.OPT.

During the process or after intermediate steps it is possible to make a reset to the original file and the original display and to execute repetitions with changed values and functions. For these procedures the buttons START, RESET, DONE and ABORT are applicable.

Using the START button means commencing the calculations and all activated functions are carried out according to the required subsequence. The progress bar indicating the actual state and provisional results is displayed on the result sheet of the dialog box. Cancellation can always be effected by clicking to the corresponding button.



Display of results

Having successfully calculated and generated the new file, the contours are immediately visible in the graphics display and the corresponding result can be checked. Subsequently it is possible to start a recalculation with modified settings or activate EXIT for adopting the results by using the buttons RESET and DONE.

Functions as required and in any possible combination

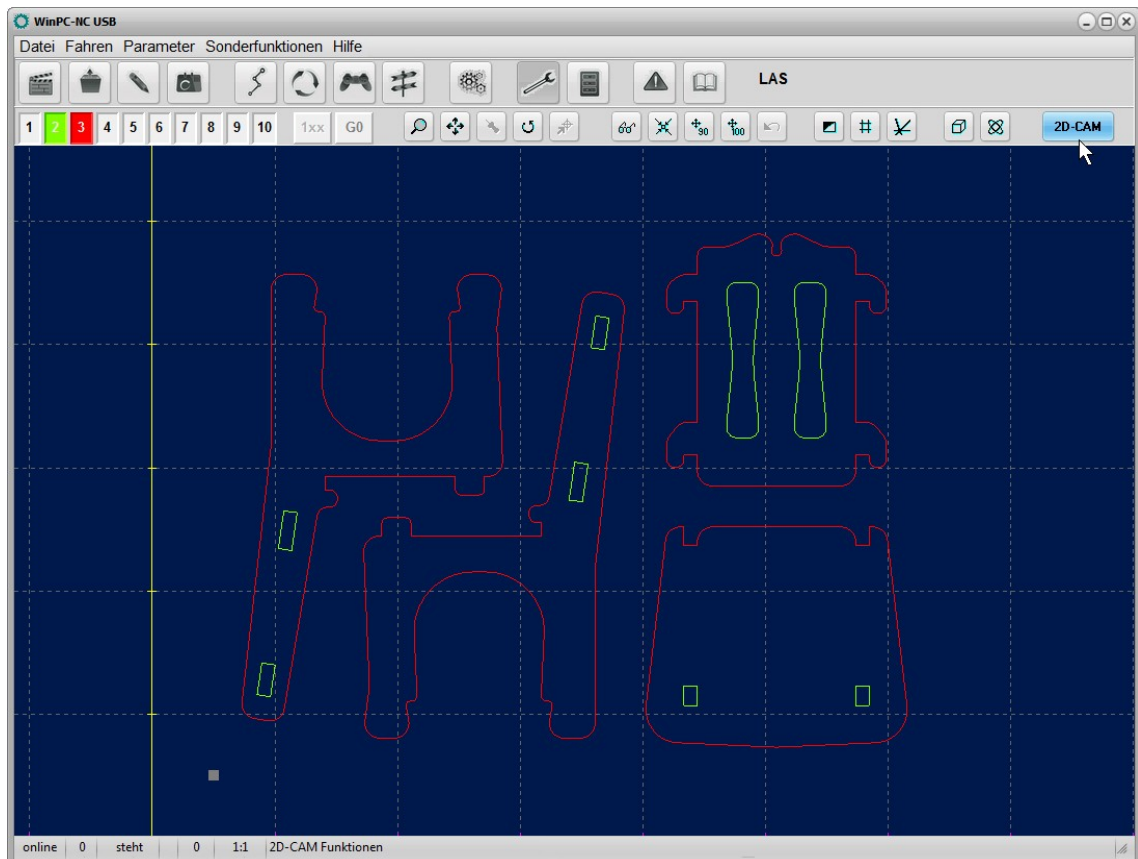
Due to the activation of individual functions it is possible to use any kind of compilation and combination according to data file and requirement.

For example concerning the drilling data of a board you have just to start the empty move optimization. In order to achieve a better surface result with millings and engravings you have to clean up the data and join contours or lines. Alternatively you just modify the processing sequence.

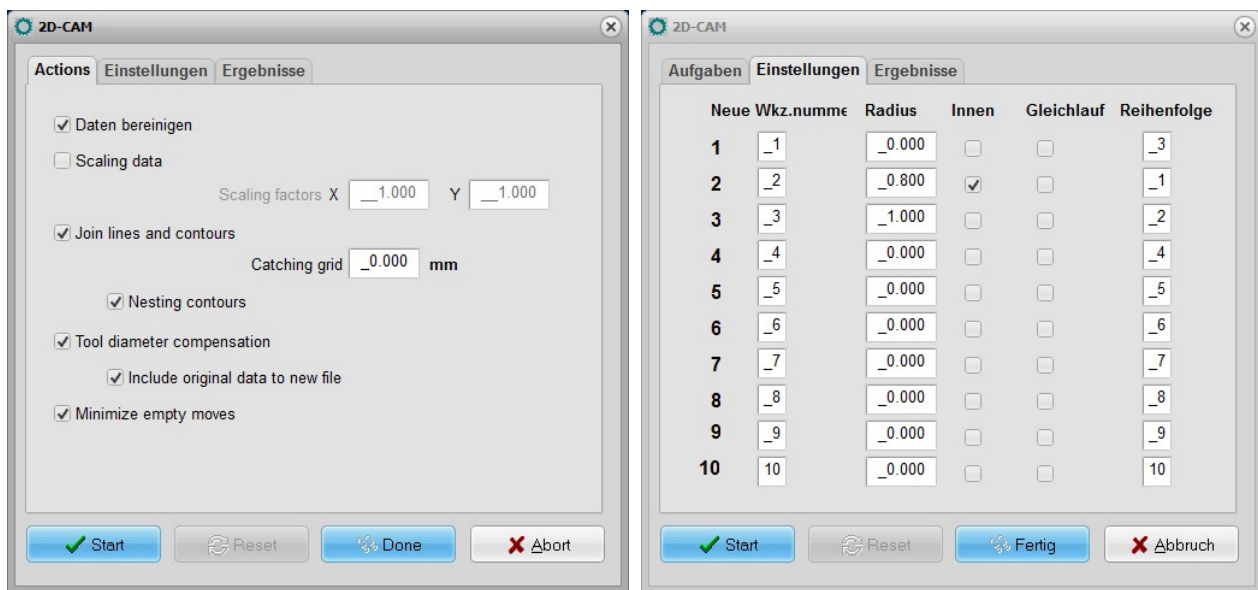
4.3. Example 1

For an efficient use of the most important functions we provide a detailed and step-by-step description on the base of examples.

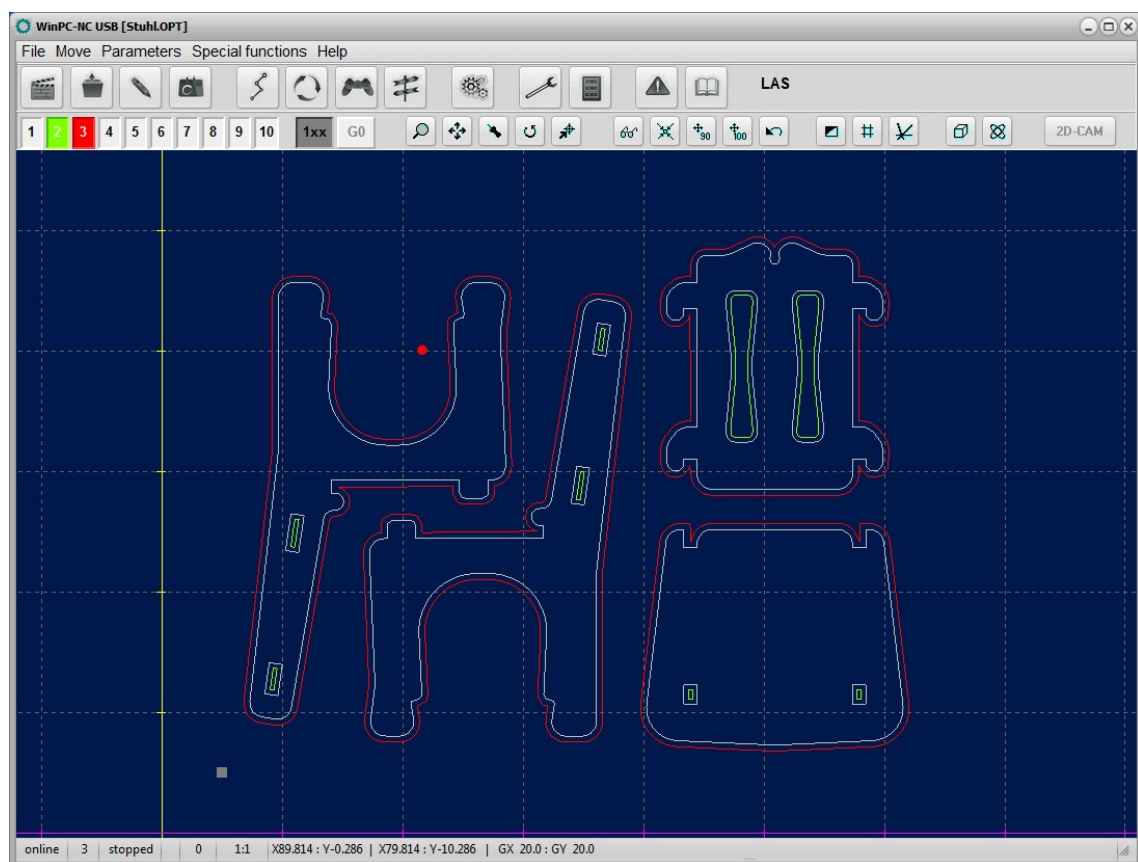
1. We are loading the chair in HPGL format and obtain the following picture



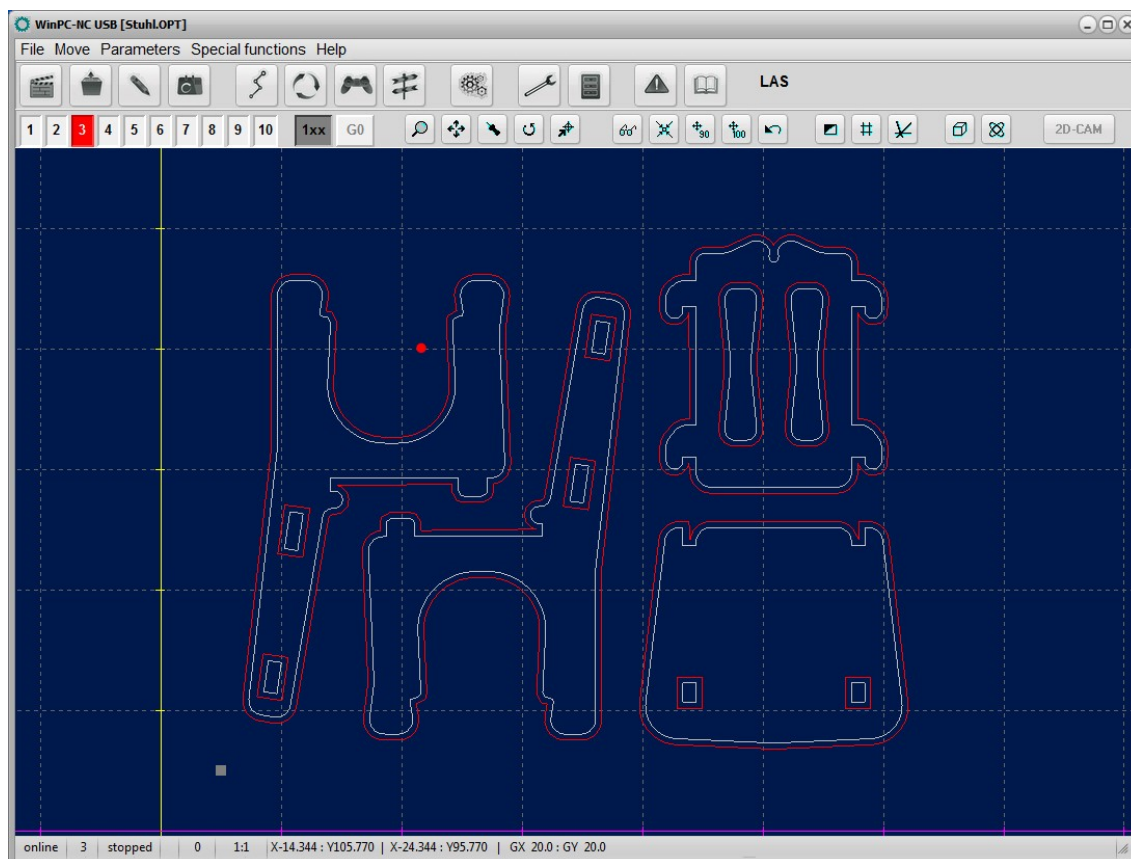
2. The contours consist of two colours and are to be milled with two tools. The inside contours are offset to the inside with a 1,6 mm milling tool and the outside contours with a 2mm milling tool. Settings are as follows:



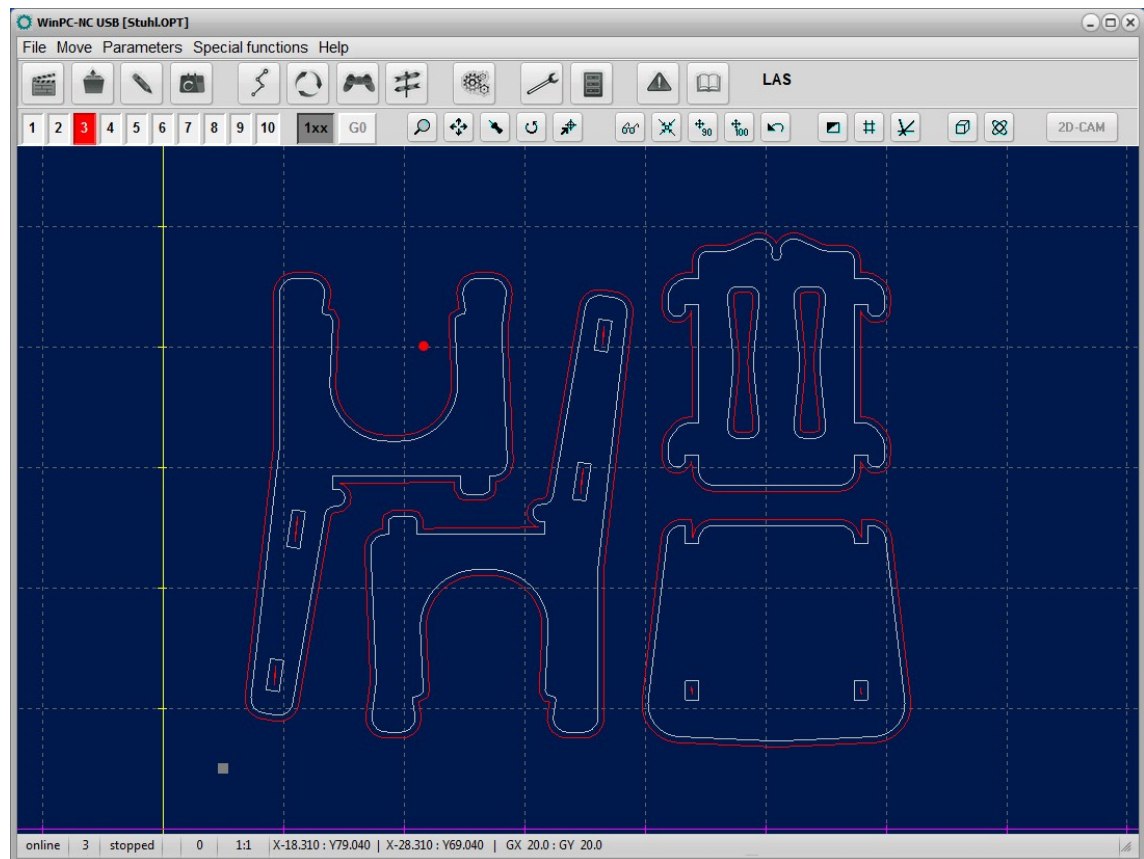
3. We just define the values for tool 2+3, start a search for contours, calculate radii compensation and optimize empty move and the result is...



4. If we want to mill workpieces by solely one tool and do not intend to carry out a tool change, calculation has to be renewed. The function is reset, change tool assignment from tool 2+3 to the new tool 3 and start a new calculation for all contours with radius.



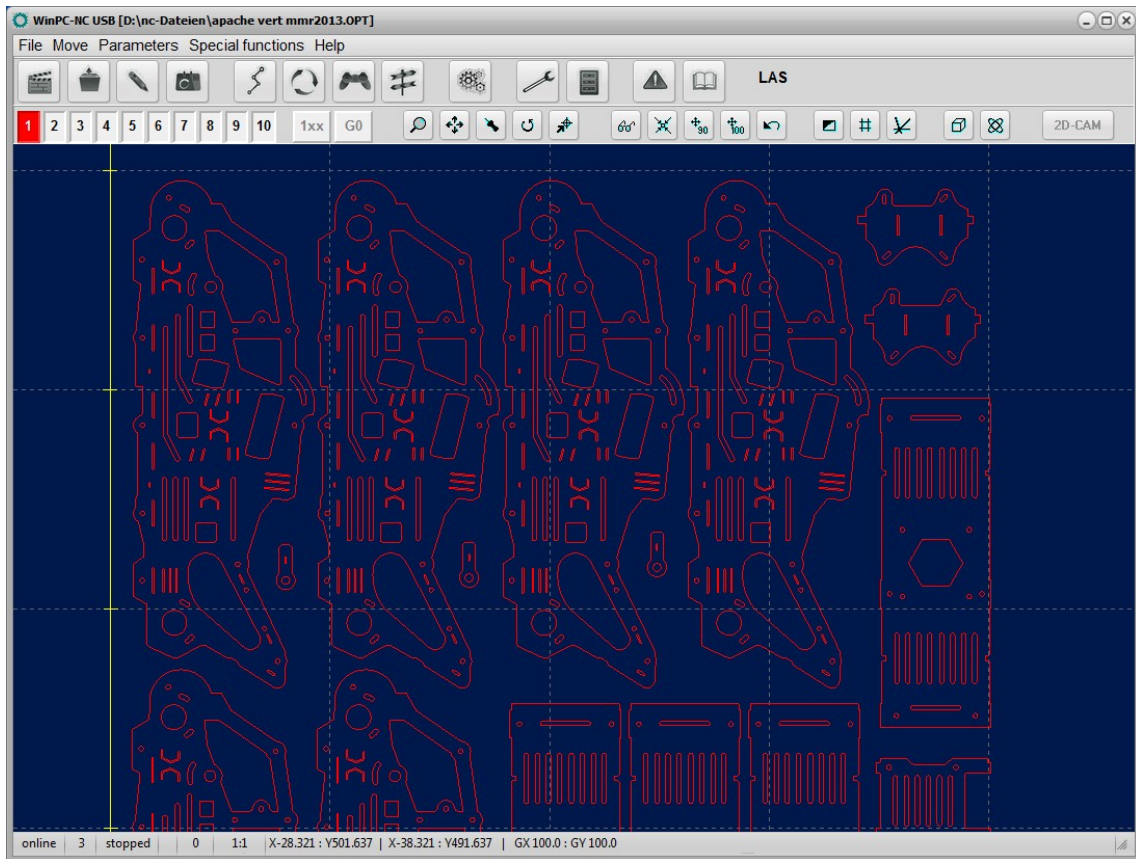
5. However, the offset of the recalculated path of the inside contour is inexact and all found contours are corrected towards to the outside. Now we still activate the button for the automatic search of surrounded contours and start calculation for the last time.



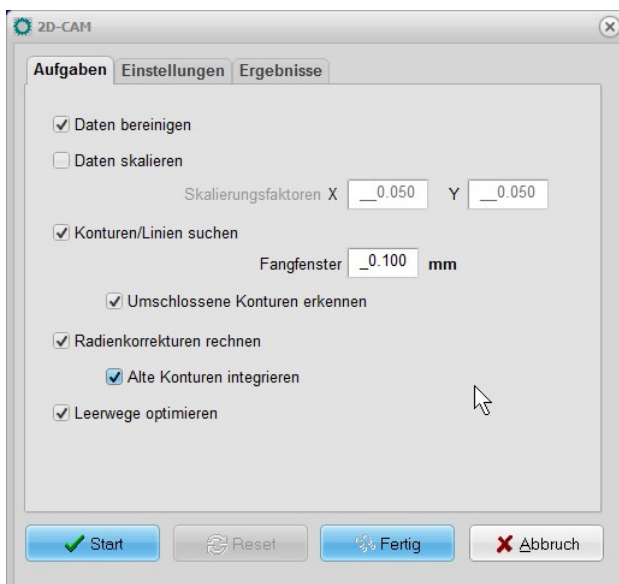
6. Now, the result fulfills our requirements and expectations, sequences are correct and all workpieces can be machined with our 2mm milling tool.

