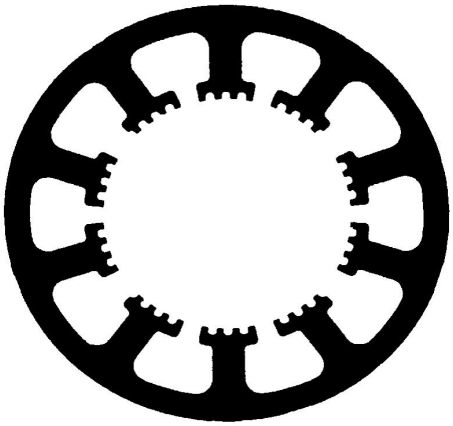
milling, plotting, engraving,
drilling, grinding, dispensing,
cutting, and much more
with...

WinPC-NC



Software that turns your
PC into a universal stepper
motor controller

Version 3.00

July 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*** 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 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 to the email address below. 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

Safety Notices

NC equipment generates magnetic fields which can cause injury, death, or equipment damage (e.g. heart pacemakers). 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 unattended 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

About this manual

The ***WinPC-NC*** manual is divided into chapters:

Chapter 1 :	The basics: computer requirements and potential uses.
Chapter 2 :	Quick start guide: installation, initial program set up, and use.
Chapter 3 :	Description of all program features.
Chapter 4 :	2D-CAM functions: Sorting and tool diameter compensation.
Chapter 5 :	Parameter and option descriptions.
Chapter 6 :	Detailed, step by step start-up procedure.
Chapter 7 :	Signal Wizard: How input and output signals are defined and assigned.
Chapter 8 :	NC data format interpreters and error messages.

Definitions

Job or NC file — A file with NC data which is read and processed by ***WinPC-NC***. The file may contain many types of data - milling, plotting, engraving, drilling, and grinding.

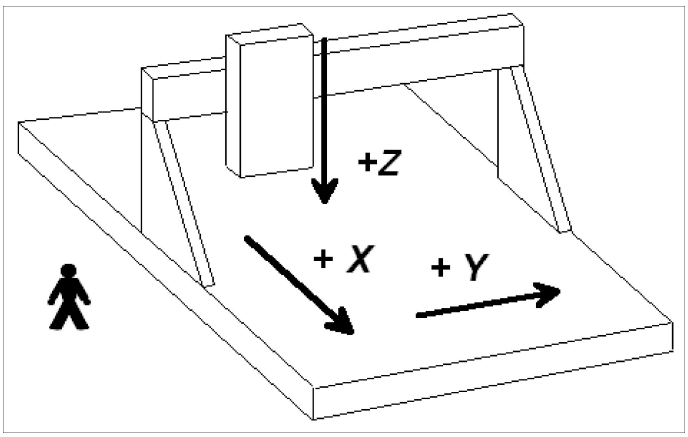
Job or process — The reading of an NC file and its resultant effects on the attached NC machine.

Command — An individual instruction in the job file which gives rise to one more actions on the attached NC machine or in ***WinPC-NC***.

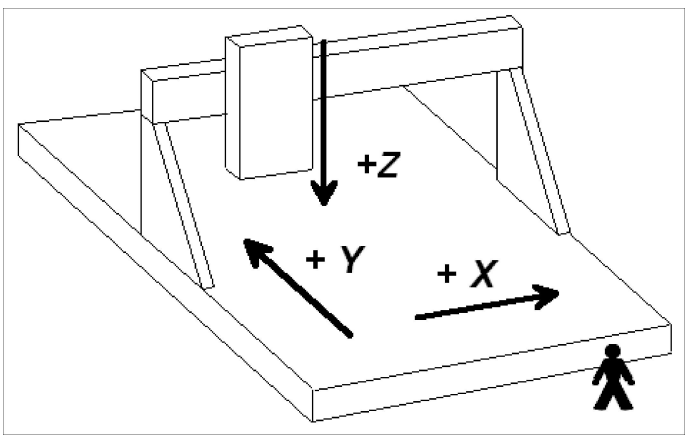
Button — Clickable field that performs a specific function.

Checkbox — Clickable box which activates or deactivates a specific parameter or function (e. g. signals). An activated checkbox is marked with a cross.

Axis assignment depends on how you view your machine:



Axis assignments with your view parallel to the gantry



Axis assignments with your view perpendicular to the gantry

Use of typography

Keyboard action	Normal script enclosed in a box: ENTER
Cursor keys	Normal script enclosed in a box: UP LEFT PGDN
Menu names	Hyphen separated words, all in upper case: **FILE-DISPLAY**
Messages	Italic script: '*Perform reference movement?* Y/N'
Function names	Capital letters: **SIGNAL TEST**

WinPC-NC Versions

WinPC-NC Light is our low-cost offering, perfect for beginners. One can engrave, mill, drill, and create PCB boards and perform 2.5D milling. Machine control is handled via ***nc100***, an external USB module capable of controlling one IEEE 1284 compatible port (LPT).

WinPC-NC USB is an enhanced version of ***WinPC-NC Light*** that adds additional NC data format interpreters and many useful features. Operations may be performed in 2.5D, as well as with 3 and 4 axes. Machine control is handled via a choice of external USB module:

- ***nc100***
  Supports one IEEE 1284 compatible port (LPT).

- ***ncUSB***
  Supports two IEEE 1284 compatible ports (LPT).

WinPC-NC Professional is our version designed for industrial use. Machine control is handled via ***CNCCONS***, our external motion controller. The ***CNCCONS*** controller offloads timing and CPU intensive operations from the PC. True real-time control with the utmost stability and reliability is possible. Custom hardware configuration, software, and mounting services are available.

Please contact ***WinPC-NC*** if there are special requirements (e.g. direct LPT control, SMC cards, custom motion controllers) for your application.

1. What can *WinPC-NC do*?

The ***WinPC-NC*** program turns a PC into a universal NC controller. Applications include:

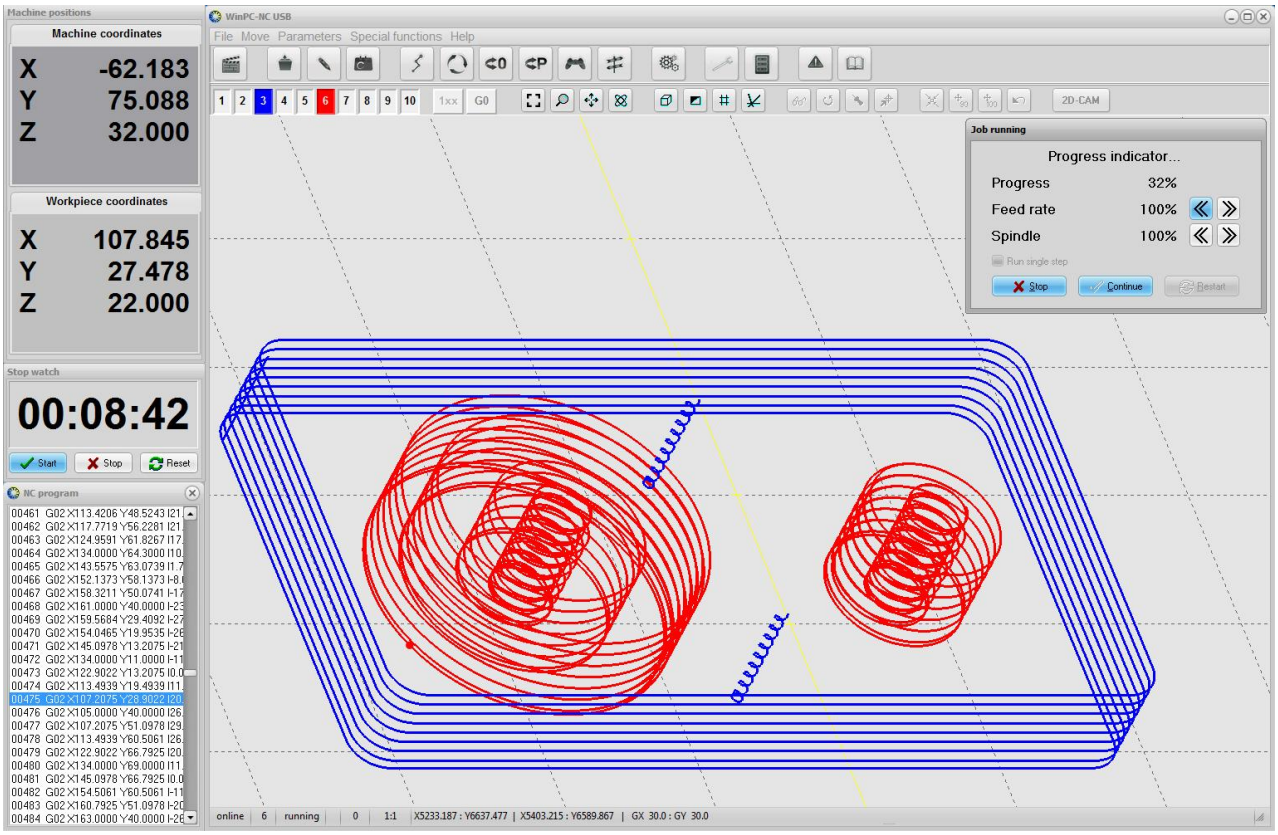
- **Milling**
- **Drilling**
- **Grinding**
- **Engraving**
- **3D Printing**
- **Fabric and plastic sheet cutting**
- **Plotting**
- **Welding**
- **Laser cutting**
- **Plasma cutting**
- **3D Scanning**
- **Tangential cutting**
- **Foam cutting**
- **Foil cutting**
- **PCB construction**
- **Dispensing**

A 3 axis milling machine

WinPC-NC:

- Offers a well thought out user interface that is easy to learn. A context sensitive help system makes getting work done easy.
- Offers an almost unlimited working range
- Handles stepping frequencies up to 80 Khz (interface dependent).
- Uses keyboard or mouse commands to precisely jog (move) around.
- Displays your NC data graphically. Scale, zoom, shift, rotate, and reflect as necessary.
- Allows axis resolutions, speeds, backlash compensation, and ramp length for the X, Y, and Z axes to be set within generous limits.
- Supports a 4th axis. It can be programmed as U, V, W, A, B, C, or tangential axis T
- Supports home and limit switches: up to 10 inputs and 8 additional outputs can be set within generous limits
- Outputs signals that can be used for synchronizing with external equipment (e.g. job start, spindle speed reached, machine ready)
- Recognizes HPGL, DIN/ISO, MultiCAM 2D and 3D, drilling formats, ISEL NCP, EPS/AI and DXF (2D).
- Has extensive tool management functions. Tools can be selected individually, colors can be assigned; feeds and speeds specified; unique characteristics handled.
- Supports automatic tool changers. Up to 10 magazine positions are supported.
- Has CAM functions for cleaning up drawings. Sorting by tool and position; tool diameter compensation.
- Handles speed settings as slow as 0.01 mm/s and as fast as 1000 mm/s.
- Offers an integrated high-performance file editor. An external editor may be specified and used as well.
- Supports speed control of drilling/milling spindles. Rotation can be clockwise or counterclockwise.
- Supports Metric and Imperial units. Dimensions may be millimeters or inches; speeds may be mm/sec, mm/min, or inches/min
- Works with multiple languages: 20 languages are available at the time of writing; more are being added.

- Has a flexible macro language. Functions can be called at various points in a job.
- Allows feed and spindle speeds to be overridden at any time.
- Supports automatic and manual tool length compensation.
- Can be remote controlled via external inputs.
- Supports hand controllers (pendants).
- Supports independent machine, job, and application profiles.
- Has Manual Data Input (MDI) capability. Enter low level commands.
- Supports job nesting



The ***WinPC-NC*** user interface

2. First Steps

2.1. Requirements

- Personal Computer (PC) - desktop, laptop/notebook, or netbook - running at a minimum of 2 Ghz clock speed.
- 50 MB of free storage space.
- 1 GB of RAM. 4 GB or more is strongly recommended. Memory requirements grow with the size of the NC file being processed.
- Screen resolution of 1024x768. A higher screen resolution is strongly recommended as this allows for increased flexibility.
- A keyboard.
- A 2 or 3 button mouse. A scroll wheel is strongly recommended.
- 32-bit or 64-bit version of MS-Windows - Windows 7 or newer.
- The OS and all its drivers should be up to date.
- Deactivation of any CPU and system power saving settings, screen savers, hard disk indexing and defragmentation programs, automatic update checking, automatic shutdown capabilities, and any CPU clock speed adjustment features.
- A free USB 2 port for an external USB module (e.g. ***nc100***, ***ncUSB***)**.** The power saving settings for USB ports and device manager must be deactivated.
- Do not have a USB hub or switch between the PC USB port and an external USB module.
- When using ***nc100*** or ***ncUSB***, be sure to use the bundled USB cable.

WinPC-NC is not limited to working with the ***nc100*** and ***ncUSB*** modules. You're free to integrate any external USB module yourself or we can provide integration services.

WinPC-NC USB ensures that correct USB driver is available for operation with the ***nc100*** and ***ncUSB*** USB modules***.***

ncUSB module and cable

nc100 module and cable

2.2. Installation

When ***WinPC-NC*** is first being installed, do not connect an external USB module (e.g. ***nc100***, ***ncUSB***) until after completing the installation procedure.

WinPC-NC is installed using a user-friendly installer program. Insert the CD into your CD/DVD drive and wait for automatic installation to start. If the installation does not start automatically, open the CD drive and start the installation by double clicking SETUP.EXE.

The installation wizard guides you through the installation procedure. When completed, the target directory will contain the following files:

WINPCNC.EXE	The ***WinPC-NC*** program
WINPCNC.WPI or WTI	Parameter file
WINPCNC.WPW or WTW	Machine tool file
WINPCNC.WPO	CAM function settings
WINPCNC.MAC	Initial or user defined macro definitions
WINPCNC.PDF	This manual in PDF format
WINPCNC.LNG and LNG2	Multilingual messages and texts
WCNCCON.DLL	Communication library for ***nc100***/***ncUSB***
README	Important, late breaking information
***.DLL**	Files required by the system folder
***.PLT *.SMM *.DIN *.NCP *.EPS *.DXF**	Sample NC files

Be sure to open and read the README file as it will contain important information!

After the ***WinPC-NC*** installation completes, plug in your external control module. For the ***nc100*** and ***ncUSB***, choose any available USB 2 port.

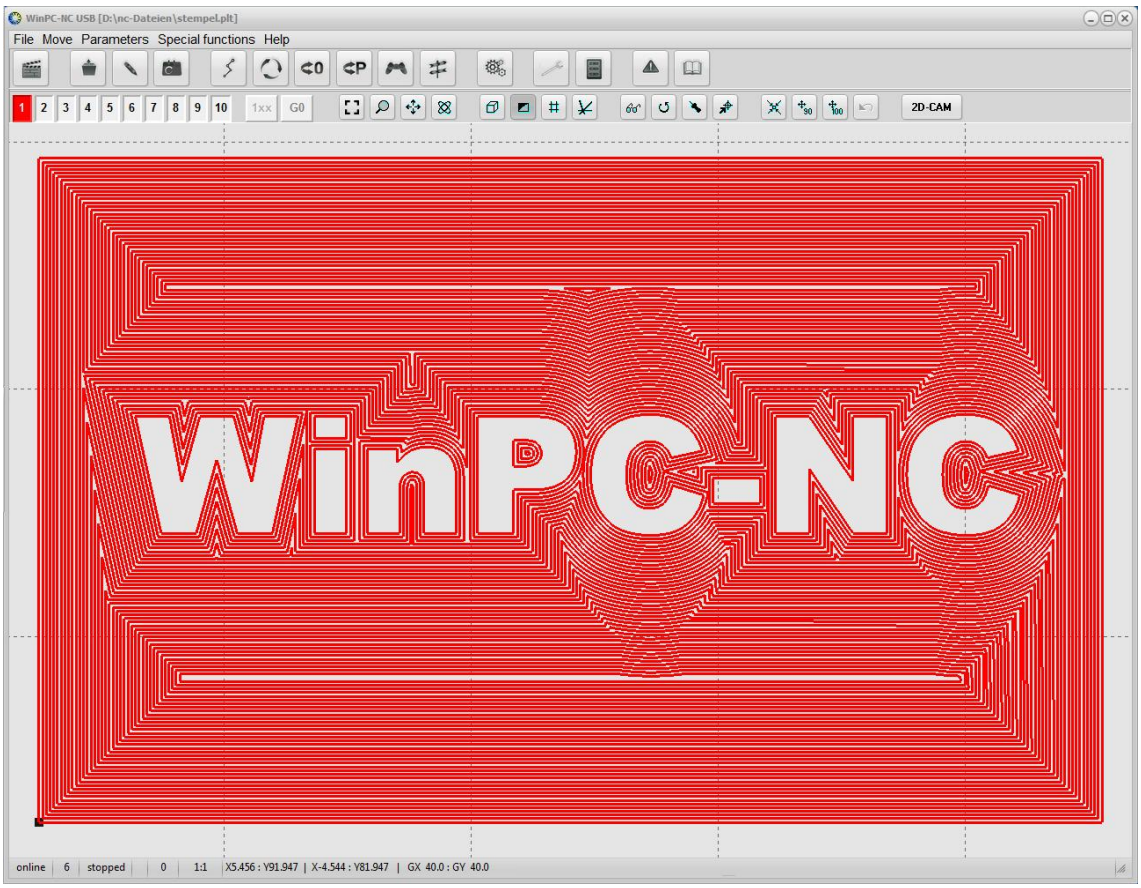
Connecting an ***nc100*** or ***ncUSB*** module will result in an immediate hardware identification message followed by driver installation. When driver installation completes, you will receive message that the module is ready for use.

2.3. Launching the program

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

If ***WinPC-NC*** came bundled with your CNC machine, the first time the program starts, a menu will appear asking you to select your machine. Your selection will ensure that the machine parameters and settings appropriate to your machine will be used.

The ***WinPC-NC*** working window will appear.



WinPC-NC main window

If the *ncUSB* or *nc100* module is not plugged or cannot be recognized, an error message is displayed, and *WinPC-NC* runs in its demonstration mode.

All buttons and menus can be activated by clicking on them with the mouse. Descriptions of the button can be found on the next page.

WinPC-NC Window section descriptions

Title bar	The ***WinPC-NC*** working window title bar.
Menu bar	Drop-down menus open from the menu bar.
Button bar	Useful buttons for rapidly performing many actions.
Tool bar	Buttons for NC data display and manipulation.
Display Area	Graphically displays the currently loaded NC file.
Status bar	Multipurpose status display area: The ***nc100***/***ncUSB*** status (online/offline). The operating status of the software. The operating status of the NC machine. The emergency stop status (in red). Error and warning messages. Helpful texts concerning the action being performed.

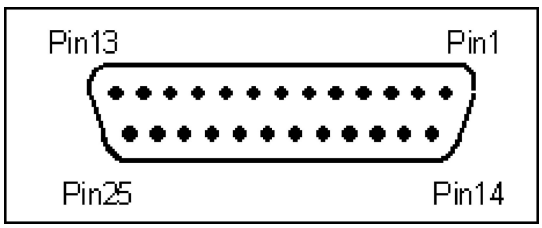
The button bar icons:

	End the program		Joystick movement
	Open an NC file		Initialize the machine
	Load current or new file in the editor		Parameter settings
	Call profile management		Handling of molett
	Start Job		Tools

	Jog		Current version information
	Move to current zero point		HELP
	Move to current park position	Last Button	Current technology function if selected

2.4. Initial setup and test run

Executing an initial test is simple. Connect the NC machine motor signal wires according to the following assignments:



IEEE 1284 parallel port (LPT) pins

***WinPC-NC* uses the following signals for motor control:**

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

Check and double check the wiring before using the MOTOR TEST or JOG function of ***WinPC-NC***. Watch the machine carefully and be ready to use the emergency stop should an unexpected motion occur.

If any unexpected motion occurs, power off the NC machine and disconnect the ***nc100***/***ncUSB*** module. Inspect the signal connections and make any necessary adjustments. Repeat this procedure until everything is working as expected.
Detailed instructions concerning the start-up procedure are given in a separate chapter.

2.5. *Exiting WinPC-NC*

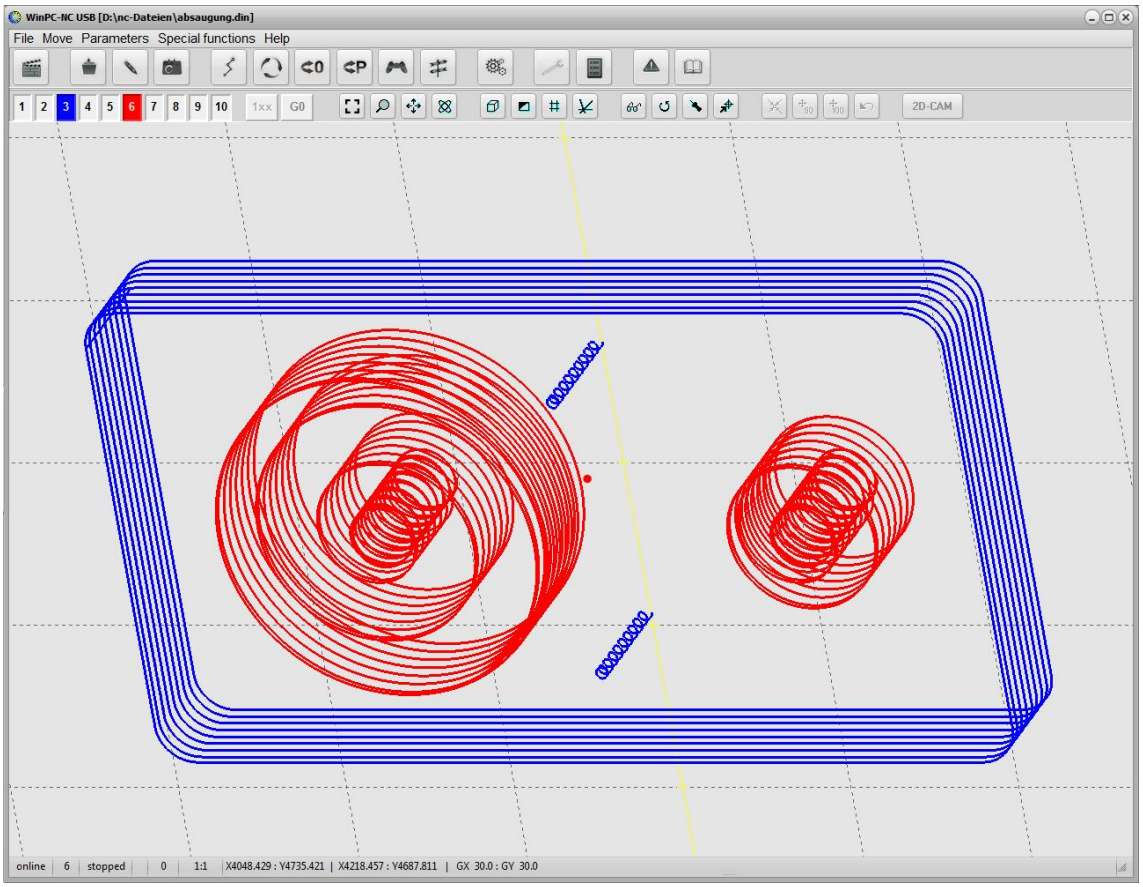
You can exit ***WinPC-NC*** at any time by clicking on the program end button or by selecting FILE-EXIT.

3. Operating WinPC-NC

3.1. Graphical display of an NC file

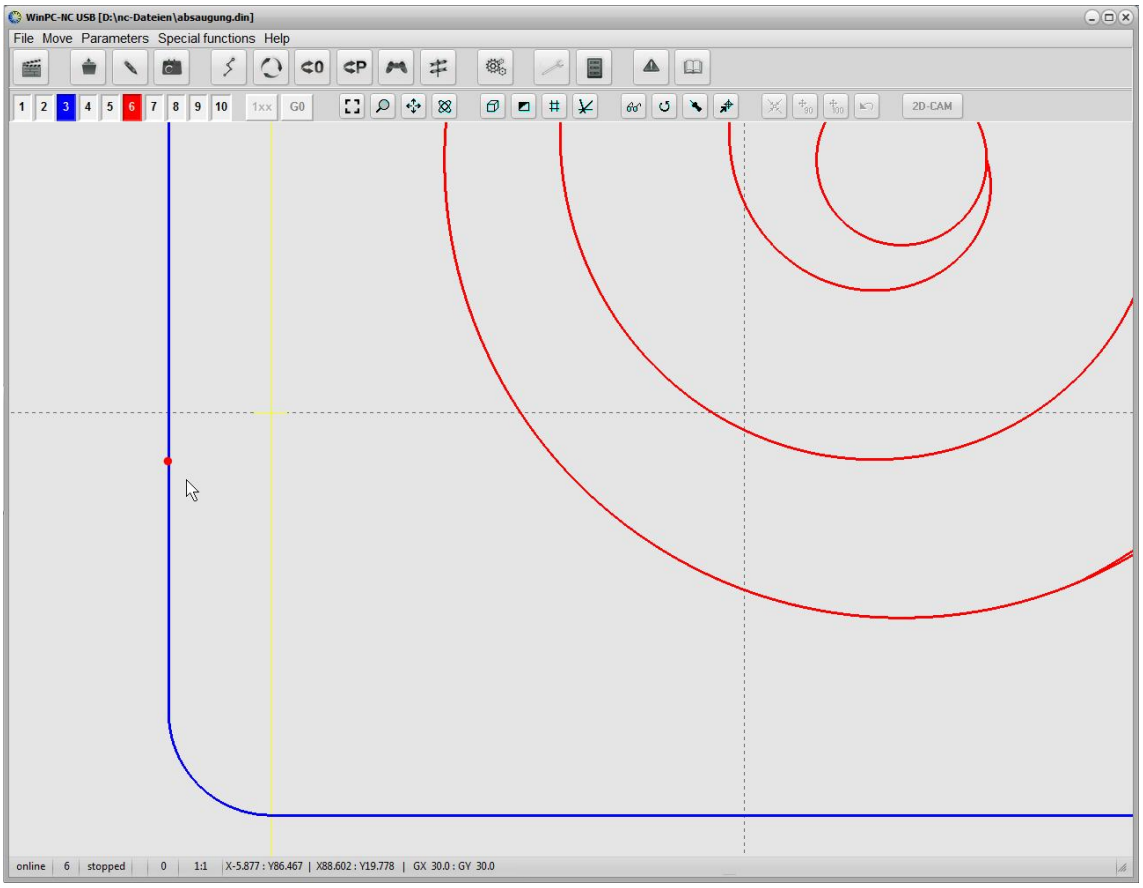
WinPC-NC fills the *Display Area* with a graphical representation of an NC file as soon as its data is loaded. All contours or vectors are displayed in color.

The initial view presents the NC data looking down at the XY plane. Other perspectives may be selected; the NC data can be rotated, scrolled, and zoomed. Measurements can be estimated by referring to the grid-lines.



Initial perspective view after loading an NC file

Here we see an NC file with its data zoomed. The work piece zero point appears as a small circle and the current position of the machine appears as a small red dot.



Zoomed NC file

The Tool bar buttons (left to right):

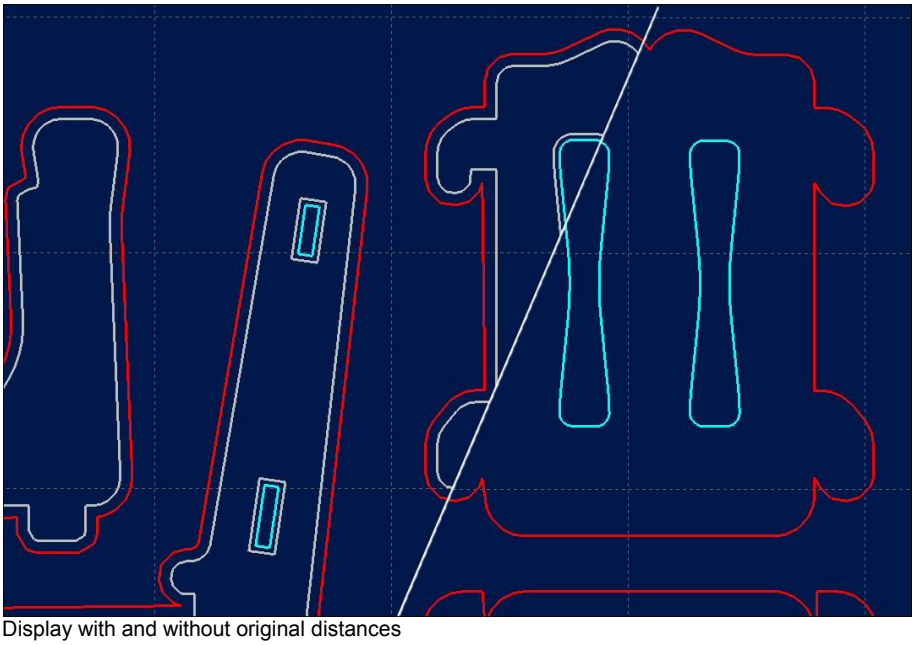
Activate/Deactivate tools

When the NC data is initially displayed, each tool involved in the job is displayed with a different color. Each tool can be activated or deactivated from the display by clicking on its number. The *Display Area* will immediate be updated with the current selections.

In the example image on the left, tools 1, 2 and 4 are activated. Tool number 3 is deactivated and tool number 5 is not involved in this job.

Display or shield original distances

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



Display with and without original distances

Toggle trial runs

Unnecessary trial runs will be displayed or suppressed.

Reset display view

Restores the Display Area to its initial condition. Any changes to the NC data will be lost. If a mode is selected, it will be deselected.

Zoom the data

The NC data may be zoomed - *magnified* - by activating this mode. The mouse cursor will change to notify you that zooming is in effect. Click the left mouse button in the *Display Area* and moving the mouse around will cause the NC data to zoom.

Zooming can also be done by moving the mouse scroll.

Scroll the data

The NC data may be scrolled – *moved around* - by activating this mode. The mouse cursor will change to notify you that scrolling is in effect. Click the left mouse button in the *Display Area* and moving the mouse around will cause the NC data to scroll.

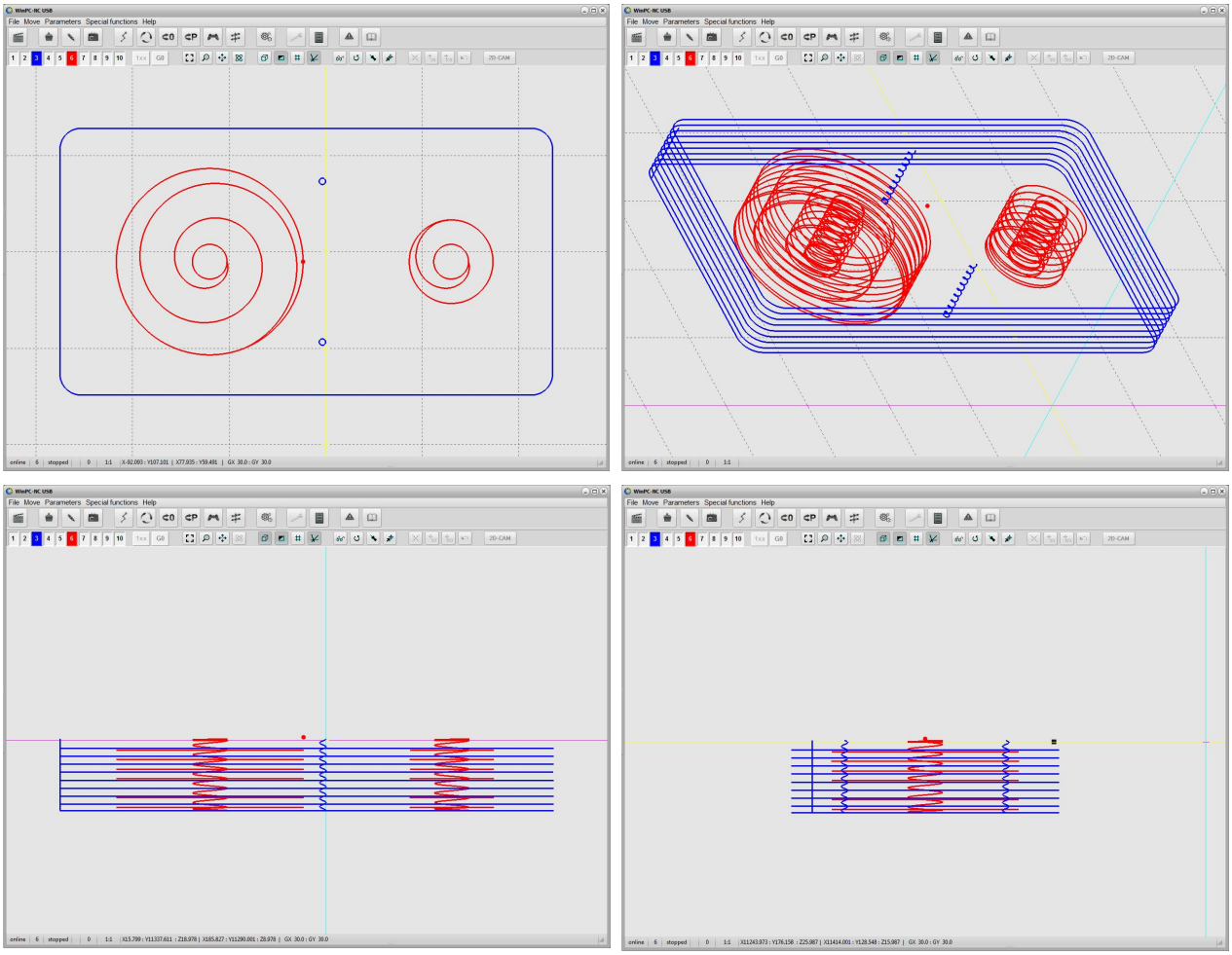
Scrolling can also be done by moving the mouse around while in the *Display Area* and keeping the right mouse button pressed.

Orbit the data

The NC data may be orbited - *rotated around* - by activating this mode. The mouse cursor will change to notify you that orbiting is in effect. Click the left mouse click in the *Display Area* and moving the mouse around will cause the NC data to be orbited.

Standard perspective views

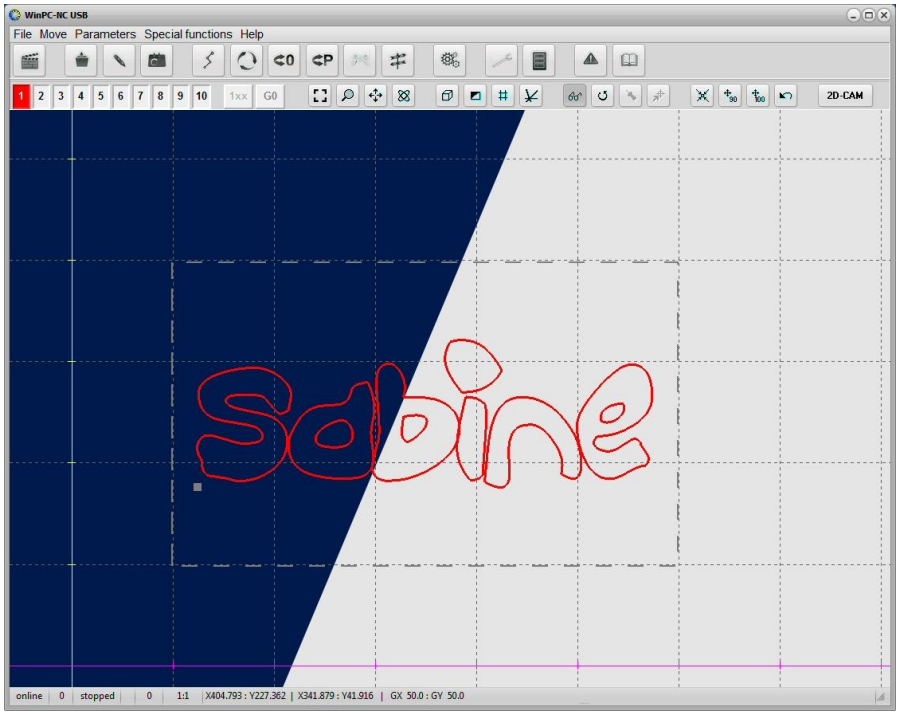
Clicking this button cycles the display through 4 useful perspectives.



Four views of the workpiece with axes visible

Toggle background color

Toggles the background color of the *Display Area* between light and dark. This is often useful when viewing difficult to see data.



Light and dark background color

Grid on/off

Activates or deactivates the grid lines which serve as a useful reference. They are automatically scaled according to the drawing size. The grid size is indicated by the GX and GY data in the *Status Bar.*

Cycle axis display

Cycles the display of the axes. The axes will be hidden, only XY will be displayed, or XYZ will be displayed.

Select the display area

Display only the piece to be produced and its corresponding data. It is also possible to display the defined work piece, or the entire machine work space.

Adjusting the display area makes it easier to visualize the data and make any adjustments.

Rotate data by 90 deg around Z

The NC data is rotated by 90°. This can quickly make sections of the data more easily visible.

Set current machine position

The current machine position is represented by a small red dot in the *Display Area*. This function moves the current position to anywhere along a tool path. Simply click on the tool path and the current position is set to that location. The file zero point will be re-calculated.

