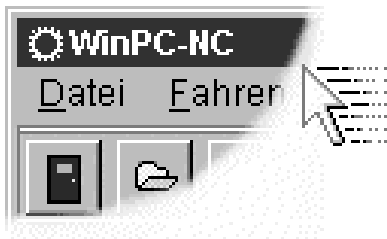**Milling, plotting, engraving
and much more besides
with...**

WinPC-NC

Starter

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

Version 2.50

April 2015

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

Please note that support assistance and reduced updates are only available for registered customers. In order to be registered please send us a mail indicating version number and serial number (either noted on the CD or visible in the program window) as well as your complete address.

Please register your licence !!!!

2.500

Table of contents

The structure of this manual ...

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

Chapter 1:	Brief explanation about *WinPC-NC*, the possibilities for using it and the hardware requirements.
Chapter 2:	Initial start-up procedure, description of how to install the program and how to adjust the initial specific parameters.
Chapter 3:	More detailed descriptions of how to operate the program and the individual functions of *WinPC-NC*.
Chapter 4 :	2D-CAM functions for sorting and calculation a tool diameter compensation
Chapter 5:	Explanation of all parameters and the setting options.
Chapter 6:	Further technical information about the program, about the implemented NC format interpreters, error messages and special versions.

Definitions

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

Job file or **NC file**	A file with NC data which is read and processed by *WinPC-NC*. Depending on the application, the file may contain milling, plotting, drilling data or other types of data.
Job process	The process of reading and processing a job file and the resultant actuation of the machine.
Command	An individual instruction in the job file which gives rise to actions by the machine or in *WinPC-NC*.
Button	Mouse clickable field to activate a certain function

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

Descriptions of the machine and the movement directions of the individual axes are made with reference to the following mechanical layout of a flat-bed machine. The assignment of axes depend on your personal position and your viewport to the machine.
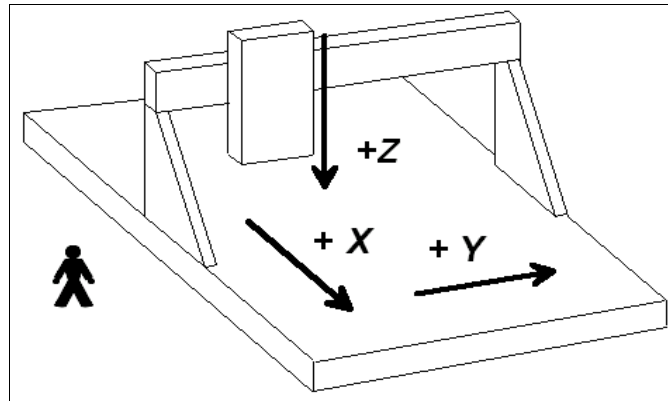
Schematic layout of a flat-bed machine with your position aside
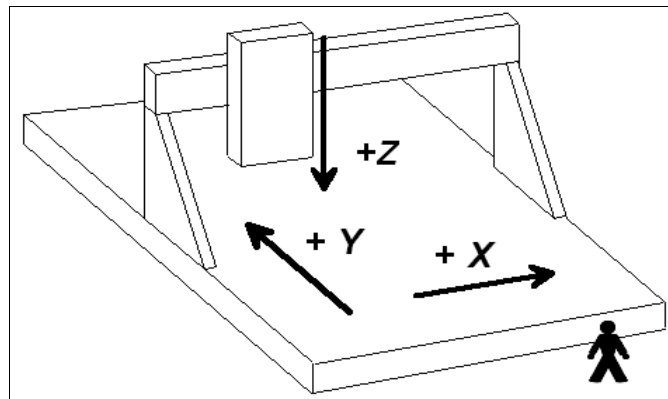
Schematic layout of a flat-bed machine with your position in front of

Use of typography

Keyboard entries   Normal script enclosed in a box, e.g. (ENTER)

Cursor keys   Normal script enclosed in a box with english defintions, e.g. (UP), (LEFT), (PGDN)

Menu functions   Capital letters with menu path, e.g. *FILE-DISPLAY*

Messages   Italic script, e.g. *´Perform reference movement? Y/N´*

Function names   Capital letters, e.g. *SIGNAL TEST*

Different versions of *WinPC-NC*

The control program *WinPC-NC* is available in 5 different versions.

WinPC-NC Starter is only for special machine with USB Hardware inside. The programm offers simple functions for engraving and 2D milling and is optimal for beginners. For this programm we offer an low-priced upgrade to *WinPC-NC USB* with a extended scope of functions and full 3D capabilities.

Our lowcost program *WinPC-NC Light* offers all necessary functions for beginners, e.g. engraving, milling, drilling and PCB drilling or manufacturing modelling parts. Controlling the machine is done via an internal LPT printer port.

WinPC-NC Economy is equipped with additional functions and offers with up to four axes a lot of special functions and provides support of extraordinary mechanical components. This program disposes of much more format interpreters as the version for beginners. Controlling also over internal LPT printer port.

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

The functions of *WinPC-NC USB* are equal to those of the Economy version. However, with *WinPC-NC USB* the machine is controlled by a small enclosed USB module and not by an integrated LPT port. The module casing is equipped with 2 connectors which are compatible to the LPT port. This USB hardware can be included into the machine as well.

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

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

If you purchased *WinPC-NC Starter* in combination with a machine you can easily upgrade to a more featured version of *WinPC-NC USB* without modifying your mechanical hardware.

All other versions of *WinPC-NC* cannot be used with your machine and the included USB hardware.