4.4. Example 2

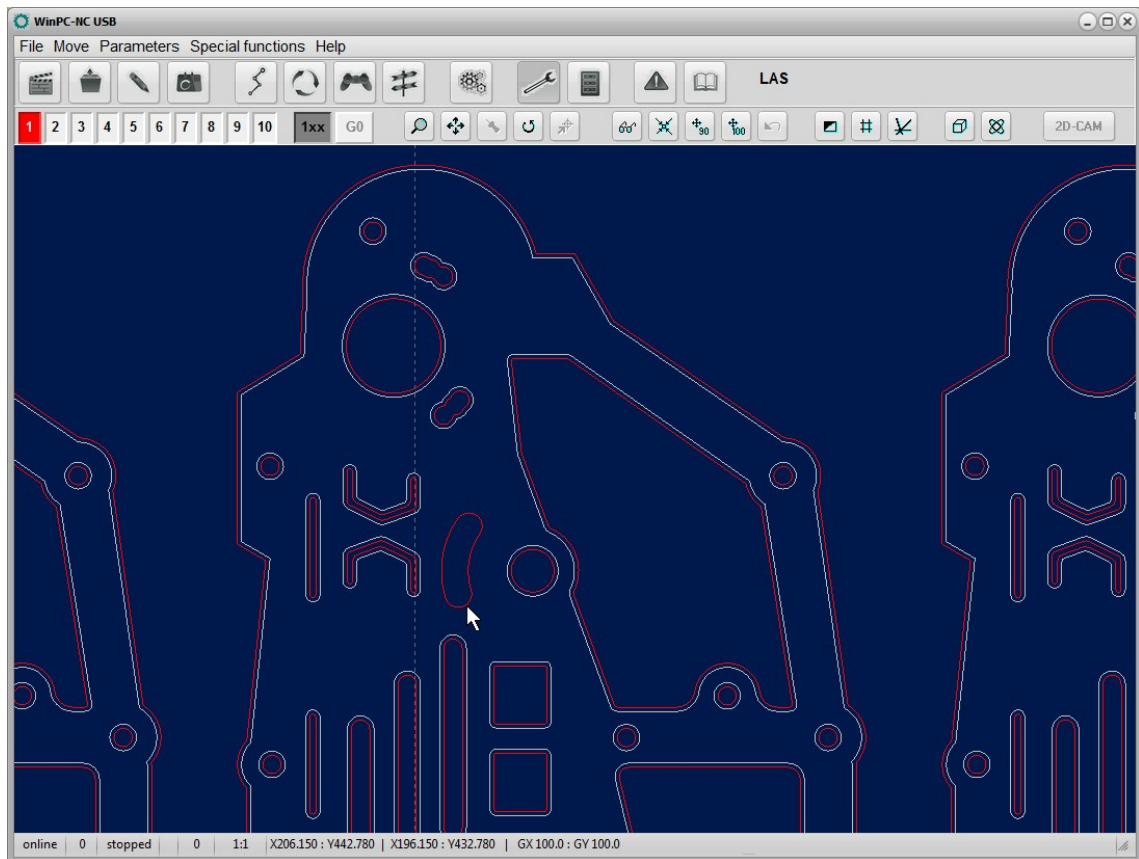
1. We are loading a more complex DXF-file and intend to mill all contours with a 1 mm milling tool and place the contours independently.



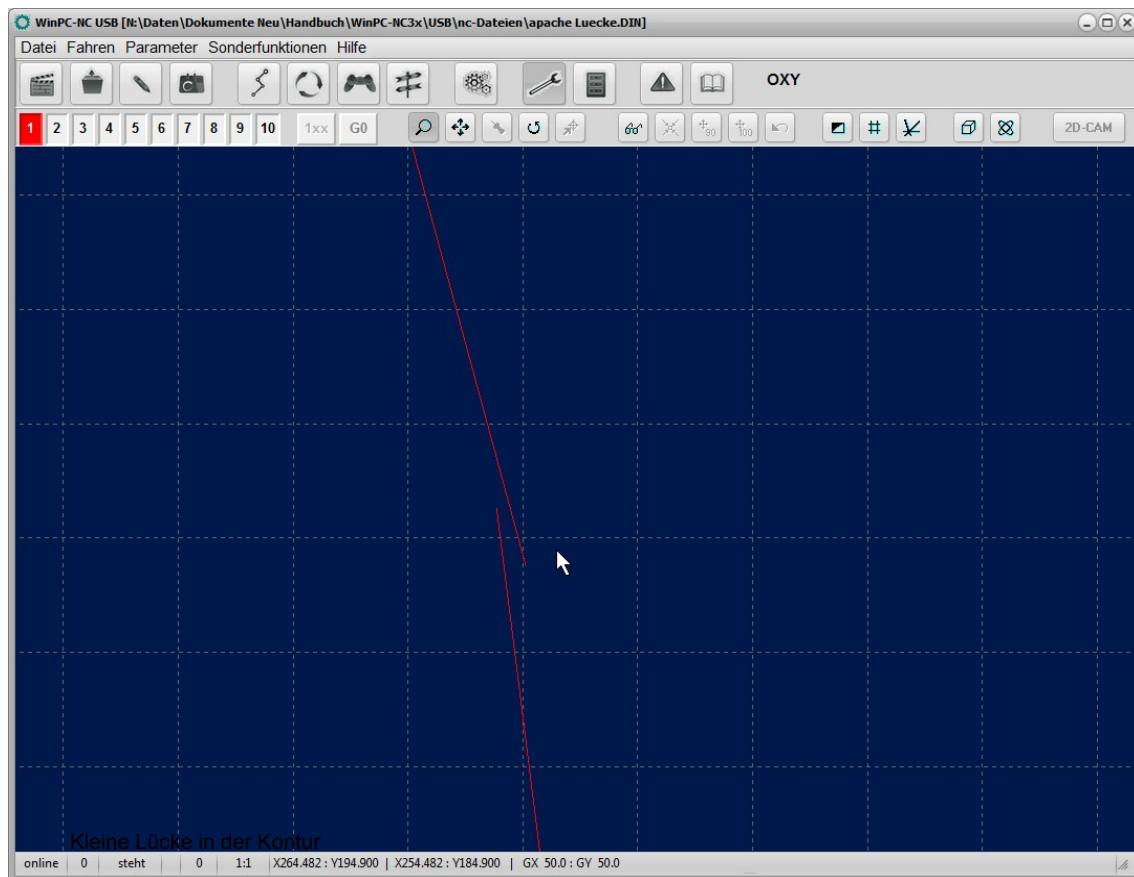
2. Settings in both dialogues are as follows:



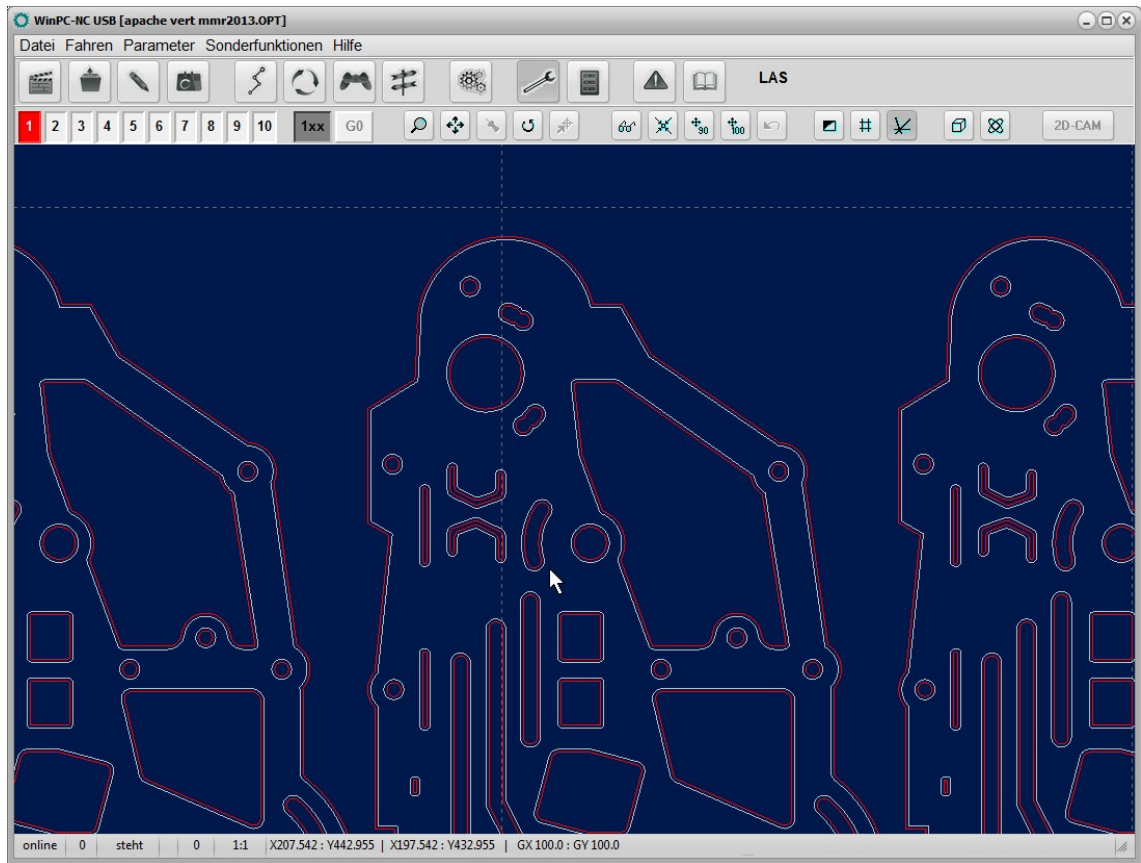
-
-
3. The result seems to be quite perfect and the surrounded contours as well as the offset paths to the inside and outside are clearly displayed. However, after zooming, it can be noticed that some contours have not been correctly recognized and calculated.



-
-
-
4. After zooming again, the incorrectness can be clarified. There is a spacing between two successive lines and it is bigger than the set catching grid.



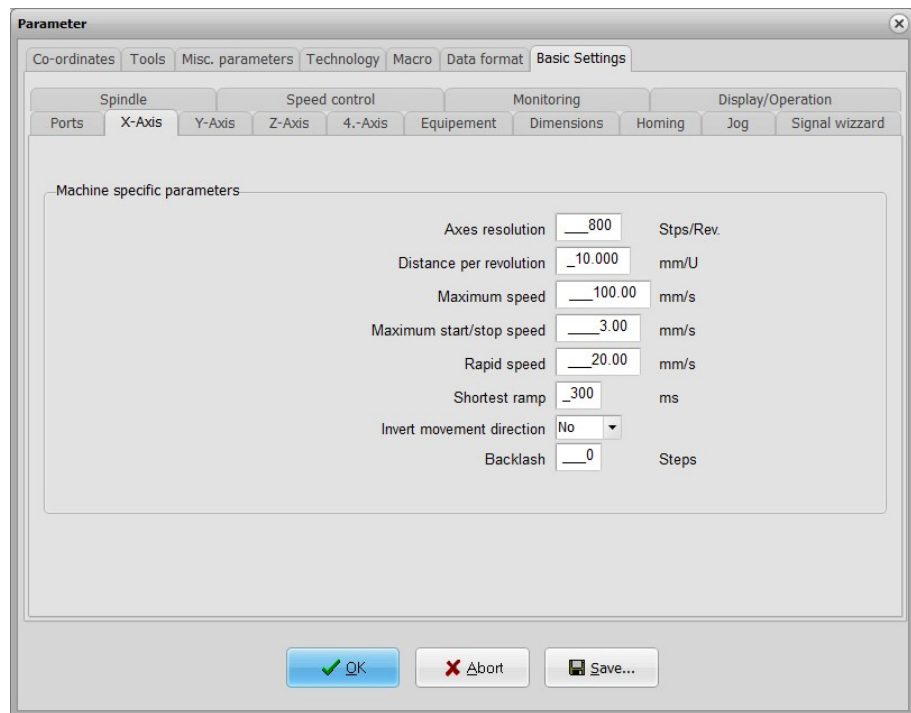
5. We extend the catching grid up to 0,2mm and start calculation again. Now we achieve a perfect result and milling can be started.



Perfect calculating of all contours

5. Parameter settings

5.1. Basic aspects



Parameter Basic settings

The parameter structure is sub-divided into two levels. The main levels comprise pages as coordinates, tools and technologies and describe the higher-level group.

Some groups are divided in additional sections and display additional windows concerning settings which are presented in sub-categories.

If the software is delivered in combination with a machine, in most cases the basic settings are properly pre-defined and chapter 5.2 can be skipped.

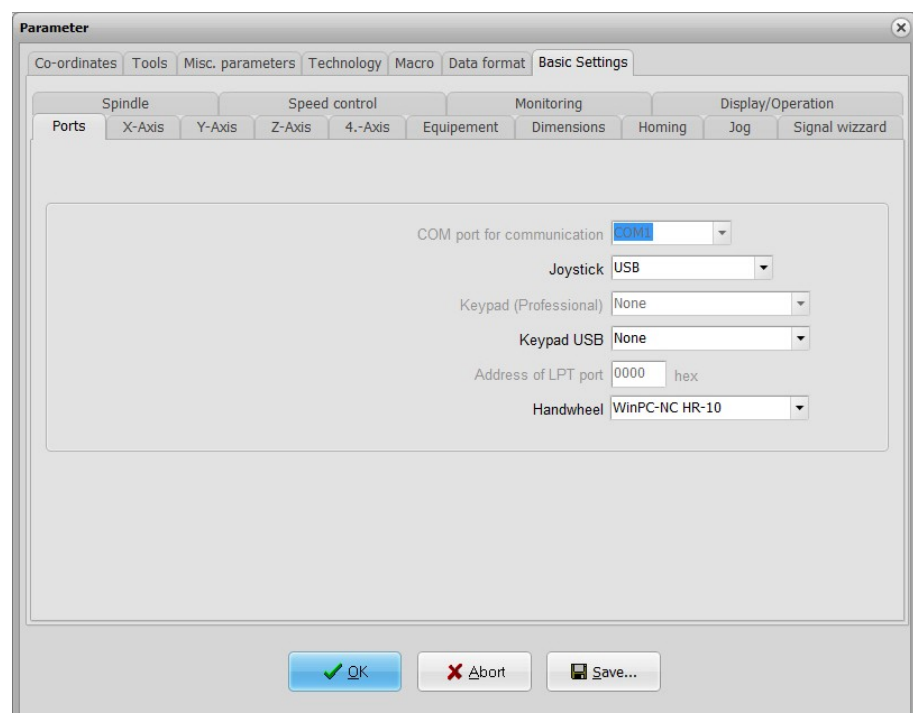
5.2. Basic settings

Usually settings are to be made once only during the setup procedure in basic settings and they comprise characteristics concerning the electronic and mechanic machine system. Preferred display properties can be set in this category.

In this category all basic settings are combined and usually they are not modified later on.

Basic settings can be called up by clicking on to parameters in the menu, as this category is usually not shown in the display.

5.2.1. Basic settings-Interfaces



Parameter basic settings interfaces

WinPC-NC does not require additional information about relevant interfaces except for the definition concerning joystick, keypad and handwheel. The used USB port is automatically determined and communication to **ncUSB** is established.

Joystick

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

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

A connected Space Mouse of 3DConnexion must be selected by different setting and all needed drivers must be installed prior to first use.

Keypad (USB)

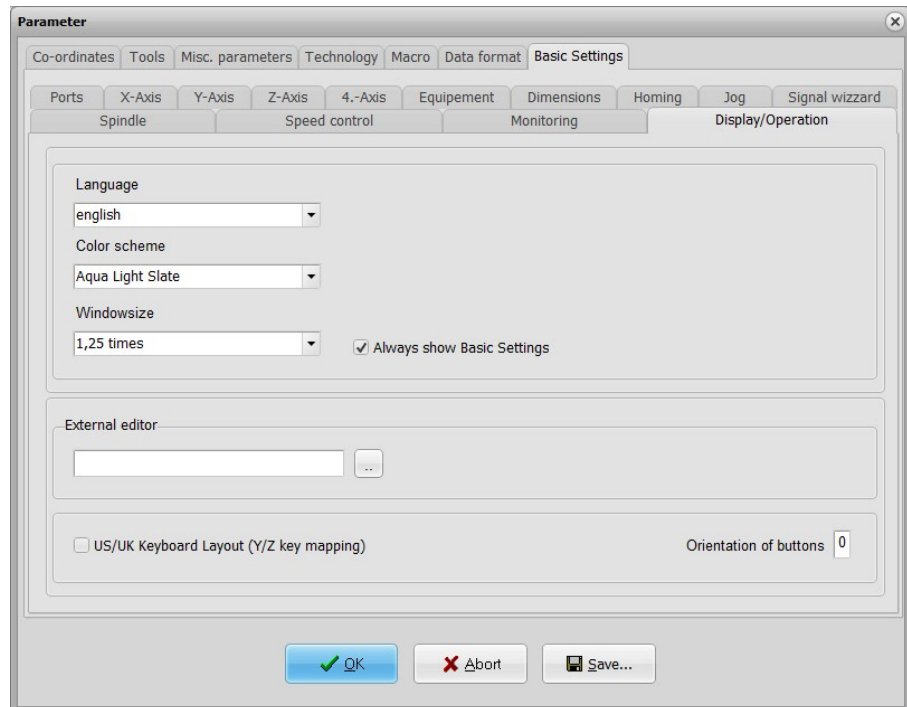
A remote control for **WinPC-NC USB** using numerical keypads is possible and can be defined with this setting. There are predefined OEM types available or a regular USB type keypad.

Handwheel

By this setting the handwheel can be activated. Handwheels are available for any versions of **WinPC-NC**. Please note, that **WinPC-NC** supports only specific handwheels



5.2.2. Basic settings Display/Operation



Parameter Basic settings Display / Operation

Language

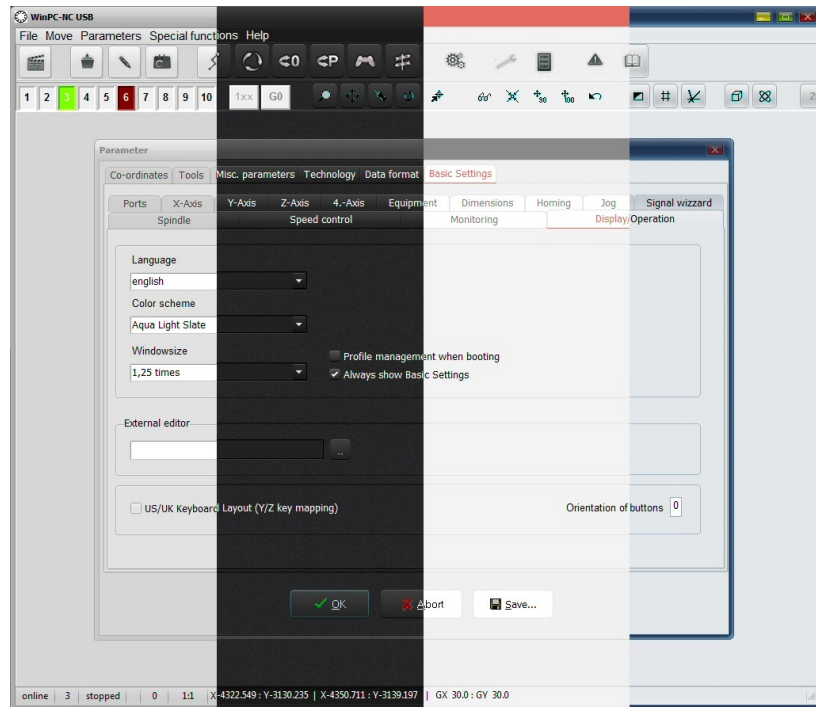
WinPC-NC is multilingual. The standard version already includes a few languages, and additional languages can easily be bolted on if required. The available languages are listed in a menu t.

According to the status of January 2016 following languages are available or will become available shortly:

germian	dutsch	croatian
English	czec	chinese (Taiwan)
french	mazedonian	chinesisch (traditional)
spanish	slovenian	serbian
polish	italian	turkish
hungarian	portoguese	bosnian
greek		

The language changeover takes place as soon as you select a new language and click Save. Some text phrases are available with certain country drivers in Windows only.

Color scheme



WinPC-NC with four various styles

WinPC-NC has more different styles for the user surface. With this setting are 11 different styles selectable.

Windows size

In **WinPC-NC** font and window size can be adapted step by step by this parameter. Independently the main screen can be zoomed as required by mouse as it is standard in Windows.

External editor

WinPC-NC is equipped by a simple text editor to load, view and modify nc files. If you need a different more enhanced or a well know editor you can easily define the name and path of a new program in this setting.

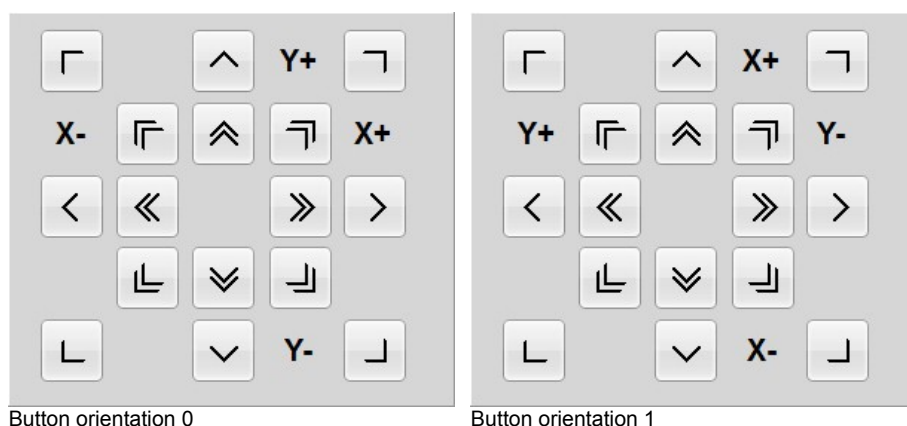
US keyboard layout (switch YZ keys)

In manual jogging you can easily move axes by pressing the (X) or (Y) or (Z) key and in combination with (Ctrl) key you even can move faster. As a difference between european and american keyboard layout the (Y) and (Z) keys are switches which makes it extremely difficult to move the machine. By checking this check-box **WinPC-NC** automatically corrects the switched key layout.