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

Move machine to position

This function moves the machine to the position indicated by the cursor. Movement will be performed using a machine rapid move command. Care must be taken when using this command so as to not cause the machine to collide with anything.

Center data in the working area

When working with engravings and millings it is sometimes advisable to center all of data to a pre-set area or on an *empty part*. The centering function accomplishes this. The zero point parameters will be re-calculated and no size change will occur.

Centering area means pre-defined work piece area. Any definitions 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.

Scale Data

Scale the work piece data by 90% or 100%.

Undo scaling and centering

Previous scaling and centering operations are reverted. The previous parameters are restored.

2D CAM functions

WinPC-NC can apply CAM functions to many 2D data formats (e.g. HPGL, EPS/AI, DXF(2D), and drilling). The 2D data can be cleaned up, sorted, and optimized. Tool diameter compensation can be specified.

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

***WinPC-NC* allows many of its windows to be *torn* from the initial window and positioned anywhere you like. The position of any *torn* windows is retained from session to session.**

3.2. Drop-down menus and function keys

WinPC-NC has an easy to use user interface. All functions may be found in drop-down menus. Many of the menu entries have an accelerator, a function key assigned to make rapid access possible.

The drop-down menus are functionally grouped - file, movement, parameters, special functions, and help.

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:

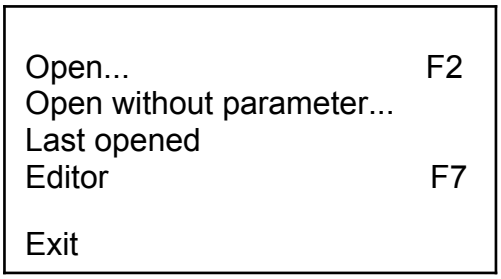
F1	Activate the help system
F2	Load NC file
shift-F2	Load parameter data
F3	Start job from beginning
shift-F3	Start job from defined point
F4	Move to XY origin
F5	Jog (mouse or pendant)
shift-F5	Jog (joystick)
F7	Load file into editor
F8	Start machine initialization
F9	Move to parking position
F10	Open pull down menu

3.3. The individual menus

Each menu and its items are explained in detail.

Some menu items may not be available at all times (e.g. joystick control when no joystick is configured).

3.3.1. FILE menu



The FILE menu is used for loading files for analysis and processing, editing files, and to exit ***WinPC-NC***.

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

FILE - OPEN

This menu item loads the selected NC file **and any previous parameter data**.

The standard file selection dialog will appear. Select the file you wish to load. Once the NC data is loaded the *Display Area* will present the data graphically.

FILE - OPEN WITHOUT PARAMETERS

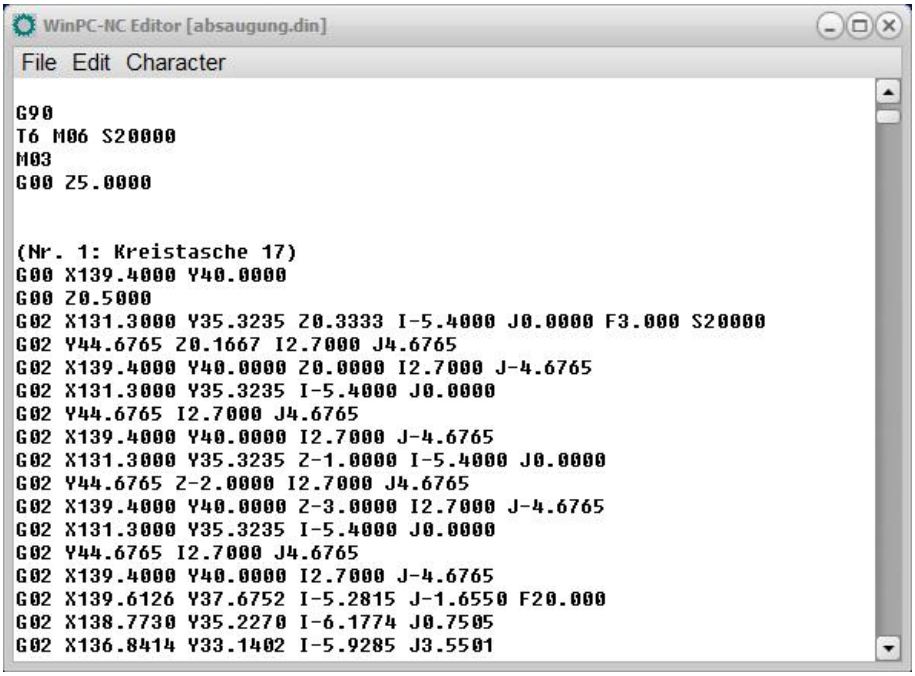
This item loads the selected NC file but **ignores any previous parameter data**.

The standard file selection dialog will appear. Select the file you wish to load. Once the NC data is loaded the *Display Area* will present the data graphically.

FILE - LAST OPENED

This item will display the last NC file that was opened. Click on the file name to load it. Once the NC data is loaded the *Display Area* will present the data graphically.

FILE - EDITOR



The **WinPC-NC** editor displaying a G-Code file

This item activates the NC file editor. The ***WinPC-NC*** editor will be used when an external editor has not been specified.

The editor will start. If an NC file is currently loaded, its data will be presented.

The editor can also be activated by hitting F7 or by clicking the editor button.

FILE - EXIT

This item exits ***WinPC-NC*.** Any NC machine activity will be terminated; all files will be closed.

Clicking the exit button has the same effect.

3.3.2. MOVE menu

Start	F3
Start from...	
Start single steps	
Zero point XY	F4
Park position	F9
Jog	F5
Joystick jogging	
Initialize machine	F8
Select tool	

The MOVE menu contains those functions which are used for controlling the machine and manipulating tools.

Press the alt-F shortcut to open the file menu.

MOVE - START

This item will start the loaded job from the beginning.

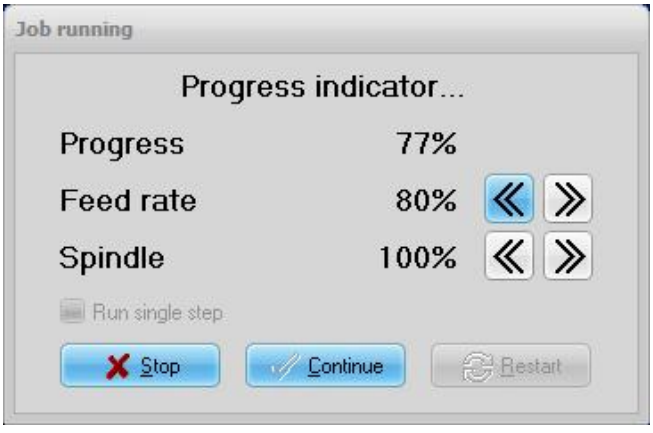
Pressing F3 or clicking the START button has the same effect.

The current tool position, marked by a red dot in the *Display Area*, will be updated in real-time.

A Job running window will appear. The percentage of the job that has been completed is updated in real-time.

The feed and spindle rates may be adjusted, up or down, in steps of 5%. The minimum rate is 10%; the maximum 200%.

Clicking the STOP button or by pressing the (ESC) key will terminate the job. The machine will immediately stop moving (no steps will be lost), the spindle will be turned off, and any coolant distribution will be turned off.



The Job running window

After STOP, a job may CONTINUE from the point it was stopped or RESTART from the beginning.

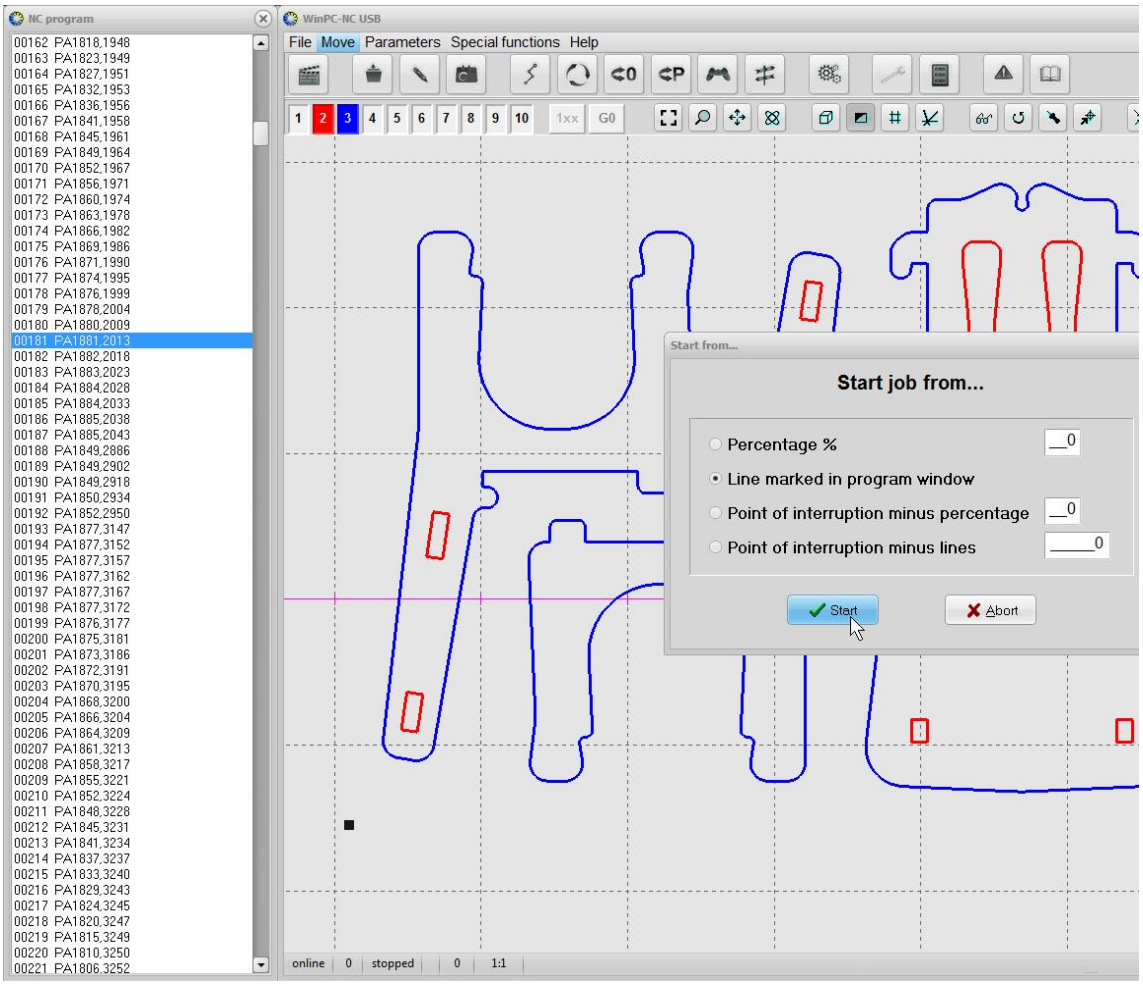
During the interruption, it is possible to change parameters, clean the tool, or jog the machine.

The ability to CONTINUE (from the Job running window) is controlled by a setting in the PARAMETERS menu.

The START-FROM menu item (see below) can be used to continue a job.

MOVE - START FROM...

This item allows the loaded job to be started from the beginning or a designated place. There are four ways to choose the designed place.



Start from... menu

- Start from the specified job percentage
- Start from the line marked in the program window

- Start from the point of interruption less a percentage
- Start from the point of interruption less a number of program lines.

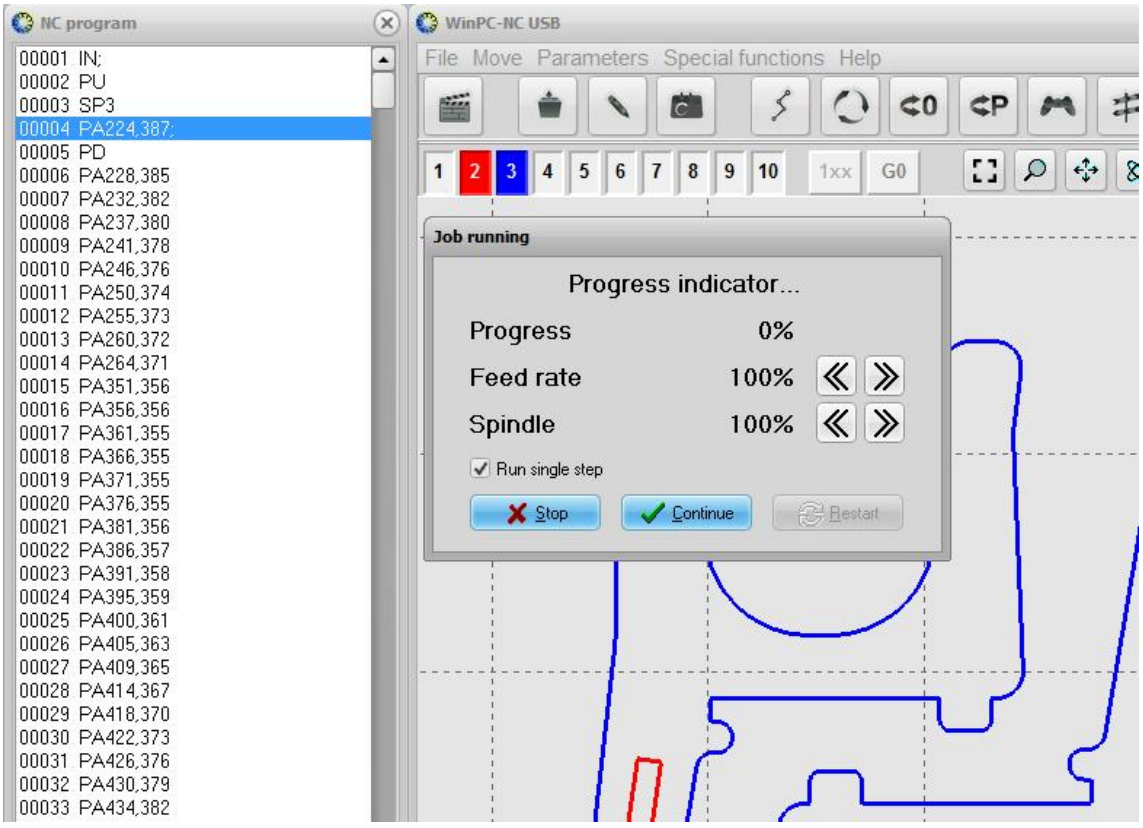
WinPC-NC will determine the point to begin, move to it, and process the job from there. If the job is being continued, the spindle and coolant will also be restored before motion begins.

START - SINGLE STEP

This item runs the loaded job one NC instruction at time. One NC instruction will be performed and ***WinPC-NC*** will wait for the CONTINUE or CANCEL button to be hit.

Moving one NC instruction at a time is useful for debugging a problem or slowing the machine down so one can see exactly what is going on.

Uncheck the single step check box and hit CONTINUE to resume full speed execution.



Start-Single step

MOVE - ZERO POINT XY
MOVE - PARK

These items cause the machine to move to the named location.

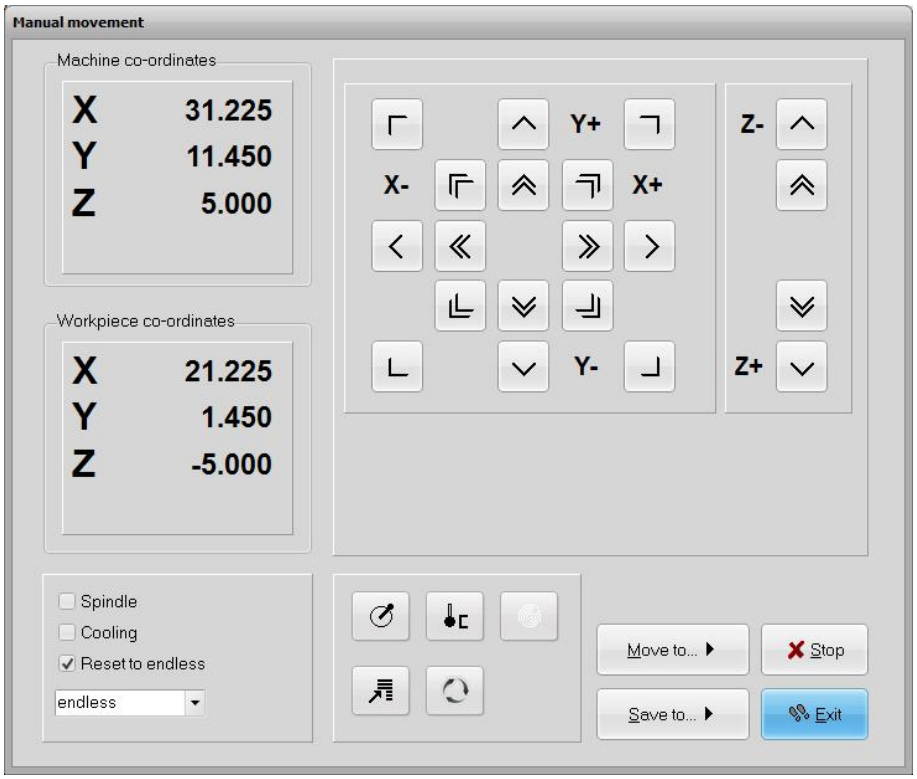
Care should be taken to ensure that a collision does not occur.

The F4 and F9 keys are *accelerators* for these locations respectively.

MOVE - JOG

The item enables manual movement of the machine.

Jogging can also be accessed by pressing F5 or the JOG button.



JOG function

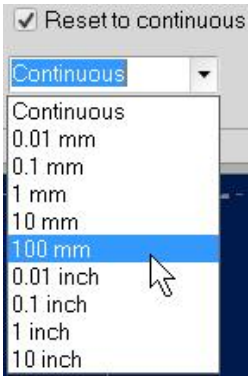
Movement may be performed via numeric key pad or mouse. Step-by-step, precise distance, or continuous motion may be specified.

Briefly pressing a key or clicking the corresponding direction button with the mouse executes a single motion command. Holding a key down or keeping the mouse button pressed causes continuous motion. The changeover time can be defined as a parameter.

The numeric keypad keys - [1] to [9] - correspond to the arrows in the Manual movement window. The [+] and [-] keys move the Z-axis.

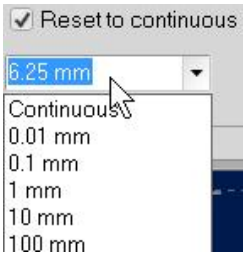
The machine position is displayed in two ways. The first is relative to the the home switches. The second is relative to the zero point of the work piece.

***WinPC-NC* maintains two coordinate systems. The first is relative to the machine and is based on the home switches. The second is relative to the zero point of the work piece. The work piece coordinates are often relative to the bottom left-hand corner of the work piece.**



How ***WinPC-NC*** will move is shown in the lower left hand corner of the jog window.

When *continuous* is displayed, motion will be in discrete steps. If a distance is displayed, motion will be in discrete distances of that size and units.



The discrete distance may be entered or selected from the pop up menu. The size and units of the menu distances are determined by the units setting in parameters (Metric or Imperial).

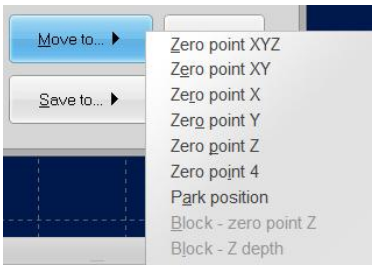
As long as a key or button is pressed, the machine will move. As soon as the key or button is release, the machine will come to a halt. No steps will be lost by this process.

Checking the ***Reset to continuous*** checkbox will immediately reset the movement to steps (*continuous).*

Be careful! Jogging can result in a collision which can damage the machine, spindle, or tool.

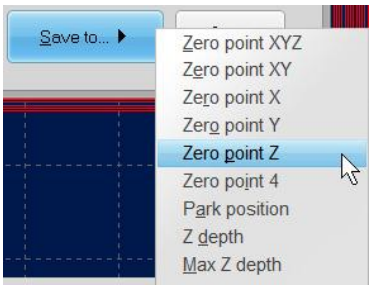
These spindle and coolant check boxes control their respective functions during jogging.

Closing the Manual movement window ensures that the spindle and coolant pump are turned off.



It is possible to move to certain well-known locations. Simply click on the *Move to* button and selection the desired location. The machine will move to location.

Similarly, the current position can be saved as one of the well-known locations. Simply, click on the *Save to* button and select the location.



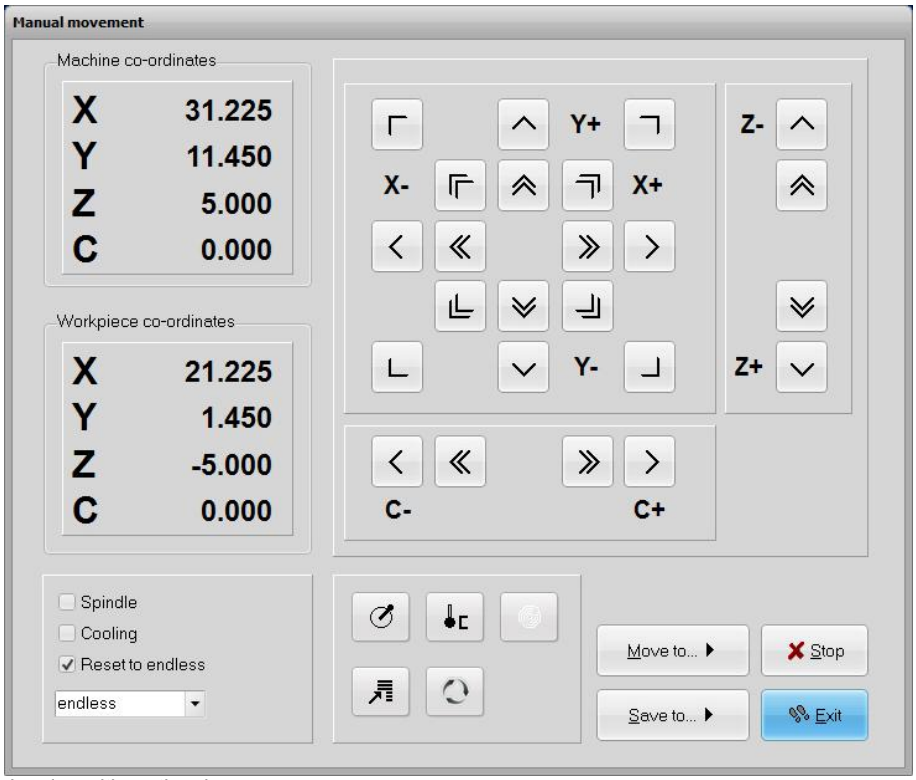
A movement in progress can be stopped by clicking the *Stop* button. Click on the *Exit* button when you're finished jogging.

WinPC-NC can automatically determine certain Z-axis heights via a probe or surface block sensor. The sensor should be connected to an input; its cable must allow easy placement anywhere in your machines work volume. The sensor height will limit the thickness of the stock that you can place in your machine.

The measuring sequence involves several steps:

- Move the machine over the point to be measured
- Place the probe or sensor on top of the work piece and underneath the tool.
- Start measuring process. ***WinPC-NC*** slowly moves the Z-axis down until the sensor contact trips. Motion stops when the sensor trips and ***WinPC-NC*** uses the stopping position and the height of the sensor (a parameter) to set the Z-axis height.

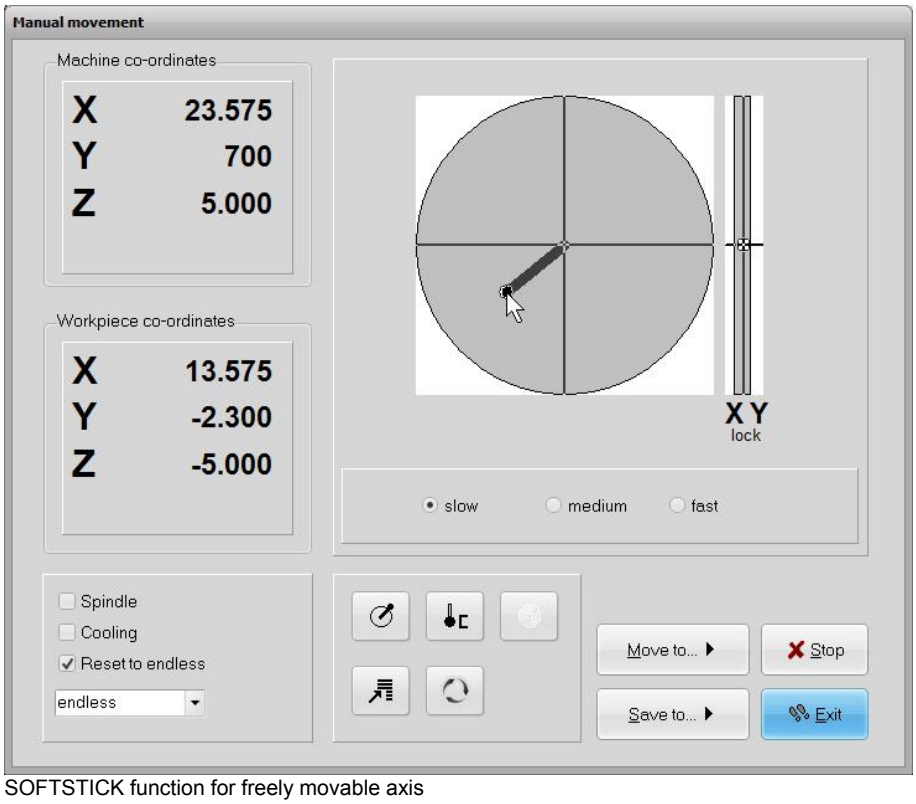
If a 4th axis is available, the appearance of the jog dialog is somewhat different. It includes the expected buttons as well as a control for the rotation. The rotation axis is defined as a parameter; the letter associated with it appears in the dialog.



Jogging with a 4th axis

SOFTSTICK is a simulated joystick. The center point is established by clicking the left mouse button. Moving the mouse with the left mouse button held is analogous to moving a joystick.

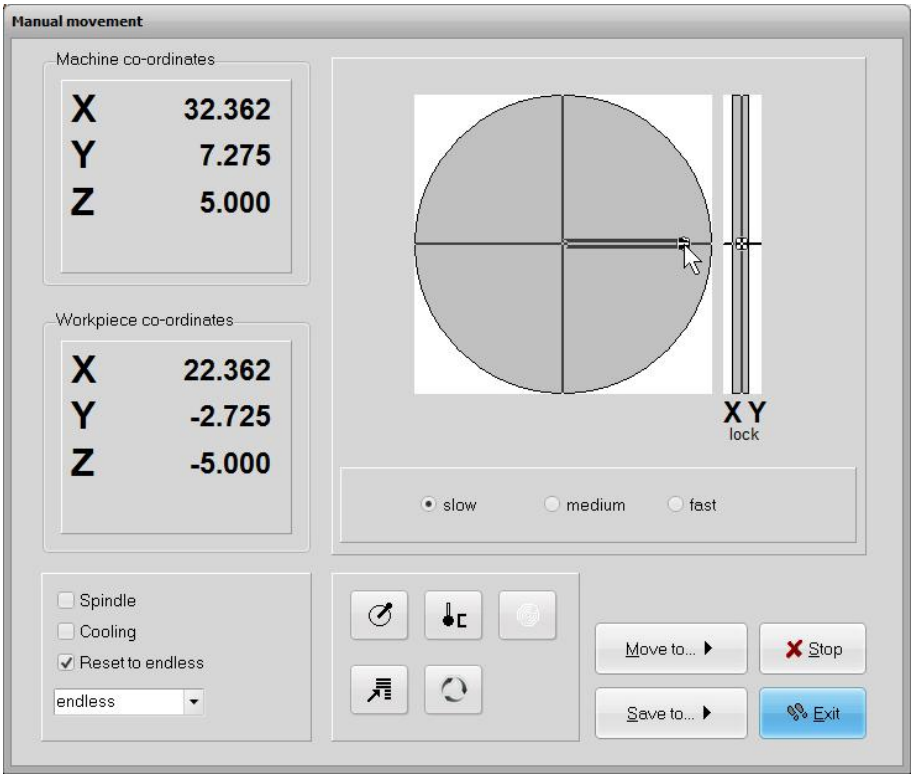
The jog speed is governed by the three speed selection buttons.



SOFTSTICK function for freely movable axis

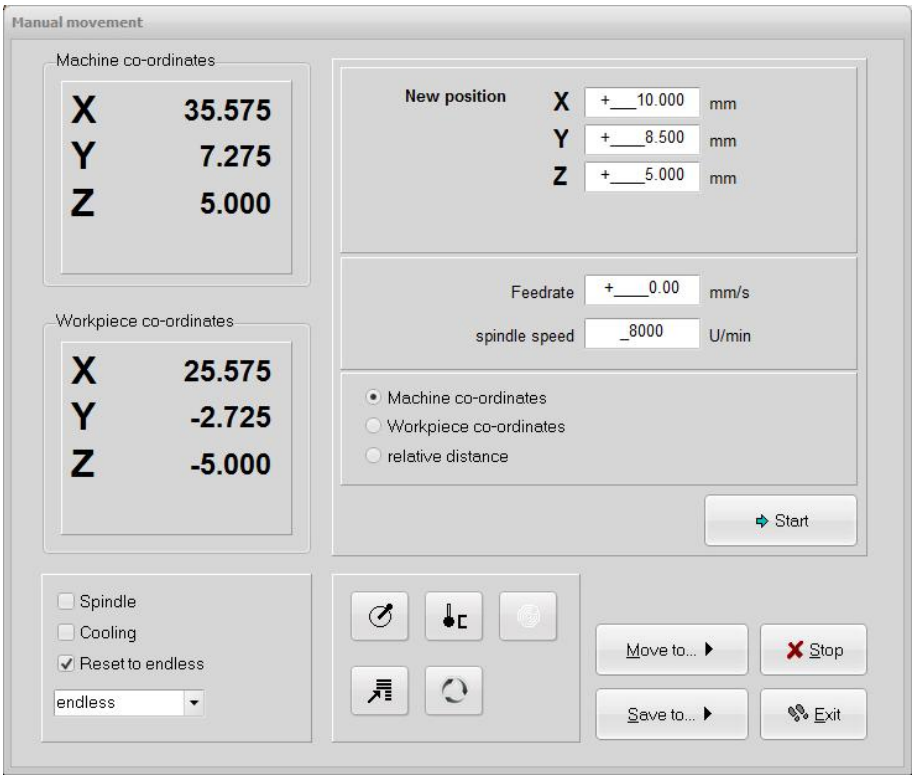
Releasing the mouse button halts any motion.

The circle controls moving in XY and the right-hand bar controls moving in the Z-axis. By clicking the lock symbols the specified axis will be fixed, preventing any further movement.



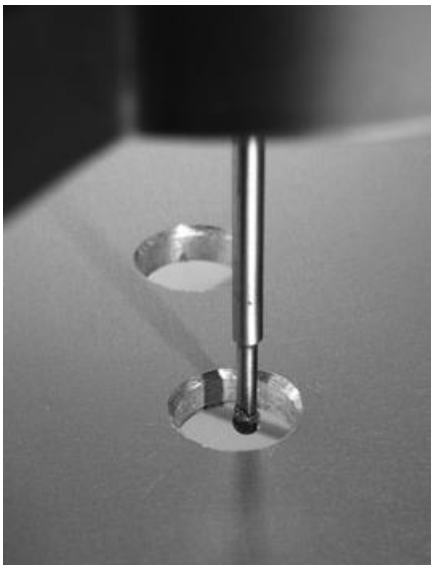
SOFTSTICK – move with fixed Y-axis (Lock-Y)

Using TARGETED MOVE distances, velocities, and spindle speed can be specified. One can easily perform motions and cuts.



Specific Movement

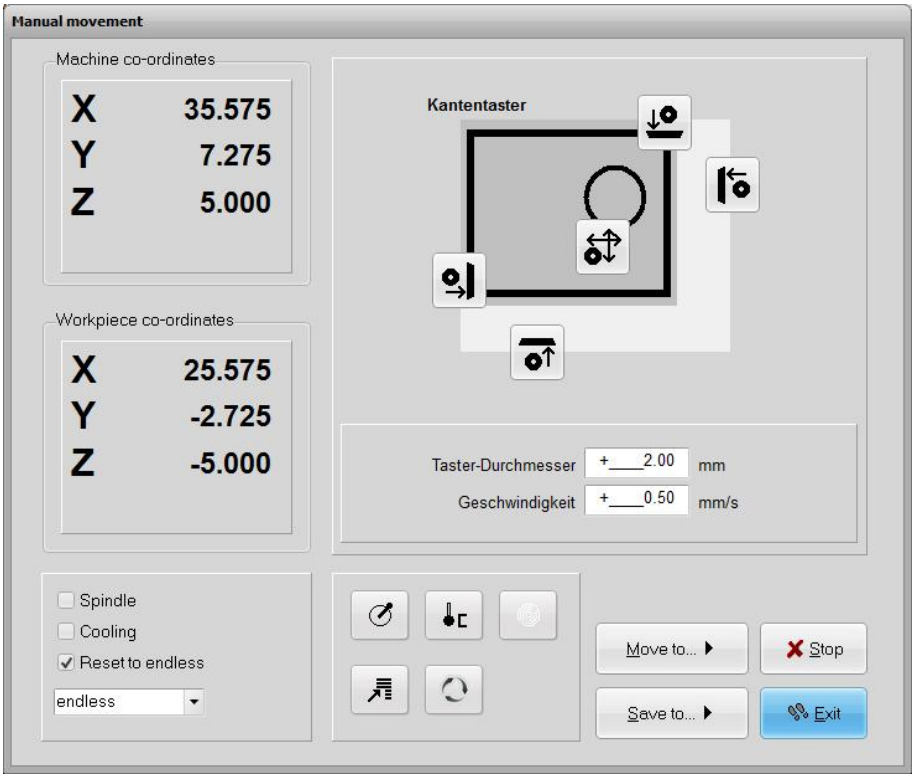
An edge probe automates locating a stock or work piece edge.



Edge probe

Select the button appropriate to the motion desired. ***WinPC-NC*** then slowly moves the edge probe laterally until it touches an edge. The motion will stop when edge probe con-

tacts an edge. The edge probe will be backed off slightly from the edge, lifted to the tool lift height, and moved such that the center of the probe is exactly above the edge. After a measurement, the new position can be saved as zero point of an axis.



JOG edge probe

To measure the center of a circle ***WinPC-NC*** moves in the X and Y direction until it establishes four points on the circumference of the circle. Once the four points have been located, the center of this circle is computed and the probe moves to the exact center of the circle.

WinPC-NC supports pendants (hand controllers). These are useful for moving a machine, establishing axis zero points, saving specific location data, and interrupting a job.

WinPC-NC directly supports the ***HR-10*** pendant. It is can be ordered from us. It is a USB device and comes with a 3M cable.

Moving around has never been so easy!

Further more information about the ***HR-10*,** contact us or find the documentation on our WEB site.

MOVE - JOYSTICK JOGGING

A joystick - rather than using a mouse, a keyboard, or a pendant - can be used for jogging.

Prior to use, a joystick must be configured in Windows system and calibrated in *WinPC-NC*.

JOG with joystick

To move in X and Y, simply move the joystick. To move the Z-axis press and hold button 1 while moving the joystick. A help point can also be stored while using the joystick. To store the help point, press button 2 at the same time as holding button 1.

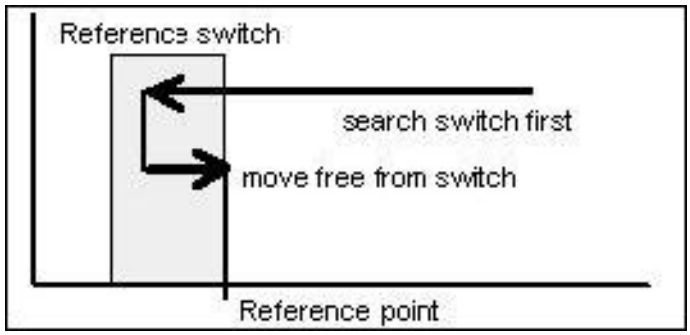
The 3D Space Mouse from the 3DConnexion company is supported. They make moving in 3 or 4 axes very intuitive. Before using one, install the latest Windows drivers. Define the type of connection the device using in the parameter menu.

Space-Mouse-Wireless from 3DConnexion

MOVE - INITIALIZE MACHINE

Before a machine can be used, it must be initialized. The initialization procedure establishes the reference positions of the machine - the points the machine will use as X0, Y0, and Z0.

The machine cycles through each axis, slowly moving until the respective home switch trips. It then moves back slowly until the home switch ceases tripping. This is the reference position for that axis.



DANGER! Machine damage can occur if the machine is not initialized before running a job.

Machine initialization can also be performed by pressing F8 or the initialize button.

MOVE - SELECT TOOL

This item brings up a dialog box for manipulating tools. It can also be activated via the Tools button.

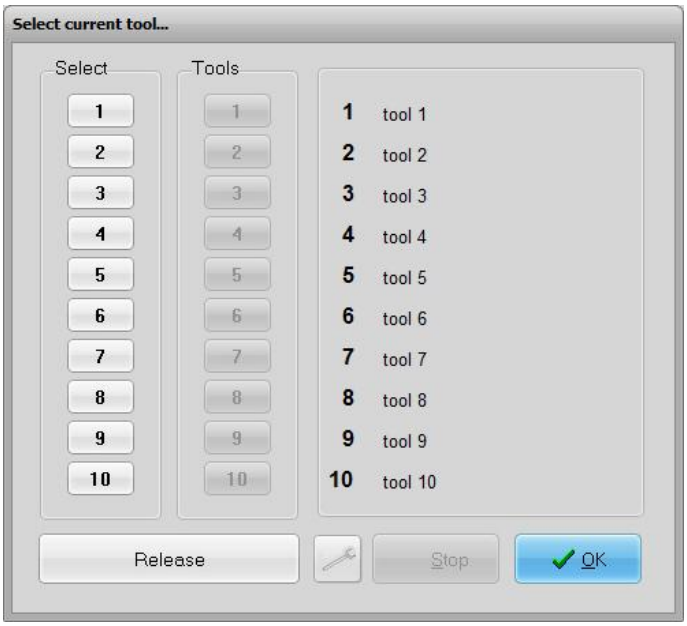
The tool dialog box can:

- Load and unload an ATC tool
- 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 last used and the status of the collet chuck. This information is retained from session to session.

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

Using an Automatic Tool Changer (ATC) requires the corresponding activation in the parameter functions, an ATC magazine, and a programmable, powered collet chuck.



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

Functions of buttons:

- Release returns the tool currently in the chuck and returns it to the magazine
- The *molett* button opens or closes or opens the chuck. There is a security prompt prior to opening it
- Stop cancels any motions (e.g. tool length measurement or moving to a magazine location)
- OK closes the dialog box

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

WinPC-NC automatically handle tool length compensation issues.

Follow this procedure:

1. Insert a tool – the one you want to use first.

2. Load your job if it isn't already.

3. Click on the tool number associated with the tool. The tool number is immediately displayed in the status bar. ***WinPC-NC*** then moves the tool to the length sensor and takes a measurement. The length of the specified tool is saved and displayed.

4. Load the job stock and define the job's X, Y, and Z zero points. The Z zero point can be established by carefully lowering the tool until it barely scratches the surface of the stock.

5. Start the job. As each new tool is required, ***WinPC-NC*** will measure the tool length automatically and make any necessary adjustments.

3.3.3. PARAMETERS menu

Coordinates
Tools
Miscellanous
Technology
Data format
Basic settings

Save profile
Load profile

Save
Save as...
Load

The PARAMETER menu contains all of the ***WinPC-NC*** settings and options. The settings and options are handled via several dialog boxes, grouped by function.

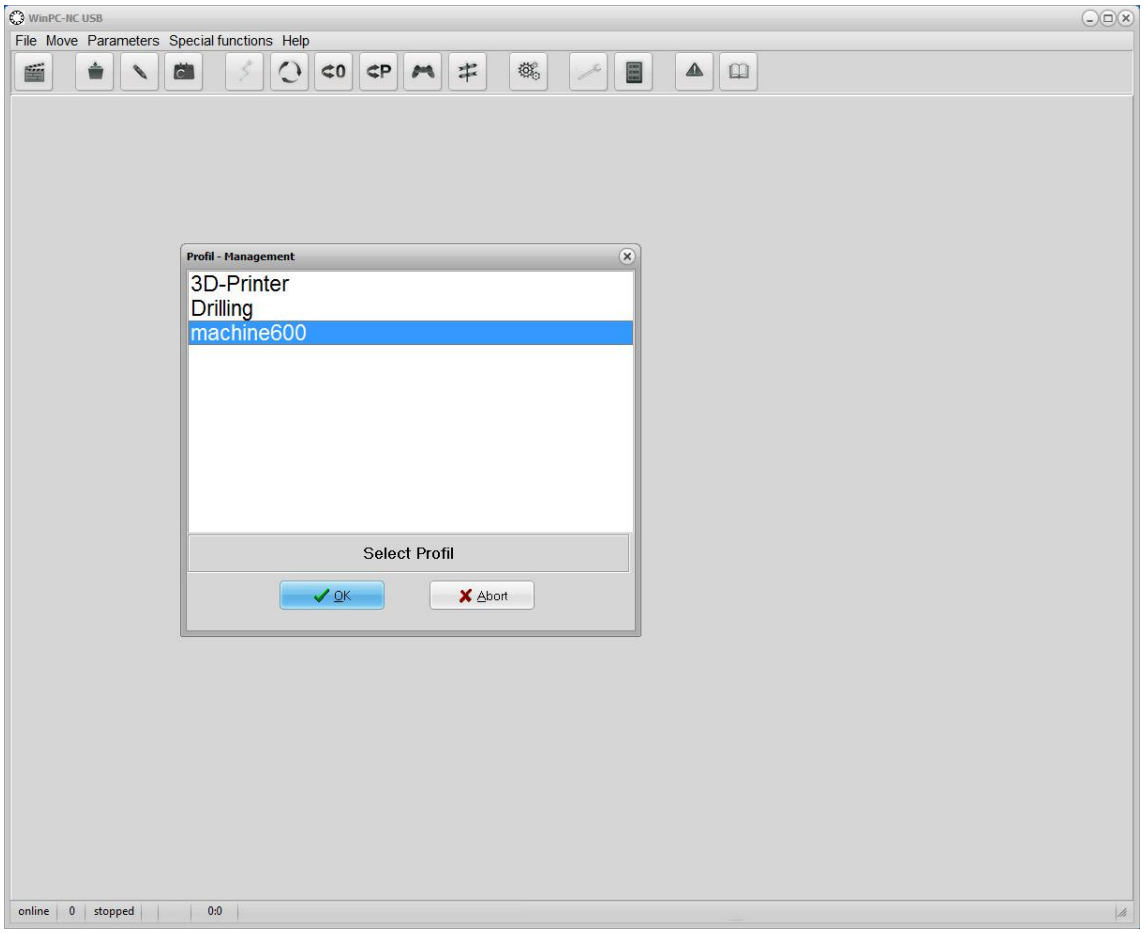
Pressing Alt-P or clicking on the PARAMETERS button has the same effect.

PARAMETERS - SAVE PROFILE
PARAMETERS - LOAD 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 profiles (machine-set-up) are marked with *.WMS and they comprise any settings concerning the machine*, **WinPC-NC,** tool settings, macros, messages and additional characteristics.



Display and selecting profile management

PARAMETERS - SAVE

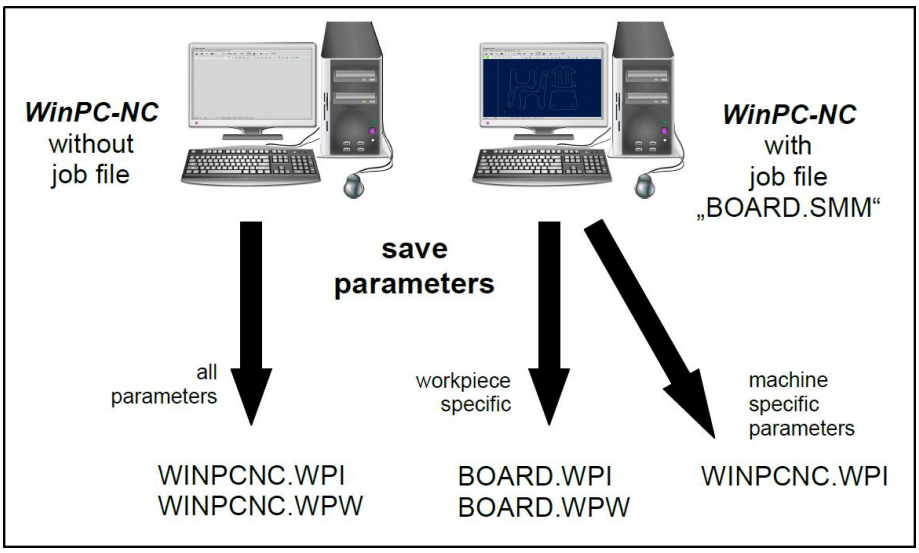
This item saves all of the parameters and tool settings associated with the currently load NC file.

All of the work piece related settings (e.g. zero point, scaling) are saved in parameter and tool files associated with the working file name. These files have the same name as the working file, but with extensions *.WPI and *.WPW.

When the NC file is loaded in the future, all the settings and tools information is restored.

The machine-related parameters (e.g. axis resolution, reference directions, interfaces used) are always stored in the standard parameter file WINPCNC.WPI.

If no working file is selected when you save, the settings are saved into the WINPCNC.WPI and WINPCNC.WPW files.



Saving parameters and tool settings

Separating the parameter files offers the advantage that the machine parameters will probably only have to be saved once; other settings only have to be saved if something changes.

PARAMETERS - SAVE AS...

This item allows you to enter the name for the project parameters file. Using names appropriate to a procedure or material is a good way to handle similar jobs in the future.

Avoid overwriting the standard parameter file WINPCNC.WPI by the using PARAMETERS-SAVE AS....

PARAMETERS - LOAD...

The LOAD function enables you to load various specific parameters 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.

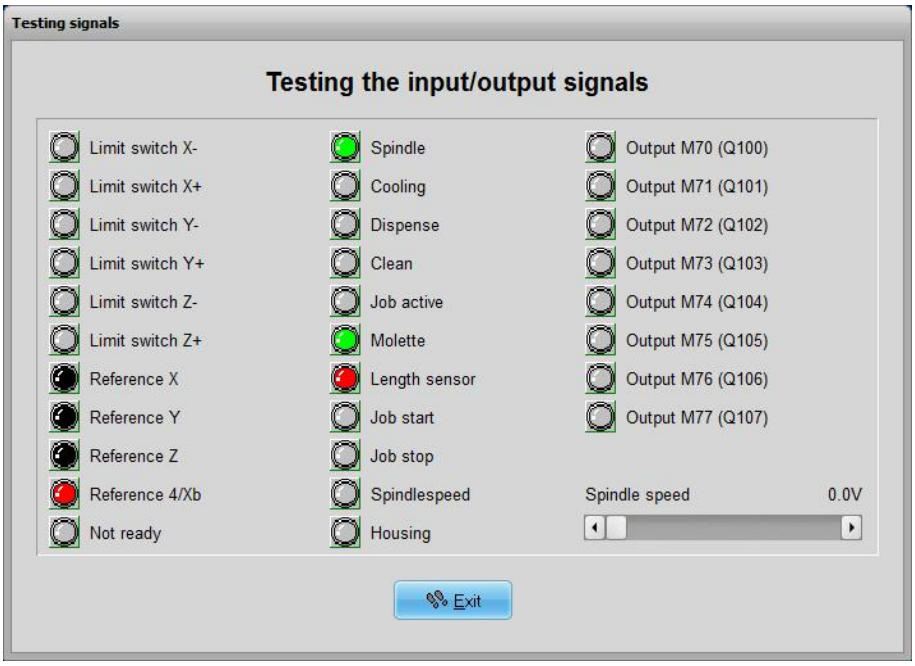
3.3.4. SPECIAL FUNCTIONS menu

Signal test Motor test Controller information Calibrate joystick Rehoming check Factory settings Teachin

The SPECIAL FUNCTIONS menu provides access a variety of function useful to test, optimization, debugging, and calibration.

SPECIAL FUNCTIONS - SIGNAL TEST

This item displays the status of many of the critical input and output signals (e.g. home and limit switches).



Signal test

WinPC-NC continuously monitors its input and output signals and displays a status of a subset of them here. This is useful for testing.

Grayed out signals have either not been defined or do not have an input/output pin assigned to them.

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

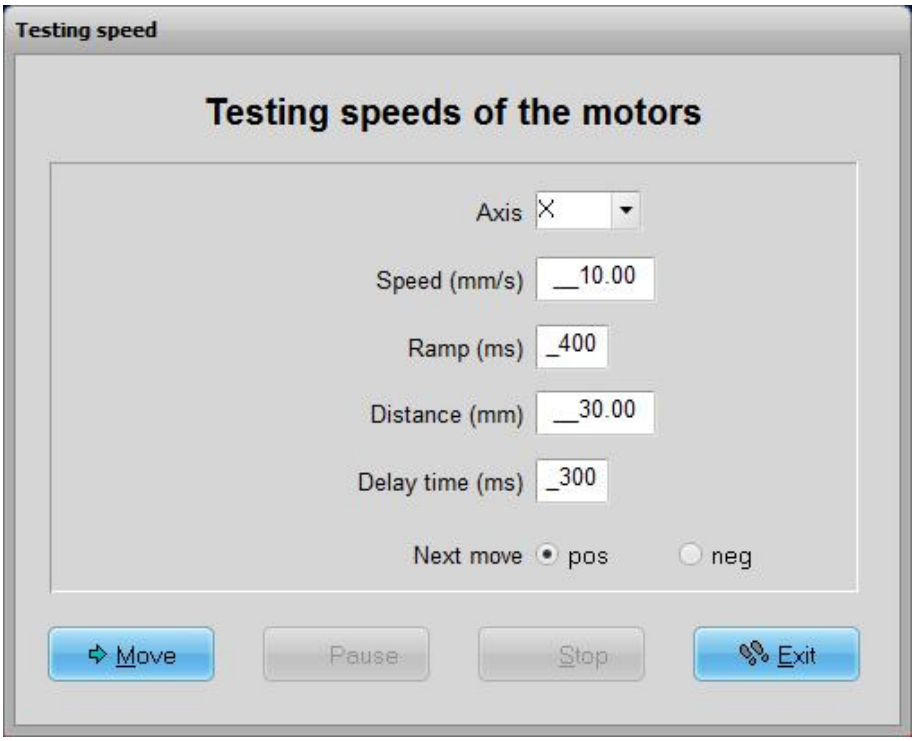
Use PARAMETERS-BASIC SETTINGS and SPECIAL FUNCTIONS – SIGNAL TEST to establish and test the function to signal assignments for all inputs and outputs.

Output signals can be tested by simply clicking on their LED symbols. Blue indicates on and green indicates off.

The slider at the bottom right-hand corner of the window is used to test spindle speed control. Depending on how ***WinPC-NC*** is configured, the slider will either vary an analog output (0-10V) or a PWM signal.

SPECIAL FUNCTIONS - MOTOR TEST

This item is used for for determining the optimum speed settings. The *Testing speed* window displays all of the parameters relevant to a stepper speed evaluation.



Motor test

Enter the parameters you wish to test with and hit the *Move* button.

WinPC-NC will move the selected axis forwards and backwards continuously. By listening to the machine and observing the movement, it is easy to tell whether the parameters are valid for the axis, or whether additional corrections are needed to the speed or ramp length.

The test run can be cancelled by pressing ESC or clicking the *Stop* button.

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

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. Reduce the speed value by 30-40%.

2. Test various ramp lengths. A good value is one where the motor starts up quickly and does not stall.

3. Increase the rapid speed **slowly**. The motor should run quickly while still developing a sufficient amount of torque.

4. Once you're comfortable with the parameters, save them.

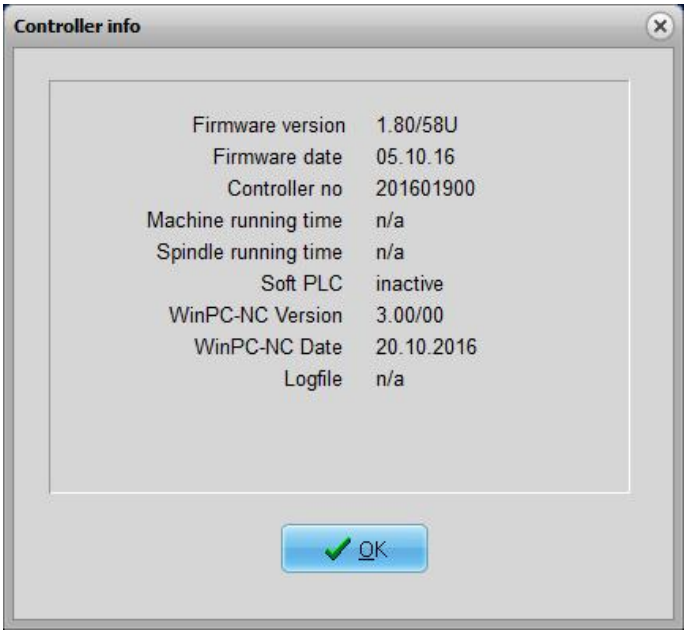
The optimum parameters for stepper motors depends on many factors, including the motor characteristics, the type of drive used (direct or belt), and the load being driven.

It is *impossible* to read machine and motor specifications and determine an optimal set of parameters. If you are unsure of how to tune a machine, stay with the factory provided defaults.

SPECIAL FUNCTIONS - CONTROLLER INFORMATION

This item will identify the external module as well as display important information about its characteristics.

Before reporting any issues - hardware or software - bring up this information, take a screen shot of it, and send the screen shot along with your questions.



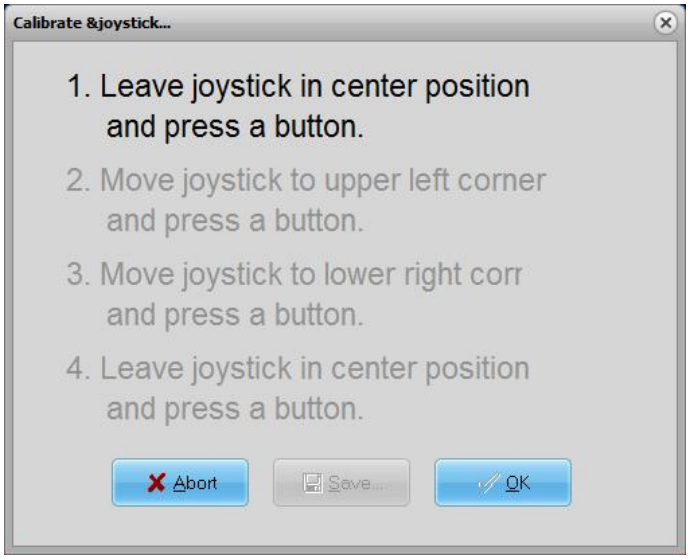
Controller information

SPECIAL FUNCTIONS - CALIBRATE JOYSTICK

This item performs joystick calibration.

Instructions for operating the joystick are displayed in a window. Please follow the procedure exactly.

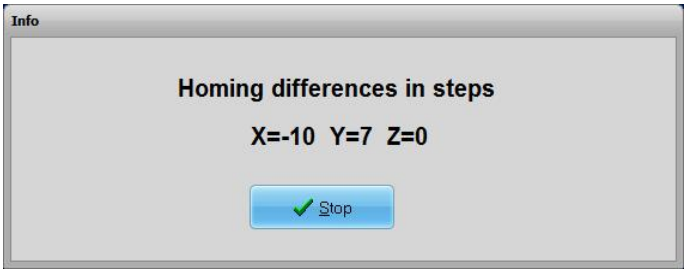
A joystick cannot be used until it is calibrated.



Joystick calibration

SPECIAL FUNCTION - REHOMING CHECK

This item is used for checking the accuracy of the reference switches.



Relatively small step discrepancies may be due to the switches. Large discrepancies, on the other hand, indicate step losses. Step losses indicate a serious problem.

Use SPECIAL FUNCTION - CHECK POSITION if:

- A collision has occurred - steps may have been lost.
- You want to determine the maximum machining speed for a tool or material, and you're concerned that loss of stepping is occurring.

- The position was changed during a tool change.

Check position can be performed only if the machine was initialized. The initialization procedure must have completed without errors (no limit switch or stop signal issues occured).

***WinPC-NC* cannot initialize a machine or check position properly unless the home switches have been properly installed. The home switches must be installed a small distance from the mechanical edges of each axis such that the initialization procedure can cause each switch to trip and then back away until the switch ceases to trip.**

SPECIAL FUNCTIONS - FACTORY SETTINGS

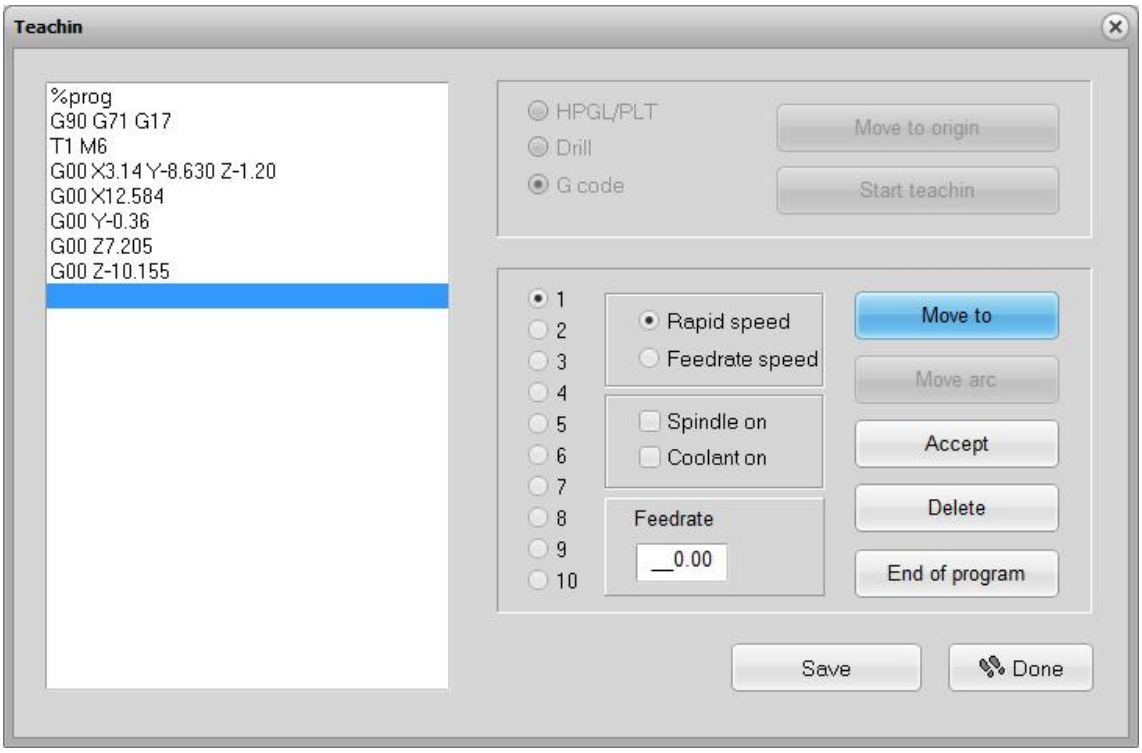
This item restores ***WinPC-NC*** to its original factory settings. Any settings made previous will be lost.

If ***WinPC-NC*** came bundled with your machine, you will be prompted to identify your machine. This will reinstate the necessary parameters for your machine.

If your machine was provided with a set of parameter files (e.g. USB key or emailed files), you will want to locate that them and load the parameters again.

SPECIAL FUNCTIONS - TEACHIN

The TEACHIN function makes it easy to create simple programs in various formats by moving and following the contours.



Special functions-Teachin

The new program is displayed in the Teachin window on the left.

The top right displays those selection that must be chosen before creating an NC program.

The bottom right displays all available actions (e.g. tool selection, speed settings, and spindle and cooling control).

The choice of acceptable commands depends on the selected data format. For instance, spindle and cooling command are not handled in the HPGL format.

Suggested procedure with Teachin:

1. Execute a reference move and call up the Teachin function.

2. Specify the data format and zero point (by jogging).

3. Start the Teachin process. The program header will automatically be set up; it will appear at the top of the program area.

4. Enter the necessary commands. Care should be taken to ensure that the command sequence is correct and the correct tool is used by the expected command.

5. Click on exit button and the program is automatically finished.

6. The new created program should be stored before leaving Teachin.

Possible actions during the Teachin process:

Movement in a line	Choose the movement speed, either the rapid rate (fast movements) or the feed (cutting) rate.
Movement in a circlular arc	A circle or arc always requires three points. The first point is the current position. The other two must be points on the circle/arc.
Accept	This function inserts the actual position of the cursor bar into the program.
Delete	Deletes the line under the cursor bar.
Exit	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. This enables the user to make up for overlooked actions.

3.3.5. HELP menu

There are three items in the help menu:

HELP - MANUAL

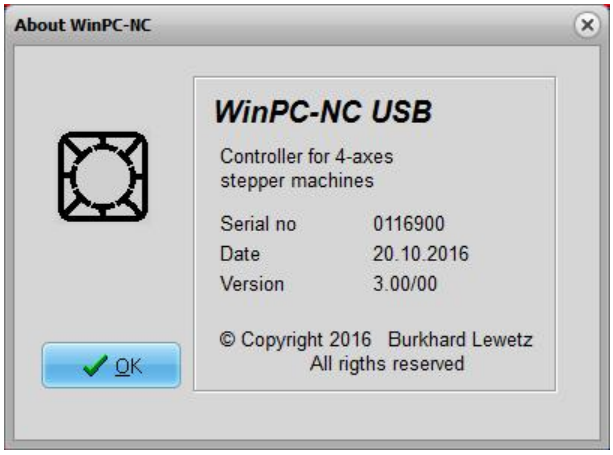
Brings up the PDF version of manual in your PDF reader.

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

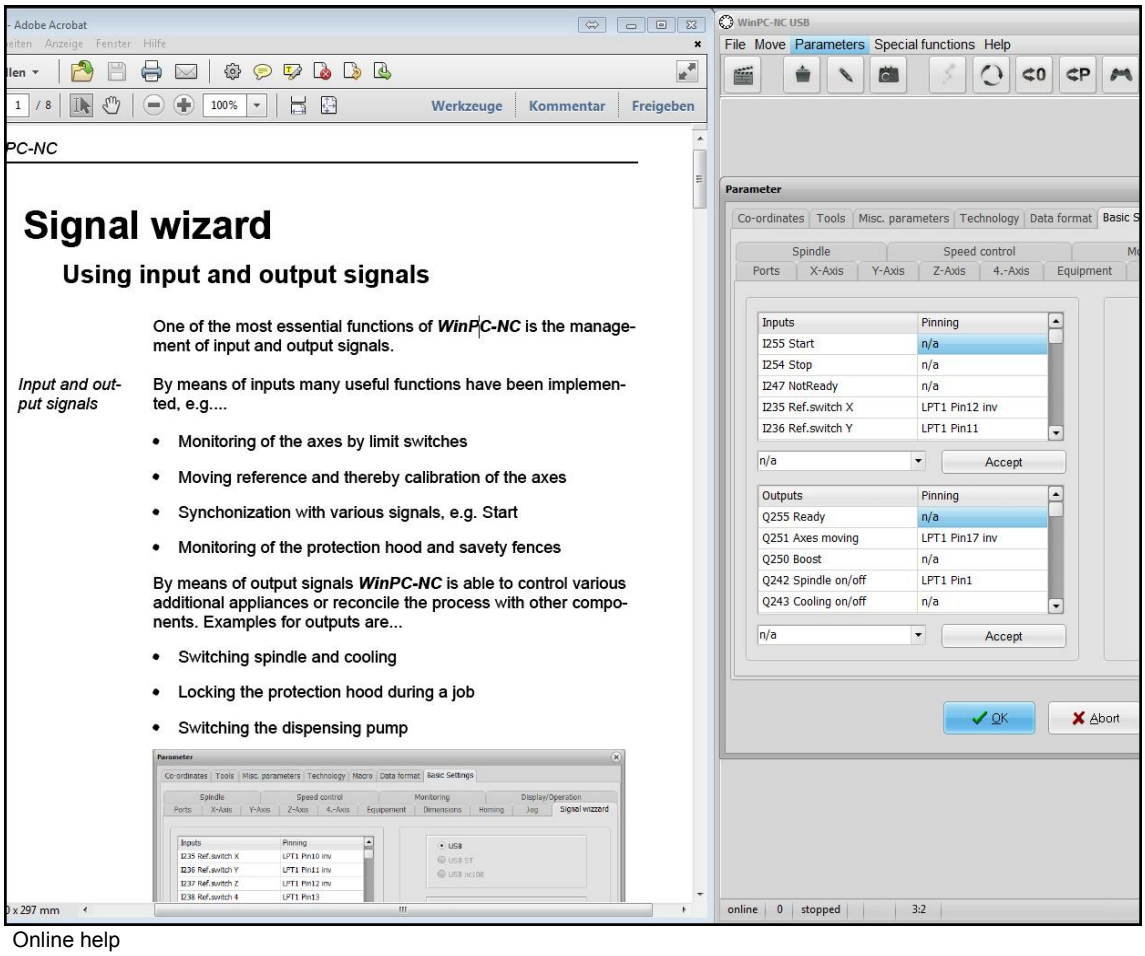
Activating this menu function displays information about the current version and revision number.



Information about the current version

Context-related help can always be summoned via F1. Help concerning the current operation will be displayed.

The ONLINE-Help will bring up this manual. The manual is included in the ***WinPC-NC*** installation and can be read via a PDF reader (e.g. Acrobat Reader, Foxit Reader).



Online help

4. 2D-CAM Functions

4.1. Overview

WinPC-NC provides special functions for sorting and preparing 2D data for subsequent processing. Following functions are available:

- Assigning tool usage
- Cleaning up data, deleting zero length vectors and duplicate lines
- Setting the output sequence according to tool number
- Optimizing empty moves
- Tool diameter offset compensation in contours

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

The CAM functions are activated by clicking in the 2D-CAM button after loading the job. A dialog box will appear. It provides all available options and functions.

4.2. Setting options

Available functions and settings:

Cleaning up data

Zero length and duplicate vectors are removed from the drawing. These modifications are usually safe and no visible changes will appear. Be aware that these changes may result in subsequent calculation failures.

Scaling data

All data and lines are scaled in size prior to any subsequent calculations. 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

WinPC-NC tries to create closed contours and continuous, extended lines out of many individual vectors. Individual vectors are sorted and linked. During the search process, consecutive lines may be discovered that do not touch each other end to end – there a small gap in the data. When a gap is detected – and only when a gap is detected – “fuzzy logic” will be attempted find a close contour and connect the ends.

During the search process, ***WinPC-NC*** will find all lines, closed contours, and contours which are contained within other contours. Each discovered line and contour will clearly be marked. This is an essential feature for the radius correction later on.

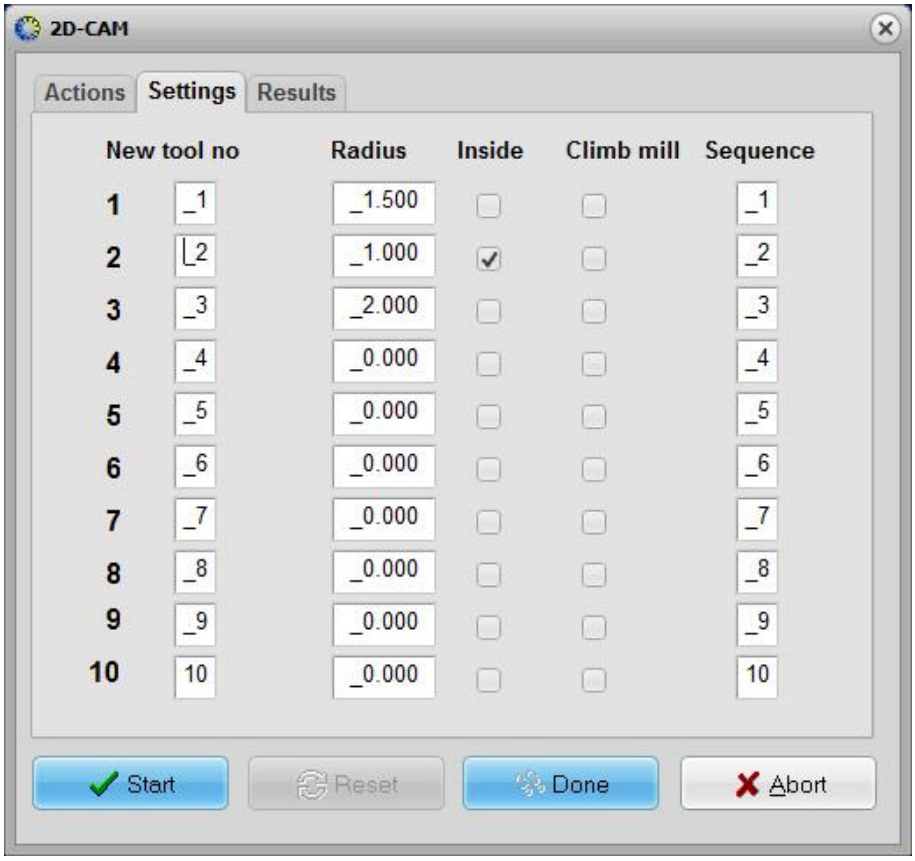
Calculating tool diameter offsets

The radii of each tool involved is used to calculate an offset appropriate to each contour it is associated with. This ensures that each contour will be precisely milled.

When a correction is made (by hitting a button), the tool number of the correction with have 100 added to it. This makes contours that have been already processed and those that require further processing easy to spot.

Optimizing empty moves

Prior to the output of the calculated data in a new NC file, ***WinPC-NC*** tries to optimize (remove) any empty moves between lines and contours. Care is taken to ensure that any element is machined only once. This saves processing time and helps to decrease the milling time.



2D-CAM settings

New tool number

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

Radius

The radius of each tool is specified for the radius compensation. The radius is used to correctly handle contours from inside and from outside.

Inside

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

Climb milling

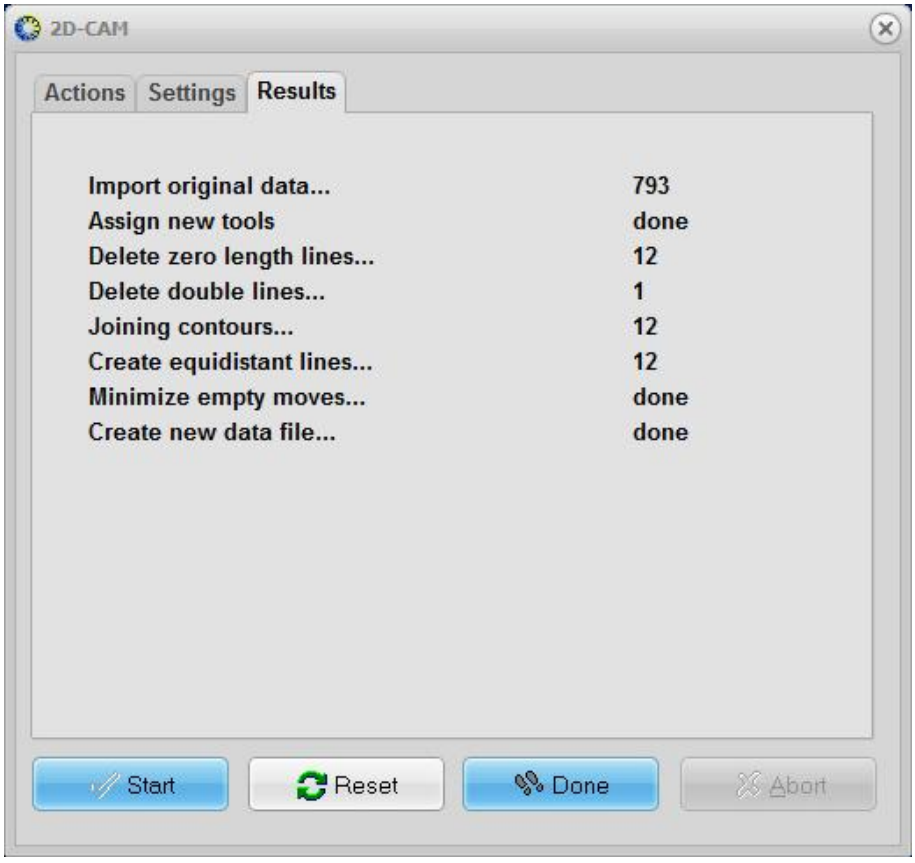
When this checkbox is check, climb milling will be used. Otherwise, conventional milling will be used.

Sequence of operations

The START button begins any (re)calculations. A progress bar will appear while the calculation process is running. The actual state of the process and the provisional results are displayed on the result sheet of the dialog box. Cancellation can always be effected by clicking to the corresponding button.

After ***WinPC-NC*** completes of all of it calculations and sorting, it creates a new 2D file in an internal format. The name of the file maintains the project name however the extension of the file will be *.OPT.

It is always possible to return the file to its original state (ABORT). For these procedures the buttons *START*, *RESET*, and *DONE* are applicable.