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 to take account of the machine orientation.



Position 2 and 3 are turning the X-Y-keys by 90 degree each:

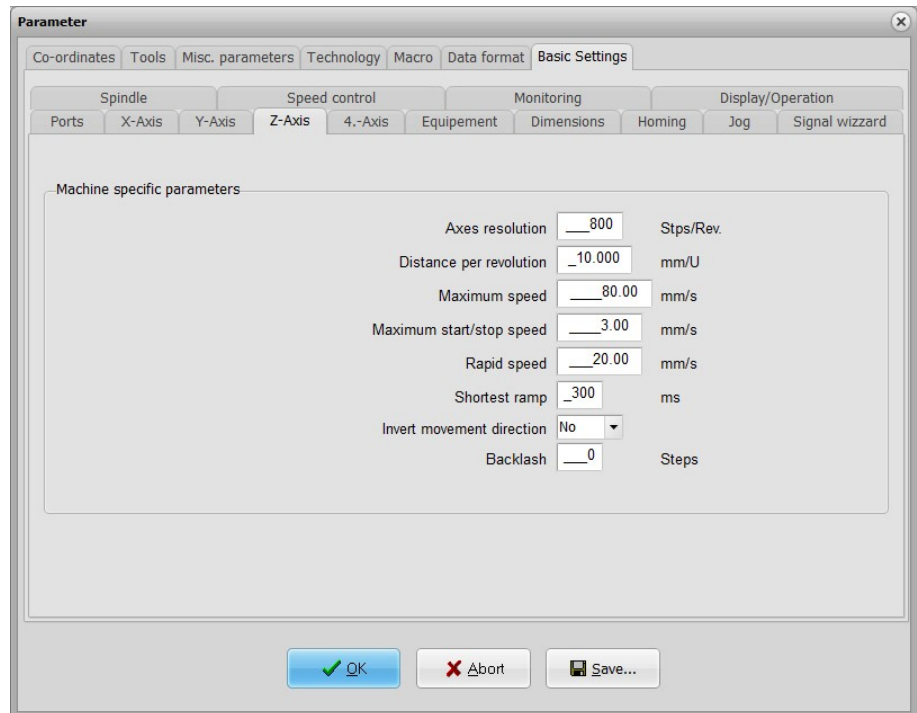
Constant display of basic settings

Basic settings are not hidden in the normal parameter dialog.

Profile selection when starting

Induces **WinPC-NC** to execute profile selecting each time when the system is started.

5.2.3. Basic settings- X-,Y-,Z-axis



Parameter Basic setting Z-axis

Axis resolution

The resolution boxes define the number of motorsteps 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 indicated in 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.

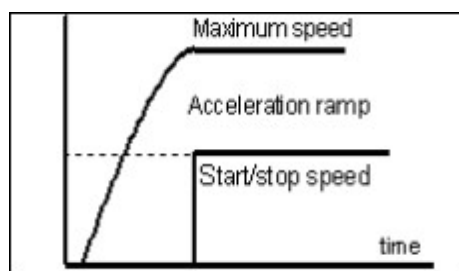
Maximum speed

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 millimeters with decimal places.

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/stopp speed and ramps

The optimum value can be ascertained with tests, e. g. by the function MOTOR TEST. Therefore it is necessary to switch off the ramp and accelerate the speed until step losses are caused. The value, detected in this way, should be reduced by 30% as a precaution.

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.

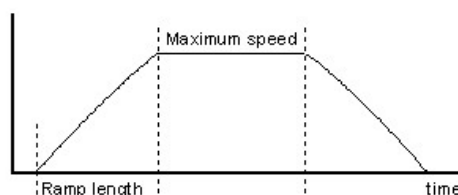
Shortest ramp

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 ramp length comes to a stop.

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



speed profile of a motor movement

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.

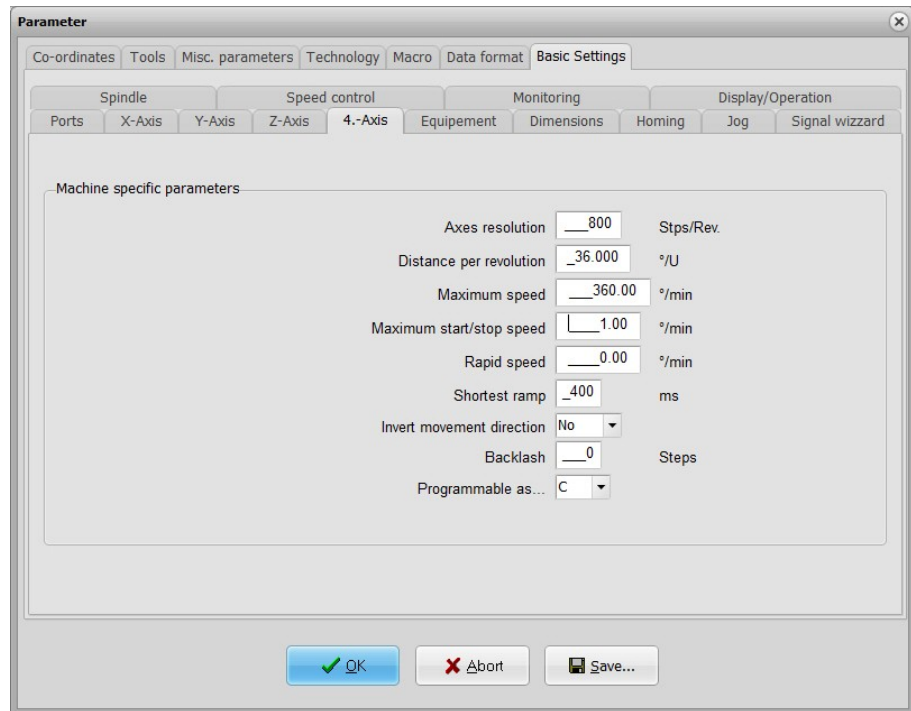
Backlash compensation

Drives which can not be adjusted absolutely close may cause insignificant differences during reversal of direction. These differences can be added up within the job process. This reversal drive can be compensated by these parameters.

Parameters for any axes are available in order to define the open circuit steps. The number of the motor steps are additionally indicated with any direction reversal.

Standard value means 0 steps and should remain unchanged if drives are closely adjusted.

5.2.4. Basic settings- 4.-axis



Parameter Basic settings 4.-axis

WinPC-NC USB is able to manage and process a 4th axis. Programming can be effected, for example, by a Gcode program. Another possibility is the automatic direction rotation of the tangential axis by **WinPC-NC**.

Prior to use the 4th axis has to be enabled. Otherwise the parameter settings are not available.

Some of the parameters as speeds, inverting the direction and reference switch are acting in the same way as for axes XYZ and not explained at this place.

Parameter settings and programming of parallel axis with letters U, V and W are made in mm and mm/sec. Settings of rotary axis with letters A, B and C are made in degree and degree/sec.

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.

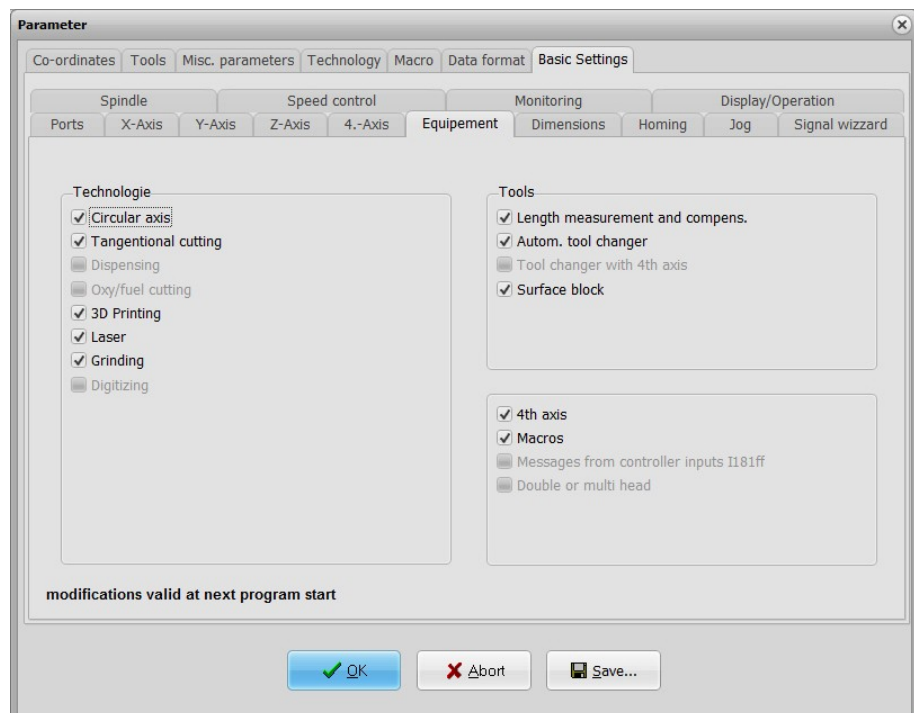
Programmable as...

Letter for programming

The 4th axis can be addressed in G code 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.

With 3D applications in most cases the 4th axis for filament feed is already programmed as E-axis.

5.2.5. Basic settings-Accessories



Parameter-Basic settings-Accessories

In this category all additional components are compiled which have been acquired in combination with the delivered machine and which you intend to use. You can select a 4th axis as well as various functions and applications.

Depending on the individual components and functions, additional parameters and setting options are available and correspondingly displayed. Some of the selected components are visible later on within the technology page. According to the specific requirements they can be activated.

Thus the machine can easily be changed over for specific tasks and the relevant technology can simply be selected, e. g. 3D printing or laser engraving. Further detailed settings can be performed.

Options with grey background are not applicable with version *WinPC-NC USB*, however they are conform to *WinPC-N-Professional*.

Tools

In this area it is possible to activate the functions for automatic length measurement and length compensation and automatic tool changer. Subsequent to this procedure a sub-level in the main level is activated. This applies to both options.

Via sensor block the Z zero point can be automatically determined with no-contact measurement.

Actuating the 4th axis means an activation or deactivation of all functions in ***WinPC-NC USB*** requiring a 4th axis.

In this area you can also activate the function for using macros. They assist in executing uncomplicated, user-defined processes and movements in various situations.

5.2.6. Basic settings -Size and dimension

The screenshot shows the 'Parameter' dialog box with the 'Basic Settings' tab selected. Under the 'Dimensions' sub-tab, the 'Machine specific parameters' section is visible. It contains a table for defining machine size limits in mm:

	X	Y	Z	
Machine size from	+30.00	+20.00	+0.00	mm
to	+250.00	+140.00	+100.00	mm

Below the table, the 'Machine area monitoring' checkbox is checked. At the bottom of the dialog are three buttons: 'OK' (with a green checkmark), 'Abort' (with a red X), and 'Save...' (with a floppy disk icon).

Parameter-Basic settings-Size/Dimension

Machine bed dimensions and area monitoring

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.

A parameter activates or deactivates monitoring of this limits.

5.2.7. Basic settings-Homing

Parameter

Co-ordinates Tools Misc. parameters Technology Macro Data format Basic Settings

Spindle Speed control Monitoring Display/Operation

Ports X-Axis Y-Axis Z-Axis 4.-Axis Equipment Dimensions Homing Jog Signal wizard

	X	Y	Z	4
Reference switch at...end	neg.	pos.	neg.	pos.
Reference point is...	+ 0.00	+ 0.00	+ 0.00	+ 0.00
			mm	°
Homing offset	+ 0.00	+ 0.00	+ 0.00	+ 1.00
			mm	°
Reference speed, search	20.00	20.00	20.00	15.00
			mm/s	°/min
Ref. speed, moving free	1.00	1.00	1.00	8.00
			mm/s	°/min

Reference sequence Z-X-Y Homing 4th axis... last

☒ Check ref. switches at reference start

OK Abort Save...

Parameter-Basic settings-Homing

Reference switch at ... End

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 Position

During homing the axis positions are normally zeroed on the reference switches, i.e. that the machine zero point is set by the reference switches.

Normally the reference switches are installed on the negative end of the axes. However, with some systems it is advantageous to attach them on the opposite side.

Reference position freely definable

By the parameter reference position WinPC-NC can easily be instructed to set this specific position on the reference switches and thus defining the desired position of the machine zero point.

Reference offset

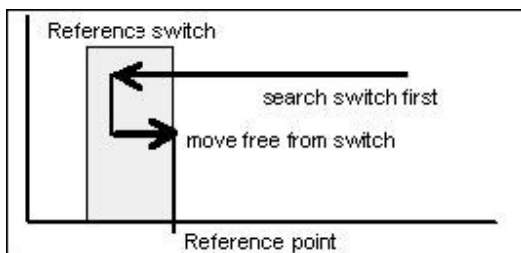
*Safe moving
free of the
reference
switch*

To avoid a stop directly at the reference switch's edge after a reference and moving free it is possible to determine an additional path with the reference offset. This has to be carried out after moving free from the switch. First after moving the offset the axis is resetted or the defined reference position is adjusted.

Typical values are 0.5 to 1mm additional distances to the switch.

Reference move, search Reference move, clear movement

Each axis starts moving with search rate 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 again.



The edge of the reference switch defines the 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.

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.

Homing sequence of 4th axis

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

Checking reference switches prior to homing

*Reference move
exclusive with
vacant switches*

Prior to a reference move the parameter induces **WinPC-NC** to check the switches. Only if each defined switch is vacant a reference move can take place.

This test makes sense if several reference switches are connected on the LPT port by a single input line and if it is not possible for **WinPC-NC** to recognize the axis currently placed on its switch.

In case if the reference move should be rejected due to one or several actuated switches, it is necessary to move the switches free by the function JOG.

5.2.8. Basic settings-JOG move

The screenshot shows the 'Parameter' dialog box with the 'Basic Settings' tab selected. The 'Jog' sub-tab is active, displaying manual speed settings for the X, Y, Z, and 4th axes. The 'Jogging, reset to endless' checkbox is checked, and the 'Changeover time' is set to 300 ms.

	X	Y	Z	4
Manual speed, fast	30.00	20.00	15.00	180.00
Manual speed, slow	3.00	1.00	1.00	5.00

Changeover time: 300 ms

Parameter-Basic settings JOG move

JOG move, fast / slow mode

Here, the speeds for JOG move can be set. Distinction is made between slow and fast mode.

Jogging, reset to endless

In jog move mode all axis can either be moved endlessly, i. e. until releasing the mouse key, cursor key or discrete distances which are selected by the dropdown menu.

The setting endless should be re-established automatically after each movement in order to avoid starting a pre-set distance inadvertently and to guarantee infinite move.

Changeover time

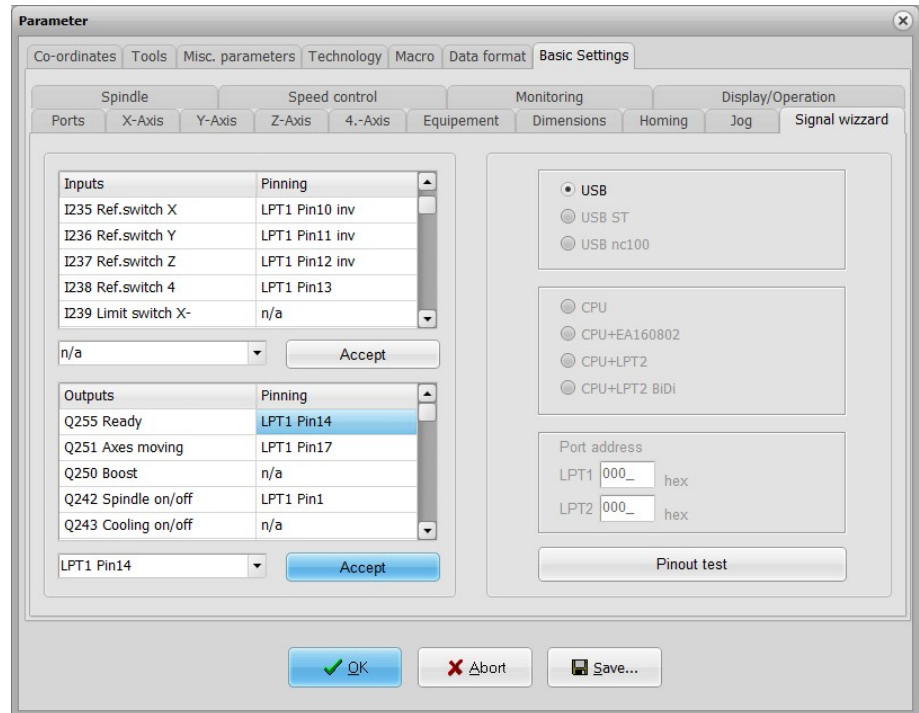
The changeover time defines the delay for the transition from single step to continuous movement in jog mode.

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 one key or button remains pressed for a longer time, that is the changeover time, the motor changes to continuous movement and is not stopped until the key or button is released.

5.2.9. Basic settings-signal wizard

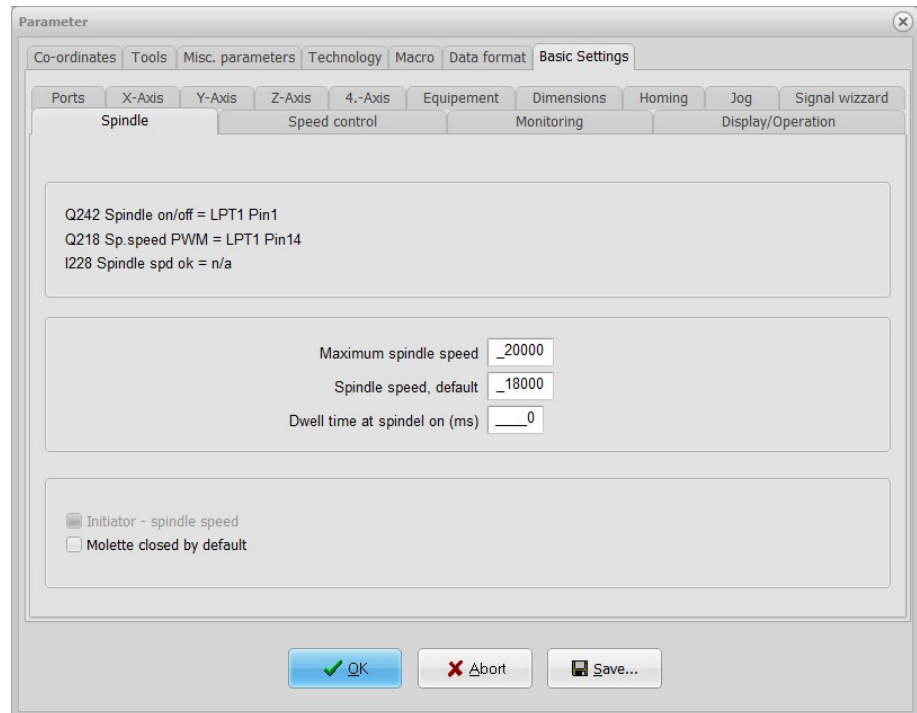
*Userfriendly
adjustment of
the signals*

By means of the signal wizard a userfriendly adjustment and definition of all inputs and outputs is ensured. As this task is of great importance, you will receive detailed information concerning possibilities in a separat chapter.



Parameter-Basic settings-Signal wizard

5.2.10. Basic settings-Spindle



Parameter-Basic settings Spindle

The upper section of the window indicates the currently assigned input and output signals. Thus it is easy to note whether the definitions have been made correctly.

WinPC-NC is able to drive the rotation speed of a drilling or milling spindle. It is possible to define a speed for each tool or the speed data is taken from the NC files.

Maximum spindle speed

*Speed control
in 256 steps*

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

The maximum value defines the reference for step 255. All lower values are calculated proportionately between 0 and this value, and are output.

The spindle speed is either generated binary coded from **WinPC-NC USB** on the data bytes (pin 2-9) of the second LPT port or as PWM value on pin 14 or pin 17 of LPT1 on **ncUSB**.

Spindle speed standard

The standard spindle speed is used whenever no divergent setting as been made in the NC program or in the tool parameters. In most cases this applies to JOG move or when starting a job process.

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 output signal is activated.

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

With additional settings it can be determined whether a signal should be sent from the spindle speed control when the desired rotation speed has been reached. Many frequency converters and speed controllers provide a corresponding signal. This kind of monitoring is advisable subject to cabling and relevant pin layout.

Collet chuck closed at machine start

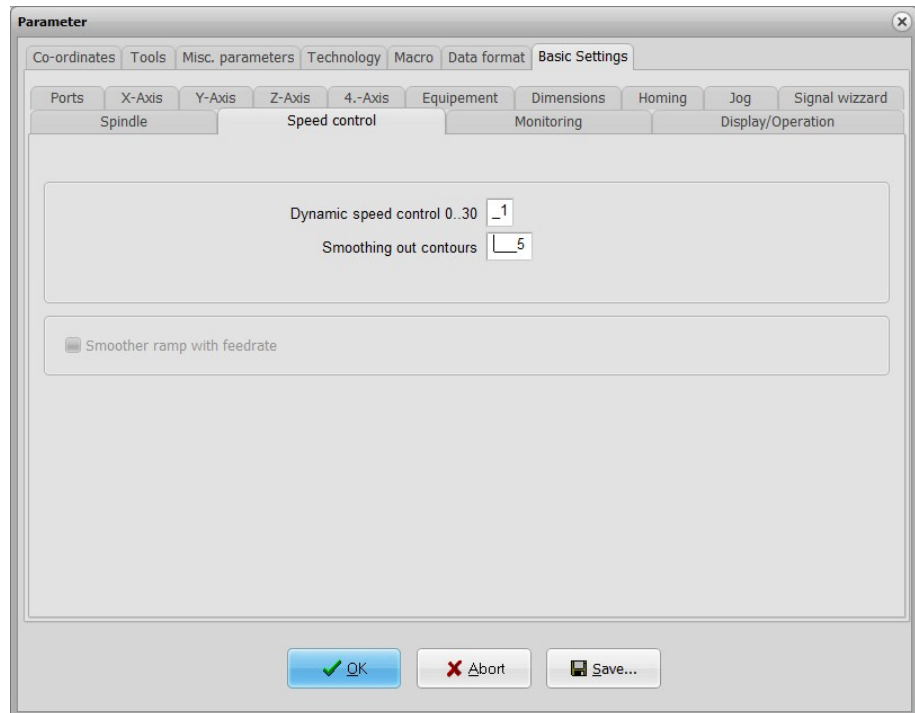
Using a changer or an automatic collet chuck this switch is responsible for determining the status of the chuck during switch on of the plant. Activating the chuck closes **WinPC-NC** remembers the last used tool from session to session.

Sensor – Spindle speed

Synchronisation when reaching rotational speed

With additional settings it can be determined whether a signal should be sent from the spindle speed control when the desired rotation speed has been reached. Many frequency converters and speed controllers provide a corresponding signal. This kind of monitoring is advisable subject to cabling and relevant pin layout.

5.2.11. Basic settings-Dynamic speed Control



Parameter-Basic settings Dynamic speed control

Factor Dynamic Speed Control

The speed control of **WinPC-NC** is a very complex function. With the help of the look ahead function it tries to recognize future contouring and to adapt the corresponding speed.

Look ahead function

The grade of quality concerning speed adaptation strongly depends on mechanical and loading conditions of the machine and the dynamic of used drives. Therefore it is not possible to give any recommendation concerning optimal settings. They are to be determined by specific tests of the individual operator.

The defined value determines the level for decelerating the current speed with small and tight radii. A value of zero deactivates speed adaptation and allows only the action of the set brake angle.

Smoothing out contours

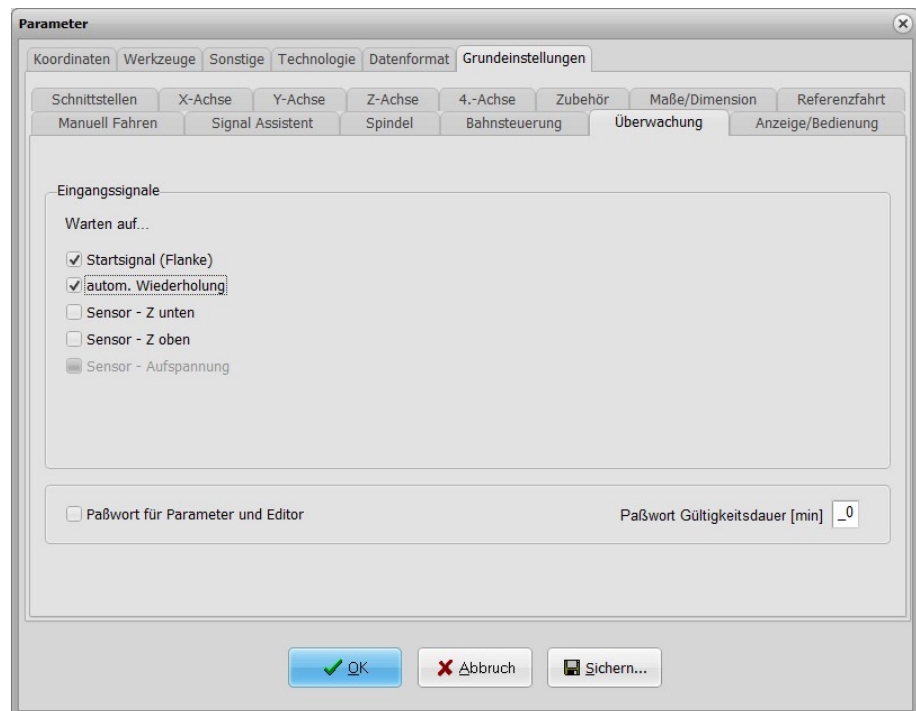
Filter for producing finer contours

WinPC-NC is equipped with a filter in order to smooth contours and sheets consisting of many small vectors. This is necessary for producing fine and neat edges.

The parameter defines a factor of 0-2000.

5.2.12. Basic settings-monitoring

All input signals and relevant parameters which can be synchronized are grouped in a specific dialog box.



Monitoring input signals

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 on definition and assignment of the signal transmissions and inputs.

The various signals and what they mean :

Start signal	Following activation of the job process, WinPC-NC waits always 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 down	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 up	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 speed starts to indicate when the required speed has been reached. WinPC-NC waits until this signal goes to HIGH before continuing the process
Clampin	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 parameter 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.

Password and period of validity

This parameter activates a password which must be entered prior to change the parameters or a loaded program. The password is fixed.

The period of validity stipulates the term how long the password is valid and how long it is not interrogated again.

5.3. Coordinates

The screenshot shows the 'Parameter' dialog box with the 'Co-ordinates' tab selected. The dialog is organized into several sections. The top section contains fields for X, Y, and Z coordinates for Park position, Scaling factors, Surface block, Toollift, and Zero point. The 'Zero point in file' is set to 'bottom left'. Below this, there are two checked checkboxes: 'Define size of working piece' and 'Work piece area monitoring'. The next section defines the working piece area with 'Working piece from' and 'to' coordinates for X, Y, and Z. The bottom section is for 'Axis 4', with a dropdown for 'Move to zero 4th axis...' set to 'first' and a 'Zero point' field. At the bottom of the dialog are three buttons: 'OK', 'Abort', and 'Save...'.

	X	Y	Z	Unit
Park position	+__0.00	+__0.00	+__0.00	mm
Scaling factors	__1.000	__1.000	__1.000	
Surface block			+__0.00	mm
Toollift			+__1.00	mm
Zero point	-__36.33	+__51.85	+__10.00	mm
Zero point in file	bottom left			

	X	Y	Z	Unit
Working piece from	+__50.00	+__50.00	+__0.00	mm
to	+__750.00	+__550.00	+__100.00	mm

Axis 4

Move to zero 4th axis...	first
Zero point	+__0.00

Parameter-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 or mouse.

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

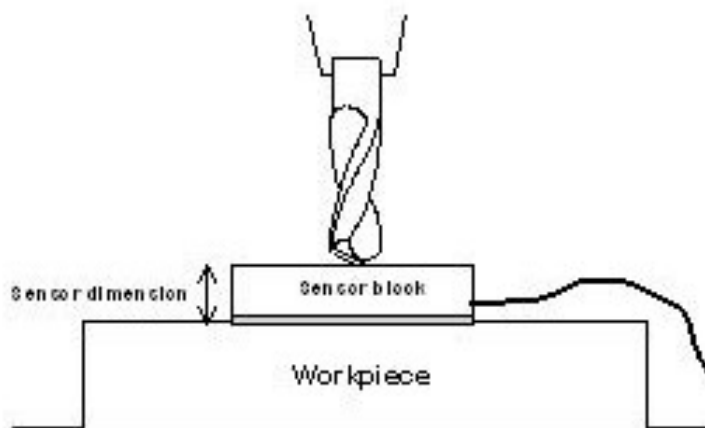
It is possible to compensate for calibration differences using the scaling factors. If both the axis resolution and the unit of measurement are set correctly but the machine does not move to the exact length nevertheless, this problem can be corrected using the scaling factors.

The values must be specified to 3 decimal places and are used for multiplying the coordinate values to which the machine is to move. It affects the imported NC data only.

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 at the LPT port of ncUSB.-



Automatic measurement of the Z-axis zero point

The measuring procedure involves several steps:

1. Place the sensor block on the workpiece surface or on the machine
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 checked and added to the defined sensor dimension, the result being stored as a parameter.

Clearing distance

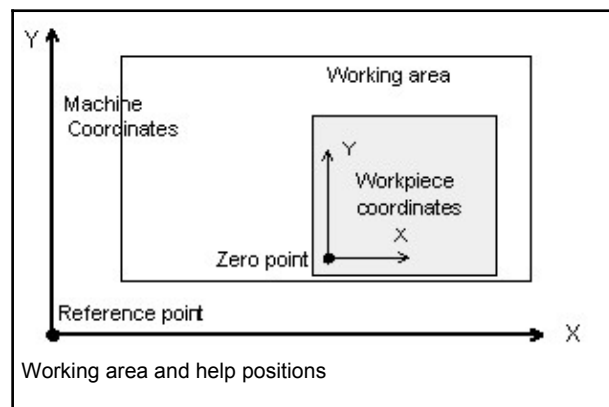
Additional distance for collision avoidance

Clearance distance can be defined as additional height of the Z-axis above the zero point level. With each job process the tool is lifted above the zero point by this distance and the new height is used as tool lift height.

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.

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.



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 machine reference point (machine coordinates).

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.

Six positions are possible:

- | | |
|------------------------------|---|
| Bottom left | The zero point is at the smallest X and Y-axis coordinates in the file, normally at the bottom left edge. Mostly used with HPGL files. |
| Origin of Coordinates | The zero point is at the coordinate origin, i.e. where the CAD programs 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 or when using G code files |
| Center | 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 |
| Bottom right | The zero point is positioned at the highest X and smallest Y coordinate of the file |
| Middle right | The zero point is positioned at the highest X coordinate and exactly between the smallest and highest Y coordinate |
| Top left | The zero point is positioned at the smallest X and the highest Y coordinate of the file |

Working area and monitoring working area

Software area monitoring- The working area defines the section, e. g. for marking material dimensions. Limits are visible in the graphical display and it is immediately recognizable whether milling or engraving can be effected with the material piece. A check is made when job starts.

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.

Easy definition by moving to corners It is very easy to move the machine to the lower left and upper right corner of the desired area and press the function keys F5 and F6.

These parameters are not to be mixed up with the values determining the size of the machine table and thereby the maximum moving area.

Measuring is made in mm and distances refer to the machine reference point (machine coordinates).

The management of an additional area is activated when you enable the corresponding workpiece by checkbox.

Homing sequence of 4th axis

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

Zero point 4th axis

Here you enter the zero point for the 4th axis. Depending on the mechanic for controlling the 4th axis, it is a position in mm or degrees. Thus the position is exactly approached during the homing run of the 4th axis.

5.4. Tools

Project-related tool management

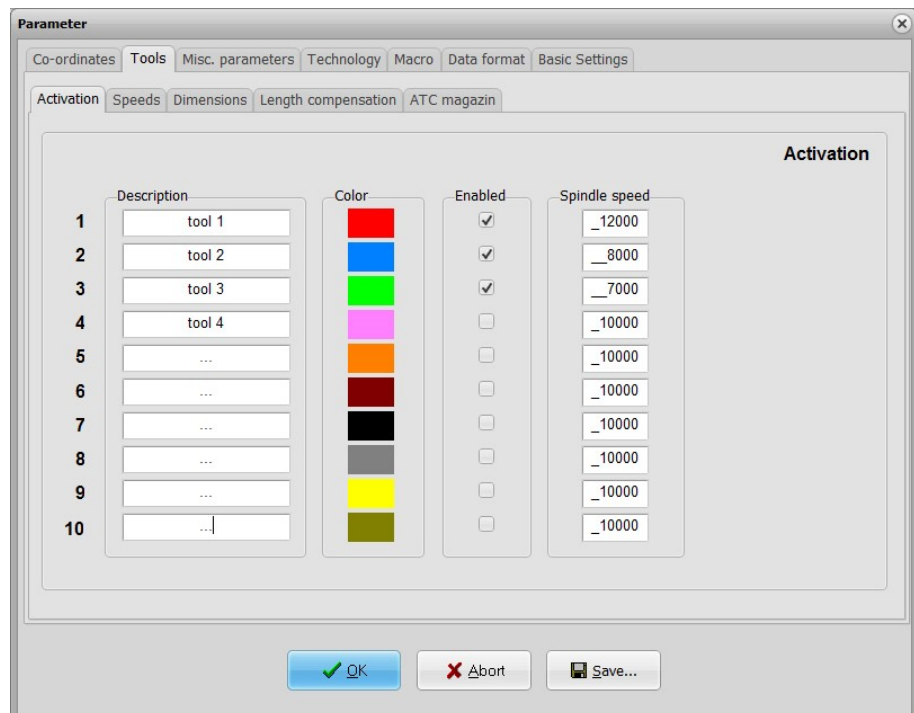
WinPC-NC has a user-friendly tool management function. It is possible to define up to 10 tools with specific values for each working project.

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.

5.4.1. Tools-Activation



Parameter-Tools Activation

Description

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. Switching tools ON/OFF is even easier by clicking with the mouse on to the colored toolbox directly above the displayed graphics.

Spindle speed

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

5.4.2. Tools-Speeds

	V-Plunge	V-Advance	V-Withdrawal	Brake angle
1	5.00	5.00	12.00	30
2	3.00	2.00	15.00	30
3	2.00	3.00	5.00	30
4	4.00	4.00	10.00	30
5	6.00	5.00	10.00	30
6	5.00	15.00	25.00	30
7	5.00	5.00	10.00	30
8	5.00	5.00	10.00	30
9	5.00	5.00	10.00	30
10	5.00	5.00	10.00	30

Parameter-Tools-Speeds

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.

Advance speed

The advance speed or feed rate 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 milling, engraving or grinding, then the maximum feed rate depends on the tool used and the material.

Withdrawal speed

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

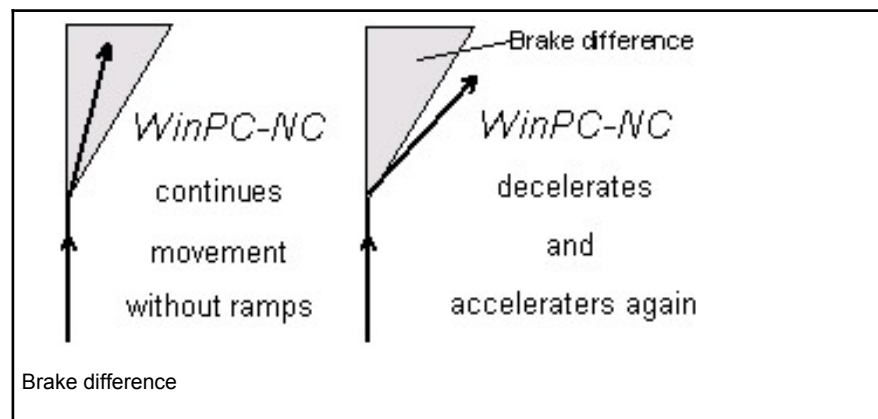
Brake angle

The brake angle 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

Between the start and finish of movements, the acceleration and braking function is only activated if the direction of the next vector deviates from the previous one by more than the defined brake angle.

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



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

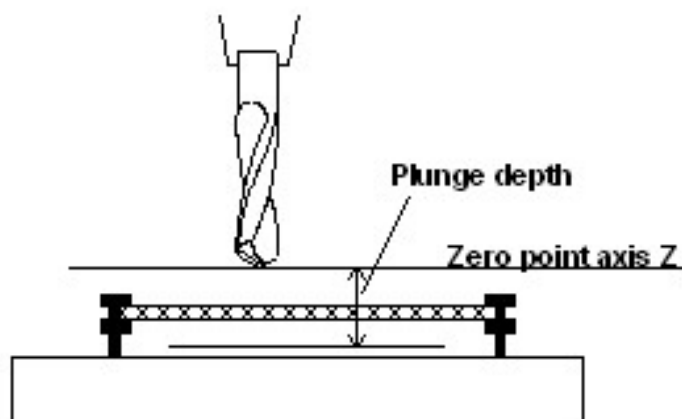
5.4.3. Tool-dimension

	Depth	Repetitions	Z feed correction
1	1.00	0	1.00
2	2.00	0	2.00
3	3.00	0	1.50
4	1.00	0	0.00
5	1.00	0	0.00
6	1.00	0	0.00
7	1.00	0	0.00
8	1.00	0	0.00
9	1.00	0	0.00
10	1.00	0	0.00

Parameter-Tools Speeds

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

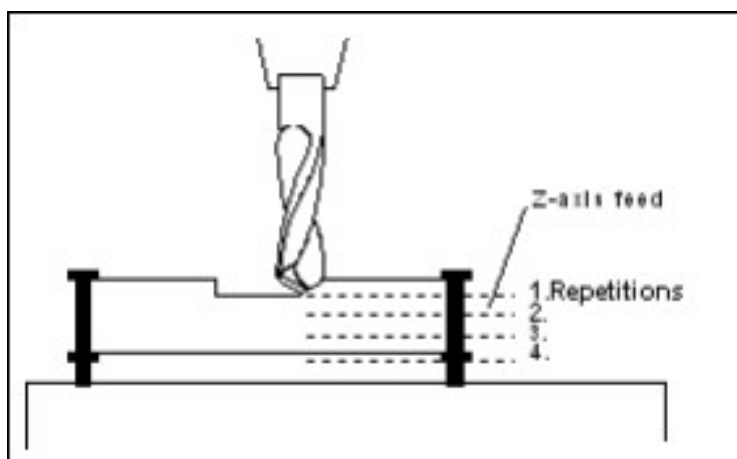
Routing in stages

Frequently, several passes are required when routing thick or hard materials. The parameters Repetitions and Z feed correction 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 correction

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



Repetitions and feed correction

5.4.4. Tool length -measurement and compensation

The screenshot shows the 'Parameter' dialog box with the 'Length compensation' tab selected. The 'Z length' section lists 10 tools, each with a value of '+ 0.00'. The 'Length sensor position' section shows 'I221 Surface sensor = LPT2 Pin11' and coordinates X: '+ 19.10', Y: '+ 119.10', and Z: '+ 60.00'. The 'autom. length compensation' checkbox is checked. Other checkboxes for 'Length check after tool change', 'Fast move to sensor with brake', and 'Save measured tool lengths' are unchecked. The 'OK', 'Abort', and 'Save...' buttons are at the bottom.

Parameter-Tools-Tool length compensation

These setting options are only visible if the function length measurement and compensation is activated in *Basic Settings-Accessories*.

Z-axis length

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

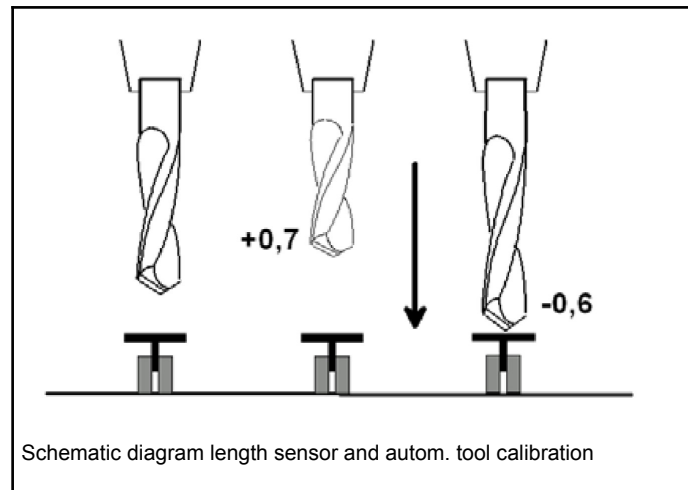
Sensor position

*Automatic
ascertainment
of length*

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

These parameters define a position above this sensor in absolute machine coordinates. When measuring, **WinPC-NC** moves to this position and then lowers the tool slowly until the sensor is switched.

The distance moved is adopted in the parameters as tool length.



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.

Length check after tool changed

This parameter enables WinPC-NC to perform a tool length check after each tool change.

Fast move to sensor with brake

To speed up the process you can activate the two step measurement and force **WinPC-NC** to move to the sensor with fast manual speed, brake the movement, reject the tool over the sensor and do the measurement in a second step precisely.

Save measured lengths as parameter

Activating this parameter means that **WinPC-NC** saves all measured lengths in the parameter file. These measured lengths are restored when the project is started again. This is useful while working with various lengths without measuring each time.

5.4.5. Tool-Changer-Magazine

These setting options are only visible subject to activation of the automatic tool changer in *Basic Settings-Accessories*.

	X	Y	Z
1	+ 1.00	+ 1.00	+ 0.00
2	+ 5.00	+ 1.00	+ 0.00
3	+ 9.00	+ 1.00	+ 0.00
4	+ 13.00	+ 1.00	+ 0.00
5	+ 17.00	+ 1.00	+ 0.00
6	+ 0.00	+ 0.00	+ 0.00
7	+ 0.00	+ 0.00	+ 0.00
8	+ 0.00	+ 0.00	+ 0.00
9	+ 0.00	+ 0.00	+ 0.00
10	+ 0.00	+ 0.00	+ 0.00

☐ No Z lift at ATC ☐ No move back to last pos.