Displaying the results

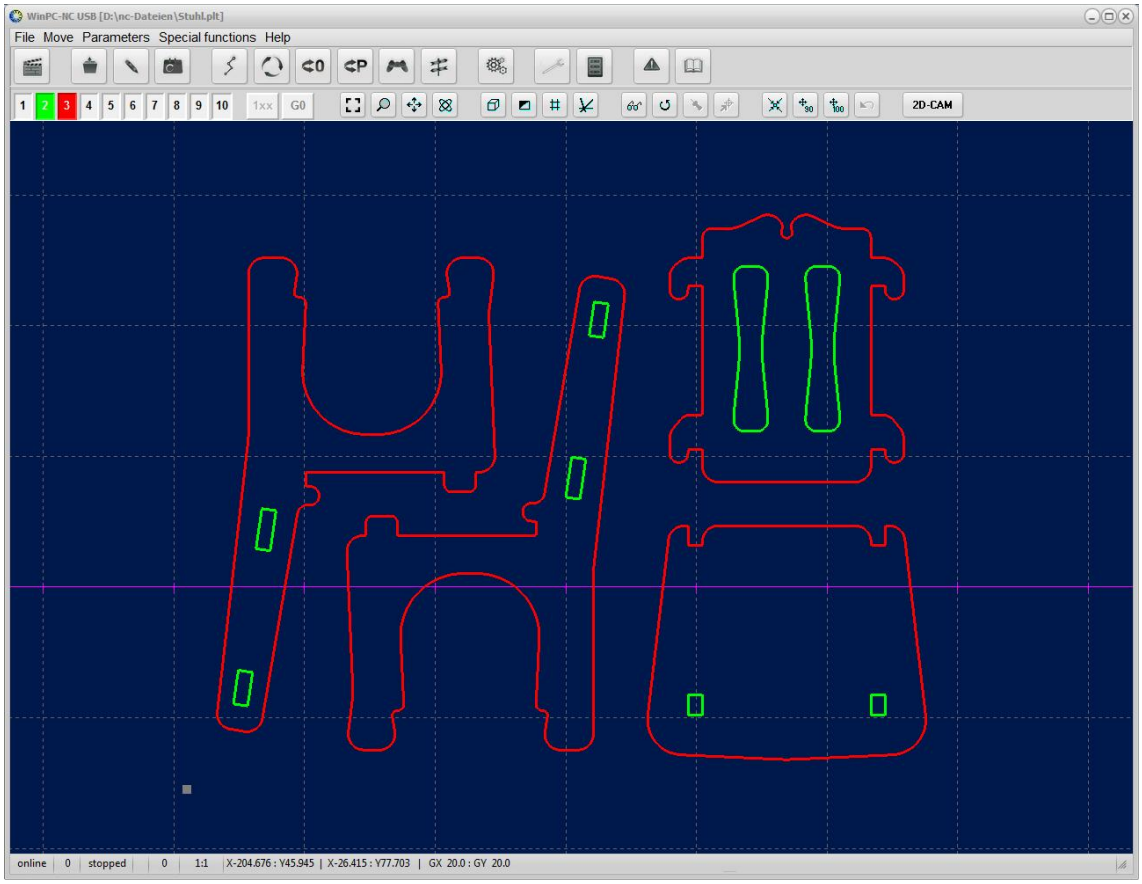
Having successfully calculated and generated the new file, the contours are immediately visible in the *Display Area*; the results can be checked. Addition changes and recalculations may be performed. Activate *EXIT* for accept the results.

Manipulating the NC data can be performed in any order.

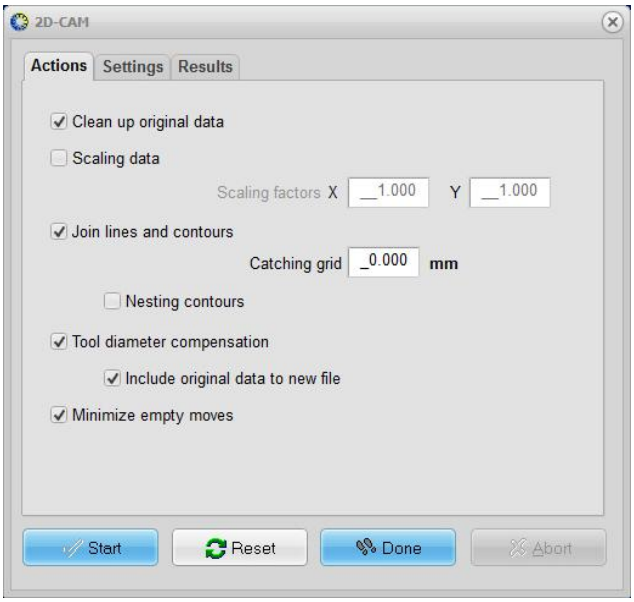
For example, in order to achieve a better surface result with millings and engravings, you should clean up the data, join contours or lines, and remove duplicate operations.

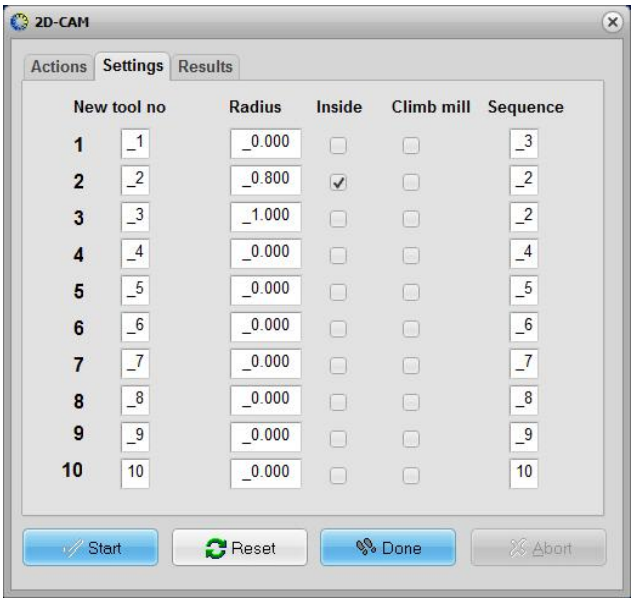
4.3. Example 1

We load a chair in HPGL format:

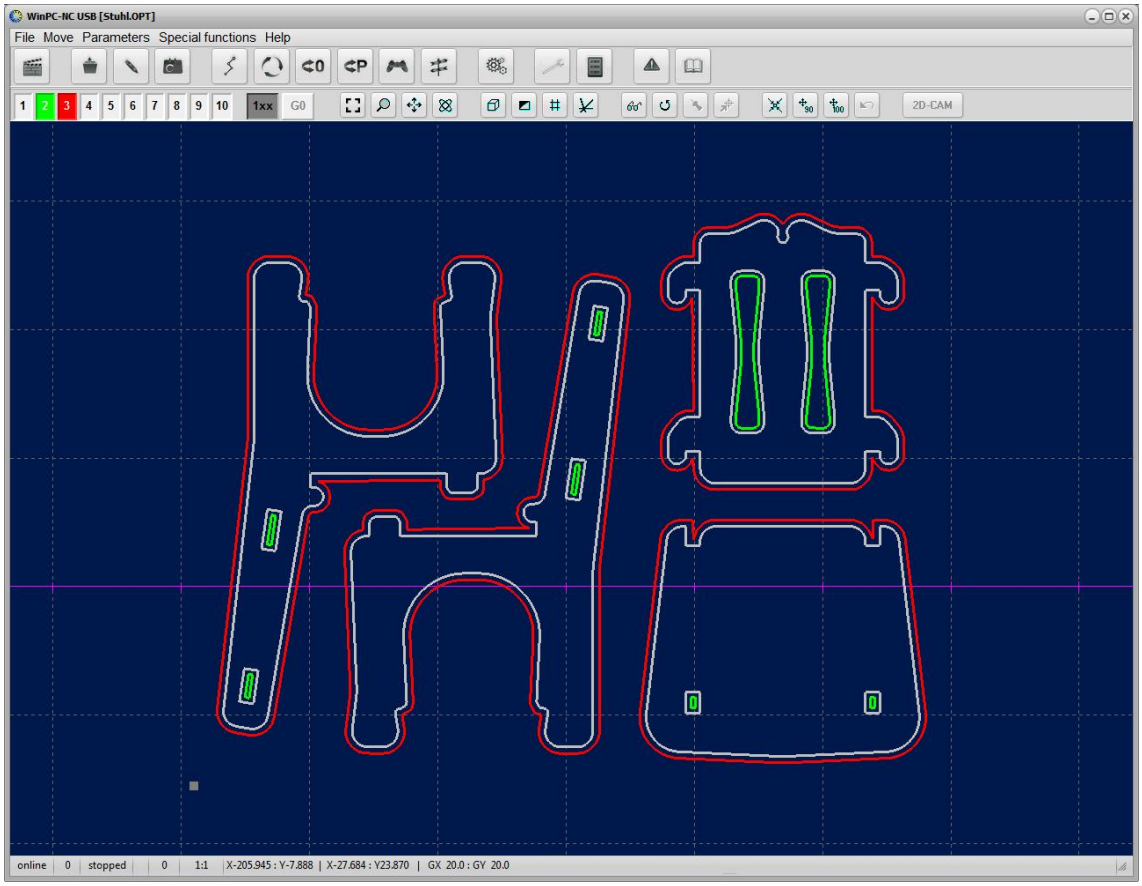


1. Two colors are present, indicating that the milling involves two tools. The interior contours are offset to the inside with a 1.6 mm tool; the outside contours will be milled with a 2mm tool. Settings are as follows:

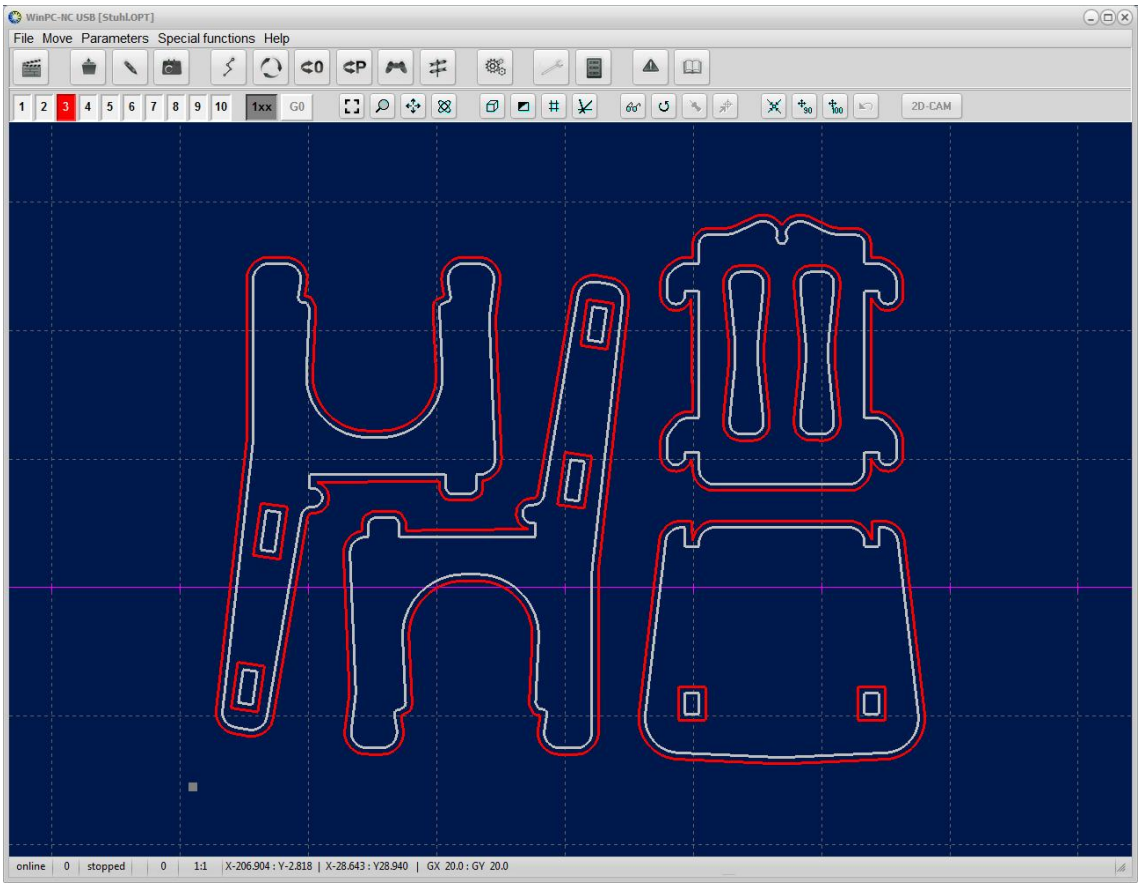




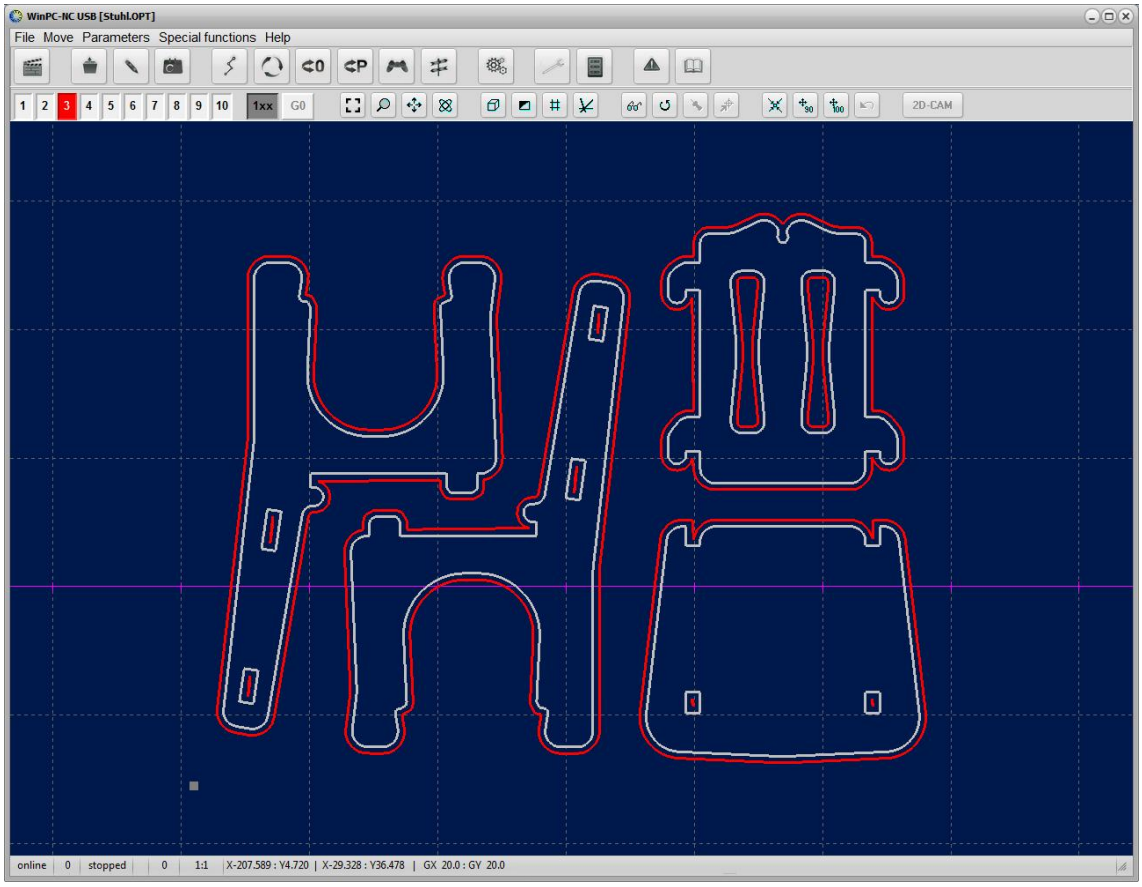
2. We define the values for tools 2+3, and start a search for contours, calculate radius compensation, and optimize any empty moves. The result is…



3. If we want to mill the job using only one tool - no tool change - calculations have to be redone. The function is reset, tool 2 is disabled, tool 3 is defined, calculations are re-done, including the radius compensation.



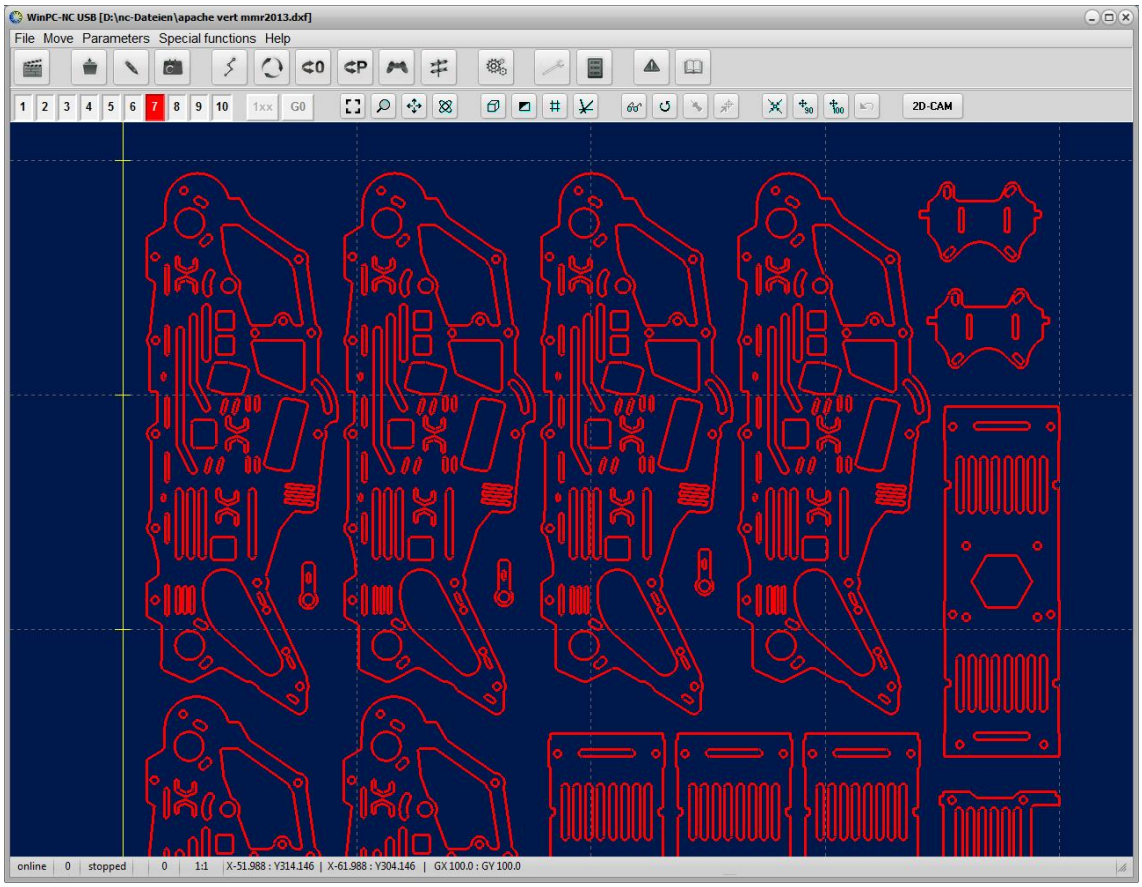
4. Notice that the interior contours are improperly milled on the outside. We still activate the button for the automatic search of surrounded contours and start calculation for the last time.



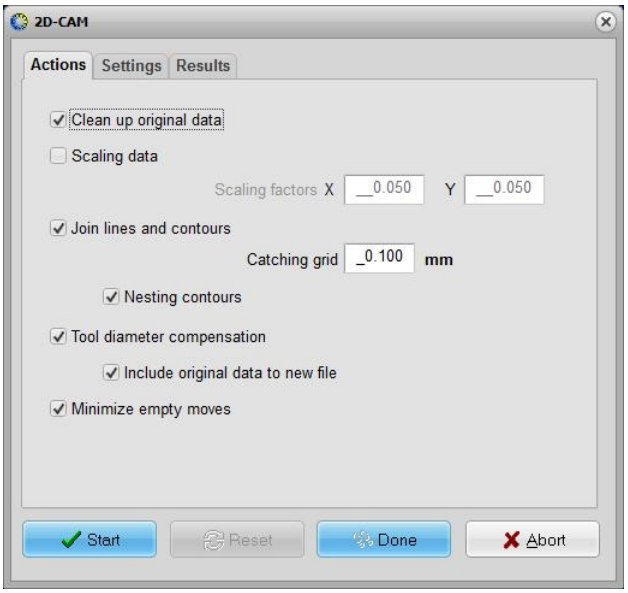
5. Now, the result fulfills our requirements and expectations, sequences are correct, and all work pieces can be machined with our 2mm tool.

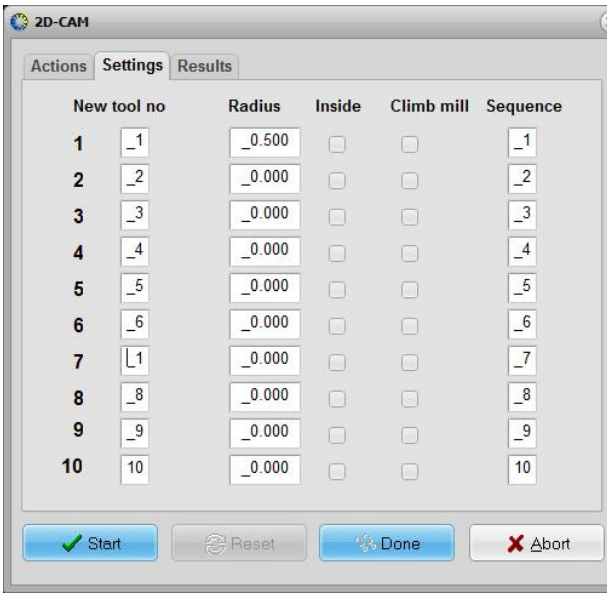
4.4. Example 2

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

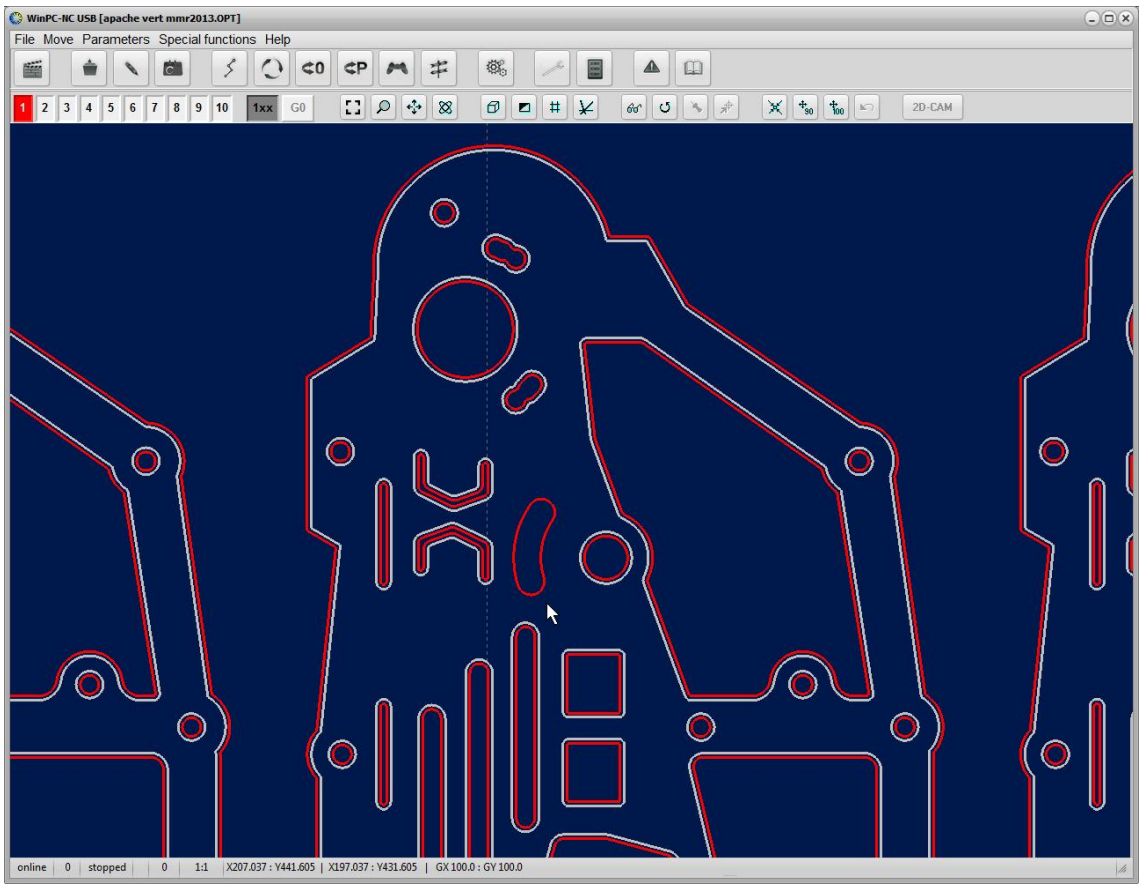


2. Settings in both dialogues are as follows:

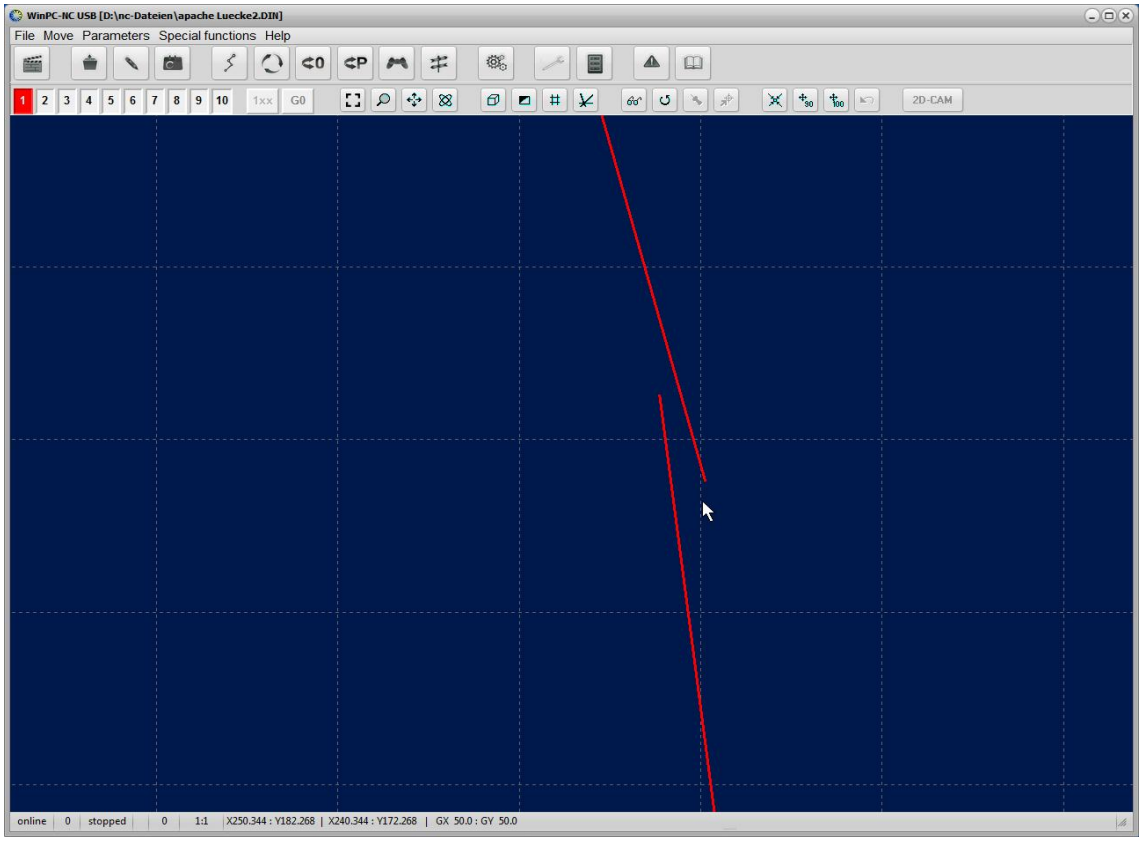




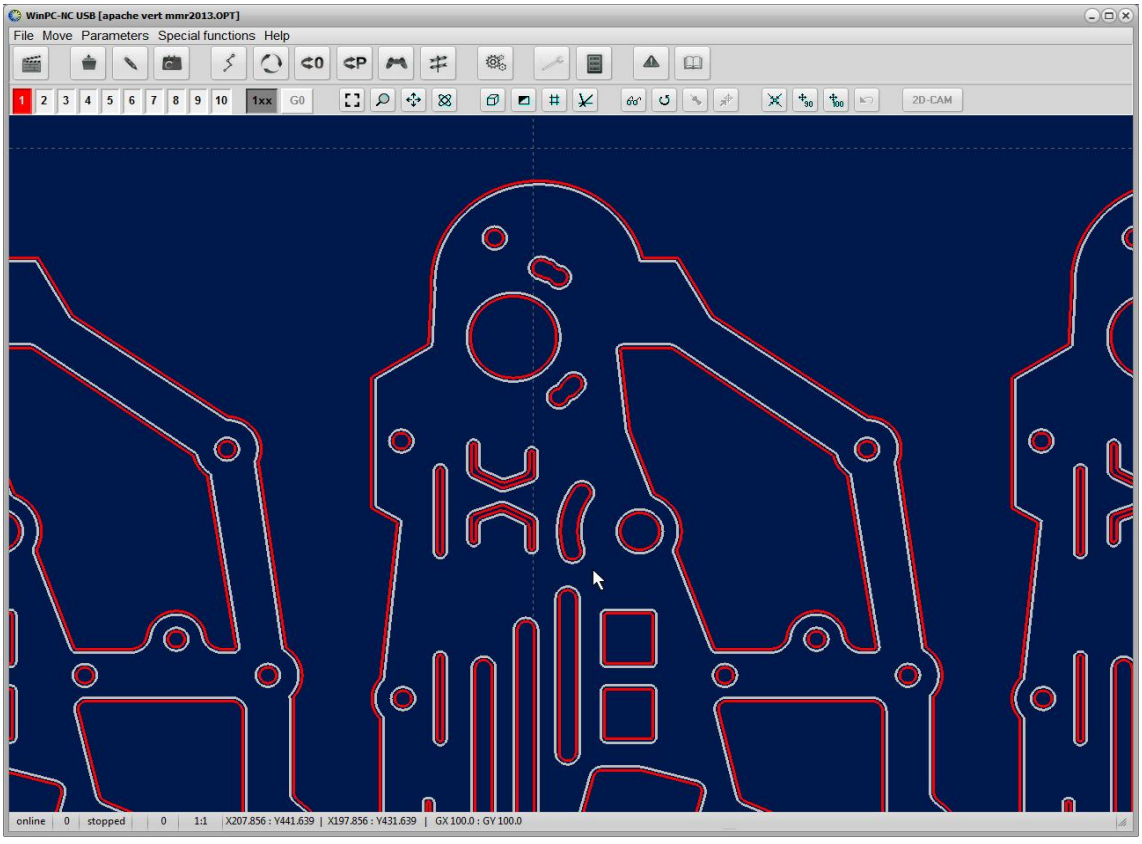
3. At first, the result seems to be perfect. The surrounded contours as well as the offset paths to the inside and outside are clearly displayed. However, after zooming, some contours have not been correctly recognized and calculated.



4. Zooming further, the cause for the failure can be identified. There is a spacing between two successive lines and it is large than the set catching grid.

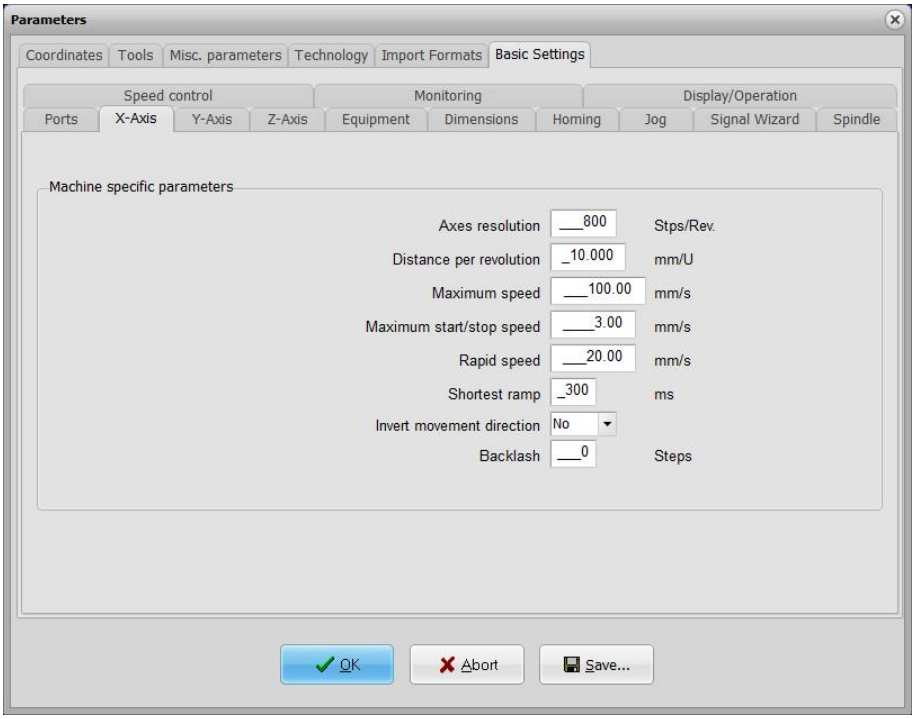


5. We extend the catching grid up to 0.2mm and recalculate. A perfect result is achieved and milling can begin.



5. Parameter Settings

5.1. General



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.

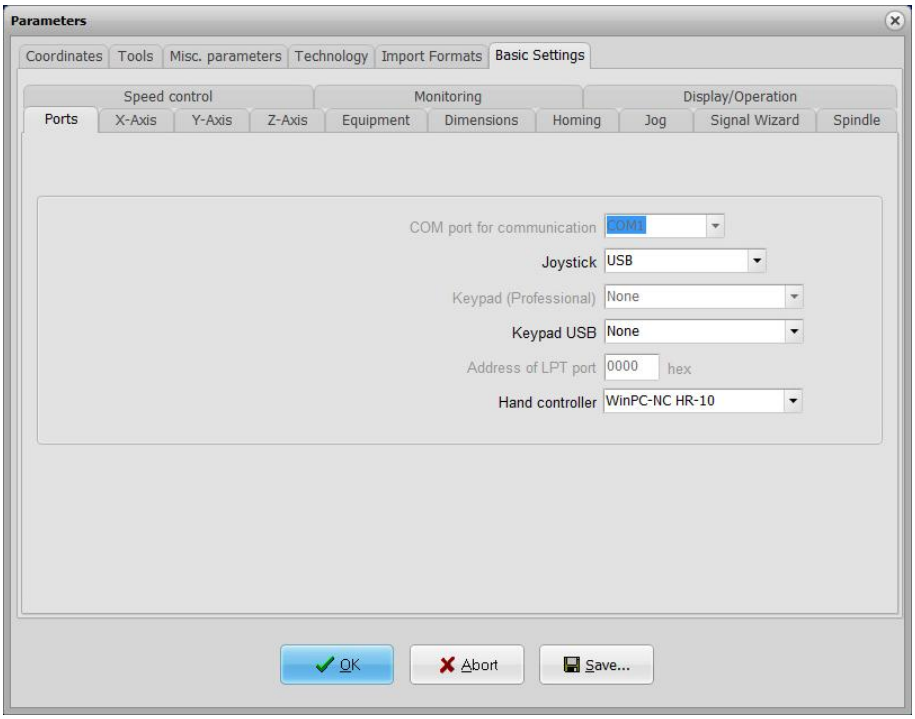
Some groups are divided in additional sections and display additional windows concerning settings which are presented in sub-categories.

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

5.2. Basic Settings

Basic settings are those parameters that are not expected to change very often (e.g. language choice, mechanical and electrical characteristics, user interface choices).

5.2.1. Basic Settings - Ports



Parameter basic settings ports

Joystick

If you want to use a joystick or 3D Space Mouse with your machine, specify the connection via the pop-up menu.

The joystick or 3D Space Mouse must have its drivers loaded prior to choosing it here. It must be calibrated via SPECIAL FUNCTIONS-CALIBRATE JOYSTICK before use.

Keypad (USB)

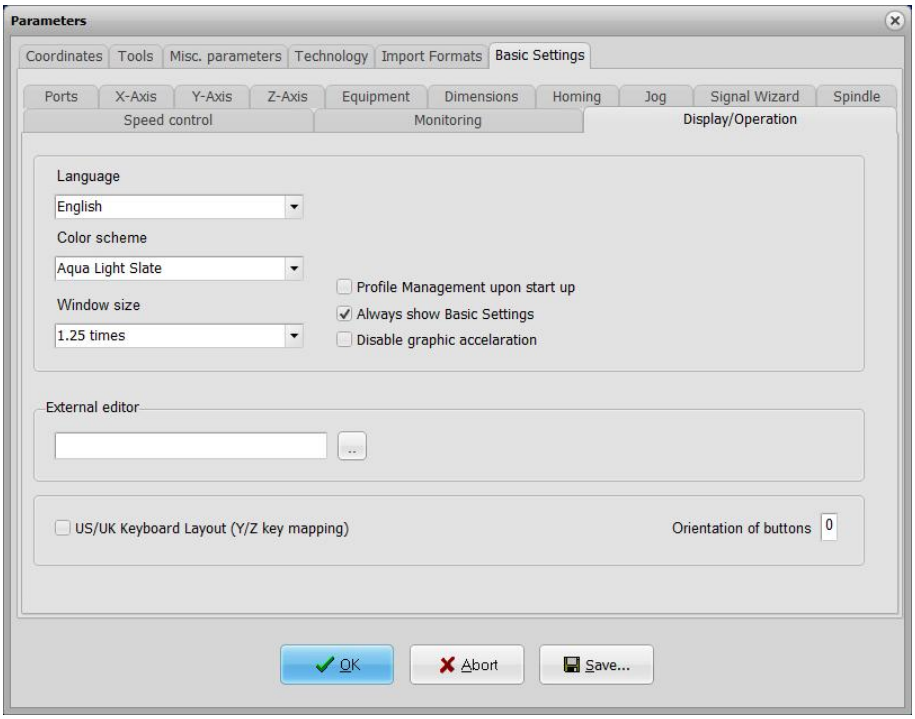
A small, numeric keypad (including generic USB keypads) can be used as a remote control for ***WinPC-NC***. Make the selection appropriate to your keypad.