OK Abort Save...

Parameter-Tools-Changer-Magazine

*Controlling
the automatic
changer*

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

It is recommended to define the changing positions with short distance to the actual pick up and the remaining distance with simple moving commands in the macros for putting down and picking up.

An electrically or pneumatically operated collect 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 releasing and picking up tools as well as movements, waiting times and switching outputs.

The procedure is described in a subsequent chapter.

Parameters for the automatic tool changer ATC are only available if the function is enabled..

Moved tool magazines, for instance retractable cabinets or circular magazines, can be controlled via 4th axes only by **WinPC-NC Professional**.

No Z lift at ATC

Movements in the tool changer normally cause a move all the way up to the zero position. If you think that these moves are too high or too long in time, you have the option to suppress it by this parameter switch.

Please take greatest care that there is no collision with the components of the tool changer by suppressed Z zero move.

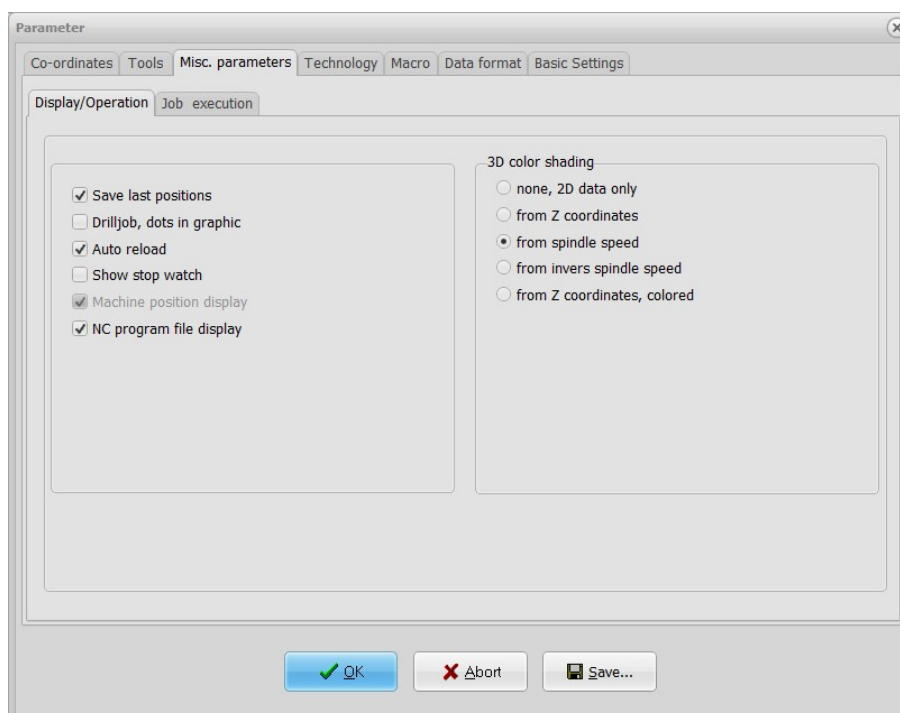
No move back to last position

Approaching to the last park position or tool changer **WinPC-NC** always remembers the previous position and returns to it again. This often results in unnecessary movements and is not desirable. This switch prevents a move to the last position.

5.5. Miscellaneous parameters

Here all settings are summarized which apply to job process or operation of **WinPC-NC**.

5.5.1. Miscellaneous-Display/Operation

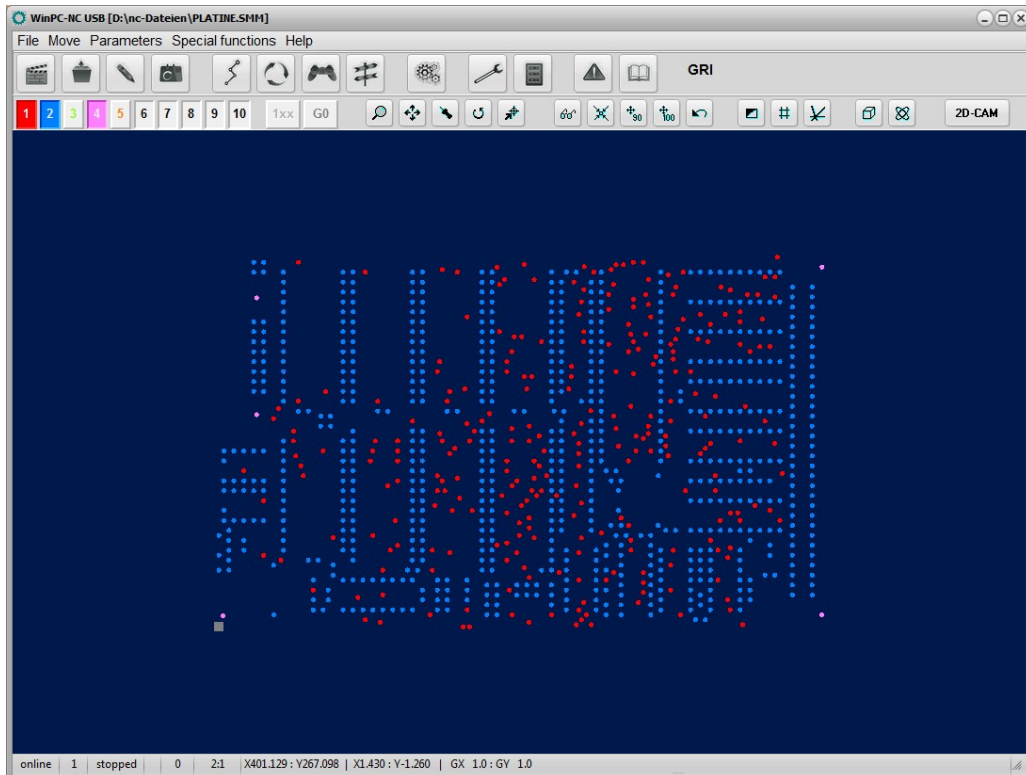


Parameter-Miscellaneous-Display/Operation

Drill job, dots in graphic

Marking insertion points

The graphical display does not show holes drilled with HPGL, MultiCAM or G code formats. Activating this parameter marks all insertion points with a small circle.



Display as drilling job with marked punctures

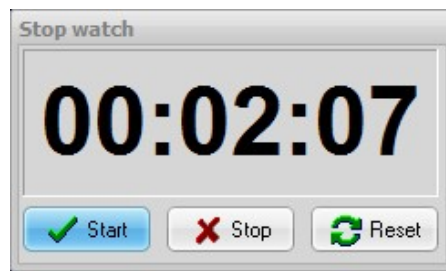
Automatic reload

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 you switch back to **WinPC-NC**, to be able to check all the changes on the screen straight away

Stopwatch

- This parameter is enabled by the function stop watch of **WinPC-NC**. The displayed stopwatch is reset with each job start and begins running in seconds. An automatic stop takes place at the job end.



Stop watch from WinPC-NC

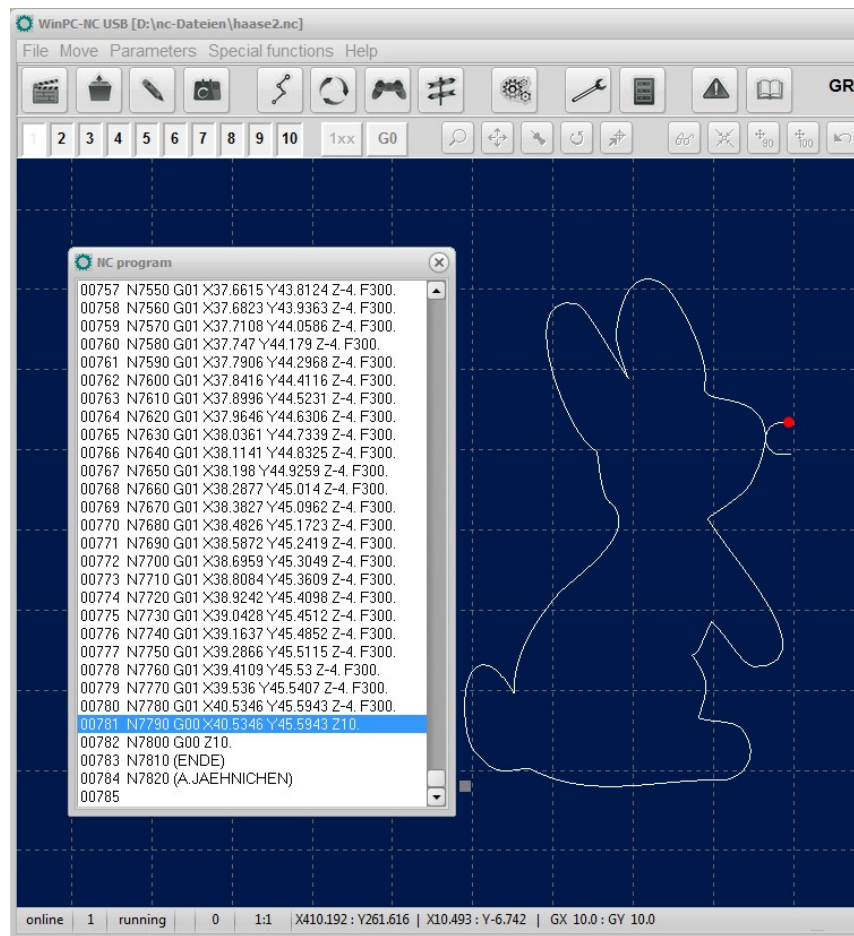
In this way it is possible to determine exact execution times of the job and for instance to account for labour costs.

Display of position

In **WinPC-NC USB** the display of the axes position in realtime is switchable and can be displayed as a small window.

Ideally you zoom and arrange the windows in the way that all relevant information is available.

NC-Program window



Program window during operation

To obtain an overview concerning the actual program section there is a file display with a cursor bar which marks the actual command during operation in realtime.

The program window is suitable for any formats except for DXF and postscript and for files with 300000 lines at most.

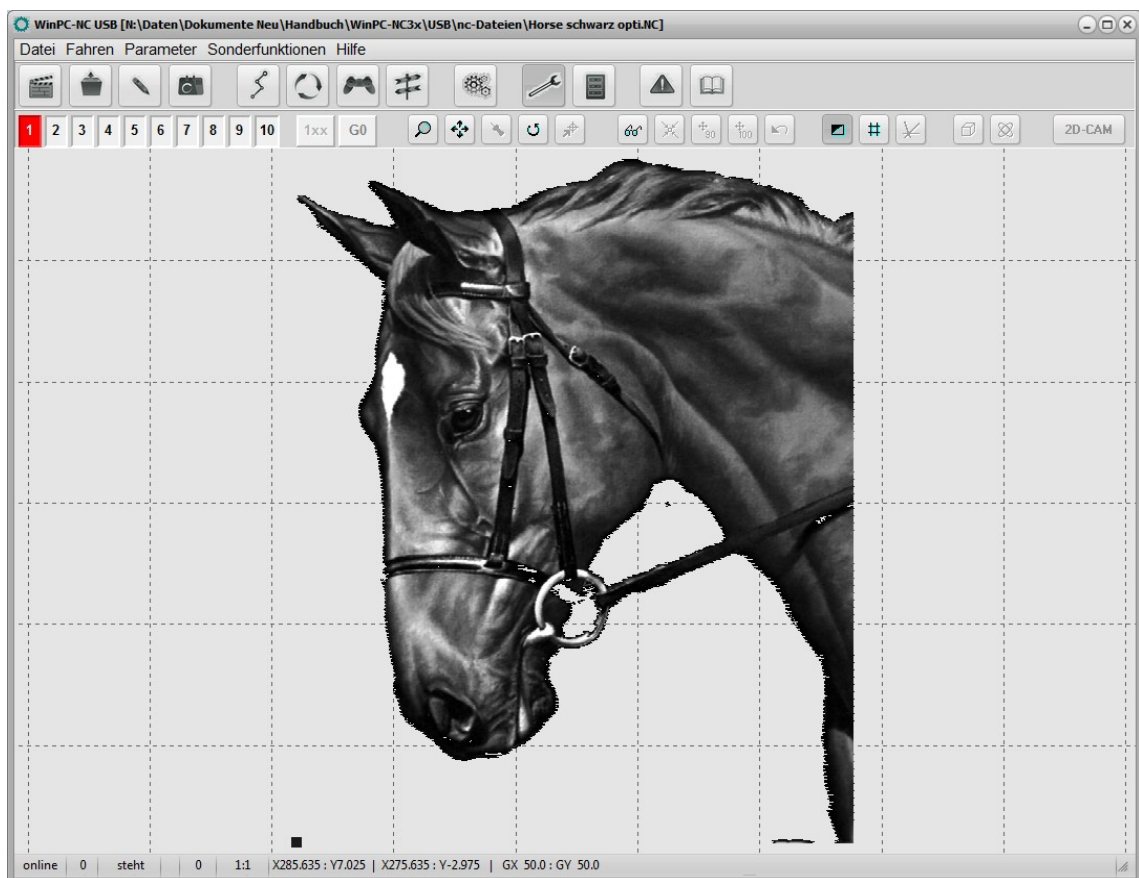
Save position

WinPC-NC can save the actual machine position after each movement and job. This is useful when there are no reference switches available or homing is not always possible. The set values remain valid even after *Exit* and *re-start*.

3D color shading

In **WinPC-NC** all loaded files are displayed in 2D and flat view only. When loading 3D files or reliefs the visual check is not quite easy and therefore you can select a color shading where the exact colors are shifted to light and dark corresponding to Z heights or PWM values.

The color shading may be selected to correspond to different values and settings. There are five possible options. With inverse display a higher speed value is displayed in brighter color, otherwise by a darker shade.



Color shade according to relief- and laser engraving or grinding applications

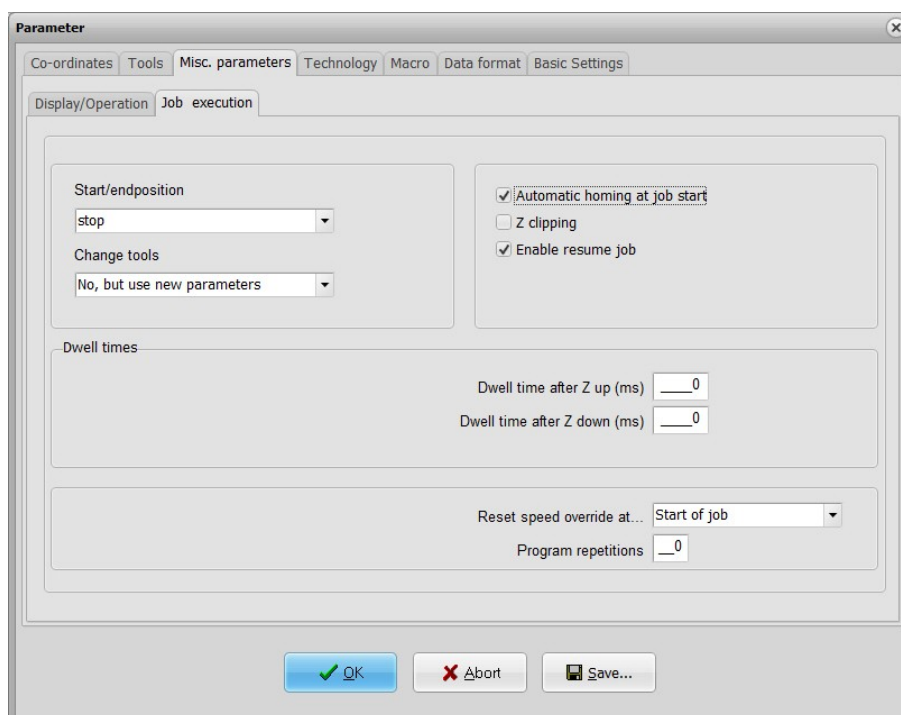
No ones, merely 2D files Here a mere 2D display is presented with lines and drillings in the selected tool color

from Z coordinates / from Z colored All colors are shifted to more light or darker corresponding to the current Z height. The colored option will shift stronger as the regular one and this may cause considerable color changes.

from spindle speed / from inverted spindle speed

this option will be a good idea at grinding applications where different spindle speeds or PWM signal levels indicate the gray shading parts. At inverted shading a higher PWM value shifts the color more to light nuances.

5.5.2. Miscellaneous-Job process



Parameter-Miscellaneous parameters-Job process

Start/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 4 possible start and end positions :

Stop

WinPC-NC stops at the reference position after reference movement, and at the last coordinate after each job process

Zero point

The machine moves to the defined zero point after reference movement and after each job process.

*Park
Position*

WinPC-NC moves to the defined parked position after reference movement and after each job process.

*Zero point
and
clearance
distance*

The machine moves to the defined zero point and lifts up the Z-axis to the defined clearance distance.

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, at the
park position*

Performs the tool change and moves to the defined park position for every tool change

*No, but use
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 by ATC

Performs automatically a tool change by existing tool changer

Homing at a job

On request a homing sequence can automatically be performed by **WinPC-NC** prior to each job. This switch is recommended in case of an exact movement to the positions or if you presume that there are stepping errors.

Z-axis clipping

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

Enable resume job

WinPC-NC can continue operation of an interrupted job exactly at the interruption point. However, this function has to be enabled by a parameter.



Interrogation at a restart or continuation of an interrupted job

Dwell 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 X and Y-axis movements. 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.

Reset override speed

The feed and spindle rate are permanently changeable while **WinPCNC** is processing a job. This parameter stipulates how long this alteration is activated.

Following settings are possible :

<i>Machinen reset or programm reset</i>	Only with restarting WinPC-NC it is possible to place back the override values to 100%
<i>New file</i>	Values are placed back by loading a new NC file
<i>Job start</i>	Reset of the values is made by the following start of a job process

Program repetitions

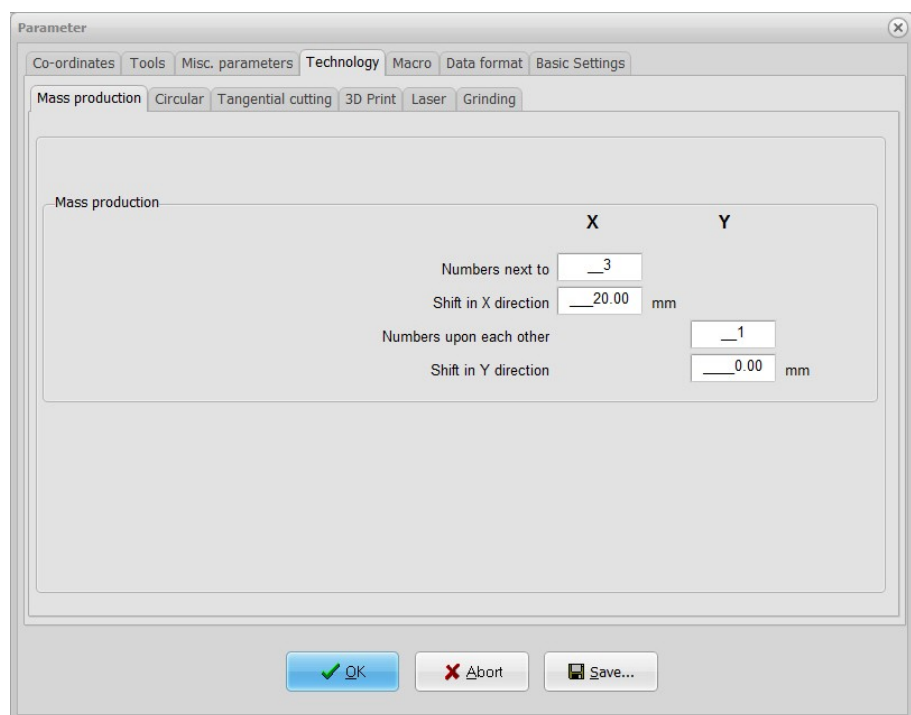
The program repetitions parameter enables you to repeat a working process up to 999 times. This enables series production to be achieved in conjunction with a start signal.

5.6. Technology

The technology page provides any information concerning selected and available applications. Depending on your needs, they can be activated or deactivated from session to session.

Only those applications are available which are selected as accessories in basic settings and which need special components.