Pendant (Hand Controller)

HR-10

WinPC-NC supports only those pendants listed in the pop-up. The ***HR-10*** pictured at the left is available directly from ***WinPC-NC***.

5.2.2. Basic Settings - Display/Operation



Parameter Basic settings Display / Operation

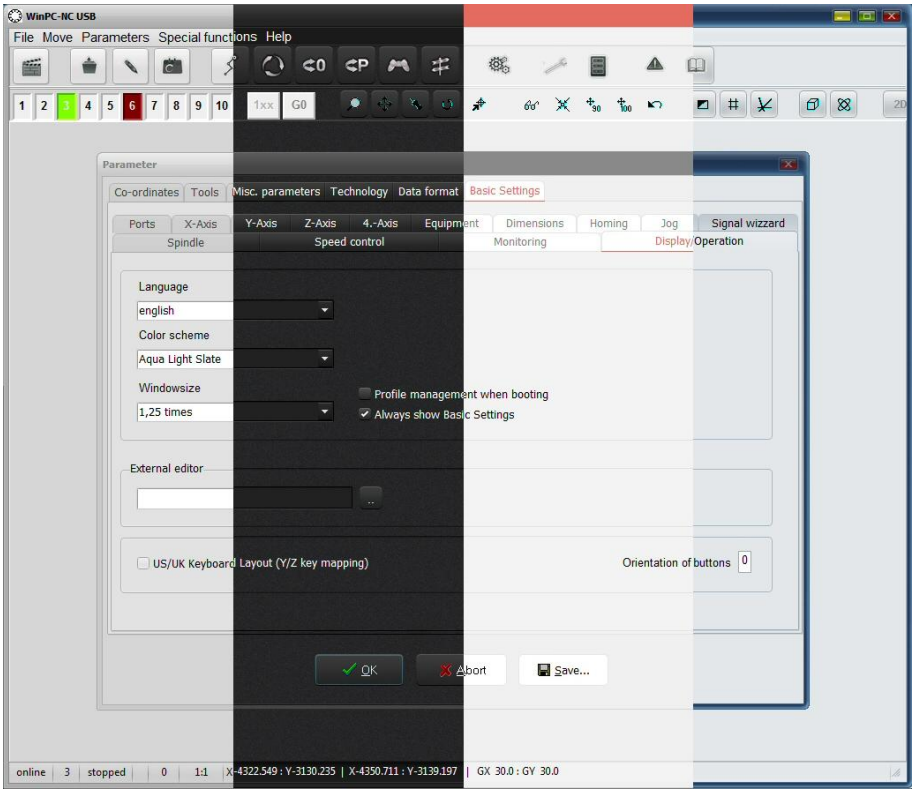
Language

WinPC-NC is multilingual. At the time writing the following languages are available:

German	Dutch	Croatian	Serbian
English	Czech	Chinese (Taiwan)	Chinese (traditional)
French	Portuguese	Slovenian	Turkish
Spanish	Italian	Macedonian	Bosnian
Polish	Hungarian	Greek	Danish
Swedish	Russian		

A language selection takes effect immediately. Click Save to make the selection permanent. Please be aware that some messages are part of Windows and may not be properly localized for the language selection.

Color scheme



Four ***WinPC-NC*** color schemes

Choose the color scheme that best suits you. Please contact ***WinPC-NC*** concerning other possible choices.

Windows size

In ***WinPC-NC*****,** the main screen can be magnified independently of any Windows setting. Select the desired magnification factor.

External editor

WinPC-NC includes a high-performance editor suitable for editing NC files. If you have a favorite or preferred editor, select your editor here and ***WinPC-NC*** will invoke it anytime an editor is required.

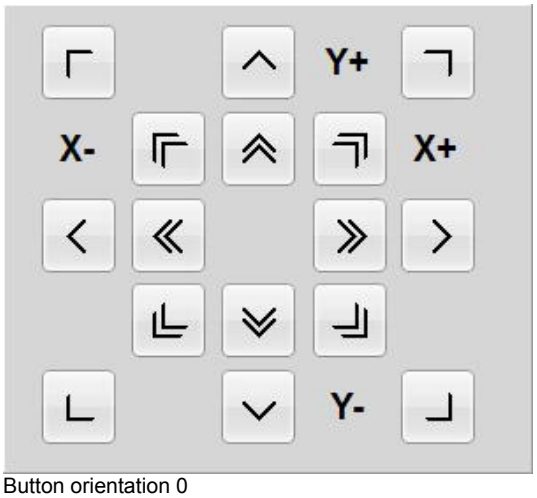
US keyboard layout (switch YZ keys)

Jogging in the direction of an axis (i.e. X, Y, Z) can be performed by hitting the letter key associated with the desired axis. Hitting Ctrl-`axisLetter` increases the jog rate.

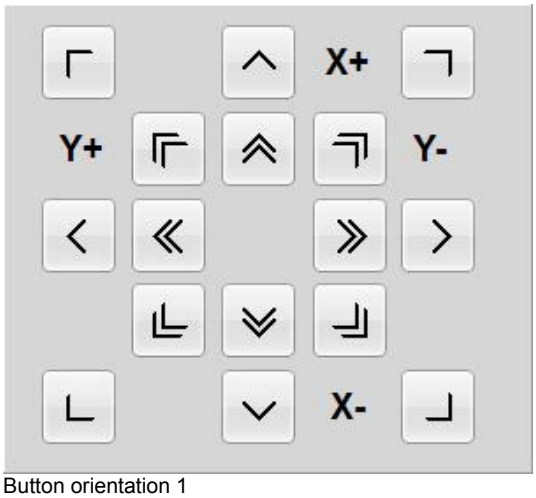
European and North American keyboards switch the position of Y and Z keys and this may make it difficult to use the keyboard for jogging. Checking this checkbox corrects for the switched layout.

Button orientation

The button orientation parameter rotates the buttons in the jog button palette by a multiple of 90 degrees. Adjust the jog button palette orientation so that it matches your machine.



Button orientation 0



Button orientation 1

Always show Basic Settings

The Basic Settings will appear in the normal parameter dialog if this checkbox is checked.

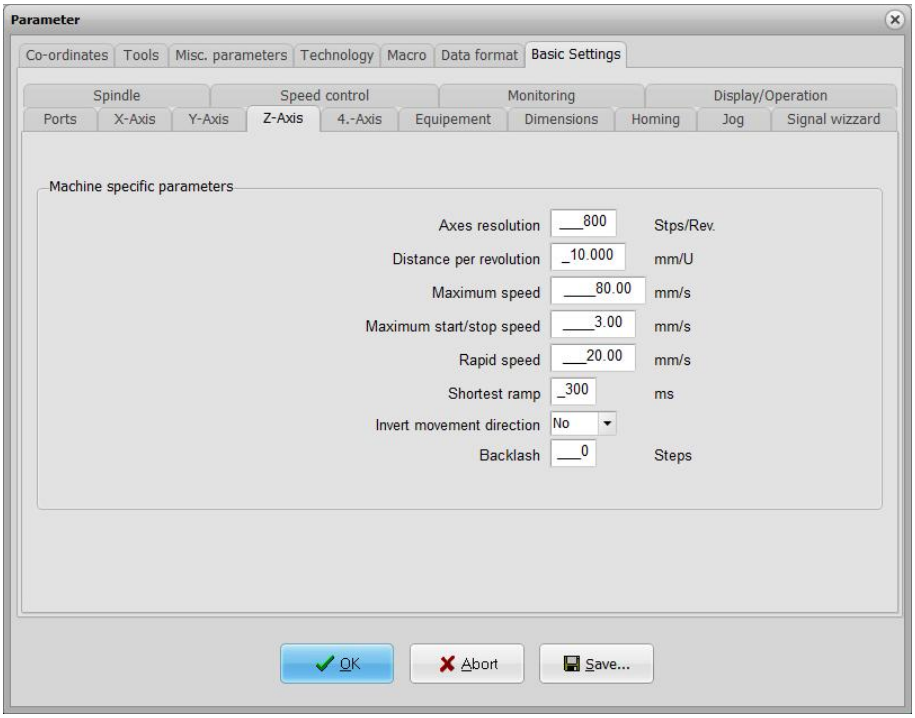
Profile Management upon start up

WinPC-NC will load and use the specified profile each time the program begins.

Disable graphic accelaration

WinPC-NC will do all graphic work in software. Only enable this if there are problems displaying NC files.

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



Parameter Basic setting Z-axis

Axis resolution

Specifies the number of motor steps per revolution. Any gearing or step up/down ratios should be calculated into this number.

Distance per revolution

Specifies the distance travelled during one motor revolution. Units are millimeters.

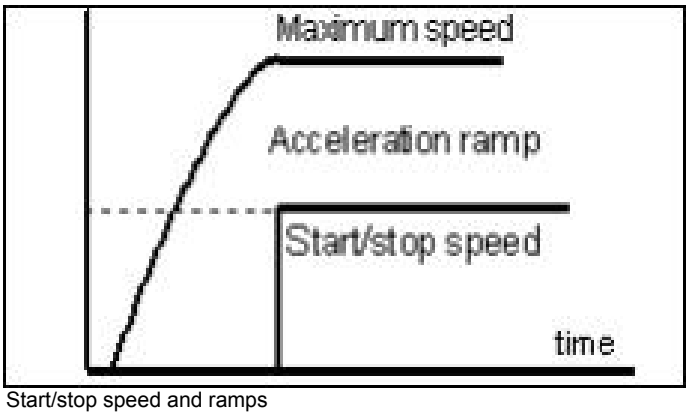
Maximum speed

Use the MOTOR TEST function to ascertain the maximum speed of each axis. This represents the absolute top speed with which an axis can be moved. The units are millimeters per second.

Maximum start/stop speed

Specifies the fastest speed (in millimeters per second) with which the stepper motor can start up without ramping. This value is used to maintain the highest possible machine throughput. Sharp edges or corners do not have to bring the machine to a standstill; movement can continue at the start/stop speed.

This speed **must** be obtained by experiment via MOTOR TEST.



Start/stop speed and ramps

Switch off the ramp and run acceleration tests until step losses occur. For safety reasons, enter 30% of the measured rate.

Rapid speed

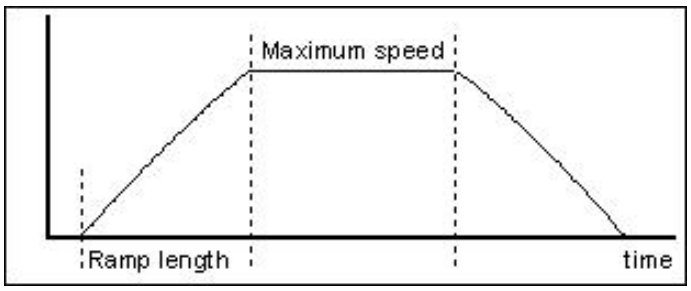
The rapid speed specifies the fastest usable speed when moving from one job location to another – and the tool is not in use.

The rapid speed will be used whenever possible.

Shortest ramp

Specifies, in milliseconds, the time it takes to go from standstill to maximum speed (or maximum speed to standstill).

Each motion is broken into three phases – ramp up, maximum speed, and ramp down.



Speed profile of a movement

Using ramping prevents step losses and safely maximizes the highest movement speed.

Each motion begins with the speed being increased until the maximum speed is reached. The maximum speed is sustained for the longest time possible then the speed is reduced until the motion comes to an end.

The slope of the ramp is constant for all movements - the ramp time is shorter for movements at slower speeds.

Invert movement direction

If a motor is moving in the opposite direction than expected, there are two solutions possible:

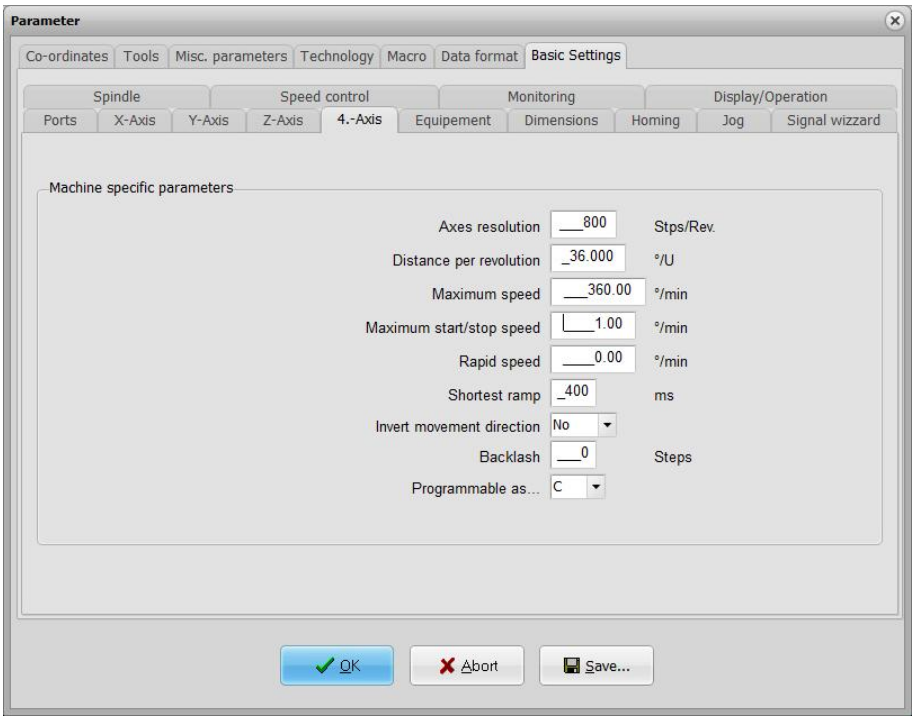
- Swap the motor winding connections
- Check this checkbox - the motor direction signal will be inverted.

Backlash compensation

In some systems, reversing direction results is a positioning error. While these errors are usually small, they can add up over time and lead to unacceptable results. This parameter can be used to compensate for this issue (backlash compensation). The units are additionally indicated steps.

If you are unsure of how to use this parameter, or have a closed loop system, leave this parameter set to zero.

5.2.4. Basic Settings - 4.-axis



Parameter Basic Settings 4.-axis

WinPC-NC is able to manage a 4th axis. Programming maybe affected by a G-code 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, such as speeds, inverting the direction and reference switch are acting in the same way as for axes XYZ are not explained here.

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 tenths of a degree.

Programmable as...

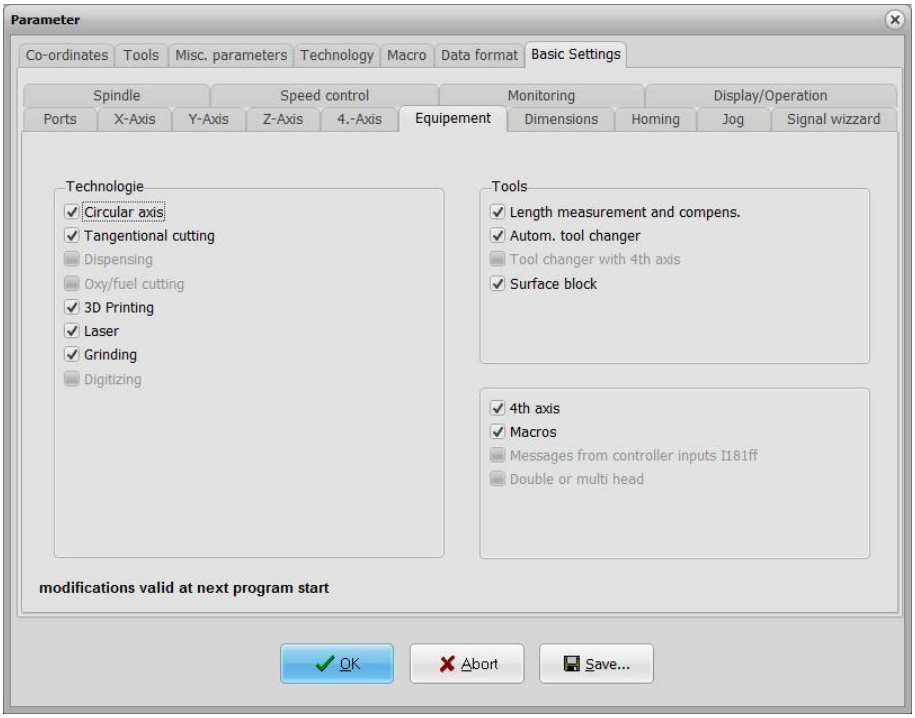
The 4th axis is addressed in G-code programs via a letter.

Axes parallel to X, Y, and Z are designated, respectively U, V, and W. Rotations around the X, Y, and Z axes are designated, respectively, A, B and C.

3D applications often use E for the filament feed.

Select that letter appropriate for your machine.

5.2.5. Basic Settings - Equipment



Parameter-Basic Settings-Equipment

Technology

The equipment settings present those ***WinPC-NC*** technologies that your machine *may* support. When a technology is enabled, the parameters and options specific to it will be visible and can be adjusted.

The specific configuration of your machine may limit the ***WinPC-NC*** technologies available to you. Some machines may only support a single technology (e.g. milling); other may be configurable for multiple technologies (e.g. 3D Printing, Laser). Check with your machine vendor for the specific details of your machine.

Enable those technologies that are applicable as you need them.

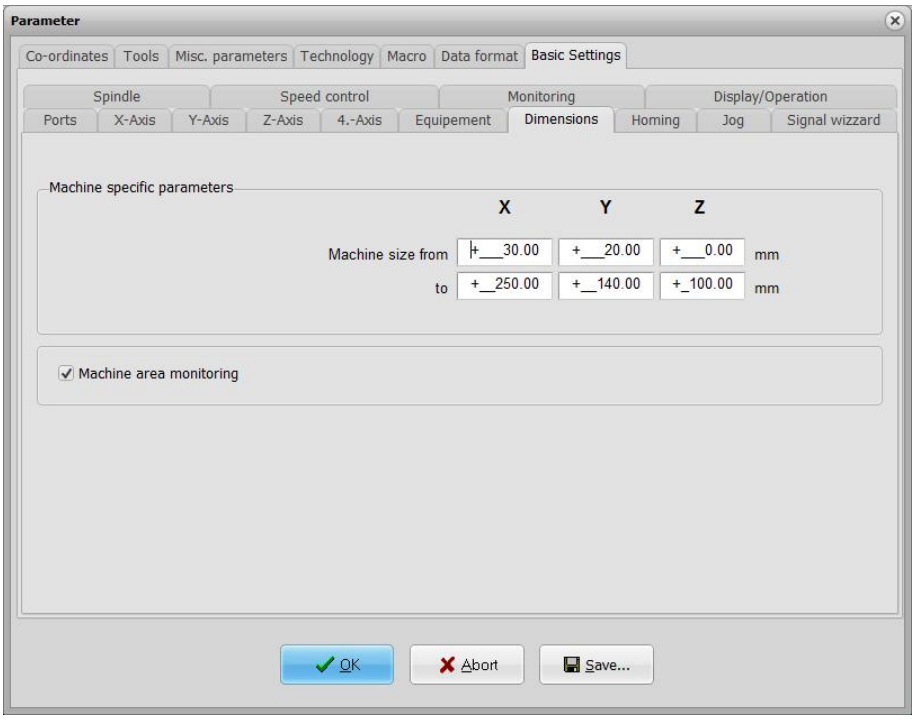
Greyed out technologies are only available in *WinPC-NC Professional*.

Tools

The tools section of the equipment settings presents the checkboxes for enabling automatic length measurement and length compensation, Automatic Tool Changer (ATC) support, and sensor block support (assistance with setting Z zero at the top of a work piece).

The final section of the equipment settings presents the checkboxes for enabling a 4th (rotary) axis, and macros (useful, user defined actions).

5.2.6. Basic Settings - Dimensions



Parameter-Basic settings-Size/Dimension

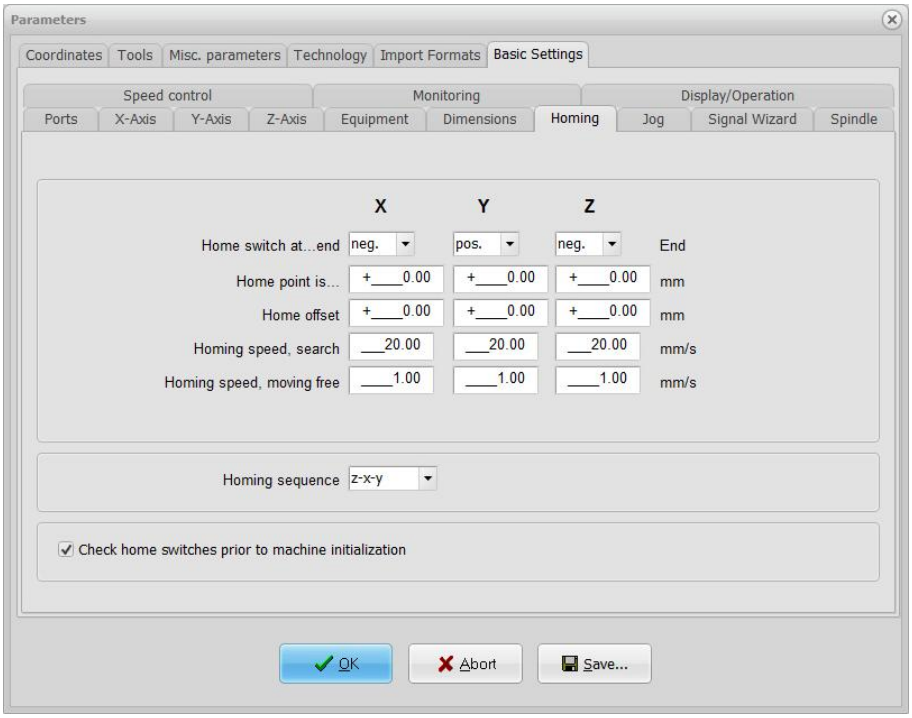
Machine size

Specify the effective movement area of the machine.

Machine area monitoring

WinPC-NC will monitor all movement, including manual jogging, against the machine size when this checkbox is checked. This will prevent any damaging motions into the extremes of the machine.

5.2.7. Basic Settings - Homing



Parameter-Basic Settings-Homing

Home switch at ... end

Specifies which end of the machine the home switches are located. Home switches are most commonly installed on the negative end of an axis, however it is not unheard of to place them on the positive end.

Home point is…

Specifies the home position. This overrides the default where the machine initialization procedure determines the zero position of each axis by searching for the home switches.

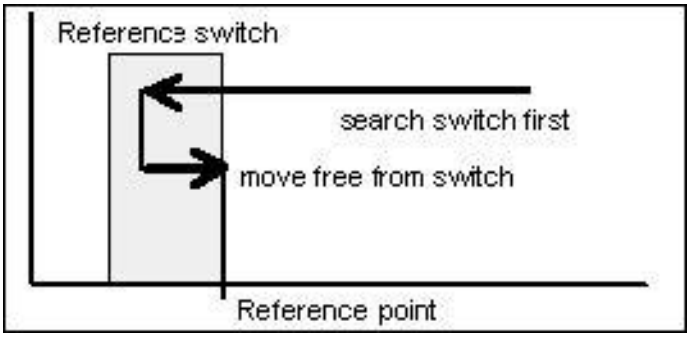
Home offset

Once the home position of each axis has been determined (see the search process description below), it is best of add a small offset to the physical home position to ensure that the home switch is not tripped by any inaccuracy in machine movement.

Typical values are 0.5 to 1 millimeters.

Homing, search speed
Homing, clearance speed

When searching for a home switch, ***WinPC-NC*** starts moving an axis at the *search* speed. Movement stops when the homing switch trips. Movement then proceeds in the opposite direction at the *clearance* speed until the homing switch ceases tripping. The point where the homing switch ceases to trip is the home position for the axis. A **slow** clearance speed is necessary to ensure the shortest possible ramp and the most accurate reference position determination.



Move homing

Homing sequence

The home position determination process cycles through each axis searching for a home switch. This setting determines the order that the axes are searched.

The most common search sequence starts with the Z-axis, moving it upwards first, to ensure that the Z-axis is clear of any fixtures or a work piece. Other sequence orders may be applicable, depending on the machine.

Homing 4th axis…

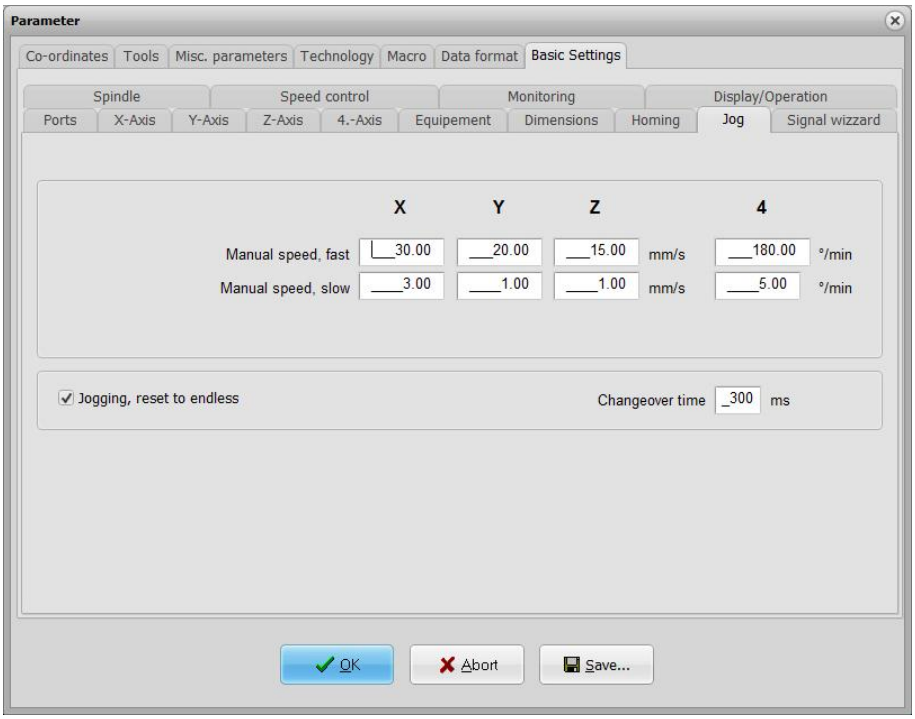
The 4th axis homing process can be performed before or after the other axes.

Checking home switches prior to machine initialization

Some machines do not have individually wired home and limit switches (the home and limit switches wired together). This makes it impossible for ***WinPC-NC*** to unambiguously determine its position (e.g. which axis is reporting a limit or home trip signal) and begin a machine initialization.

When this checkbox is checked, ***WinPC-NC*** will refuse to initialize the machine until all home and limit switches are not tripping. Jog the machine to a location that is far from all ends, then begin machine initialization.

5.2.8. Basic Settings - Jog



Parameter-Basic settings-Jog

Manual speeds - fast and slow

Specifies the jogging speeds – fast and slow.

Jogging, reset to endless

In jog mode, all axes can either be moved endlessly or discrete distances which are selected by the drop-down 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.

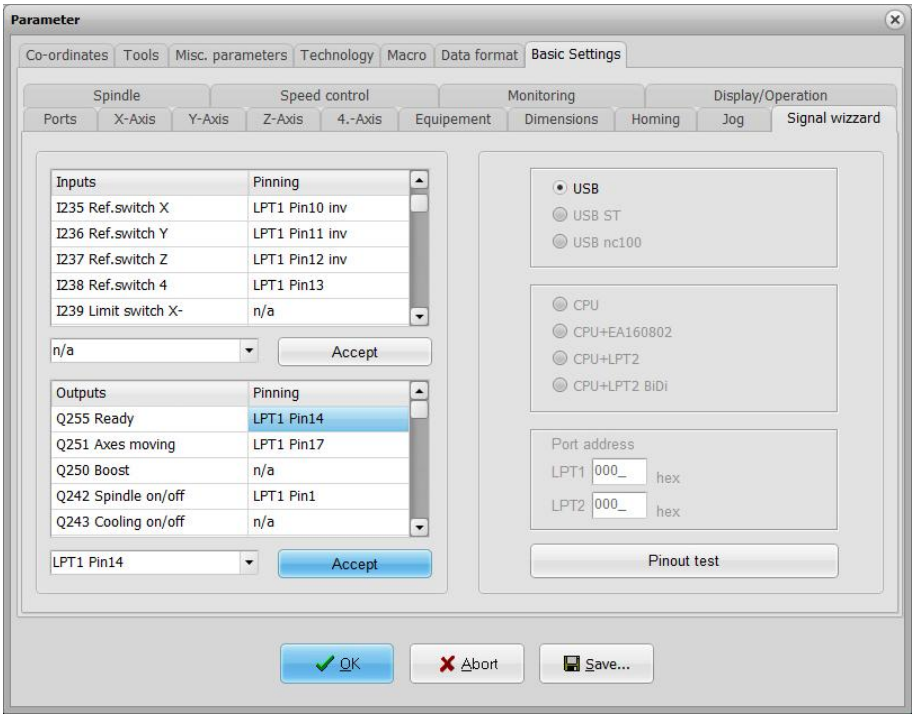
Change over time

Specifies the time, in milliseconds, to delay when transitioning from single step to continuous movement in jog mode.

When a key is hit briefly or a mouse button is hit, ***WinPC-NC*** outputs individual steps. If the key or mouse button is held for more than this time, motion switches to continuous mode and stays this way until the key or mouse button is released.

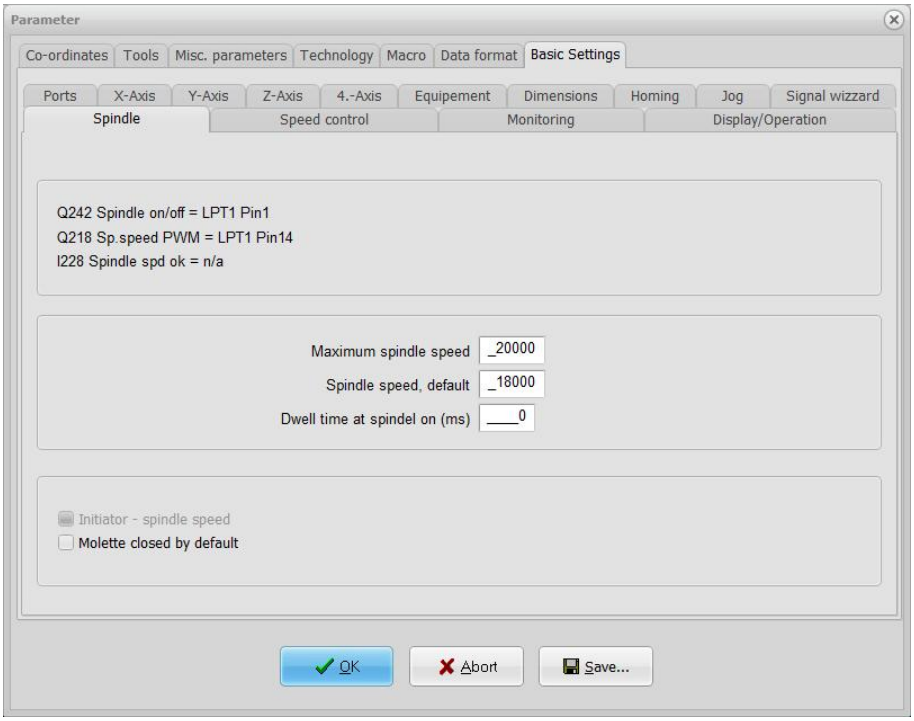
5.2.9. Basic Settings - Signal Wizard

The *Signal Wizard* is a user-friendly way to define the necessary inputs and outputs. Setting up these signals can be complex task. A chapter dedicated to setting up a machine using the *Signal Wizard* may be found later in this manual.



Parameter-Basic Settings-Signal Wizard

5.2.10. Basic Settings - Spindle



Parameter-Basic Settings Spindle

The upper section of the window displays the input and output signals associated with spindle control… a reminder to check that the spindle signals have been properly assigned.

If necessary hardware is available, ***WinPC-NC*** can programmatically control the rotation speed of a spindle.

The spindle speed is encoded as an 8-bit *step number*. Step 0 specifies that the spindle is off, step 255 specifies that spindle is running at its maximum speed. The *step number* is output, encoded, via one of two methods:

1. Pulse Width Modulation (PWM)
   Output via pin 14 or 17 of ***nc100***/***ncUSB*** LPT1.

2. Binary number
   Output via pins 2 through 9 of ***ncUSB*** LPT2**.**

Maximum spindle speed

Specifies the maximum speed of the spindle in revolutions per minute. Check with your NC machine and/or spindle vendor for this information.

Default spindle speed

Specifies the default speed of the spindle in revolutions per minute. The default spindle speed is used whenever no spindle speed information is specified in an NC file or is associated with a tool.

Dwell time after spindle on

Specifies the time, in milliseconds, to wait for the spindle to reach its specified speed when it is first powered on or after a tool change.

Check with your NC machine and/or spindle vendor concerning the spindle dwell time.

Molett (chuck) closed by default

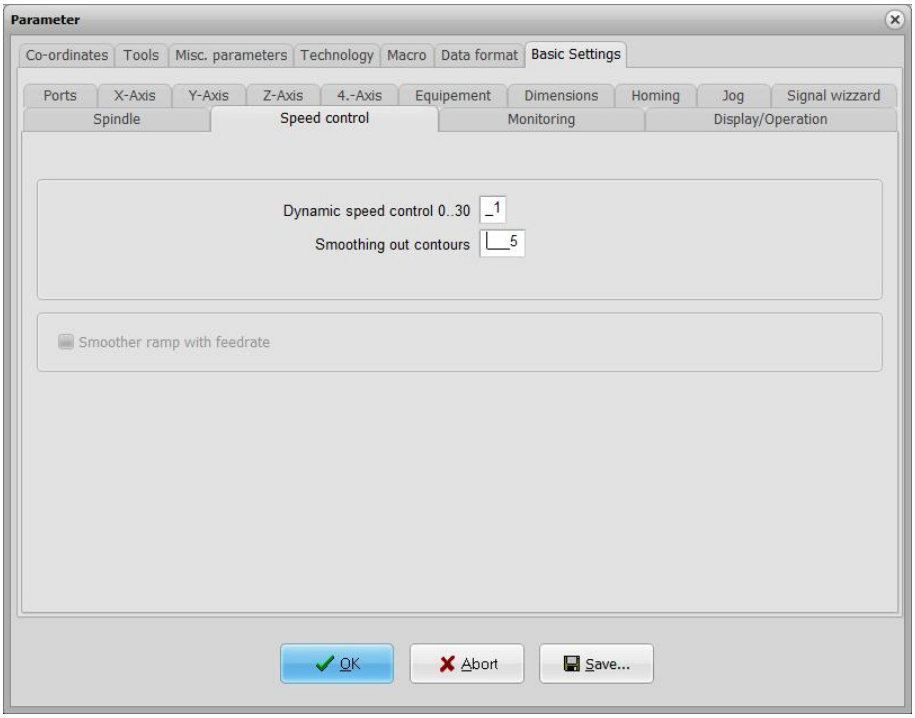
When ***WinPC-NC*** is used with an Automatic Tool Changer (ATC), a spindle with an electrically or pneumatically operated chuck is required. Checking this checkbox ensures that the chuck is closed when the machine is initialized. ***WinPC-NC*** can then maintain knowledge of the last used tool from session to session.

Sensor - Spindle speed

If your spindle exports an “at speed” signal and the necessary hardware is available, it is highly recommended that ***WinPC-NC*** be configured to wait for this signal rather than use a dwell time.

Check with your NC machine and/or spindle vendor concerning the “at speed” signal information.

5.2.11. Basic Settings - Speed control



Parameter-Basic settings Dynamic speed control

Dynamic speed control

The speed control functionality of ***WinPC-NC*** attempts to recognize future contouring (“look ahead”) and to dynamically adjust the cutting speed accordingly. The results are smoother edges and tighter corners.

How well speed adaptation can improve a job strongly depends on the mechanical properties of your machine and spindle. Optimal settings **must** be determined by experiment.

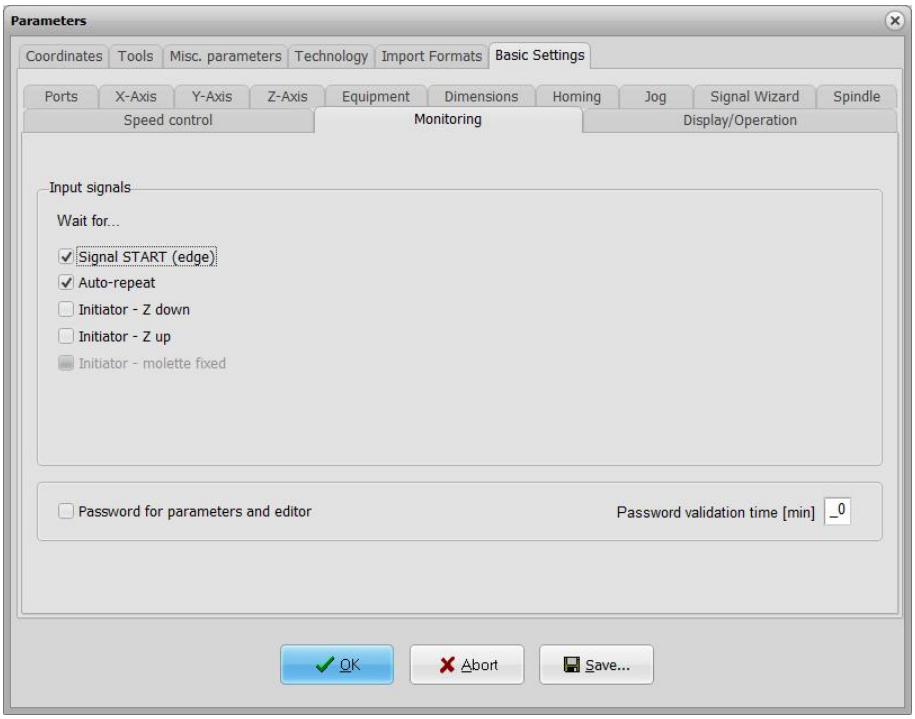
The speed control parameter controls how rapidly the machine will decelerate when it encounters small radius features. A value of zero deactivates speed adaptation; only the action of the set brake angle has an effect.