5.6.1. Technology- Function for mass production



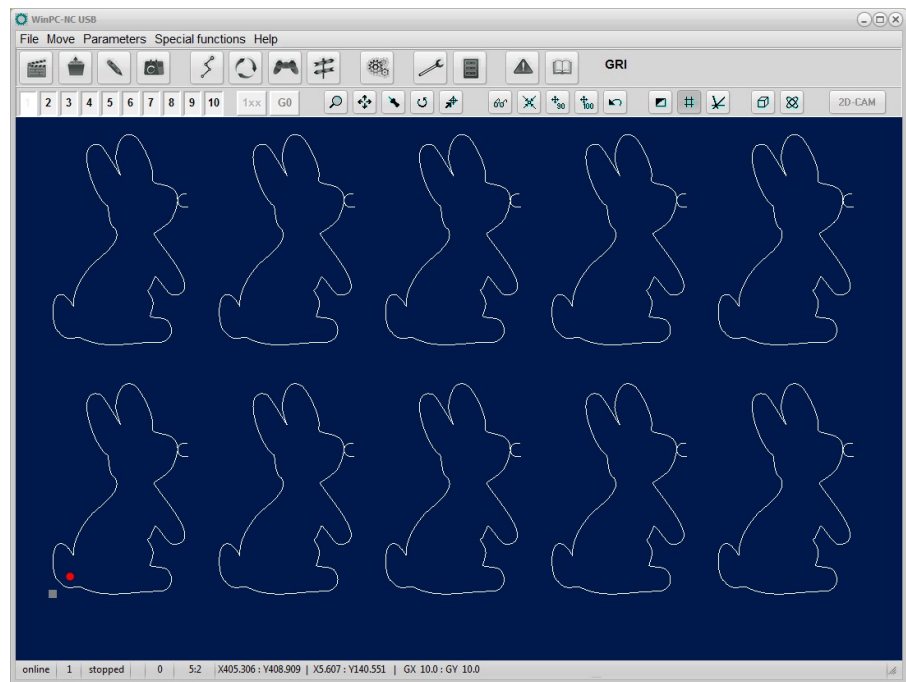
Parameter-Technology- Function for mass production

Number of pieces next to and upon each other

These parameters determine the numbers of pieces in rows upon and in column next to each other.

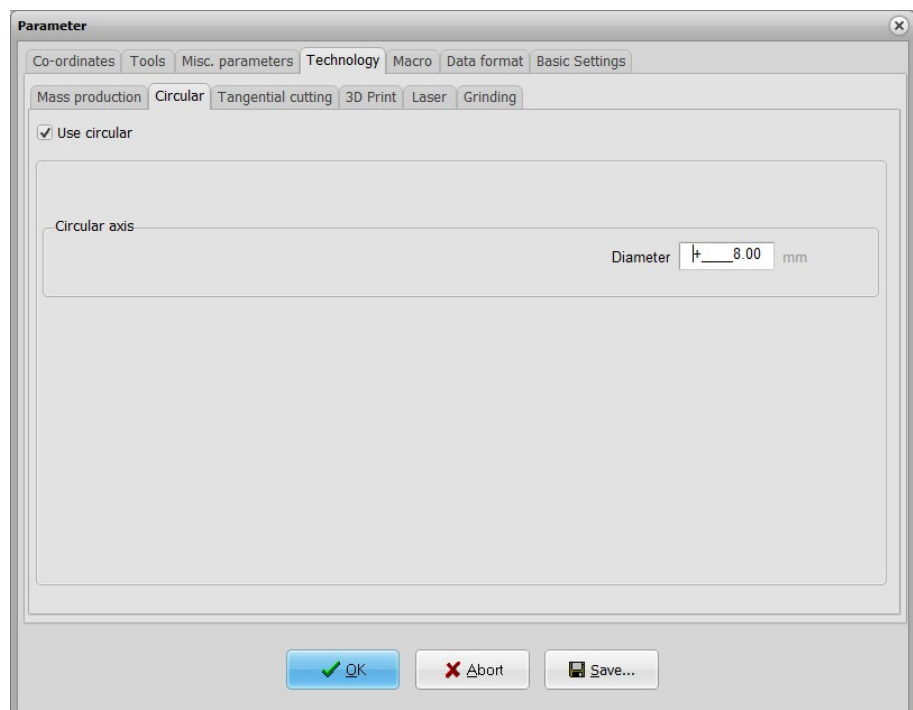
Offset zero points

This parameter defines the distance between the working pieces. The offset has to be slightly greater than the dimensions of the pieces.



Mass production with 5 by 2 pieces

5.6.2. Technology-rotary axis



Parameter-Technology-rotary axis

Use circular

This parameter generally switches on or off the function for circular engraving.

Diameter

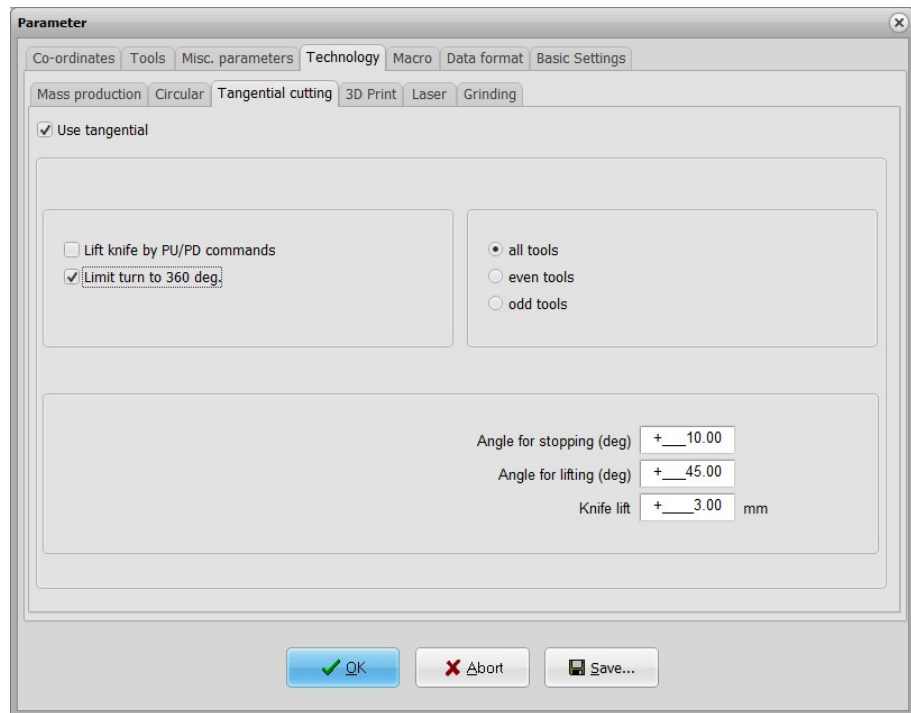
The diameter of the cylindric body is essential for executing calculation of the Y-coordinates and should be entered with greatest exactness.

Round engraving is only possible with data in HPGL format.



Sample of circular engraving

5.6.3. Technology-Tangential-cutting



Parameter-Technology-Tangential-cutting

Enable tangential axis

The parameter *Enable tangential axis* is generally responsible for switching on or off the tangential axis function. If it is activated this function will also be considered during reference move.

Tangential cuts are only possible if the 4th axis is equipped with a defined reference switch or if a reference move has been carried out. Otherwise the user is asked to execute a reference move.

This parameter is also part of this activation on the right side section for choosing tools. There are various options for actuating the tangential function: either by all available tools or by even or odd ones. While a job is running, it is easy to use tangential blades, gouge balls and perforating tools.

Lift with PU/PD

Lifting the cutter during rotations is normally directly effected by the program.

However, if macros are used which are working solely with

Lift and *Lower* commands, lifting of the cutter can be forced by this parameter with PD and PU commands integrated in HPGL data.

This parameter is only applicable using macros.

Rotation of max. 360 degree

With certain tangential heads it is impossible to carry out rotations over 360 degree because there can be strokes or overstripped cables.

Limitation of cutter rotations

An activated parameter carries out rotations always within the limits of 0 to 360 degree. Movements requiring cutter rotations over this limit are stopped, the cutter will be resetted and then movement will be continued.



Sample for foil cutting with tangential axis

Angle for stop, lift an lift height

During the process of tangential cutting it is important to avoid too large or too wide cutter rotations into the material. By means of both angle paramaters it is possible to determine exactly the moment of stopping a movement concerning direction change or cutter rotations or even when the cutter has to be lifted.

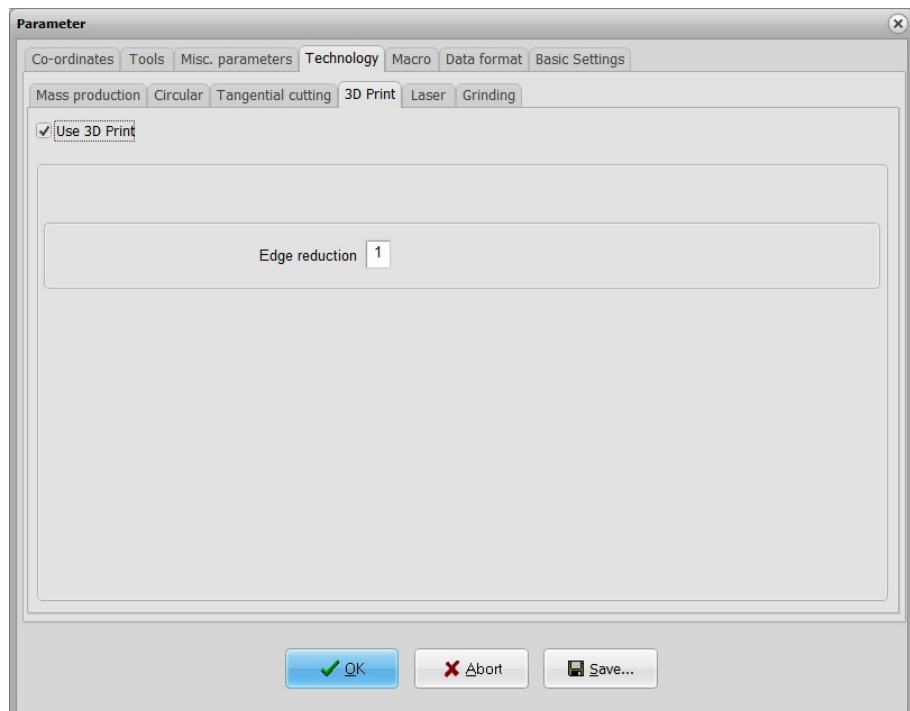
External cutter rotations

A movement will be previously stopped when the cutter rotations are greater than the stop angle, then the cutter will be rotated and subsequently the movement will be continued. If rotations are greater than the lifting angle the cutter will be lifted after a movement stop and then it is rotated and put down again.

With the parameter knife lift you can determine the height in millimeters, in which the cutter is lifted prior to rotation. This should be set in the way to avoid damages of the material to be cut.

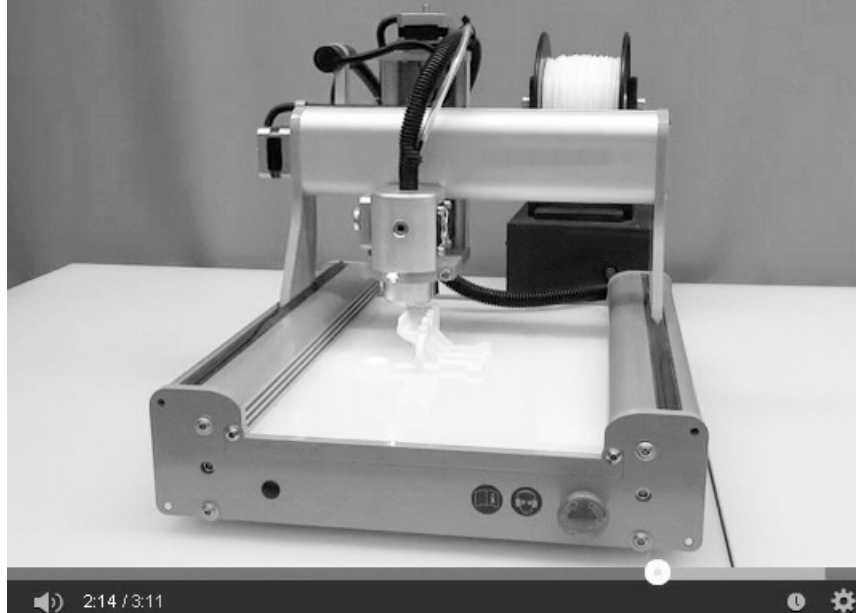
5.6.4. Technologie-3D-print

For executing the special function *3D-print*, **WinPC-NC** makes use of some inputs/outputs for controlling the print head and some functions are modified during the control of axes or during the job process. This means for the filament feed, that the 4th axis can only be moved with slow speed into the feed direction. A break during the job is made by a special process in order to avoid damage of the printed piece.



Parameter-Technologie-3D-print

The handling of 3D printing in detail can be learned in documentation and manual of the printing head and addon parts.



Machine with 3D-print head and running process

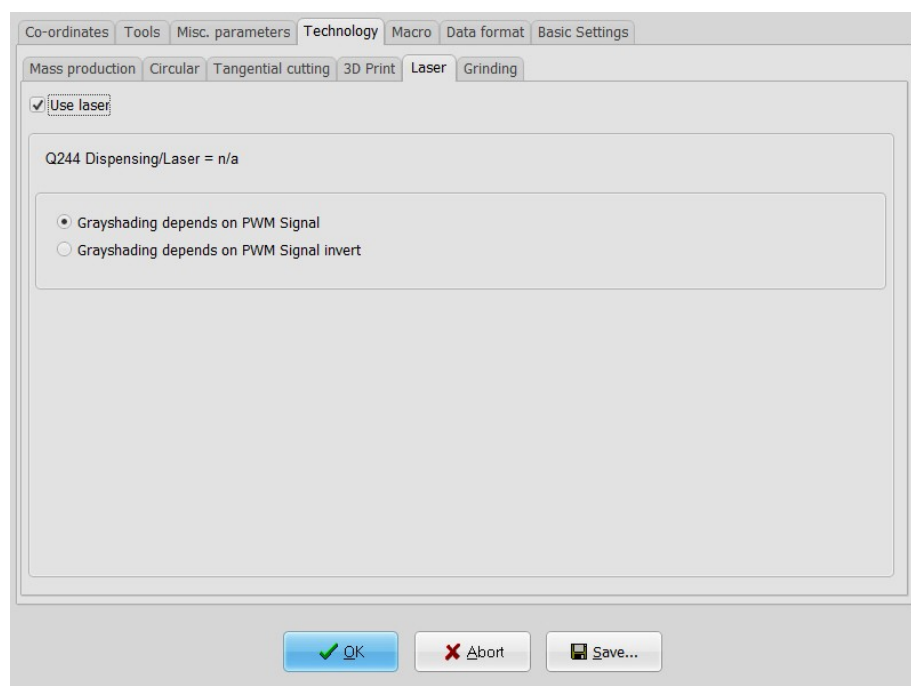
Edge softening

By the factor which is responsible for edge softening, the filament feed can be reduced step-by-step in corners and on edges. According to the filament state an unduly order can be avoided or reduced.

5.6.5. Technology-Laser

In **WinPC-NC** it is possible to use both, milling spindle and laser for realizing engravings and cuttings. Ideally output Q244 should be used for switching the laser on/off and it always switches on when moving along contours.

The power control of the laser is realized by the PWM output signal and values between 0 % and 100 %. These values are set in the NC file or are defined as rotational speed in *Tools-Parameters*.



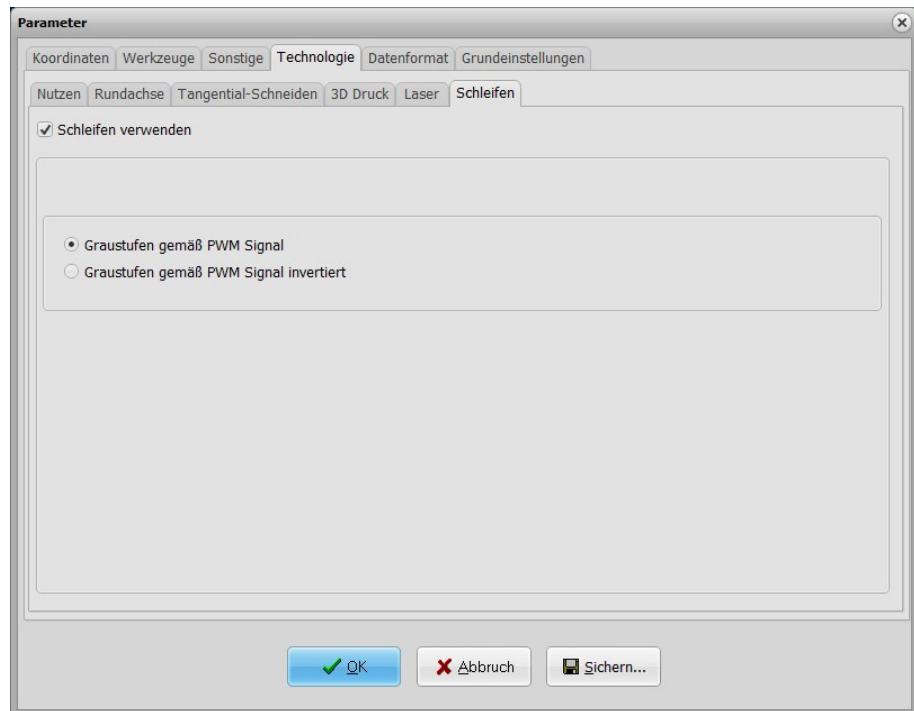
Parameter-Technology-Laser

Grey scales according to PWM signal/inverted

Engravings which are executed with laser line by line can be presented much more effective when the display is organised in grey scales. Laser cutting does not require a display in grey scales.

5.6.6. Technology-Grinding

Grinding of smooth surfaces with various grinding pressure is also available. For example, this method is used for processing grave stones or other stone surfaces like marble.



Parameter-Technology-Grinding

Grey scale image according to PWM signal/inverted

To improve the graphical display it is also possible that the NC file can be effected in grey scales according to the defined spindle speed value. The grinding pressure of the grinding head is controlled by the programmed rate of rotation.



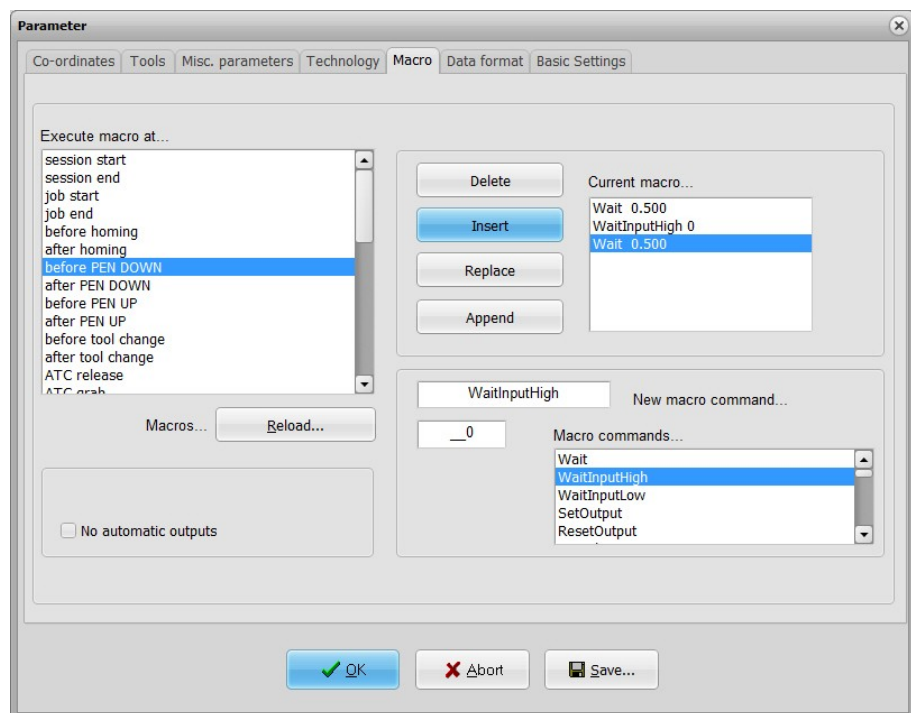
5.7. Macros

The macros function in **WinPC-NC** offers you a powerful tool for individual adaptations to an extremely wide range of different mechanics 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 as a special function.

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.



Macro definition

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 list box 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 to be deleted, replaced or a new command can be inserted and append

Executing of macros must be assigned to certain program points and situations. You can assign a macro to be activated before a reference move or after a tool change or even when a special tool is selected.

In addition there are macros which can be activated by specific commands in a G-Code or HPGL file. As an example the programmable macro1 can be called up by M90. Further information are provided in later chapters concerning G-Code and HPGL files.

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 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.

Macro commands

Most of the macro commands are self-explanatory and herefore we want to treat solely some special ones.

Wait time	Performs a wait time in ms
Wait InputHigh no	Waits for HIGH level of input no
Set OutputHigh no	Sets output no HIGH
Speed spd	Speed setting for future moves in mm/secs.
MoveX xxx	Moves a relative distance
MoveAbs xxx	Moves to absolute position xxx, i.e. in machine coordinates
Spindle ON/OFF	Switches the additional signal Spindle
Cooling ON/OFF	Other outputs are functioning analogously
Wait Spindle Stop	Waits until input 'Spindle Stop' is activated and the milling spindle has definitely been stopped (subject to installation and wiring)
WaitStart	Waits for a start signal on input I255 Start
OffsetX	not available in WinPC-NC USB
Block Z zero point	not available in WinPC-NC USB
Spindle Speed	sets up a new spindle speed

Defining inputs and outputs always requires the indication of the corresponding numbers. Do not indicate the pin number of the LPT port. Typical numbers to be used are I100 to I107 or Q100 to Q115. In advance it is necessary to assign to these inputs and outputs a pin numbers and wires via signal wizard.

Example 1
*Release tool***Example for releasing 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 t 5,00	Set speed slower
Wait Spindle Stop	Wait until spindle has stopped
MoveY 10,00	Move the tool into the chuck holder
MoletteOpen	Open the molette
Wait 500	Wait 500 ms for pneumatic system
Speed 2,00	Set very slow speed
MoveZ -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	switch on oxygen with output 100
Wait 500	Wait another 500 ms before movement

Please note that certain macro commands like offset or macros for certain additional functions, e. g. Block zero Z, are only usable by WinPC-NC Professional.