Contour smoothing

The contour smoothing parameter specifies how aggressively ***WinPC-NC*** will attempt to convert a sequence of small vectors into a smooth curve. A value of zero deactivates contour smoothing.

5.2.12. Basic Settings - Monitoring

All input signals and their associated parameters are grouped together.



Parameter-Basic settings Monitoring input signals

Input signals

WinPC-NC can synchronize the current job with various input signals. Check with your NC machine and/or spindle vendor concerning which signals may be available.

The signals and their definitions:

Start signal	***WinPC-NC***, after the current job been started, waits until this signal (“external hardware is ready”; “you may start”) to transition from LOW to HIGH before beginning processing.	**Clampin**	***WinPC-NC***, after the current job been started, waits for this signal (“work piece clamping is activated”) to transition from LOW to HIGH before starting processing.

Sensor Z up	***WinPC-NC*** waits until this signal (“Z-axis is up”) to transition from LOW to HIGH before continuing processing.	**Spindle speed**	***WinPC-NC*** waits until this signal (“spindle has reach specified speed”) to transition from LOW to HIGH before continuing processing.
Sensor Z down	***WinPC-NC*** waits until this signal (“Z-axis is down”) to transition from LOW to HIGH before continuing processing.		

Automatic repeat

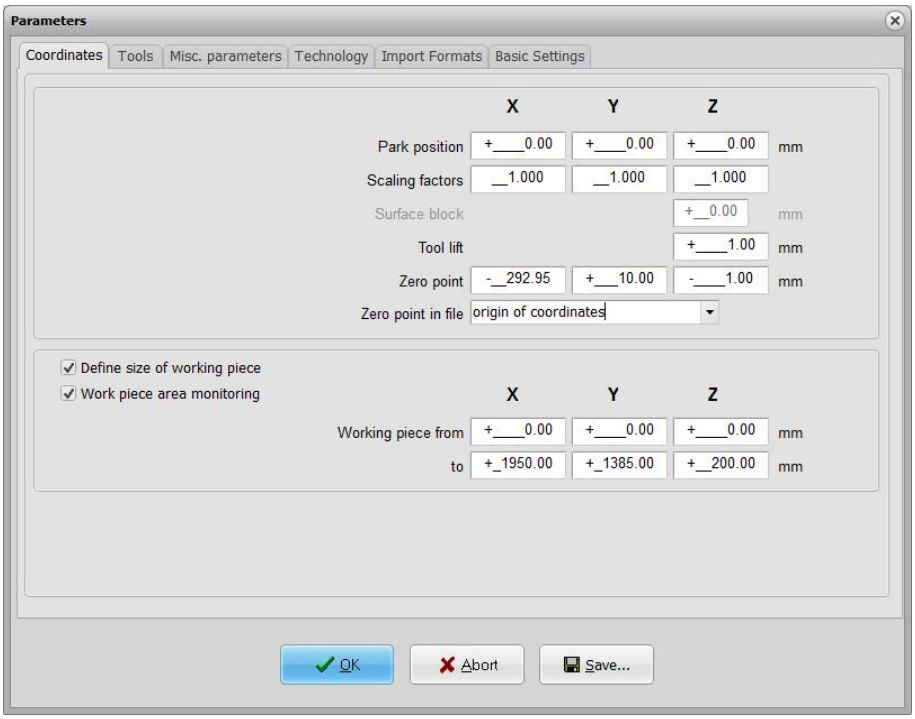
When this checkbox is checked, ***WinPC-NC*** will check for the *Start* signal, and if it is HIGH, restart the currently loaded job. No keyboard or mouse input is required.

Password and period of validity

This parameter specifies a password which must be entered prior to changing any parameters or a loading an NC file.

The period of validity specifies how long the password will remain in effect.

5.3. Coordinates



Parameter-Coordinates

Park position

The park position is a convenient location that the machine can be quickly moved to (be parked). For example, a location where it is easy to manually change a tool or “out of the way” when placing stock in a clamp.

The park position can be selected as the target of a jog or can be used as part of a tool change operation.

The position is in **absolute machine coordinates** (distances relative to the home switches); units are in millimeters.

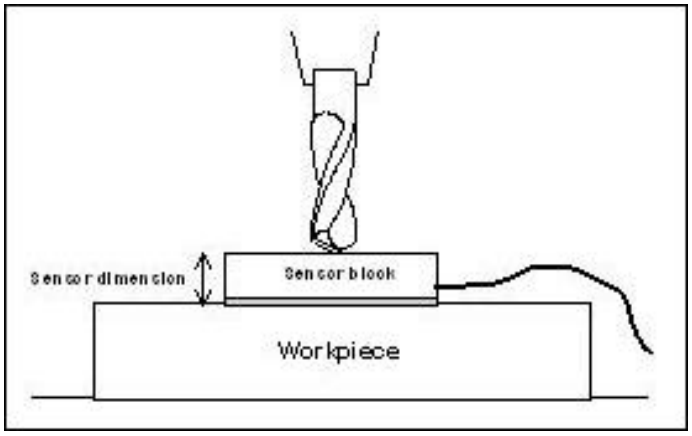
Scaling factors

The scaling factors make it possible to compensate for machine initialization difficulties. If the axis resolution and the unit of measurement are set correctly but the machine does not consistently move to exact position expected, a scaling factor can be used correct this issue.

These values are multiplied times the machine coordinates of the imported NC data.

Surface block

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



Automatic measurement of the Z-axis zero point

The measuring procedure involves several steps:

1. Place the sensor block on the work piece surface or on the machine
2. Jog the tool over the sensor block.
3. Start the measurement using the MOVE-JOG menu function
4. ***WinPC-NC*** slowly moves the tool down to the sensor block and stops when it makes contact. The position is checked and added to the defined sensor block height, the result being stored as a parameter.

Tool lift

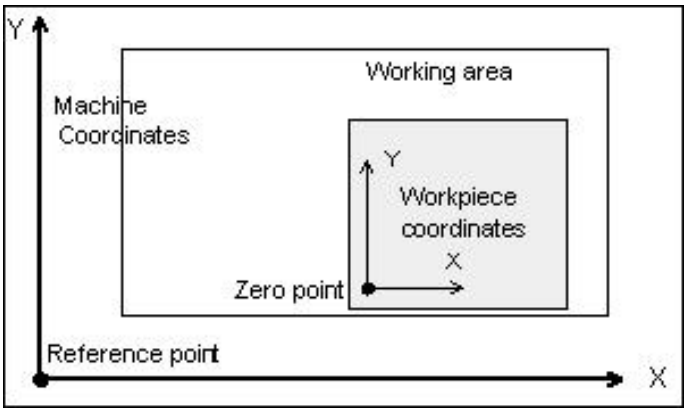
The clearance distance specifies an additional height of the Z-axis above the zero point level. With each job process the tool is lifted above the zero point by this distance and the new height is used as tool lift height.

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 origin of the coordinates of the work piece within an NC file. All distances are measured relative to this point.

The zero point in an NC file must be assigned to a physical point within the working area of the machine before a job can begin.



Workpiece and machine coordinates

Assigning the zero point to a position within machine work area is a manual process.

The machine position is in **absolute machine coordinates** (distances relative to the home switches); units are in millimeters.

The assignment is made by jogging to the desired position and saving the position as the zero point. A zero point can be specified for X, Y, Z, XY, and XYZ. Once a zero point has been assigned for X, Y, and Z, the job can begin.

Zero point in file

Chose one of the six possible positions for the NC file zero point.

Bottom left	The zero point is at the smallest X- and Y-axis coordinates in the file, normally at the bottom left edge. Use this choice when working with HPGL files.	**Coordinate origin**	The zero point is at the coordinate origin assign by the CAM program that produced the NC file. Use this choice when working with G-code files.
Center	The zero point is exactly in the center of the coordinate dimensions in the X- and Y-axis directions of the work piece. This setting is useful for processing round work pieces (e.g. plates).	**Middle right**	The zero point is positioned at the highest X coordinate and exactly between the smallest and highest Y coordinate.
Bottom right	The zero point is positioned at the highest X and smallest Y coordinate of the file.	**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 millimeters and distances refer to the machine reference point (machine coordinates).

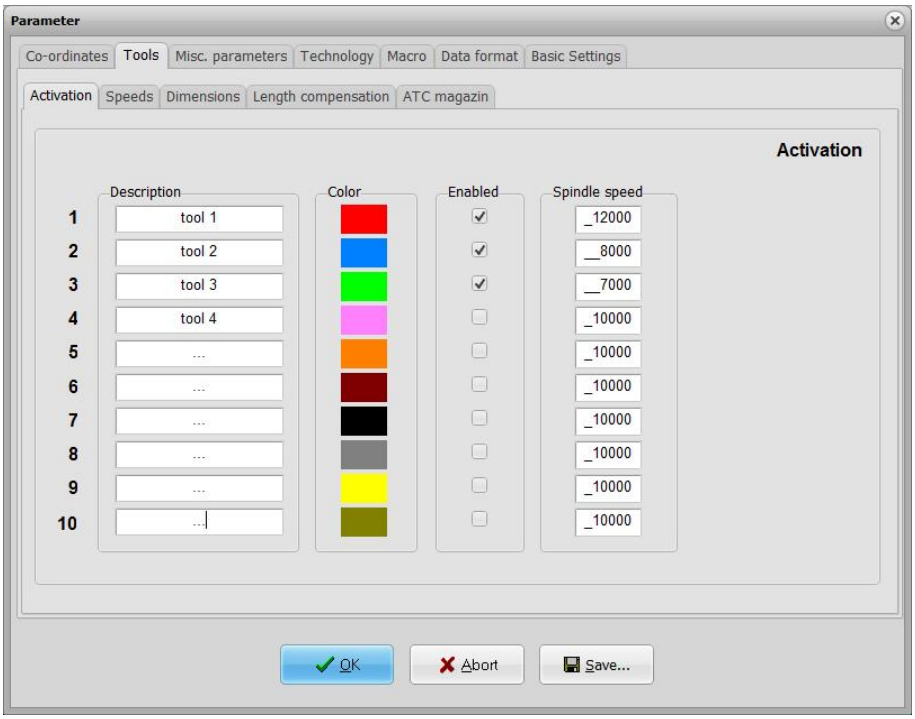
The management of an additional area is activated when you enable the corresponding work piece by checkbox.

5.4. Tools

WinPC-NC has a user-friendly tool management function. It is possible to manage up to 10 tools - each with unique parameters - for each 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

Each tool is identified with a name which is used when a tool change prompt appears.

Color

Clicking the color box associated with each tool opens a color selection dialog. Choose a color that suits your requirements or harmonizes with the CAD program you are using. This is useful for those with color blindness as they can specify a color palete that works best for their specific difficulties.

Enabled

Each tool can be individually enabled or disabled. When a tool is disabled any command involving it will be ignored and any toolpath involving it will not be shown in the *Display Area.*

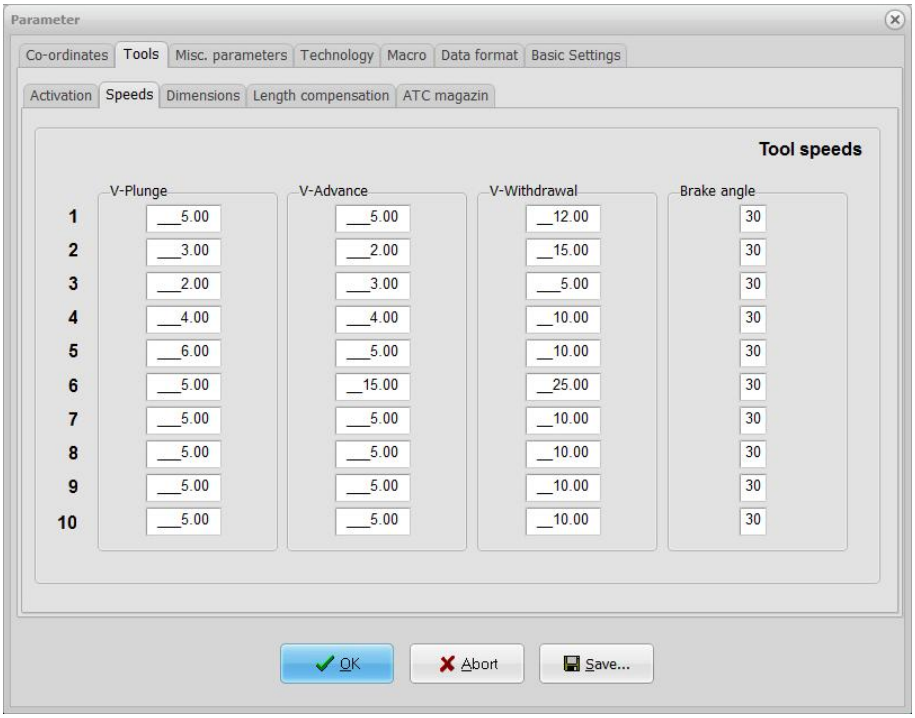
Tools can also be enabled and disabled via the numbered tool area at the left of the *Tool Bar*.

Spindle speed

Defines the spindle speed for the specified tool should:

- The spindle speed not be specified from within an NC file.
- ***WinPC-NC*** is directed to ignore the NC file spindle speed directives.

5.4.2. Tools - Speeds



Parameter-Tools Speeds

Plunge speed

Specifies the speed that each tool is pushed (plunged) into the work piece.

This number is dependent on the material being used, the tool involved, and the mechanical properties of the machine.

Advance speed (feed rate)

Specifies the speed that each tool is pushed (fed) across the work piece. This value is not used in straight forward (hole only) drilling applications.

When ***WinPC-NC*** is used for milling, engraving or grinding, this number is dependent on the material being used, the tool involved, and the mechanical properties of the machine.

Withdrawal speed

Specifies the speed that a tool with withdrawn (raised) from the work piece.

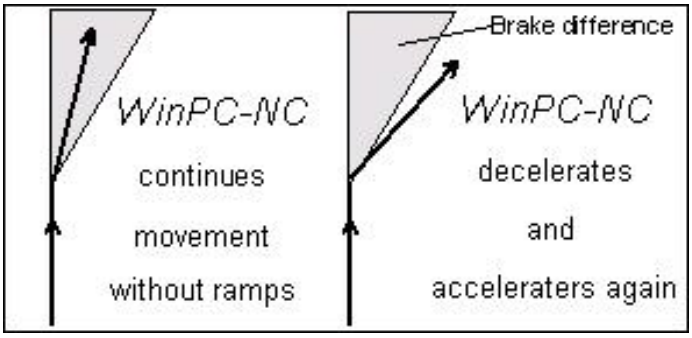
Brake angle

Specifies the maximum angle, in degrees, between two linear movements where movement can (still) take place at full speed.

This parameter is not used in straight forward (hole only) drilling applications.

Each linear motion normally has an acceleration phase, a constant speed phase, and a deceleration phase.

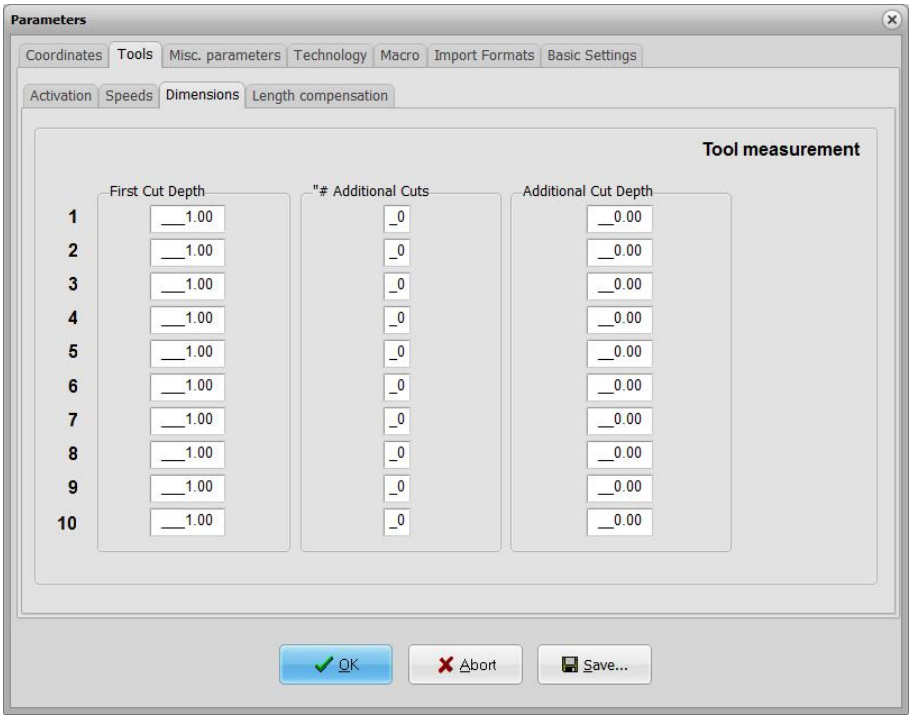
If the angle between the current motion and the next is less than the break angle, no deceleration is necessary - the constant speed phase can continue.



Brake angle

An example of how this works can be seen in movement around a circle. A circle is approximated via large number of individual vectors. The angle between any two of the vectors is very small. As a result, the circle can be completely traversed at maximum speed.

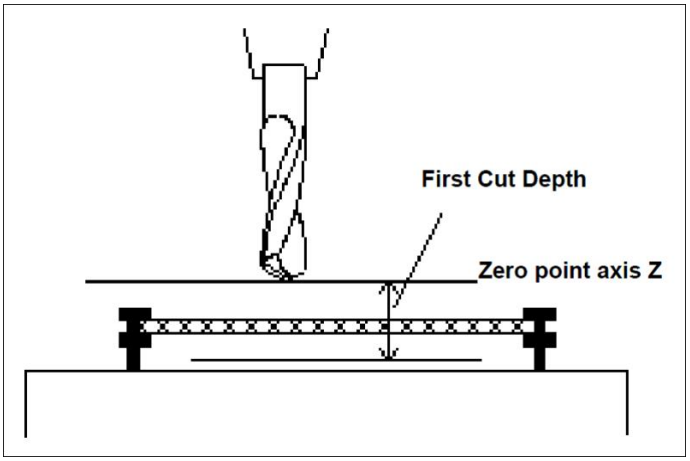
5.4.3. Tools - Dimensions



Parameter-Tools Dimension

First Cut Depth

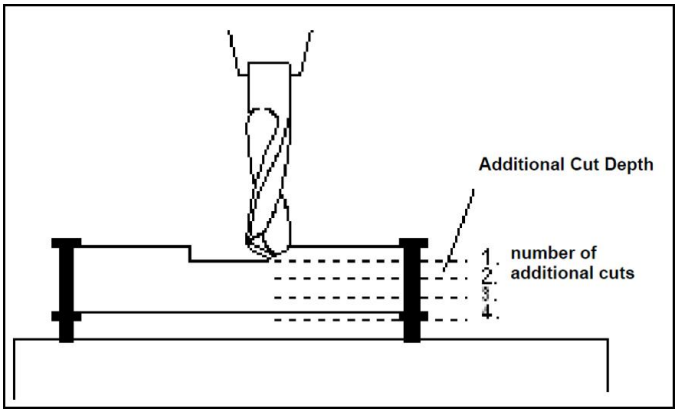
Specifies the depth of the first cut from the Z-zero point. Additional cutting depth is handled separately, via the # Additional cuts and the Additional Cut Depth parameters (see below).



First cut depth measured from the zero point of the Z-axis

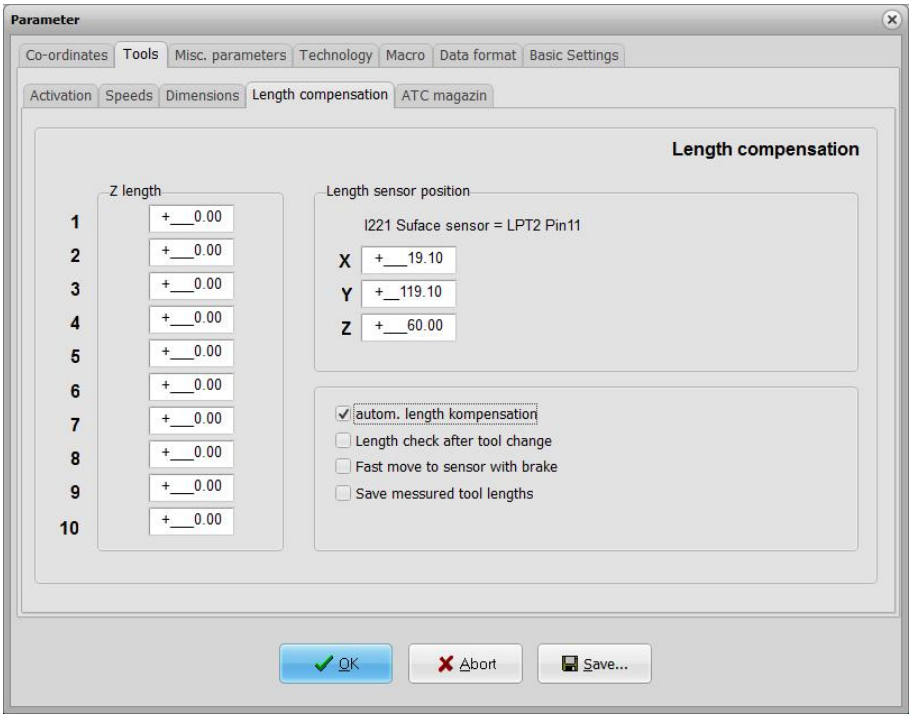
Additional cuts and Additional Cut Depth

Additional cuts specifies the number of cuts to perform *after* the first cut. Each additional cut adds an Additional Cut Depth to the depth of the previous cut.



Additional cuts and Additional Cut Depth

5.4.4. Tools length - measurement and compensation



Parameter-Tools-Tool length measurement and compensation

These setting are only visible if the tool length measurement and compensation functions are activated in *Basic Settings-Accessories*.

Z-axis length

The length of a tool is specified here. Normally, these boxes cannot be edited; the values are registered automatically when tools are measured. The length difference used by the compensation function is calculated on the basis of these values.

Sensor position

The tool length sensor is a switch, button, pressure plate, solid state contact sensor, or other device which transitions, unambiguously upon tool contact.

This parameter defines the position of the tool length sensor in **absolute machine coordinates** (distances relative to the home switches) and a Z-axis height; units are millimeters. To measure a tool, ***WinPC-NC*** moves to this position and Z height, then lowers the tool slowly until the sensor transitions (off to on or on to off).

The distance from the Z height to the height the sensor transitions is used as the tool length.

Automatic length compensation

Automatic tool length compensation is enabled/disabled.

Length check after tool change

Enable/disable tool length compensation after a tool change.

By default, ***WinPC-NC*** does not perform automatic tool length compensation after a tool change.

Fast move to sensor with brake

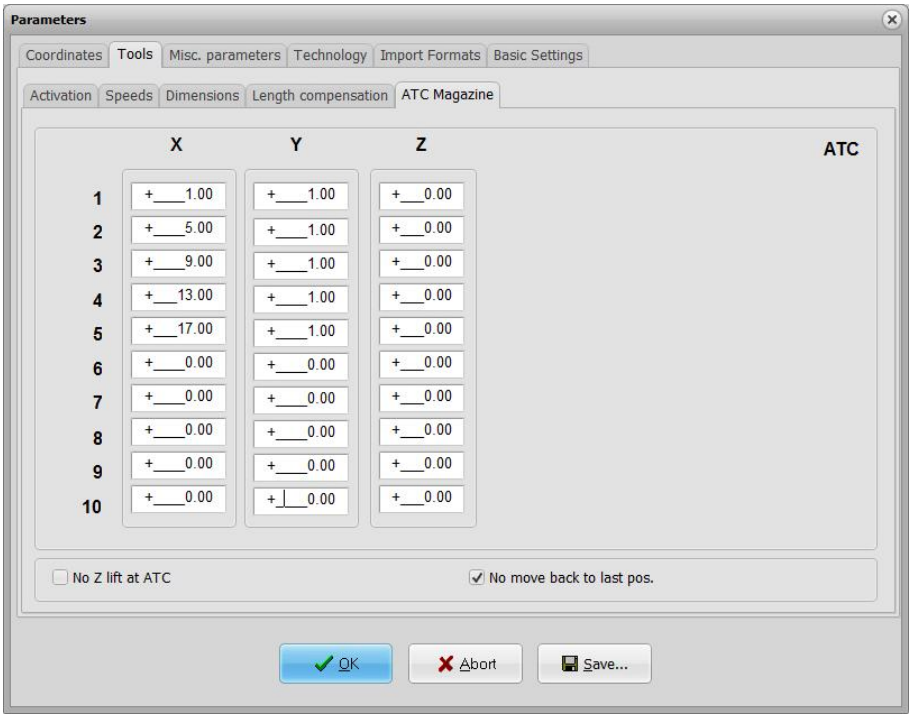
The automatic tool length compensation process can be sped up by enabling this feature. When enabled, ***WinPC-NC*** will move to the sensor at the fastest possible speed, accelerating and decelerating as necessary.

Save measured tool lengths

WinPC-NC will save all measured tool lengths in the parameter file when this feature is enabled. In applications that constantly use the same tools, this can speed things up - the tools do not have to be measured again.

5.4.5. Tools - ATC Magazine

This setting is only visible when the Automatic Tool Changer (ATC) capability is enabled in *Basic Settings-Accessories*.



Parameter-Tools-ATC Magazine

WinPC-NC can control an Automatic Tool Changer (ATC) with up to 10 magazine positions.

A spindle with an electrically or pneumatically operated chuck is required when using an ATC. This chuck must be controlled via input/output controls.

The position of each magazine location should be specified at a short distance to the actual pick up location. Macros should be used to handle the exact sequence of picking up and releasing tools as well as handling the final movements, waiting times, and any input/output controls.

The macro procedure is described in a subsequent chapter.

Moving tool magazines (e.g. retractable cabinets, circular magazines) require 4th axis control only available in ***WinPC-NC Professional***.

No Z lift at ATC

By default, ***WinPC-NC*** does a lift to the maximum Z height before moving to the appropriate magazine location.

Enabling this feature will disable the maximum lift, saving time.

CAUTION: Enabling this feature may increase the chance of a collision with a fixture or other components.

No move back to last position

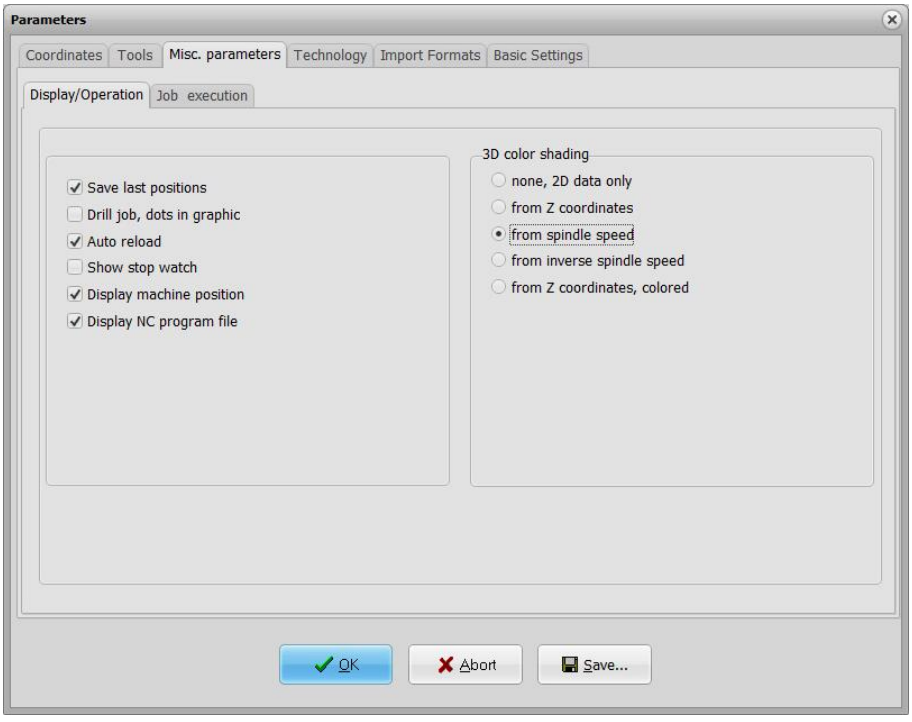
Enable/disable remembering the last position before a tool change.

Moving to a tool changer magazine position, ***WinPC-NC*** will remember the previous position and return to it after a tool change. This often results in unnecessary movements and may not be desirable.

5.5. Miscellaneous parameters

Settings which apply to a job process or ***WinPC-NC*** operation.

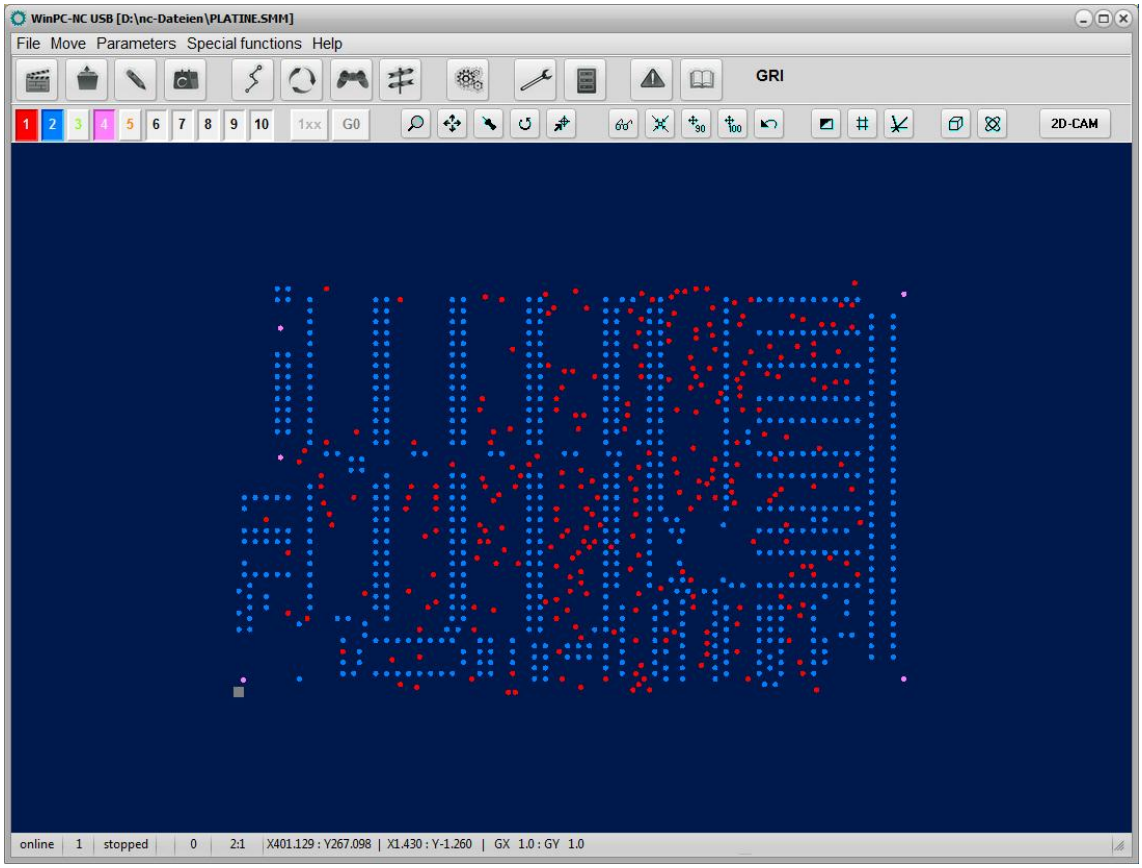
5.5.1. Miscellaneous - Display/Operation



Parameter-Misc. parameters-Display/Operation

Drill job, dots in graphic

By default, the *Display Area* does not show the location of any drill holes. Activating this parameter places a small circle at the hole insertion points for HPGL, MultiCAM, or G-code files.



Drilling job with hole locations indicated

Automatic reload

When this feature is enabled, ***WinPC-NC*** continuously monitors the currently loaded file and reloads it whenever the file modification time is newer than that of the copy currently loaded.

It is possible to edit an NC file in the drawing program, switch back to ***WinPC-NC***, and immediately see the result of the changes.

Show stop watch

Display the job stop watch. The clock starts when the job starts and stops when the job stops. The time is displayed in hours, minutes, and seconds of elapsed time.

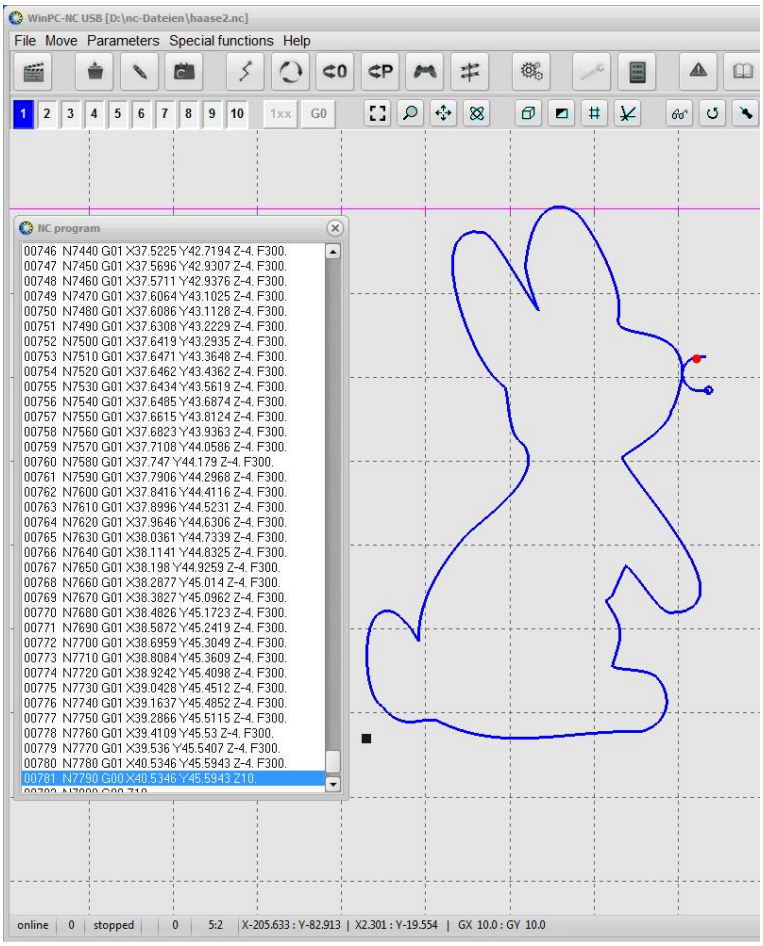
WinPC-NC stop watch

Knowing the exact length of job is often useful. For instance, it is a critical piece of data necessary for costing commercial work.

Display position

In ***WinPC-NC,*** the display of the real-time position data can be enabled/disabled. The data appears in a small window.

NC-Program window



Program window during operation

When this feature is enabled, a window displaying the currently loaded NC program will appear. A *cursor bar* is updated in real-time to indicate the command currently being executed.

Postscript and DXF files cannot use this feature. The maximum number of lines that can displayed is 300000.

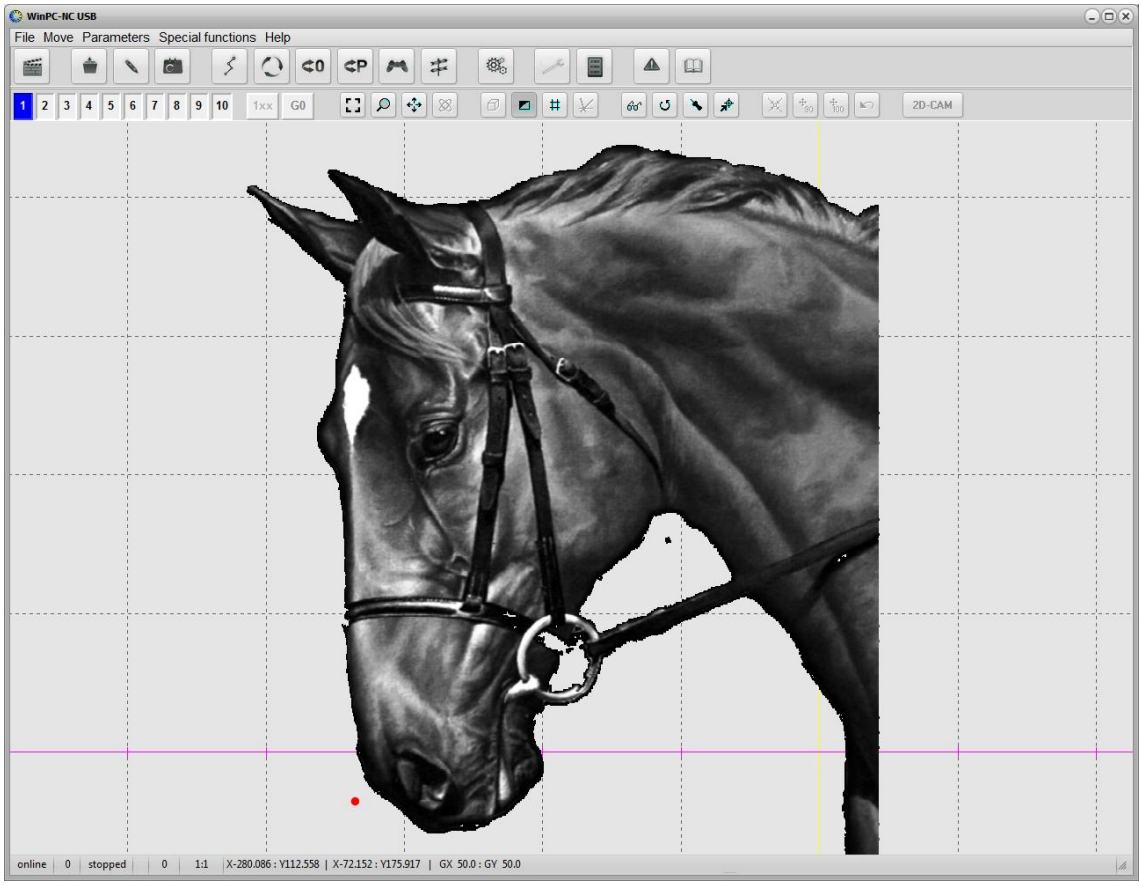
Save last position

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 saved values remain valid even after *Exit* and *re-start*.

3D color shading

In ***WinPC-NC*****,** the default view after loading a 3D file or relief is looking down onto the data. This can be visually confusing.

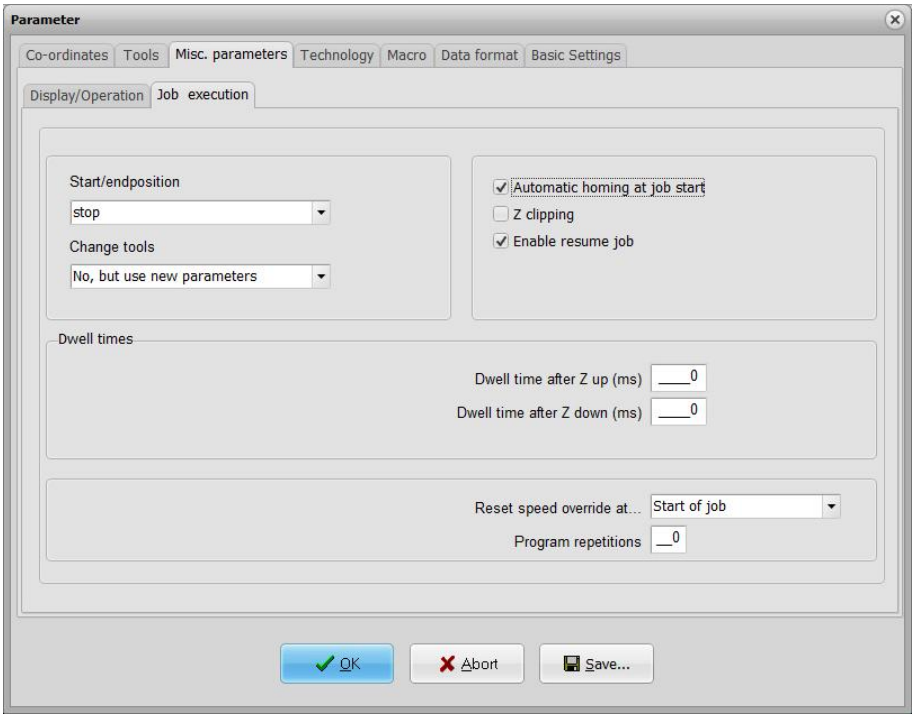
How the data is shaded – presented – can be selected. Choose one of the 5 possible methods that best suits you:



Color shade according for relief- for laser engraving or grinding applications

From spindle speed / from inverted spindel speed	This option is often useful with grinding applications where different spindle speeds or PWM signal levels indicate the gray levels. Inverted shading a higher PWM value shifts the color towards lighter nuances.
none, 2D data only	Here a 2D display is presented with lines and drillings in the selected tool color
From Z coordinates / from Z coordinates colored	Colors are shifted - lighter 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.

5.5.2. Miscellaneous - Job process



Parameter-Miscellaneous parameters-Job process

Start/End position

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

There are 4 possible start and end positions:

Stop	***WinPC-NC*** stops at the reference position after initialization and at the last coordinate at the end of each job.	**Zero point**	The machine moves to the zero point after initializtion and at the end of each job.
Park Position	***WinPC-NC*** moves to the park position after initialization and after at the end of each job.	**Zero point and clearance distance**	The machine moves to the job zero point then lifts 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.	**No, but use new values**	No tool change is performed, although new values for plunge depth, plunge speed, and feed speed are adopted.
Yes	Performs the tool change; remains stopped in the current position for every tool change.	**Yes by ATC**	Performs the tool change via an ATC.
Yes, at the park position	Performs the tool change; moves to the park position for every tool change.		

Homing at a job

When enabled, a homing sequence is automatically performed by ***WinPC-NC*** prior to each job. This is **highly recommended** since rehoming will recalibrate the machine, eliminating any accumulated stepping errors.

Z-axis clipping

When enabled, Z-axis moves will be clipped. No Z-Axis move below the working area will occur.

Enable resume job

WinPC-NC**,** by default, does not allow an interrupted job to continue (resuming interrupted jobs often leads to errors). Care should be taken when enabling the ability to resume a job.

Dwell time after Z up and Z down

In many applications, it is a good idea (or a requirement) to pause briefly after the tool has been lowered or raised or before starting X-axis and Y-axis movements. For example, when working with flexible materials or when a tool takes time to come up to speed (or to stop).

The dwell time is specified in milliseconds.

Reset override speed

The feed rate and spindle speed may be adjusted while a job is in progress. A change stays in effect until it is manually changed or one of the following occurs:

Machine reset or program reset	Values are returned to their 100% values after a machine or program reset.	**Job start**	Values are returned to their 100% values when a job is started.
New file	Values are returned to their 100% values when a new NC file is loaded.		

Program repetitions

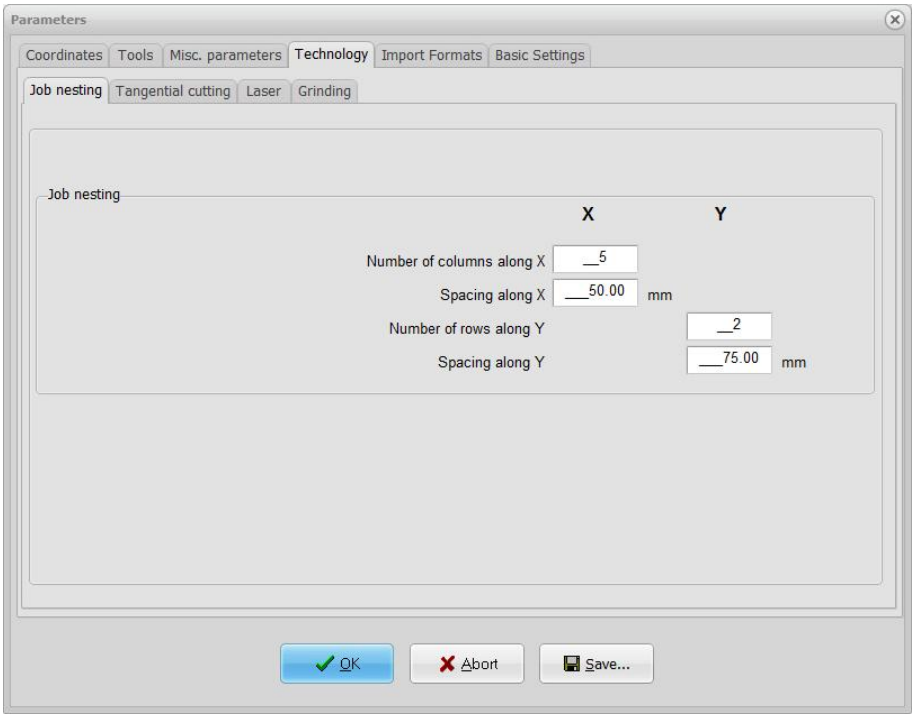
The program repetitions parameter enables you to repeat a working process up to 999 times. Batch production is easily facilitated by this parameter and using the *Start* signal.

5.6. Technology

The technology tab displays all of the available technologies (applications) that ***WinPC-NC*** supports**.** These technologies can be enabled and disabled, as necessary, according to your needs.

Special hardware is a required by some of these technologies.

5.6.1. Technology - Job nesting



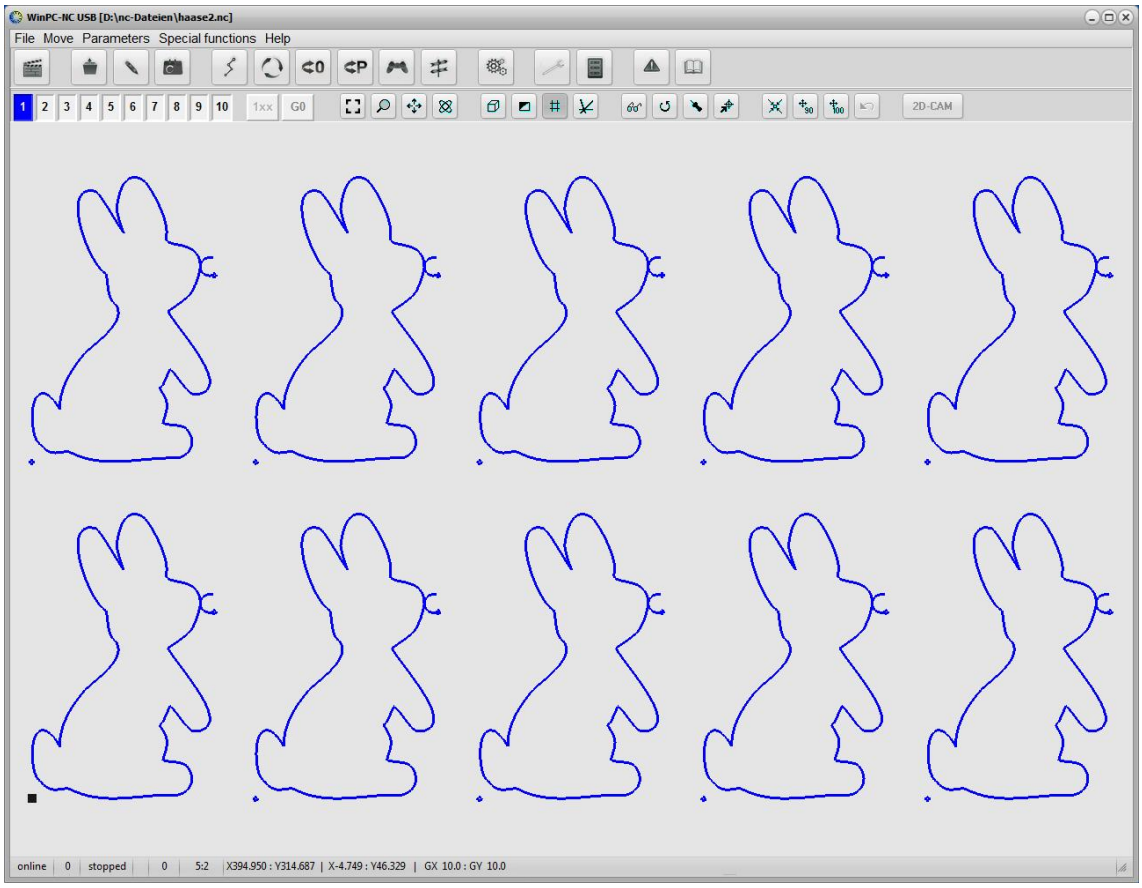
Parameter-Technology-Job nesting

Number of columns along X
Number of rows along Y

These parameters specify the numbers of pieces in each row and column. The product of these numbers is the total number of pieces to be produced.

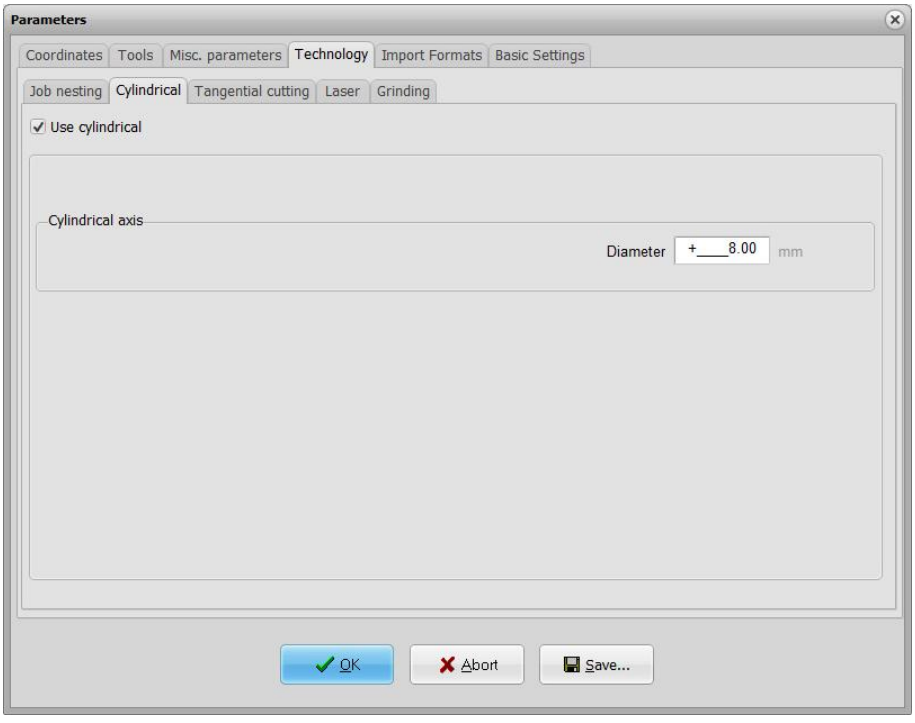
Spacing along X
Spacing along Y

These parameters specify the distance, in millimeters, between the working pieces in the X and Y directions. These spacings must large enough to prevent any tools from touching consectutive pieces.



Nested job, 5 by 2 pieces

5.6.2. Technology - Cylindrical axis



Parameter-Technology-Cylindrical axis

Use cylindrical

Enable or disable cylindrical engraving.

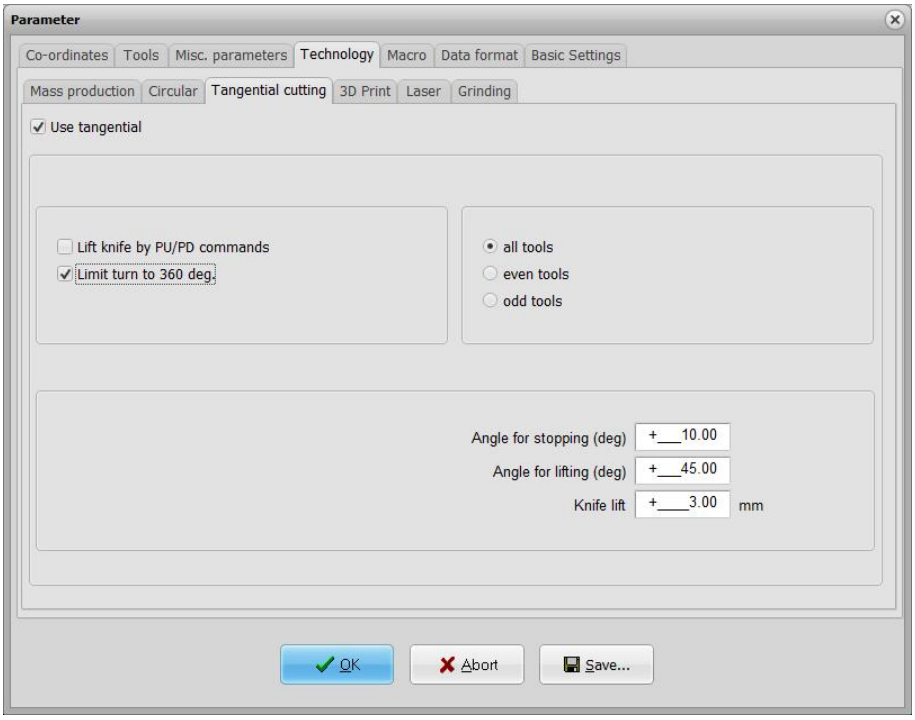
Diameter

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

Cylindrical engraving is possible only when using HPGL formatted NC data.

Cylindric engraving example

5.6.3. Technology - Tangential cutting



Parameter-Technology-Tangential cutting

Use tangential

Enable or disable tangential cutting.

Tangential cuts are only possible if the 4th axis is equipped with a 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 perforation tools.

Lift with PU/PD

Lifting the cutter during rotations is normally controlled by the NC program.

When this checkbox is checked, the PD and PU HPGL commands will be handled entirely via macros.

Rotation of max. 360 degree

Certain tangential heads cannot rotate more than 360 degrees without damaging themselves. Check this checkbox if you own such a tangential head.

Foil cutting with tangential cutter

Angle for stop/lift and lift height

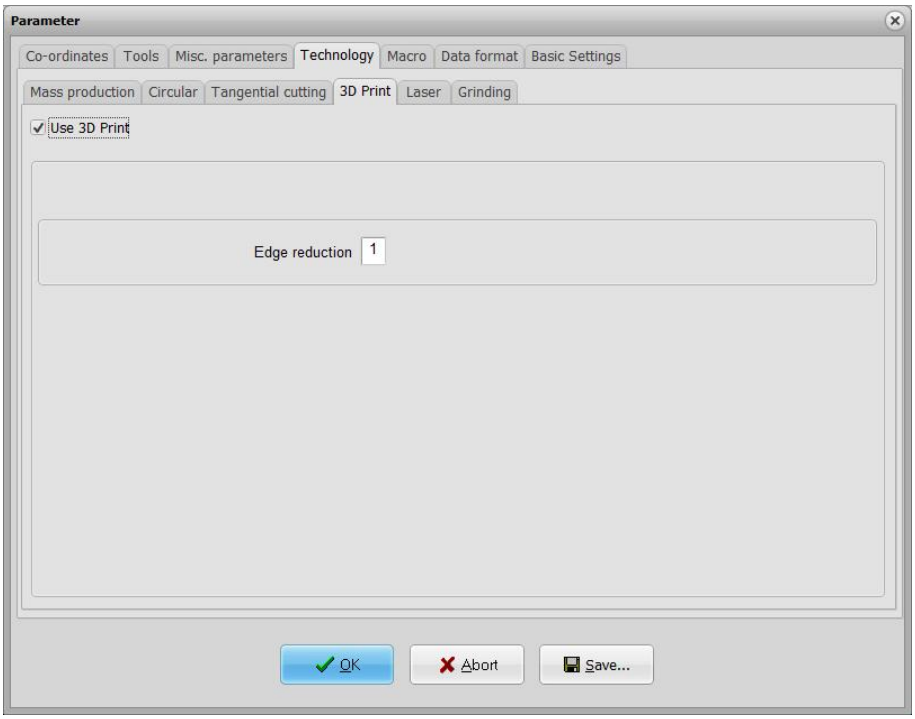
When a rotation larger than the stop angle is requested, the cutter will stopped, and a reset will be performed before the job continues.

When a rotation larger than the lift angle is requested, the cutter will stopped, the knife lifted, and a reset will be performed before the job continues.

The knife lift specifies the height, in millimeters, to which the knife must be lifted before a rotation. The height should be set such that no material damage occurs.

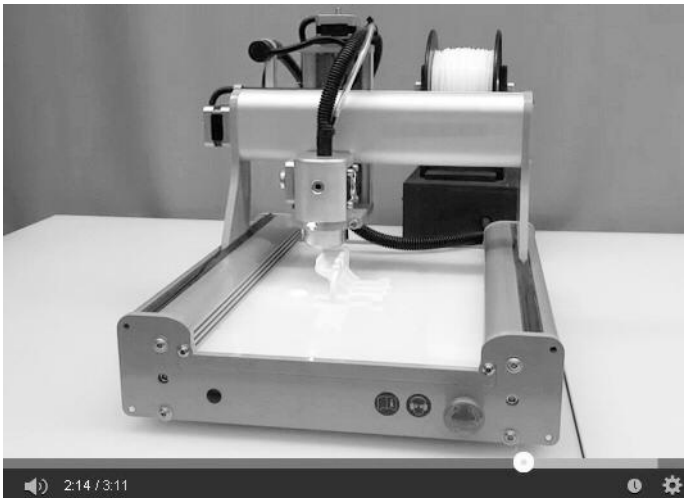
5.6.4. Technology - 3D-Printing

Enable or disable 3D printing.



Parameter-Technologie-3D-print

3D printing requires a special *print head*. The *print head* filament feed control must be connected to ***WinPC-NC.*** This requires using a 4th axis and connecting other inputs and outputs. Care must be taken to restrict speeds so as to not damage the *print head* or create a mess.



Machine with 3D-print head running a job

Edge reduction

Specifies the edge reduction factor. The filament feed speed will be reduced step-by-step in corners and on edges. This can help avoid messy problems in these areas.

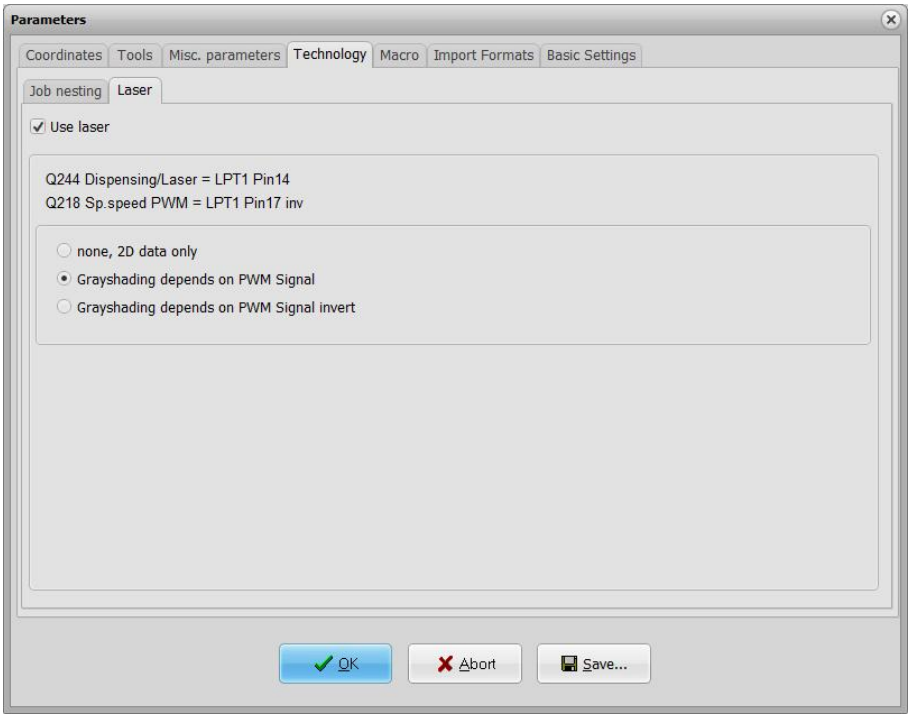
5.6.5. Technology - Laser

With ***WinPC-NC*** it is possible to have a milling spindle, a cutting/engraving laser, or both at the same time.

The laser is controlled by two outputs:

- Laser on/off is controlled by the Q244 output
- Laser power is controlled by the Q218 output.

The Q218 output is a PWM signal that controls laser power, from 0% to100%.



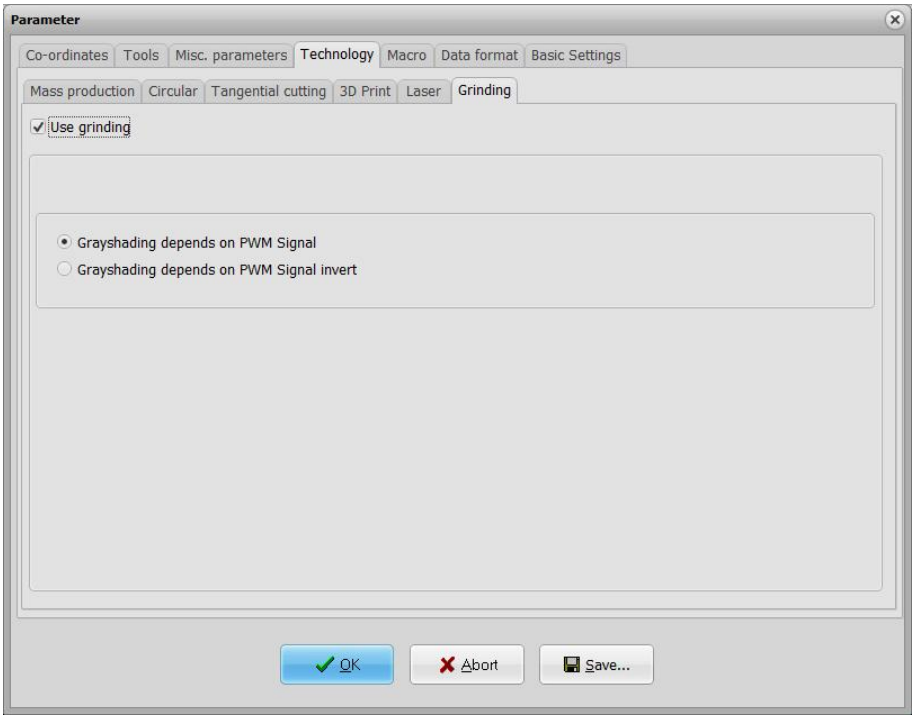
Parameter-Technology-Laser

Grey scale depends on PWM signal
Grey scale depends on PWM signal inverted

Laser engravings quality can be improved by using grey scale data. The grey scale data can be PWM encoded directly or as inverted data.

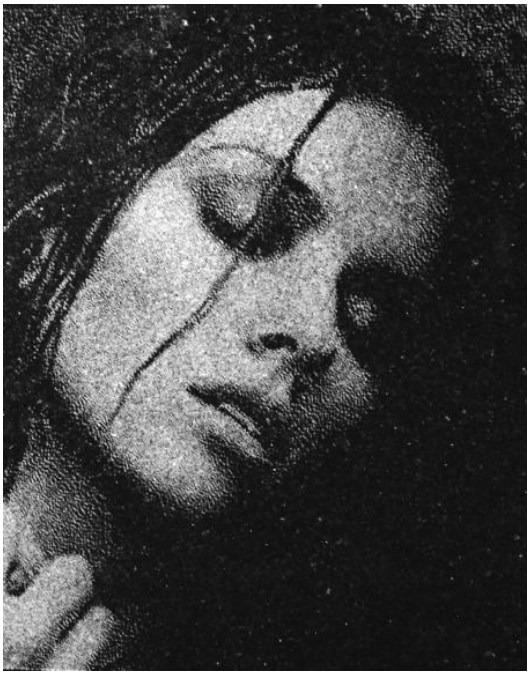
5.6.6. Technology - Grinding

Grinding with control of the grinding pressure is available. For example, grave stones or other stone surfaces and be inscribed and/or images can be embossed.



Parameter-Technology-Grinding

Grey scale depends on PWM signal
Grey scale depends on PWM signal inverted



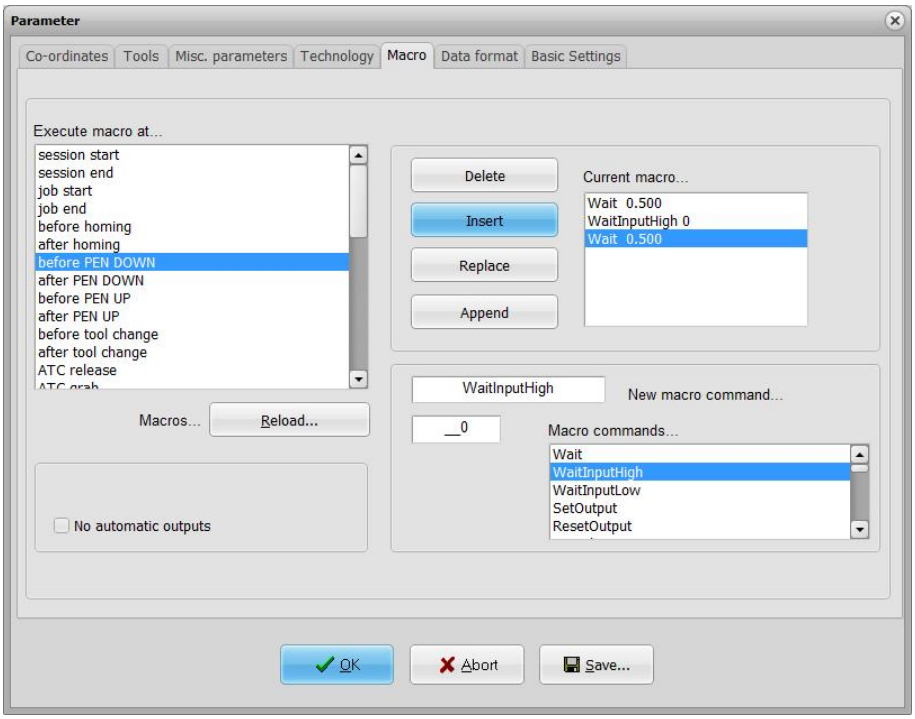
Images can be embossed as grey scale data. The grey scale determines the spindle speed.

5.7. Macros

WinPC-NC has a powerful, user-definable, macro facility designed to enable or enhance a wide range of work flows and applications. Custom hardware requirements are easily handled via this facility.

Macros must be enabled before they can be used in *WinPC-NC*.

WinPC-NC always stores its macro definitions in the WINPCNC.MAC file when the parameters are saved. They are always valid and apply to all projects.



Parameter-Macro

The *Reload* button will load most recently saved macros from the WINPCNC.MAC file.

The dialog box for macro handling is divided into several parts:

Execute macro at … box	The list of ***WinPC-NC*** events where a macro can be used.	**Current macro… box**	The current macro. A new macro is assembled in this box.
Editing boxes	The selected macro commands are combined with the necessary parameter values here (e.g. dwell time after a move).	**Editing buttons**	The current macro command may be deleted, replaced, or edited. A new command can be created.

Macro commands box	The list of all possible macro commands suitable for the current macro.

A macro must be assigned to a specific ***WinPC-NC*** event (e.g. job start, after a tool change, when a special tool is selected). It will be executed when that event occurs during a job.

Some of the macros are only available in specific contexts (e.g. G-code or HPGL NC files). For example, the macro1 macro can be executed when the M90 M-command is found in a G-code NC file. Further information about the specific context macro capabilities is provided in later chapters.

Macros manipulation procedure:

- Select an event by double-clicking in the **Execute macro at …** box. This is the event which the macro is to be associated with. The **Current macro…** box will display “No macro defined” for a new macro or display the commands of an existing macro.

- Edit the selected macro by double-clicking the lines you want to change. The selected command will appear in the editing box. Once the command appears in the editing box, values can be assigned. Click *Replace* or *Insert*, as necessary, to change the selected macro.

- To delete a macro command, move the bar to the corresponding command and click the *Delete* button.

Most of the macro commands are self-explanatory. A few of the more unusual ones are listed here:

Wait time	Wait. Time is in ms	**Cooling ON/OFF**	Cooling is turned on/off
Wait InputHigh num	Waits for HIGH to appear of input ‘num’	**Wait Spindle Stop**	Waits for HIGH to appear on the ‘Spindle Stop’. Only available with certain spindles that are wired appropriately.
Set OutputHigh num	Sets output ‘num’ to HIGH	**OffsetX**	Not available in ***WinPC-NC USB***
Speed spd	Speed setting for future moves in mm/secs.	**Block Z zero point**	Not available in ***WinPC-NC USB***

MoveX xxx	Moves a relative distance	**Spindle Speed**	Sets the spindle speed
MoveAbs xxx	Moves to absolute position xxx (i.e. in machine coordinates)	**WaitStart**	Waits for HIGH to appear on the Start signal (input I255).
Spindle ON/OFF	Spindle is turned on/off		

Inputs and outputs have formal names (e.g. I100, Q115). Do not use an LPT LPT port pin number. The formal inputs and outputs must be associated with specific hardware ports and pins before they can be used. Use the Signal Wizard to create the necessary associations.

Example 1 Release tool

Macro releasing a tool from a 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 molett
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

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

Not all macros are available in all versions of *WinPC-NC*.

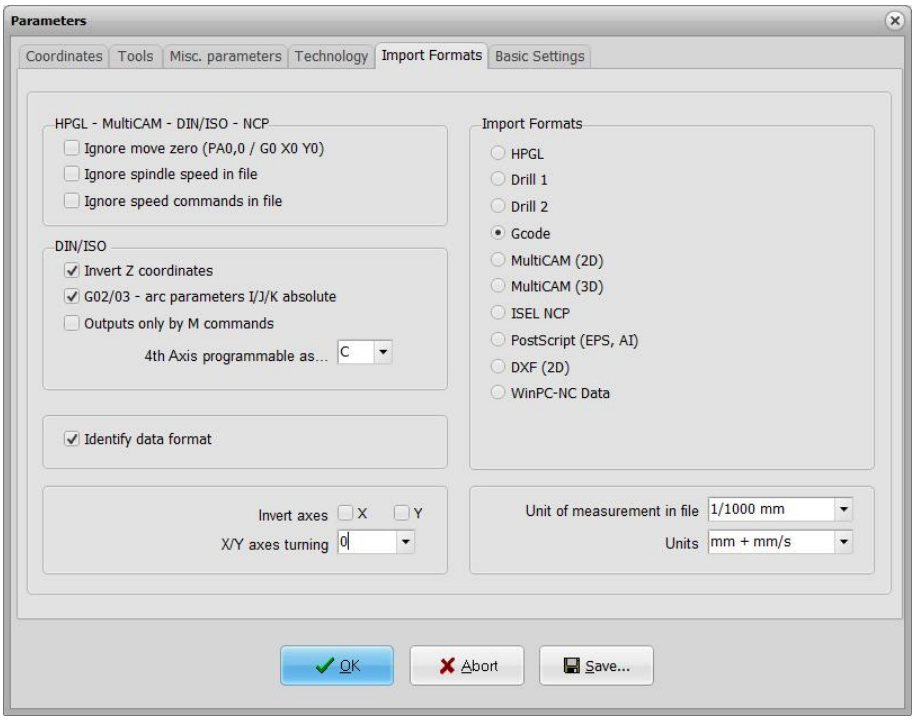
Signals only via macros

By default, ***WinPC-NC*** generates certain output signals as it runs (e.g. 'Job running' while a job is running, spindle control signals, coolant pump on/off).

Clicking this checkbox disables the automatic output signal generation. Output signals must be generated explicitly from within a macro. This makes it possible to control, explicitely, every machine operation.

5.8. Import Formats

WinPC-NC understands many NC data formats. These data can be displayed and used for controlling a machine.



Parameter-Import Formats

The supported data formats are HPGL, MultiCAM, G-code (DIN/ISO), DXF (2D), ISEL NCP, EPS and AI Postscript data, Sieb&Maier1000, Sieb&Maier3000, and Excellon.

HPGL is widely used graphic format from Hewlett Packard. MultiCAM (also referred to as Extended HPGL format) is a widely used, extended version of HPGL. Sieb&Maier1000, Sieb&Maier3000, and Excellon are drilling formats.

See the appendix for further information.

Automatic identification of the format

By default, ***WinPC-NC*** will attempt to determine the type of NC data as it is loaded (automatic recognition). Rarely, this identification process can fail. If this occurs, explicitly select the NC format of the data to be loaded and try loading the data again.

Ignore movement to zero (PA0,0)

In HPGL and MultiCam files a PA0,0 command is often found at the end of the program. Checking this checkbox will suppress interpreting this command.

Ignore feed rate and spindle speeds

HPGL, MultiCAM, and G-code formats may contain commands to set the spindle speed and feed rates. Checking this checkbox will cause any NC speed control information to be ignored; tool data will then be taken from the ***WinPC-NC*** tools parameters.

Invert Z-axis

By default, ***WinPC-NC*** interprets negative Z-axis positions as “up”. Most CAM programs interpret negative Z-axis positions as “down” (plunge). Checking this checkbox will invert the default **WinPC-NC** behavior.

G02/G03 - I/J/K relative

There are several dialects of G-code. In one, the G03/G04 (arc) parameters I/J/K are specified as absolute coordinates, while in the other the I/J/K are specified as distances relative the current position. This parameter enables you to distinguish between the two.