Signals only via macros

Outputs automatically or by macros

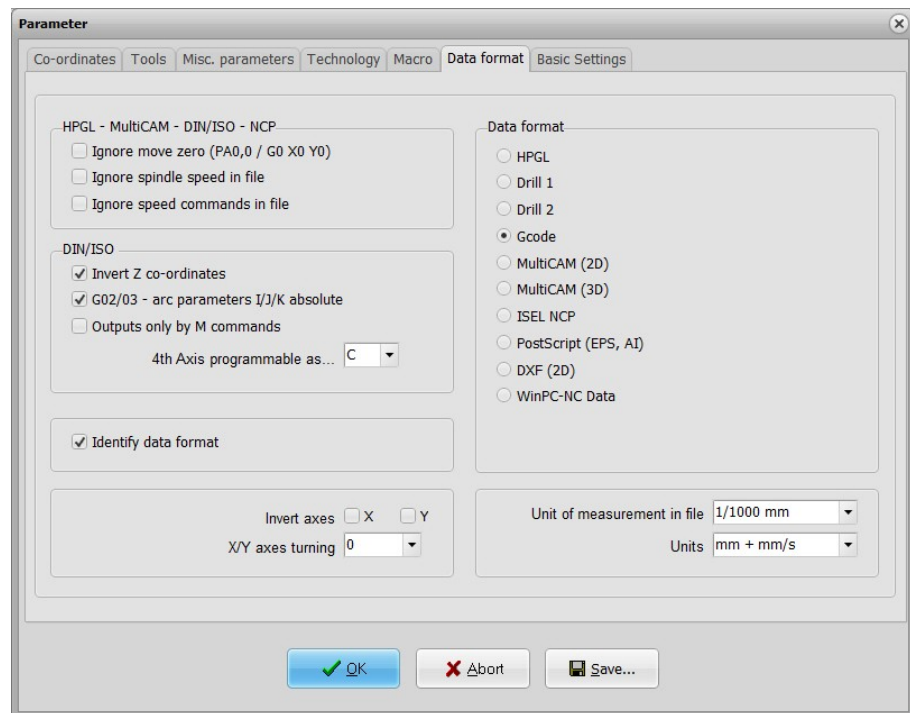
Usually some output signals are automatically performed in **WinPC-NC**. For example, the output 'Job runs' is set when starting up and it is reset at the end or with break off. The same applies to the signals for drilling spindle, coolant pump, dispensing, etc.

Automatic operation mode must be deactivated in this parameter, if it is more convenient to manage signals by macros or if operation times are to be changed.

All signals can solely be used by macros.

5.8. 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.



Parameter-Data format

At present, the commonly used plotter format HPGL, MultiCAM, two drilling formats, G code (DIN/ISO), DXF (2D) and ISEL NCP are possible. Furthermore WinPC-NC is able to recognize vector informations of postscript EPS and AI files, e. g. as created by various Adobe programs.

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, industrial NC machines are programmed in the DIN/ISO format by G codards

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 analysed using this description and revised with the editor.

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

Automatic identification of the format

Usually most of the supporting NC formats are automatically recognized by **WinPC-NC** during file opening. It rarely happens that errors are produced in this way. If it should be the case they can be easily eliminated by switching off the identification function and by manual adjustment of the correct format.

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 G code formats also contain commands to set the spindle speed and the feed rate. 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 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.

Output signals with M commands

In G code 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.

4. Axis 4 programmable as...

Letter for programming

The 4th axis can be addressed in G code 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.

Mirror/Invert 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.

Rotation of X/Y axes

The NC data can be rotated around the zero point, e. g. for a better placement on the material. Rotation always happens in 90 degree steps.

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 1/100mm or also 1 mil.

EPS and AI files are automatically set when opening the file. The same applies to HPGL and DXF.

Units in the NC 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 millimeter/second (mm and mm/s)
- Millimeters and millimeter/minute (mm and mm/min)
- Inch and inches/minute (inch and inch/min)

6. Initial start-up with the machine

Machine adjustment

After software installation on your PC it is necessary to carry out some start-up processes and corresponding adjustments. This procedure is only necessary with the initial start-up procedure and generally relates to the adjustment of your machine.

In case if you have acquired **WinPC-NC USB** together with the machine either most of the parameters are correctly preadjusted or you have received an additional disk with the corresponding adjustments. If this is the case the following described procedure can be disregarded.

The accessory module ncUSB is equipped with two ports which are pin compatible and similar to LPT. To simplify matters, we continue using the term LPT port in this document.

6.1. Connecting ncUSB and the machine

After software installation and connection of the accessory module ncUSB there should be the corresponding hardware identification. After a short time you will receive the message that the newly identified hardware is ready for use. Connection of the machine and the drives is made on the first LPT port of ncUSB.

Attention !

All Windows operating systems provide the possibility to deactivate USB ports or to reduce the 5V power supply on the USB ports for energy saving reasons.

Therefore please make sure that the sleep mode of USB ports is permanently switched off in the device manager.

Connection of the machine and the drives is made on the first LPT port of ncUSB..

ATTENTION !

Greatest care has to be taken when connecting all cables to the housing. Following actions can cause serious damage: Incorrect assignment of the signal cable, plug incorrectly inserted, cables incorrectly connected.

Starting the entire unit is not allowed until all necessary and state specific safety rules have been accomplished and checked. The user is responsible for unit operation.

ATTENTION !

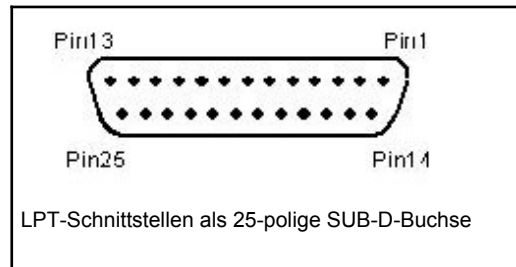
For further interrogation of the status emergency stop or ready for operation it is essential to use one of the inputs. In this way it is possible for the control to stop actuation of output signals if necessary. Detailed information is provided in the documentation concerning controlling software. Please observe all instructions concerning machine safety.

Assignment and application of the possible input/output signals concerning the existing port pins is characterized in the following chapter. For an initial start-up, you do not require any signal input or, to simplify matters, you can use the predefined signals..

PLEASE NOTE !

The software WinPC-NC USB is a controlling component only and used within a unit. In order to obtain a real working machine the software has to be completed by PC equipment, working appliance, drive systems and mechanics. The user is responsible for operation safety.

6.2. Pin assignment LPT ports at ncUSB



Pinning LPT1

All signals on the LPT printer ports present 5V TTL level.

Pinning of the first LPT port is as follows:

Pin 2	OUTPUT	direction motor X
Pin 3	OUTPUT	clock motor X
Pin 4	OUTPUT	direction motor Y
Pin 5	OUTPUT	clock motor Y
Pin 6	OUTPUT	direction motor Z
Pin 7	OUTPUT	clock motor Z
Pin 8	OUTPUT	direction motor 4 (i.g.tangential axis)
Pin 9	OUTPUT	clock motor 4 (i.g. tangential axis)
Pin 1	OUTPUT	drilling spindle on/off (default)
Pin 14	OUTPUT	coolant pump on/off (default)
Pin 16	OUTPUT	current reduction (default)
Pin 17	OUTPUT	job running (default)
Pin 10	INPUT	reference switch X (default)
Pin 11	INPUT	reference switch Y (default)
Pin 12	INPUT	reference switch Z (default)
Pin 13	INPUT	sensor/surface block (default)
Pin 15	INPUT	free
Pin 18-25		signal mass (0V GND)

Pinning LPT2

Pinning of the second LPT port is as follows:

Pin 2-9	OUTPUT	analog output, 256 steps binary coded
Pin 1	OUTPUT	free
Pin 14	OUTPUT	free
Pin 16	OUTPUT	free
Pin 17	OUTPUT	free
Pin 10	INPUT	free
Pin 11	INPUT	free
Pin 12	INPUT	free
Pin 13	INPUT	free
Pin 15	INPUT	free

Pin 18-25

signal mass (0V GND)

*User definable
additional
in-/outputs*

All inputs on pins 10, 11, 12, 13 and 15 as well as the additional pin outputs 1, 14, 16 and 17 are free definable and assignable to the desired signals. When the software is delivered above mentioned signals are assigned.

The additional signal Q218 Spindle Speed/PWM, which is a very important one, can only be connected on the pins LPT1:14 or LPT1:17.

The safety signal Q219 Toggle/Chargepump is available on the pins LPT1:16 or LPT1:17 only.

6.3. Determination of axis resolution

For calculating the required distances and speeds it is necessary to tell **WinPC-NC** precisely the definition of the axis resolution.

Two parameters for avoiding calculation inaccuracies

Please open the parameter dialog box parameter - machine and determine the exact data of your mechanics and drives for each axis by the first two parameters.

The parameter axis resolution defines the number of steps or increments per rotation for the corresponding motor. Please consider the electronic settings relating to the macro/micro stepping operation and a possibly integrated reduction ratio.

Motorsteps and distance per rotation

The required value is the number of motorsteps **WinPC-NC** has to create in order to carry out exactly one rotation round the spindle or the shaft.

The second parameter distance per rotation defines the distance which is made exactly by the number of above mentioned motorsteps. With spindles it is the spindle rise or with belts or gear racks it is the graduated circle size of the pinion.

Incorrectly setted axis resolutions cause dimensional inaccuracies and inexact speeds.

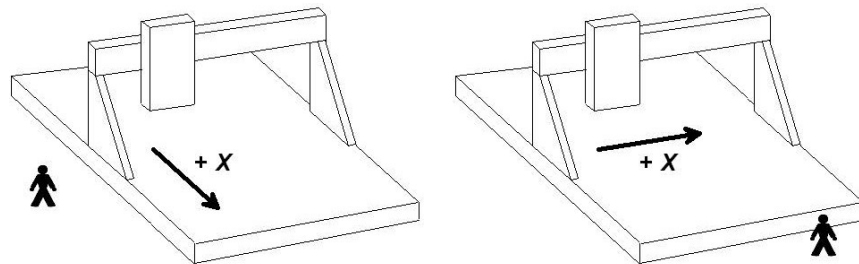
6.4. Determination of direction

Call the function JOG MOVE and move all axes with the cursor button or by mouse clicking to the arrow buttons and check machine direction.

Each button corresponds to a certain axis or direction. Please start with axis X.



Arrow keys showing to the right side are to start a positive movement of the X-axis, i.e. following movement to the right of the machine.



Arrow keys showing to the left side are to move the machine left in opposite direction.

If this is not the case, one of the following errors may have occurred...

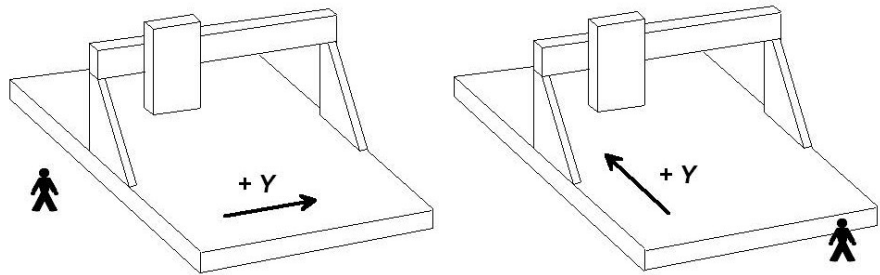
Error	Possible Cause	Solution
Machine does not move	Incorrect port Incorrect port pinning Maschine off	Port selection at ncUSB Control of pinnings Check whether motors are energized and provide supply
Movement of the wrong axis	Incorrect port pinning	Control and adjust the port pinnings

Axis X moves always in opposite direction, i.e. moves to the left by activated right-hand arrow and the other way round	Direction signal works with incorrect logic	Change the parameter INVERT MOVE DIRECTION under machine parameters
---	---	---

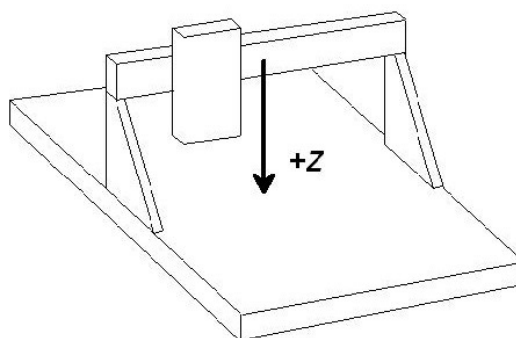
After checking the X-axis a further test of the both other axes is advisable. If necessary, please correct move direction by parameter.



The machine has to move backwards when arrow keys showing to the top.



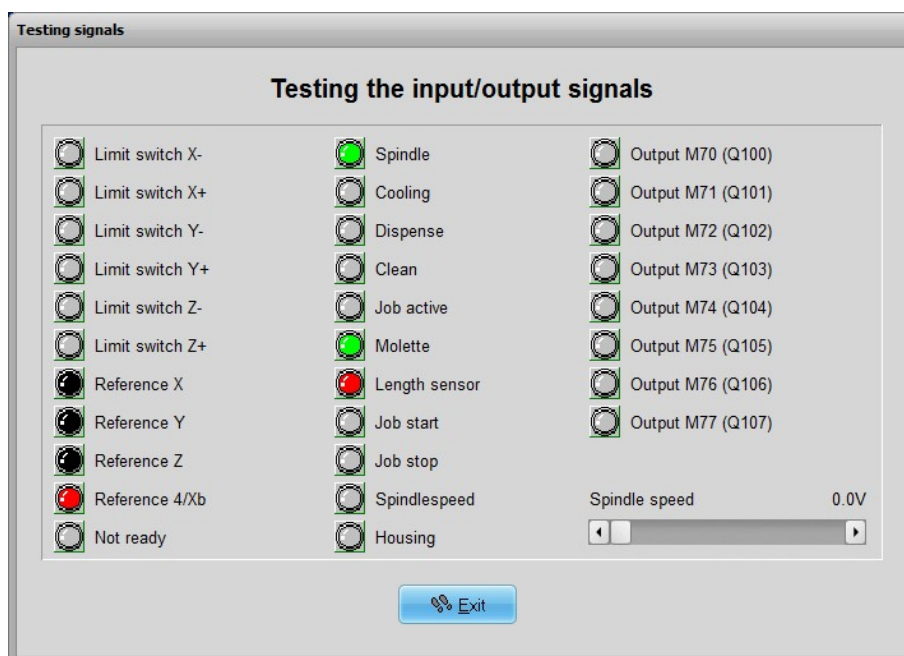
The Z-axis moves down when arrow keys showing downwards.



For a direction reversal of one axis it is only necessary to change the parameter INVERT MOVE DIRECTION.

The move directions to be determined are always relating to the movement of the tool above the workpiece. Please pay attention to the correct setting of move directions by mechanics with moved table. The moved table runs always opposite to the arrow keys in order to guarantee a correct move direction of the tool above the workpiece.

6.5. Adjustment of the reference switches



The way of function of the limit and reference switches can easily be checked with the special function SIGNAL TEST. After activating this function a window appears with LED display for the most important inputs.

Manual activating of the switch

After manual activating the switch of your machine please observe the reactions on the screen. Each change of switch must be clear displayed. The LED flashes of the reference switch should change the color from black to red.

The definition of the reference switches has been correctly determined when the LED is black in unpressed condition and changes to red in pressed condition.

Depending on the used switch (NO contact – NC contact) the switching logic may be misapplied, i.e. red colored with the unpressed switch and black colored with the pressed switch. If this is the case, a change of the switching logic of the reference switch by parameter is required.

Determination of switching logic

Please carry out the following steps:

1. Open parameter dialog box parameter-machine-signals
2. Select the corresponding reference switch in left-hand window
3. Select the inverted entry in the drop-down menu below and accept the setting, e.g. change from LPT1 PIN 10 to LPT1 PIN10 INV
4. Save the latest setting and recheck by function SIGNAL TEST

A detailed description of the signal wizard and the possibilities for defining and setting the inputs and outputs is listed in the next chapter.

6.6. Sequence and direction of reference move

In the following step settings for reference move of the machine are stipulated.

Please carry out the following steps:

1. Open parameter dialog box parameter-machine

2. Under reference switch at pos./neg. end you have to adjust on which axis side the switch is located. If it is not possible to be directly ascertained, change again to the function JOG MOVE and check by moving and observing the position counters and note to which position the axes move to in the moment when they are approaching the switch.
3. After the setting of all switches there is nothing else to do than selecting the reference sequence. The desired sequence can be stipulated in the drop-down menu. Normally it is always the Z-axis which starts moving upon to it's switch followed by the other axes.
4. If you intend a reference move of one or several axes into positive direction, it makes sense to place the machine zero point not at the switch but define another position by parameter.

6.7. Control of settings

After respecting all items a reference move with your machine should not cause any problems. Subsequent to the execution and saving of all parameter settings please press the function key **F8** or select the corresponding function by menu.

Reference move for control

During the reference move all axes run upwards to their switch according to the adjusted sequence and then downwards the switch with reduced speed. Exactly at the switching edge all axes are stopping and marking the reference point.

If this procedure should cause problems or stop with indicating error messages some of the settings have been incorrectly carried out. The tabulation below provides additional information concerning different kinds of errors and corresponding solutions.

Error	Solution
Axis moves into wrong direction	Define the reference switch at the other end
Axis moves into correct direction, but very slowly	Switching logic has been adjusted incorrectly and has to be inverted and define the reference switch at the other end
Axis moves slowly into correct	Switching logic has been adjus-

direction, but stops on the switch	ted incorrectly and has to be inverted and define the reference switch at the other end
Axis moves very slowly into wrong direction	Switching logic has been adjusted incorrectly and has to be inverted

6.8. Additional steps

The most important tasks have been finished by connecting the machine and setting the parameters. Additional actions are recommended in the following sequence:

1. Determination of the optimal ramps and speeds of all axes by the special function MOTOR TEST.
2. Determination of further speeds for JOG MOVE and REFERENCE MOVE by these values.
3. Desired functions can be enabled under parameter-machine-functions.
4. Connecting additional desired and required input and output signals definable by the signal wizard.

7. Signal wizard

7.1. Using input and output signals

One of the most essential functions of **WinPC-NC** is the management of input and output signals.

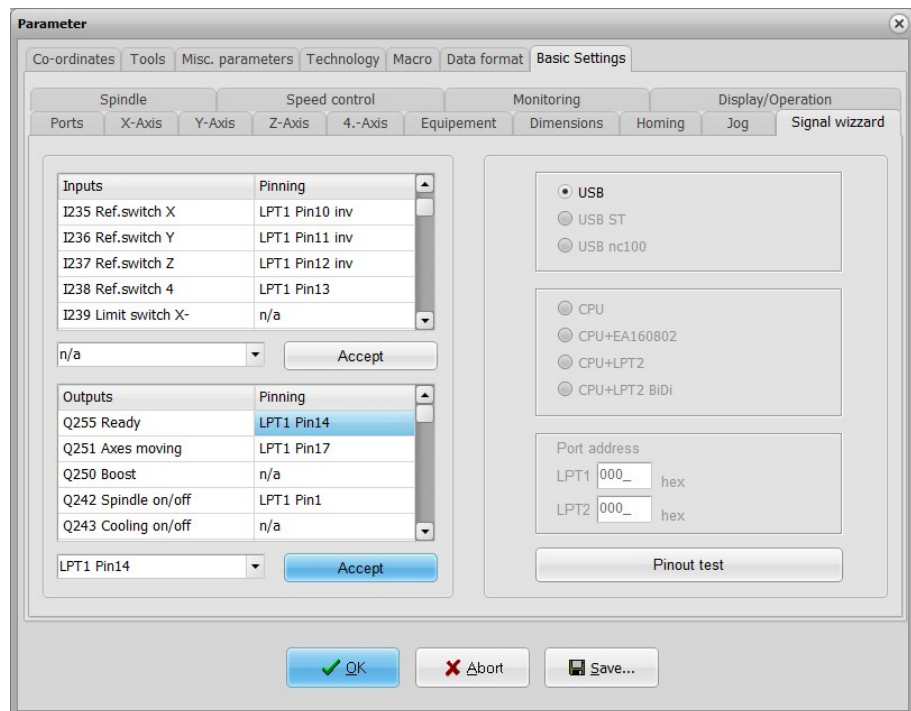
Input and output signals

By means of inputs many useful functions have been implemented, e.g....

- Monitoring of the axes by limit switches
- Moving reference and thereby calibration of the axes
- Synchronization with various signals, e.g. Start
- Monitoring of the protection hood and safety fences

By means of output signals **WinPC-NC** is able to control various additional appliances or reconcile the process with other components. Examples for outputs are...

- Switching spindle and cooling
- Locking the protection hood during a job
- Switching the dispensing pump



Integration of the signals

WinPC-NC is able to manage a lot of different signals, certainly all inputs and outputs are not required for each task. In this way it is quite simple to assign the required signals to the available pins of the LPT ports at **ncUSB** using the signal wizzard.

Each LPT port disposes of 5 input lines at the pins 10, 11, 12, 13 and 15 and 4 additional output lines at the pins 1, 14, 16 and 17.

You can connect the required signals at any possible pin. The assignment is subsequently effected by the signal wizzard which is activated by parameter-machine-signals.

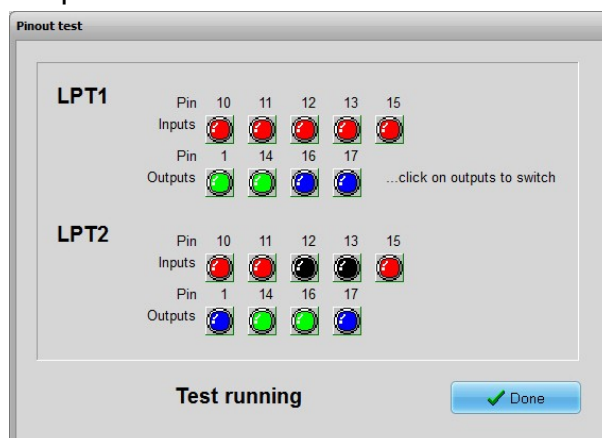
Attention: Please take greatest care when assigning and testing the inputs and outputs. Be always aware of uncontrolled actions of the output signals.

7.2. Assignment of inputs

Assignment of inputs

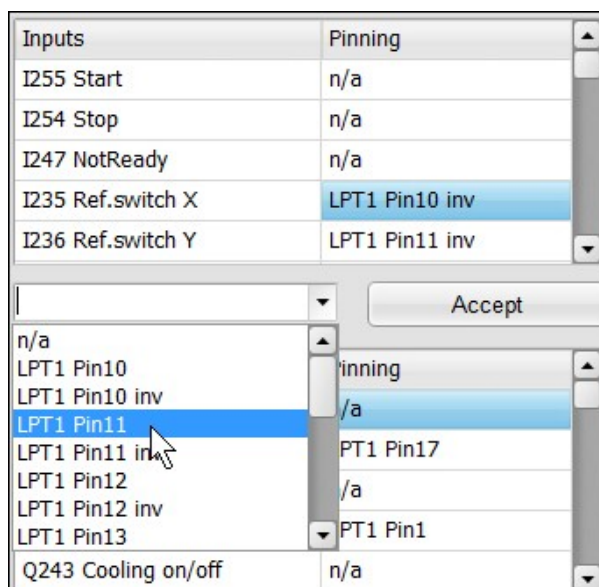
Please execute the following steps for assigning the input lines:

1. Connect the required switch or sensor to the available pin
2. First click to Pinout test and observe the LEDs in the next window which are symbolizing the status of the individual input lines



3. Note or notice the pin number and the switching logic after manual pressing the individual switches. A normally open (NO) contact is characterized when the LED light changes from black to red during pressing. However, a normally closed (NC) contact is characterized when the LED light changes from red to black.

4. Click to the function Done and change again into the window of the signal wizard. Select now the corresponding input signal in the lefthand window (e.g. I235 reference switch X) and open the drop-down menu below in order to assign the pin number (e.g. LPT1 Pin11).



5. With normally open contacts (color change from black to red) always select the normal pin number. With normally closed contacts (color change from red to black) select the inverted pin number, e.g. LPT1 PIN11 Inv
6. After clicking the button Accept the selected pin number is displayed in the assignment table.
7. Proceed in the same way with all other required input signals.
8. Save the parameter settings and if necessary, check once more all assignments with the function SPECIAL FUNCTION-SIGNAL TEST. Herewith a pressed switch must always produce a red LED light, i.e. a HIGH level.

With inputs WinPC-NC reacts only to a HIGH level. Please take care that the switching logic with the possibility for inverting is assigned in the way that an activated switch generates a HIGH level.

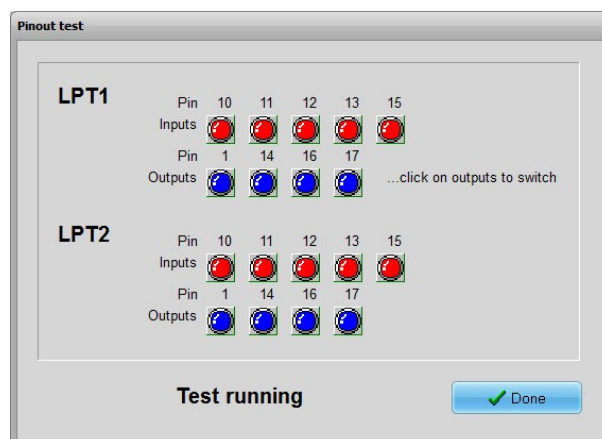
7.3. Assignment of output lines

Assignment of outputs

The assignment of outputs is similar to the assignment of inputs.

Please carry out the following steps :

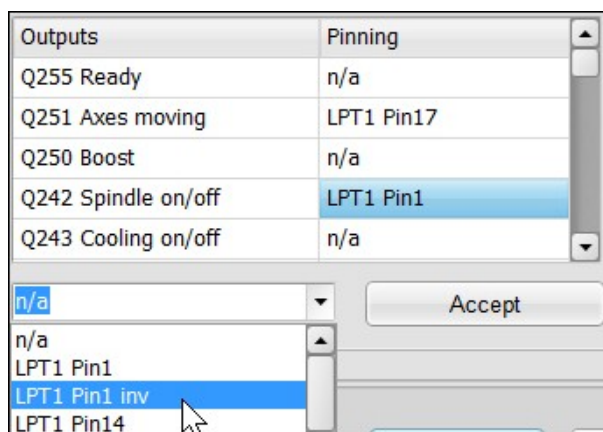
1. Connect the required cables to the available pins and use for control purposes a LED light or a lamp.
2. Click to Pinout test and observe the LEDs in the next window which are symbolizing the status of the individual output lines.



3. Click by mouse on to the possible outputs and observe the level of the cables. A blue LED light signalizes a LOW level or an output switched off. A green LED light signalizes a HIGH level or an output switched on. Please note or notice the corresponding pins as well as the switching logic.

4. Click to the button Done and change again into the window of the signal wizard. Select now the corresponding output signal in the right-hand window (e.g. Q242 Spindle on/off) and open the dropdown menu below in order to assign the pin number (e.g. LPT1 PIN1).

5. After clicking the button Accept the selected pin number is displayed in the assignment table.



Assignment of the pin number for output lines

6. Proceed in the same way with all other required input signal and finally save the settings.

7.4. Input signals available

WinPC-NC is able to manage following input signals:

Possible inputs

Input signal	Assigned action
I255 Start	Starts loaded job
I254 Stopp	Stops immediately each running action with HIGH level, subsequently another reference move has to be carried out
I247 Not ready	Monitors whether the machine is ready and the drives. With HIGH level each running action is immediately stopped. Here the emergency-stop signal is monitored
I235-238 Reference switch	Reference switch of all 4 axes
I239-242 Limit switch-	Limit switch positioned at the negative end of the axis
I243-246 Limit switch+	Limit switch positioned at the positive end of the axis
I234 ZinpositionUp	Synchronizes a semi-automatic drilling job and with HIGH level it can move to the next position at this input
I233 ZinpositionDown	Synchronizes a semi-automatic drilling job. The signals Zinposition top and Zinposition down must be activated by turns.
I229-232 Limit switch summary	Limit switch inputs, if only one switch is appropriate for both ends of the axes, e.g. a flexible switch moving on two cams. Using these inputs, WinPC-NC is not able to recognize the move direction and obviate continuation of the move.
I228 Spindle speed reached	Signal of the milling spindle, when the desired rotation speed has been reached and the job can be continued.
I222 Protection hood	Safety function for protection hoods, light screens and access controls
I221 Length sensor/ Surface probe	Input for surface sensors or tool length sensor

*Possible inputs***WinPC-NC is able to manage following input signals:**

Input signal	Assigned action
I180 JobSingle	Starts a loaded job in step by step mode and stops after each movement
I179 JobStart	Starts a loaded job from the main menu, previous check concerning a change of the job data by the function automatic reload
I178 JobStop	Stops running job
I177 JobResume	Continues interrupted job
I160-169 free I100-109 free	Signals freely to dispose, with macro programming

All additional input signals which are indicated in the table are reserved to special functions or only available in **WinPC-NC Professional**.

7.5. Output signals available

*Possible Outputs***WinPC-NC is able to manage the following output signals :**

Output singal	Assigned actions
Q255 Ready	Indicates whether the controlling software is ready with HIGH level. In the case of error the output turns to LOW.
Q251 Axes moving	Indicates moving (HIGH) or stopping (LOW) axes. This signal can be used for reducing current at the rating class limits.
Q242 Spindle	Spindle Switches on/off the drilling-/milling spindle.
Q243 Cooling	Switches on/off the coolant pump or the spray cooling
Q244 Dispensing/Laser	Switches on/off the dispensing signal or a connected Laser
Q245 Purging with	Indicates with HIGH level an upwards

compressed air	moving and thus a withdrawing Z-axis
Q246 Job works	Job working in HIGH level and thus indicates automatic operation.
Q247 JobEnd	Signals briefly the end of done job
Q248 Collet chuck/molette	Controls an automatic chuck in conjunction with tool changer
Q249 Spindle left-hand/righthand	Switches the rotation direction of the milling spindle, e.g by M-functions M3 and M4 in G code programs
Q219 Toggle/Chargepump	Toogle output continuously switching with 12kHz, can be considered as ready signal for machine control
Q218 Spindle speed /PWM	PWM-signal (pulse width modulation) for speed step display
Q100-115 output M70-87	Additional outputs, programmable with M-functions in G code programs.
Q220-230 free	Signals freely to dispose, with macro programming

All additional output signals which are indicated in the table are reserved to special functions or only available in **WinPC-NC Professional**.

8. Additional Information

8.1. Interpreter

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

- HPGL
- Drilling data
- Multicam
- G code (DIN/ISO)
- ISEL NCP
- Postscript (vector information in EPS and AI files)
- DXF(2D)

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.

Problemlos können Zeichnungen von...

- AutoCAD®
- AutoSketch®
- Eagle®
- HCAM®
- CADdy®
- Google Sketch®
- QCAD®

*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®

...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 following plotters: HP7475, HP Color Pro or HP Draftmaster.

HPGL Syntax **The following commands represent a valid HPGL data file :**

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
FCt;	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
FC300,m;	Activates a programmable macro mf

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;)

HPGL example PU; Start by lifting tool
PA1000,1000; Then move to 1000/1000

SP1;	Select tool 1
PD;	Lower tool
PA1 000,2000;	Move square
PA2000,2000;	
PA2000,1000;	
PA1000,1000;	
PU;	Lift tool

MultiCAM-Interpreter

HPGL extended 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;	Tool lift 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.

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][Tx]  
.....  
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

```
%  
T1  
X1000Y1000  
X1200Y2340  
X2700Y2950T2  
Y1000  
M30
```

Remark lines

```
Start of programm  
Select tool 1  
Drill hole at 1000/1000  
Drill hole at 1200/2340  
Tool 2 hole at 2700/2950  
Drill hole at 2700/1000  
End of program
```

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

*Sample of
drilling data in
format 1*

Short drilling program
in format 1

```
%  
T1
```

Remark lines

```
Start of programm  
Select tool 1
```


X01Y01	Drill hole at 1000/1000
X012Y0234	Drill hole at 1200/2340
X027Y0295T2	Tool 2 hole at 2700/2950
Y01	Drill hole at 2700/1000
M30	End of program

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 NC machines.

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
	G28	Shift of zero point
	G54..59	Zero offset
	G70	Dimension in inch
	G71	Dimension in mm
	G81	Drilling and withdrawal in slow speed Z-depth R-toollift P-dwell time
	G82	Drilling and withdrawal in rapid speed Z-depth R-toollift P-dwell time
	G90	Absolute position information
	G91	Relative position information
	G98	Define subroutines
M-Funktionen	M00	Program stop
	M02	End of program
	M03	Spindle on, clockwise rotation
	M04	Spindle on, counterclockwise rotation
	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..77	Set output 100..107 to HIGH
	M-70..-77	Set output 100..107 to LOW
	M80..87	Set output 108..115 to HIGH
	M-80..-87	Set output 108..115 to LOW
	M90..99	activate programmable macros1-10

	M106	set output HIGH for 3D printer head
	M107	set output LOW for 3D printer head
Misc commands	N	Record number
	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 parametersr
	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
	R	toollift for G81/82 drilling
	P	dwell time at G81/82 drilling
	L	Call subroutines (1..20)

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

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

- The programs must be written using an editor or an external program
- The program name with % sign introduces the real program code, all preceding lines are remark lines
- At least one tool has to be selected and changed, e.g. with T1 M6 in the program head (M6 is absolutely necessary)

- Speeds are adjustable with F commands in mm/sec. or mm/min. defined by parameters
- 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°.
- Up to 20 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.
- In the course of the zero shift with G54ff the definition of the zero point is made by the call line
- G54Xxxx Yyy by defining the relative offset to the current zero point by the coordinate values.
- Activating the zero point is then made by the only command G54 without coordinates

*Example drilling
cycle as
subroutine*

```
%prog2
N001 G90
N002 G00 X110 Y100
N003 L1
N004 G00 Y110
N005 L1
N006 G00 Y120
N007 L1
N008 G00 X0Y0
N009 M30
N020 G98 L1
N021 G01 Z20
N022 G00 Z0
N023 G01 Z25
N024 G00 Z0
N025 G01 Z30
N026 G00 Z0
N027 G98 L0
```

Start of the program
Absolute coordinates
Rapid traverse to 1st position
Call drilling cycle
Next position
etc...

End of program
Define drilling cycle

End of subroutine

ISEL NCP Interpreter

The format ISEL NCP is common to many users and mainly used by machines and units of the companies ISEL and MES from the Iser group. **WinPC-NC** is able to interpret commands of NCP programs, excepting track data and track sections.

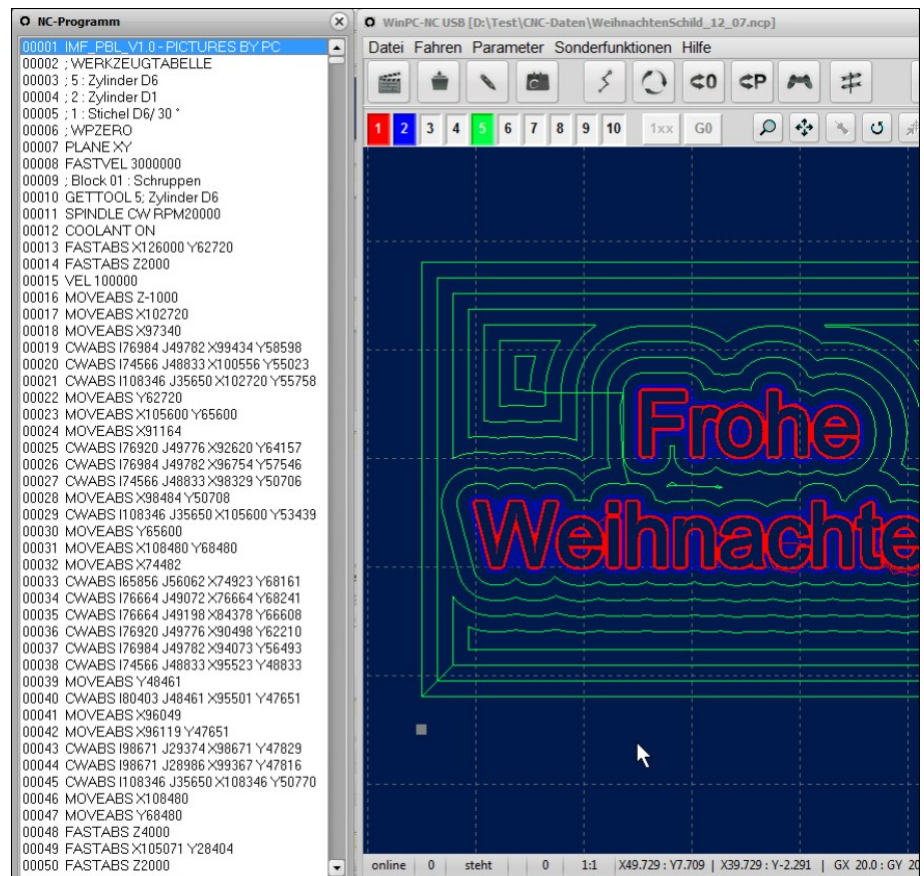
*Simple change
from ISEL
machines*

With this option it is quite simple to process by WinPC-NC current NCP programs with an alternative machine stieg

Interpreted commands are for example :

**FASTVEL
MOVEABS
MOVEREL
PLANE
CWABS
CWREL
GETTOOL
SPINDLE
DRILLDEF**

**VEL
FASTABS
FASTREL
CCWABS
CCWREL
WAIT
HALT
COOLANT
DRILL**



Postscript Interpreter

As an additional interpreter a postscript import filter has been implemented in **WinPC-NC** which filters vector information out of common postscript files, e.g. as it is created by Adobe Illustrator® or Corel Draw®. Other postscript informations in the files are uninteresting for CNC processing and are disregarded.

Please create PS files always without header. Unfortunately, a realtime display of the program of postscript files can not be indicated.

DXF import

WinPC-NC can import files in DXF format but will accept included 2D informations only. DXF is a common used exchange format for drawings and technical constructions and can be created by nearly all drawing programs. Before creating a DXF file for **WinPC-NC** please omit polylines and splines and other complex data elements and unchain all existing groups to single elements.

Please use merely DXF files in version R12 and R14.

8.2. Error messages

Error messages are displayed in a small window in the middle of the screen during operations in **WinPC-NC**. 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 moved on to one of the limit switches. It is necessary to move back from the switch and perform reference run again before continuing work.
- **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**.
- **VConnection canceled**
The connection to the DLL communications module is interrupted. This can happen when other window programs or drives interfere with the realtime tasks. Mostly this message is

made by the power saving function in the Windows system which deactivates the used USB port and thus connection is blocked.

- **Incorrect parameter**
A parameter in the WPI or WPW file is incorrect or the parameter file has been corrupted.
- **Stop signal found**
The stop signal has been detected. Please rectify the fault.
- **Go to reference position first**
WinPC-NC and the controller always need to know what their current position is. Please always perform reference travel after restarting the program.
- **Error xx from controller**
An internal error has occurred in **WinPC-NC** during communication with the DLL module. 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. Another cause for failure may be that the format identification is switched off or that an incorrect data format has been adjusted.
- **Tool not yet defined**
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.
- **Command rejected by controller**
An internal error has occurred in **WinPC-NC** during communication with the DLL module. Please report this error to the program authors, providing as many details as possible.
- **Maschine not ready**
The machine has not been switched on or there is a fault. The NOT READY input on the axis controller signals this condition.

- **USB communication failed / Timeout**
Communication of **WinPC-NC** interferes with the realtime-DLL. Please make sure your machine and USB hardware is connected with the original cable and there is no USB hub between. Disable all power saving functions for computer and especially from the USB port in your hardware manager of Windows.
- **Protection hood not closed**
The protective hood or another protective device has not been closed and therefore no process can be started. The machine can still be set up.
- **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.
Please also check in the generated NC program the height of the Z-axis. The value should not exceed a specific height. Often a clearance distance of 100 mm height has been defined but the machine's passage height is 50-60mm.
- **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 or DIN/ISO 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.

- **Maschine positioned on the reference switch**

You intend to execute a reference move, however there is at least one axis already positioned on the reference switch. Start moving back all axes from the switches by manual operation.

- **Out of memory**

There is insufficient vacant main memory space for processing actions by WinPCNC. Please extend the virtual memory in the system control and restart **WinPC-NC**.

- **Incorrect realtime-DLL**

There are communication problems between the actual real-time module and **WinPC-NC** which are probably caused by another licence.

- **DIN-syntax error / DIN-circular arc error-**

There is an incorrect command in the indicated line of the DIN/ISO file or the programmed circular arc does not match the indicated center point..

- **Error in speed**

The defined speed does not match the corresponding machine parameters.

- **Keypad error**

A defined key pad does not communicate any more or cabling is incorrect.

- **WinPC-NC does not run with this DLL**

Both program files, WINPCNC.EXE und WCNCCON.DLL are not compatible with each other or are not issued under the same licence. Please re-install and activate the program again.

- **Profile- file incorrect / incompatible**
The selected profile file is incorrect or not compatible with this **WinPC-NC** licence.
- **Failure in loading profile**
An error has occurred while loading the selected profile and thus the procedure could not correctly be finished. Please check all parameters and settings prior to further use of the system.
- **G00/01 after G54 only one axis**
Zero point setting in G-code or DIN/ISO programs and using G54ff commands require a command to move and continuation has to be effected by all axes.
- **3D-mouse not found / not installed**
The 3-mouse of 3DConnexion has been defined but could not be found in the system. Please check drivers and relevant connection.
- **Invalid licence file**
The licence file WINPCNC.LIC is corrupt or not according to the used USB hardware. Please contact us by mail to solve this problem.

8.3.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.

We have taken great efforts already in the past to provide an optimum of solutions concerning special functions and options. As a result we have launched successfully our professional version, called **WinPC-NC Professional**. This application is absolutely qualified for individual operation purposes and for controlling specific accessory components.

Here are some examples:

- Dispensing plants for glass inserts and reconstructed motif glasses
- Laser welding plants and laser marking devices
- Plasma cutting plants and oxy cutters
- Laboratory applications for complex measuring tasks
- and there are many other applications besides

In conjunction with the axes controller of **Win-PC-NC Professional** you have also the possibility for direct control out of external programs. The communication report of the axes controller is carefully documented and can be provided on request. Thus the user is able to apply any other host systems independent of the **WinPCNC** user interface.