Output signals with M commands

In G-code programs, external control is handled via “M” commands (e.g. spindle, coolant, tool changes). When this checkbox is checked, ***WinPC-NC*** will perform the actions associated with M commands as they encountered. Otherwise the actions will occur at the beginning of the job.

4th Axis programmable as...

The 4th axis is addressed in G-code programs via a letter. Axes parallel to X, Y, and Z are designated, respectively U, V, and W. Rotations around the X-, Y-, and Z- axes are designated, respectively, A, B and C.

Mirror/Invert axes

The X- and Y-axes and their coordinates can be independently inverted. The change is immediately visible in the graphical display.

Rotation of X-/Y- axes

The NC data can be rotated around the zero point for a better stock placement. Rotation is in 90 degree steps.

Unit 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 units for HPGL, DXF, EPS, and AI files are automatically set when the file is loaded.

Units may be millimeters or inches. HPGL files are usually in units of 1/40 mm or 1 mil; drilling data are usually in 1/100mm or 1 mil.

Units in the NC program

Specifies how NC file distances and speeds are understood by ***WinPC-NC***.

There are three choices:

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

6. Setting up a New Machine

Once the ***WinPC-NC*** software has been installed, a new machine must be set-up and tested before it can be used for production work. The set-up procedure generally only has to be done once.

If your machine came bundled with ***WinPC-NC*****,** the set-up procedure can be skipped. Follow the instructions that came with the machine to load the parameters appropriate to your machine.

If your machine uses an external USB module (e.g. ***nc100***, ***ncUSB***):

- Windows must be configured to not perform any power saving actions on its USB ports – disable the sleep mode for USB ports in the device manager.

- Do not connect the external USB module to the PC until after the ***WinPC-NC*** software has been installed and is running. After a short time, ***WinPC-NC*** will notify you that the newly identified hardware is ready for use.

WinPC-NC **is software that controls hardware. The hardware must be properly connected to the PC and configured in *WinPC-NC* for correct machine operation. The user is responsible for making all connections, performing all safety procedures, and operating the machine.**

Care must be taken to correctly connect and test each wire, switch, sensor, or output. Everything should be checked multiple times; having another person review things while you work is a good idea.

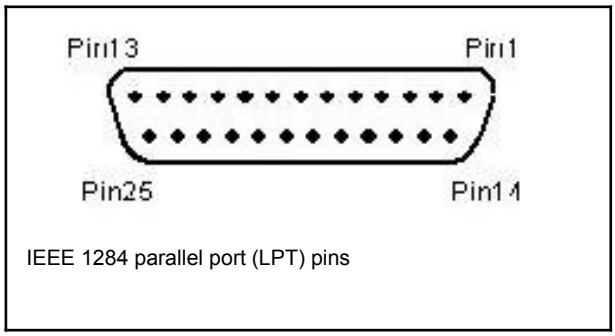
Care must be take to avoid damage due to static, incorrect signal assignments, and incorrectly inserted plugs/cables/wires.

Do not attempt to set-up the machine until you have familiarized yourself with this manual and the set-up procedures, are rested, and have plenty of time to set-up the machine.

IT IS HIGHLY RECOMMENDED THAT THE FIRST SIGNAL CONNECTION IS THE EMERGENCY STOP (E-STOP). THIS WAY, ANY UNEXPECTED ACTIONS CAN BE STOPPED BEFORE DAMAGE OCCURS.

6.1. IEEE 1284 port (LPT) pins

All IEEE 1284 ports adhere to the 5V TTL standard for inputputs and outputs.



IEEE 1284 parallel port (LPT) pins

LPT1 is available on the *nc100* and *ncUSB* modules. The default pin assignments are:

Pin 1	**OUTPUT**	**drilling spindle on/off**
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 14	**OUTPUT**	**coolant pump on/off**
Pin 16	**OUTPUT**	**current reduction**
Pin 17	**OUTPUT**	**job running**
Pin 10	**INPUT**	**reference switch X**
Pin 11	**INPUT**	**reference switch Y**
Pin 12	**INPUT**	**reference switch Z**
Pin 13	**INPUT**	**sensor/surface block**
Pin 15	**INPUT**	**free**
Pin 18-25		**Ground (0V)**

LPT2 is only available on the *ncUSB* module. The default pin assignments are:

Pin 1	**OUTPUT**	**free**
Pin 2-9	**OUTPUT**	**analog output, 256 binary coded steps**
Pin 16	**OUTPUT**	**free**
Pin 17	**OUTPUT**	**free**
Pin 14	**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		**Ground(0V)**

Each IEEE 1284 port (LPT) has pins 10, 11, 12, 13, and 15 available for input and pins 1, 14, 16, and 17 available for output.

The Q218 signal (i.e. Spindle Speed/PWM) may only be connected to pins LPT1:14 or LPT1:17.

The Q219 signal (i.e. Toggle/Charge pump) may only be connected to pins LPT1:16 or LPT1:17.

6.2. 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.

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.

The required value is the number of motor steps ***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 specified motor steps. With spindles it is the spindle rise; with belts or gear racks it is the graduated circle size of the pinion.

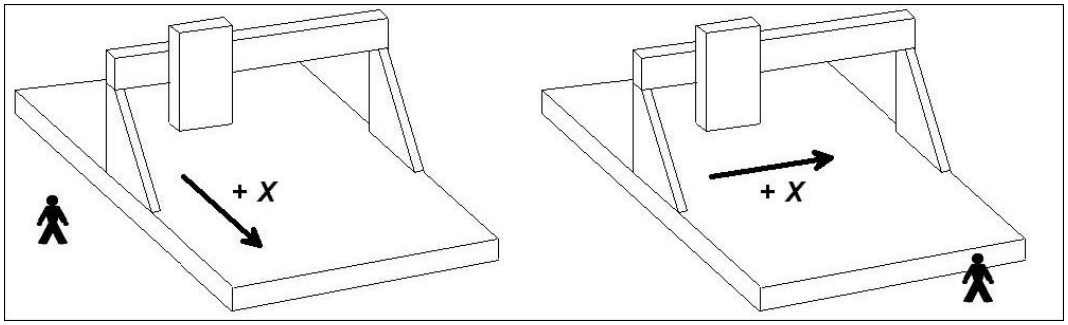
Incorrectly set axis resolutions will cause dimensional inaccuracies and inexact speeds.

6.3. Determination of direction

Select JOG MOVE and study the buttons. Each axis is to be **carefully** tested for forwards and backwards motion. Each motion **must** move exactly as expected.

Start with X-axis.

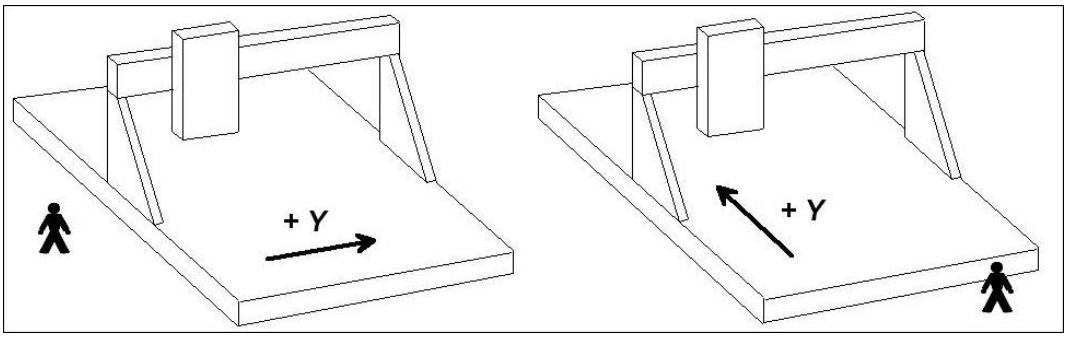
Arrow keys on the right are for motion in the positive direction; on the left for negative motions. Check your machine documentation for where to stand:



Carefully attempt a positive motion first. If successful, attempt a negative motion. If the expected motions do not occur, here is a table of common errors, causes, and solutions:

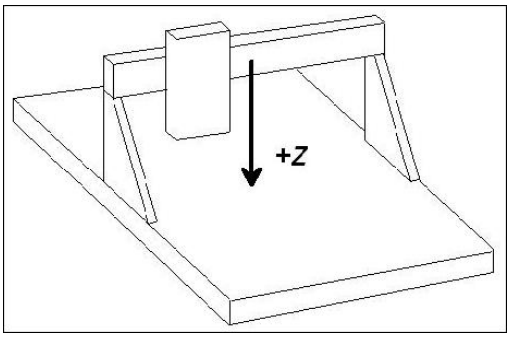
Error	**Possible Cause**	**Solution**
Machine does not move	Cable not connected	Connect the cable
	Cable not connected to the correct LPT port	Connect the cable to correct LPT port
	Incorrect pin assignment	Correct the pin assignment
	Machine off	Check if the machine and/or its motion controller are powered on.
Unexpected movement (wrong axis moves)	Incorrect pin assignment	Correct the pin assignment
Axis moves in opposite direction expected	Direction signal works with inverted logic	Change the parameter INVERT MOVE DIRECTION under machine parameters

Once the X-axis appears to be working as expected, repeat the testing procedure on the Y-axis. Correct any issues until the X- and Y- axes appear to be working as expected.



Now test the Z-axis.

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

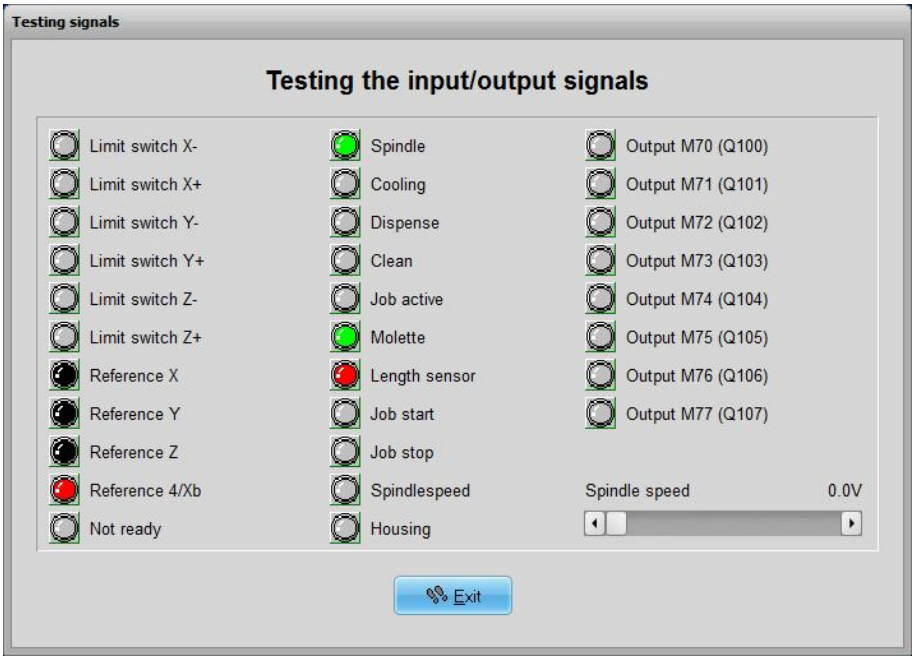


For a direction reversal of one axis it is only necessary to change the parameter INVERT MOVE DIRECTION.

Movement directions are relative to the movement of the tool above a work piece. Movement is opposite that of the arrow keys in order to guarantee the correct movement direction of a tool above the work piece.

6.4. Home and limit switch adjustment

The home and limit switches can easily be checked with the SPECIAL FUNCTION-SIGNAL TEST.



Testing signals

Manually activate each switch (home and limit) and check the signal LED in the *Testing signals* window. The expected LED should be black until it is activated; red when it is activated.

Depending on the type of switch used (NO contact – NC contact) the expected result may be inverted (i.e. red when not activated; black when activated). If this is observed, return to the Signal Wizard, locate the signal and change the pin assignment (e.g. “LPT1 Pin11” to “LPT1 Pin11 Inv”). Rerun this test until all of the switches behave as expected.

6.5. Setting up the Home Switches

Now that the home and limit switches are installed and work as expected, it’s time to let ***WinPC-NC*** know where they are, how fast they can be located (search and free speeds), and in what order the home switches are located.

Please follow the following procedure:

1. From the Parameters menu, select the BASIC SETTINGS tab and the HOMING tab.

2. Specify the home switch location for each axis of your machine. This will at the positive or negative end of your machine. If you’re unsure which is the positive or negative end of your machine, change to JOG MOVE and move your machine in both directions, observing the position counters and noting which direction the axes move as they approach a home switch.

3. Set the appropriate search and free speeds. These should be on the slow side as one does not want to collide with the switches too fast, potentially damaging them.

4. Set the appropriate axis search order. It is usually best to stay with the default (Z-Axis first).

5. Set the machine zero point. It's best to set this at a small distance from the reference position.

6. Start the machine initialization sequence. One axis at a time will move.

6.6. Control of settings

Before attempting to initialize the machine for the first time, double check all of your settings. Press function key F8 or select the corresponding function from the menus, and save your settings.

During the initialization procedure, your machine will move one axis at a time. Initially, each axis will move towards its home switch, until it activates. It will then move away from its home switch at a reduced speed until the home switch ceases to activate. This is the reference point for that axis.

If everything has been done correctly, the initialization procedure of your machine should initialization procedure finish without any problems. If an error occurs or your machine moves in an unexpected way, review your settings, make the necessary adjustments, save your parameters, and try again.

The table below identifies common errors and a suggested solution.

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 direction, but stops on the switch	Switching logic has been adjusted 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

6.7. Additional steps

Once the basic connections and motions have been made and verified, it's time to finish the set-up procedure:

1. Determination the optimal motor ramp and traverse speeds for each axis using SPECIAL FUNCTION-MOTOR TEST.

2. Connect any remaining input and output signals and verify them via the Signal Wizard.

3. Review all of parameters and adjust them to your liking.

6.8. Input and output signal management

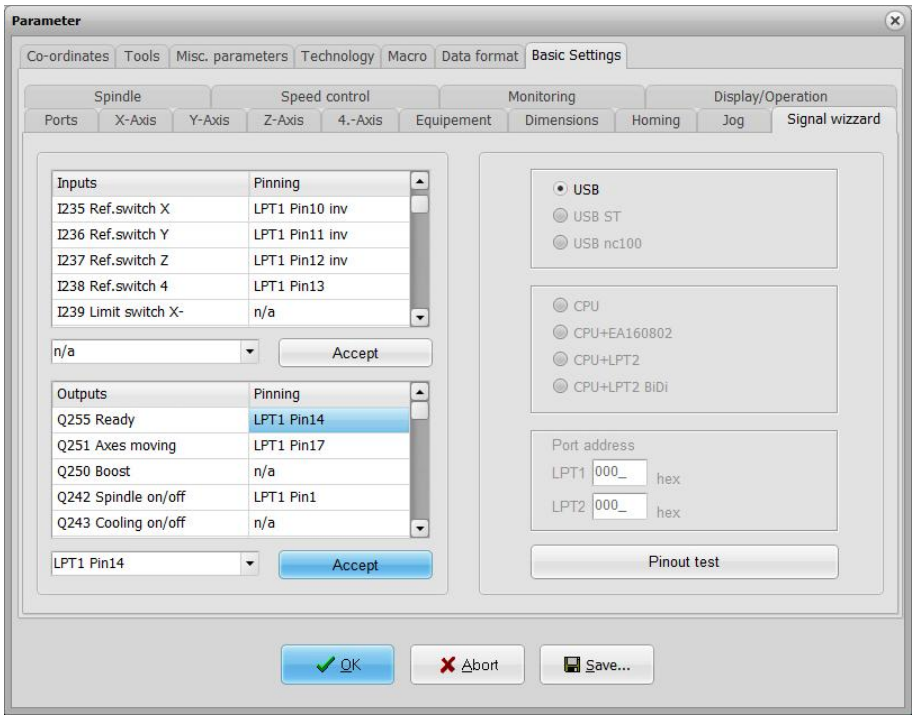
The management of the input and output signals is fundamental to ***WinPC-NC***.

Input signals make it possible to handle things like:

- Home and limit switch monitoring
- Machine initialization and automatic calibration
- Synchonization with external hardware (e.g. Start).
- Safety device monitoring (e.g. doors, hoods, fences).

Output signals make it possible to control things like:

- Spindle power and speed.
- Cooling and lubrication devices.
- Safety device locks (e.g. doors, hoods, fences).
- Dispensing pump power and speed.
- 3D print head power and feed rate.



Signal Wizard

WinPC-NC supports *many* different signals. Only connect and assign those signals that are applicable to your machine and application.

The Signal Wizard makes it is quite simple to assign a signal to any available pin.

Each IEEE 1284 port (LPT) has pins 10, 11, 12, 13, and 15 available for input and pins 1, 14, 16, and 17 available for output.

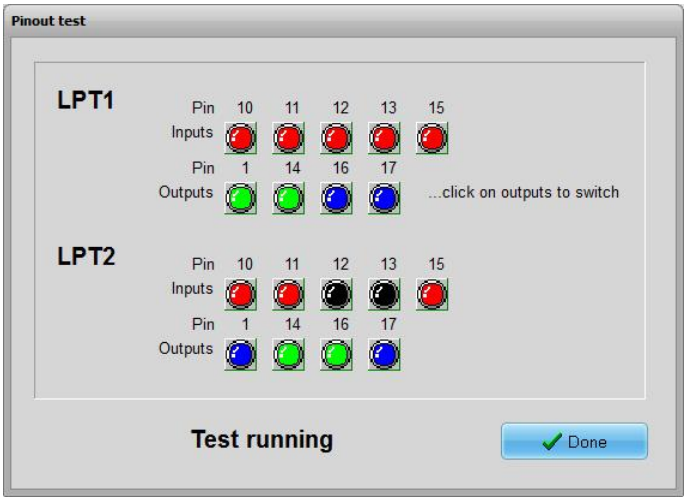
CAUTION: Please take great care when assigning and testing inputs and outputs. Be vigilant, watching for any unexpected motions or actions.

6.9. Assignment of inputs

Use the following steps for assigning the input lines:

1. Connect a required switch, wire, or sensor to an available pin.

2. Click the *Pinout test* button. The Pinout test window will appear.

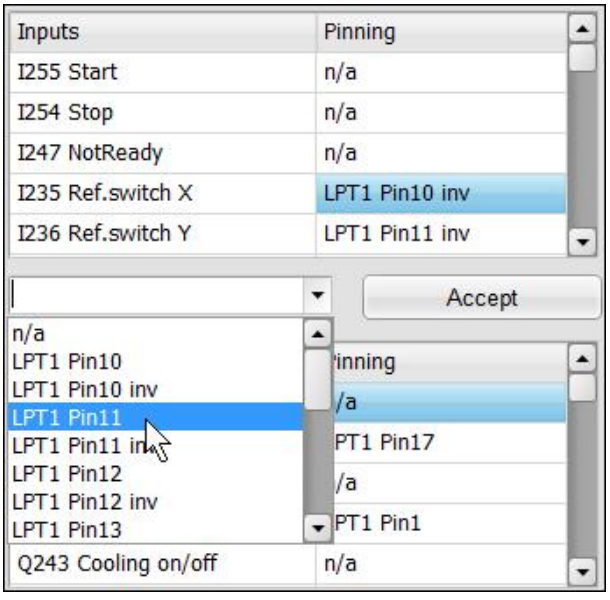


Pinout test

3. Perform an action which activates the switch, wire, or sensor.

   A normally open (NO) contact is indicated when the LED light changes from black to red when it is activated.

   A normally closed (NC) contact is indicated when the LED light changes from red to black when it is activated.



4. Click *Done*. Control now returns to the Signal Wizard.
   Select the required signal from the input list on the left (e.g. I235 reference switch X) and select the pin (e.g. LPT1 Pin11) from the pop up menu next to the *Accept* button.

5. With normally open contacts, select the pin number (e.g. LPT1 PIN11). With normally closed contacts, select the inverted pin number (e.g. LPT1 PIN11 Inv).

6. Click the Accept button.
   The selected signal and pin number will be displayed in the assignment table.

7. Repeat this process for all required inputs.

8. Save the parameter settings.

   It is a good idea to use SPECIAL FUNCTION-SIGNAL TEST and/or SPECIAL FUNCTION-MOTOR TEST to verify that everything is working properly.

***WinPC-NC* inputs only react to a HIGH level. Pins that use inverted logic must be assigned properly such that their state generates a HIGH level.**

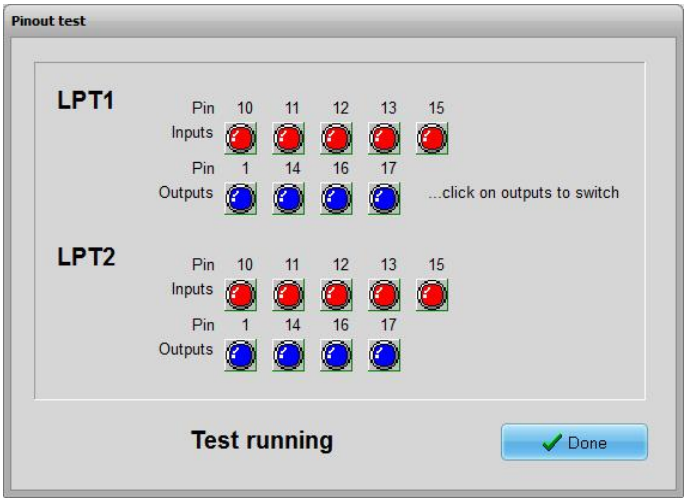
6.10. Assignment of output lines

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

Please carry out the following steps:

1. Connect a required relay, wire, or control to an available pin.

2. Click the *Pinout test* button. The Pinout test window will appear.

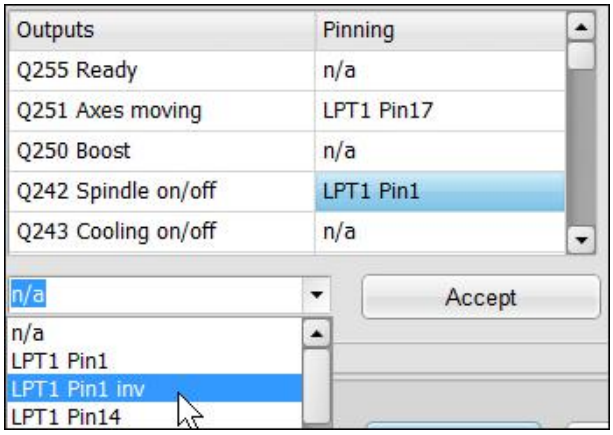


Pin number assignments for output lines

3. Click by mouse pin and observe the level of the signal.

   A blue LED light indicates a LOW level or an output that is switched off. A green LED light indicates a HIGH level or an output that is switched on.

   Does the expected action occur?



4. Click *Done*. Control now returns to the Signal Wizard.

   Select the required signal from the output list on the left (e.g. Q242 Spindle on/off) and select the pin (e.g. LPT1 Pin11) from the pop up menu next to the *Accept* button.

5. Click the *Accept* button.

6. The selected signal and pin number will be displayed in the assignment table.

7. Repeat this process for all of the required outputs.

8. Save the parameter settings.

It is a good idea to use SPECIAL FUNCTION-SIGNAL TEST and/or SPECIAL FUNCTION-MOTOR TEST to verify that everything is working properly.

6.11. Supported Input Signals

WinPC-NC supports the following input signals:

Input signal	Assigned action
I255 Start	Loaded job may begin
I254 Stop	Immediately stops the running job when this signal transitions from LOW to HIGH. After this signal is detected, the machine must be (re-)initialized before further use.
I247 Not ready	Immediately stops the running job when this signal transitions from LOW to HIGH. This is the signal used for the emergency stop (E-Stop). After this signal is detected, the machine must be (re-)initialized before further use.

Input signal	Assigned action
I228 Spindle “at speed”	Spindle has reached its specified speed.
I222 Protection hood	Safety interlock. Job will not begin until this signal is HIGH.
I221 Length sensor/ Surface probe	Surface or tool length sensor contact acknowledge.

I235-238 Home switch	Home switch signal for all 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 ZinPosition-Up	Synchronization for a semi-automatic drilling job. When this transitions from LOW to HIGH level, it is permissible to move to the next position. ZinPositionUp and ZinPositionDown are mutually exclusive.
I233 ZinPosition-Down	Synchronization for a semi-automatic drilling job. When this transitions from LOW to HIGH level, it is permissible to move to the next position. ZinPositionUp and ZinPositionDown are mutually exclusive.
I229-232 Limit switch summary	Only one limit switch signal is available for both ends of an axis. Since these signals are ambiguous, ***WinPC-NC*** cannot detect the direction of a motion and change direction.

I180 JobSingle	Starts the loaded job in step by step mode; stop after each NC command.
I179 JobStart	The job loaded from the main menu has started.
I178 JobStop	Stop running job.
I177 JobResume	Continue interrupted job.
I100-109 free I160-169 free	Signals available for custom use; macro programming accessible

All other input signals are reserved.

6.12. Supported Output Signals

***WinPC-NC* supports the following output signals:**

Output signal	Assigned actions
Q255 Ready	The controlling software is ready (HIGH). If an error occurs, the output transitions to LOW.
Q251 Axes moving	Indicates movement (HIGH) or stopped (LOW). This signal can be used to adjust current flow.
Q242 Spindle	Spindle on/off.
Q243 Cooling	Cooling/lubricant on/off.
Q244 Dispensing/Laser	Laser/dispensing on/off.
Q245 Purging with compressed air	An upwards Z-axis motion (HIGH).
Q246 JobRunning	Job is running (HIGH).

Output signal	Assigned actions
Q247 JobEnd	A 100 ms long pulse *(*HIGH*)* indicating that the job has ended.
Q248 Collet chuck/molette	Controls the automatic chuck in conjunction with an Automatic Tool Changer (ATC)
Q249 Spindle CCW/CW	Spindle rotation direction control (e.g. M04/M03 in G-code NC programs)
Q219 Toggle/ChargePump	Continuous 12.5kHz square wave clock signal.
Q218 Spindle speed/PWM	Pulse Width Modulation (PWM) signal used for spindle speed control.
Q100-115 M code	Maps, respectively, to the M70-M86 codes in G-code programs.
Q220-230 free	Signals available for custom use; macro programming accessible

All other input signals are reserved.

7. Additional Information

7.1. Interpreters

WinPC-NC has interpreters (import filters) 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

The HPGL interpreter recognizes the most of the commands in the HPGL plotter language.

Successful tests have been carried out with various CAD and drawing programs:

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

Furthermore, the interpreter can also work with HPGL data generated by the Window HP-Plotter driver.

Programs such as...

- CorelDRAW®

...and other Windows drawing programs use the driver for creating plotter data.

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 extension are also supported:

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

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

- Coordinates are always pairs of numbers - 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

The MultiCAM format is an expansion of the HPGL standard that adds 3D capabilities. All HPGL commands are supported however extension for handling 3D may be enabled.

A differentiation is made between 2D and 3D modes. There must always be 3 coordinates in 3D mode.

ZZ0/1;	Activates 2D or 3D mode
ZU*z*;	Tool lift height of the Z-axis over the zero point (overwrites the safety clearance parameter)
ZD*z*;	Plunge depth (overwrites the depth values in the parameters)
SF*v*;	Feed rate
ZO*d*;	Spindle speed

The syntactical rules of HPGL language apply analogously to MultiCAM.

Drilling data - Interpreter

The drilling data interpreter recognizes data that follow the Excellon, Sieb&Maier1000, and Sieb&Maier3000 standards.

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 tool numbers.

The syntax is as follows:

% or %%
[X*number*][Y*number*][T*x*]
.....
M30

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

In format 2, leading zeros are discarded and the coordinate values therefore comprise different numbers of digits. The drilling formats prescribe the following syntactical rules, amongst others, which must be observed when creating or revising the data

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

Sample of drilling data in format 2	Short drilling program	Comment
	%	Start of program
	T1	Select tool 1
	X1000Y1000	Drill hole at 1000/1000
	X1200Y2340	Drill hole at 1200/2340
	X2700Y2950T2	Tool 2 hole at 2700/2950
	Y1000	Drill hole at 2700/1000
	M30	End of program

Sample of drilling data in format 1	Short drilling program	Comment
	%	Start of program
	T1	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 (DIN/ISO) 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 move at machine rate (rapid)
	G01	Linear move at specified rate (machining)
	G02	Clockwise circular movement
	G03	Counterclockwise circular movement
	G04	Dwell time, milliseconds, via F command
	G17	Plane XY
	G18	Plane XZ
	G19	Plane YZ
	G28	Shift of zero point
	G54..59	Zero offset
	G70	Units are Imperial (inches)
	G71	Units are Metric (millimeters)
	G81	Drilling and withdrawal in slow speed Z-depth R-tool lift P-dwell time
	G82	Drilling and withdrawal in rapid speed Z-depth R-tool lift P-dwell time
	G90	Poisitions are absolute

	G91	Positions are relative
	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 macros 1-10
	M106	set output HIGH for 3D printer head
	M107	set output LOW for 3D printer head
Misc commands	N	Line number
	S	Spindle speed in revolutions per minute
	F	Pin number (with M16 command)
	F	Dwell time (with G04 command)
	F	Feed rate (mm/s, mm/min, in/min)
	I,J,K	Circle parameters
	T	Select tool
	X,Y,Z	Coordinates
	U,V,W	Coordinates for 4th axis, if activated
	A,B,C	Coordinates for 4th axis, if activated
	R	Radius of arcs
	R	tool lift for G81/82 drilling
	P	dwell time at G81/82 drilling
	L	Call subroutines (1..20)

Example Square with rounded corners	%prog2	Start of programm
	N001 G90	Absolute coordinates
	N002 G71 T1 M6	Dimensions in mm, tool 1
	N003 G00 X110 Y100 Z10	Speed to 1st position
	N004 G01 Z11	Plunge movement in 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 G-code program file must be a text file, not a word processing file.
- The first line should be a percent sign (%) followed by the name of the program.
- At least one tool has to be selected e.g. with T1 M6 in the program head (M6 is absolutely necessary)
- The feed rate units are set in PARAMETERS-UNITS
- The command number can be defined using N commands
- The arc commands (G02/G04) support both the I, J, K (position) and R (radius) variants. They cannot be mixed. Positive radii produce an arc less than 180° and negative radii an arc greater than 180°.
- Up to 20 subroutines may defined after M30 (the end of program). A subroutine begins with G98 Lx and ends with G98 L0. A subroutine call made by invoking Lx.
- G54…G59 define job zero points. A definition of the zero point combines the position data on the line with the current zero point.
- Activate one of these zero points by usings its number (e.g. G54) on a line by itself (no other parameters).

Example drilling cycle as subroutine

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

ISEL NCP Interpreter

The ISEL NCP format is largely associated with companies in the ISEL group (e.g. ISEL, MES). ***WinPC-NC*** is able to interpret NCP programs - excepting the track data and the track sections.

The commands supported by this interpreter are:

FASTVEL	**VEL**
MOVEABS	**FASTABS**
MOVEREL	**FASTREL**
PLANE	**CCWABS**
CWABS	**CCWREL**
CWREL	**WAIT**
GETTOOL	**HALT**
SPINDLE	**COOLANT**
DRILLDEF	**DRILL**

Postscript Interpreter

WinPC-NC is able to interpret vector data contained in Postscript files generated by many graphics programs (e.g. Adobe Illustrator®, Corel Draw®). All other data will be discarded as they have no affect on NC processing.

Always create PS files without a header.

A realtime display of the program or postscript files is not available.

Please restrict EPS/AI files to version 3 or 8.

DXF import

WinPC-NC can import files in 2D DXF format. DXF is a widely used data exchange format. A DXF file for ***WinPC-NC*** must not contain any polylines, splines, and other complex data elements. Unchain all existing groups to single elements.

Please restrict DXF files to version R12 or version R14.

7.2. Error messages

Error messages are displayed in a small area just below the *Display Area*. Messages in a gray box are status messages, no fault is indicated.

The following error messages may occur:

- **Limit switch reached**
  The machine moved until one of the limit switches tripped. The calibration of the machine is now in question. When this occurs, the jog away from the offending limit switch and reinitialize the machine.

- **HPGL Syntax error**
  There is an invalid HPGL command in the plotter file.

- **Connection canceled**
  The connection to the DLL communications module has been interrupted. This happens when other Windows programs or drivers interfere with real-time tasks.

  If this message appears, ensure that all Windows power saving features have been deactivated, including those used for USB ports.

- **Incorrect parameter**
  A WPI or WPW file has been corrupted. One or more values are unacceptable.

- **Stop signal found**
  The stop signal has been detected. Carefully determine what has occurred and rectify the problem.

- **Machine has not been initialized**
  WinPC-NC and all related hardware must be initialized before any machine operations can occur. Failure to do so can result in machine, spindle, or tool damage.

- **Error xx from controller**
  Communication between ***WinPC-NC*** and one of its DLL modules has failed. Please report this error to the program authors and providing as many details as possible.

- **Data cannot be displayed**
  Graphical display of the current NC data is not possible. The data may be corrupted, not be in a format that ***WinPC-NC*** understands, or in a format that is currently disabled.

- **Tool not yet defined**
  The NC data is referring to a tool number that has no data specified for it. The appropriate parameters must be defined prior to use.

- **Required tool not defined**
  The NC data is referring to a tool number that has no data specified for it. A plunge depth and speed must be defined for every tool; HPGL files require a feed speed to be defined.

- **No tool file found**
  No tool file can be found. ***WinPC-NC*** will continue using its default values.

- **Machine is positioned on a limit switch**
  An attempt was made to initialize the machine when the machine is already positioned at one or more of the limit switches. Jog the machine to a position that is well away from all of the limit and home switches and then initialize the machine.

- **Machine positioned on a home switch**
  An attempt was made to initialize the machine when the machine is already positioned at one or more of the home switches. Jog the machine to a position that is well away from all of the home and limit switches and then initialize the machine.

- **Command rejected by controller**
  An internal error occurred when ***WinPC-NC*** was communication with a DLL module. Please report this error to the program authors and provide as many details as possible.

- **Machine not ready**
  The machine has not been switched on or a fault is present. Carefully check the machine and its interfaces and resolve the issue.

- **USB communication failed / Timeout**
  WinPC-NC was unable to communicate with the external USB module. Check the following:
    - Check all the cabling. Only the cable provided with the USB module should be used. No USB hubs or switches should be interposed between the PC and USB module.
    - Ensure that all Windows power saving features have been deactivated, including those used for USB ports

- **Protection hood not closed**
  A protective hood or a similar protective device has not been closed; no process can be started until the hood is closed.

- **File with invalid format**
  The selected NC file contains data that does not match the selected NC data format. Change the NC data selection to match that of the file.

- **Help file not found**
  The ***WinPC-NC*** help file cannot be located. It is called WINPCNC.HLP and must be located in the same folder as the program.

- **Work piece exceeds the size of the working area**
  One or more parts of the work piece extends beyond the boundaries of your machine.

  See if it is possible to relocate the work piece to avoid extending beyond one or more of the boundaries. Pay particular attention to the Z-axis - the work piece, tool, and spindle must fit with the Z-axis limitations of your machine.

- **Next position outside working area**
  While the current position is within the boundaries of your machine, the next NC command is requesting a move that is outside the boundaries. See if it is possible to relocate the work piece to avoid extending beyond one or more of the boundaries.

- **Unknown command**
  The NC file contains an unknown command. Processing ceases. The job can be cancelled or continued from command immediately following the unknown command.

- **Cannot create file**
  WinPC-NC tried, unsuccessfully, to create a file. The selected location is on a write protected drive, the drive is full, or you lack sufficient access rights. Correct the problem or switch to another location.

- **No park position defined**
  Moving to the park position is only possible if it was previously defined. Define a park position.

- **Tool moves in too far**
  The Z-axis move would cause the machine to exceed the defined maximum depth - or a Z-axis depth alarm has been raised.

- **G-code or DIN/ISO syntax error**
  There is a syntax error in the G-code file being processed. Please edit the file, repair this issue (which may occur more than once), and restart the working process.

- **Out of memory**
  WinPC-NC cannot process any further data as it has exhausted all of the memory available to it. Please consider adding more RAM or extending Windows virtual memory. ***WinPC-NC*** **must be restarted**.

- **Incorrect realtime-DLL**
  There are communcation problems between the actual realtime module and ***WinPC-NC*** which are probably caused by another license.

- **DIN-syntax error / DIN-circular arc error-**
  There is an invalid command at the indicated line of the DIN/ISO file; The circular arc specification does not match the indicated center point.

- **Error in speed**
  The defined speed does not match the corresponding machine parameters.

- **Keypad error**
  A specified key pad cannot be located. Check the cabling.

- **WinPC-NC does not run with this DLL**
  The files WINPCNC.EXE and WCNCCON.DLL are not compatible with each other or are not issued under the same license. Please re-install and activate the program again.

- **Profile- file incorrect / incompatible**
  The selected profile file is incorrect, corrupted, or incompatible with the license for the version of ***WinPC-NC*** currently running.

- **Failure in loading profile**
  An error occurred while loading the selected profile. Please check all parameters and settings prior to further use of the system.

- **G00/01 after G54 only one axis**
  The zero point setting in G-code or DIN/ISO programs using G54-G59 commands requires a command to move and continuation has to be effected by all axes.

- **3D Space Mouse not found / not installed**
  A 3D Space Mouse is configured but it could not be found. Please check drivers and cables.