1. *What can WinPC-NC do ?*

Universal program

WinPC-NC is a software program which takes any standard personal computer and turns it into a universal NC control system for up to 3 axes.

WinPC-NC Starter requires a modern PC with at least 2 Ghz clock-pulse rate and 32 or 64 bit operating system Win2000, Windows XP, Windows Vista, Windows 7 or Windows 8 and 8.1

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

- Drilling
- Plotting
- Dispensing
- Milling
- Engraving plates

Clearly structured operator interface

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

Runs on any modern PC

Running **WinPC-NC Starter** requires a modern standard PC with hard disk, USB ports, any kind of graphics card and a 32 or 64-bit Windows operating system. Notebooks and netbooks with corresponding equipment can also be used.

Special features

Special features of **WinPC-NC Starter:**

- almost unlimited working range

- jog mode in exact steps using the cursor keys or the mouse

- graphical display with scale, zoom, shift functions

- runs from Win2000 to Windows8.1 under 32 bit and 64 bit systems without restrictions (date 4/15)

- recognizes HPGL, PLT, EPS/AI and DXF (2D) file formats
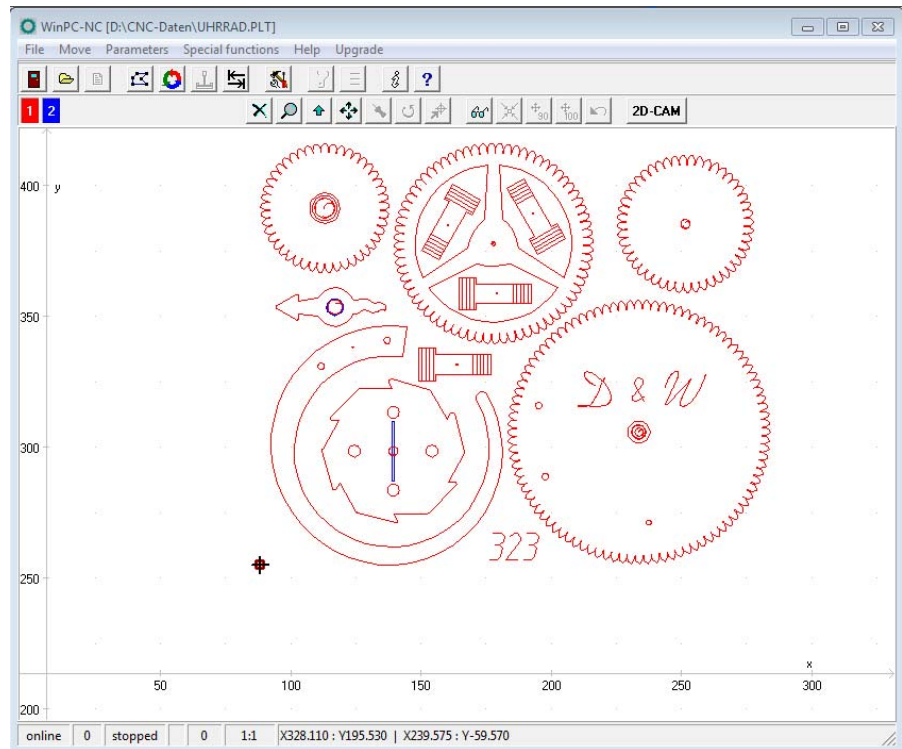
WinPC-NC main screen

- extensive tool management function, tools can be selected individually (max two tools)

- speeds, axes resolutions, acceleration settings are predefined to your machine and there is no need to modify them

- multilingual, other languages can easily be retrofitted, 17 languages selectable from menu (date 3/13)

- NC files of nearly unlimited size can be worked out

- CAM functions for cleaning up drawings, sorting by tool and position and calculating tool diameter compensations

- homing switches predefined, easy reproduction of saved positions after power on

- easy and valuable upgrade to the full featured version of **WinPC-NC USB**, which can be used on your machine without any modification

2. First steps

2.1. Requirements to hardware

WinPC-NC Starter is able to perform a direct control of a connected CNC machine. The required USB driver and the software are to be installed on the computer.

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

- modern CPU with at least 2 GHz clocking frequency

- 32 or 64 bit Windows operating system like Windows 2000, Windows XP, Windows Vista, Windows 7 or Windows 8 / 8.1

- a direct USB 2.0 port, no passisve USB hub if possible

- standard graphical card, keyboard, mouse, hard disk and other common PC implements

- it is recommended to deactivate any kind of power savings, e. g. screen saver, hard disk shutdown and reduction of clock-pulse rate

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

2.2. Installation

User-friendly installation

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

The installation wizard then guides you through the entire procedure.

WinPC-NC Starter is delivered together with a certain CNC machine and cable. Please connect it to the computer after the software has been completely installed and thus the driver is loaded.

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

List of files   **The following files are installed to the target directory :**

WINPCNC.EXE	Control program
WINPCNC.WPI or WTI	Parameter file
WINPCNC.WPW or WTW	Tool file
WINPCNC.WPO	Settings for CAM functions
WINPCNC.PDF	this manual in PDF format
WINPCNC.HLP	Help texts
WINPCNC.LNG	Messages and texts, multilingual
WCNCCON.DLL	*machine* driver
README	Latest changes to the manual
*.DLL	Some files needed in the system folders
*.PLT *.HPG *.EPS *.DXF	Sample NC files
CDMSETUP.EXE	Driver for USB communication

2.3. Launching the program

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

After a short time the window of the communication module appears in the top left area displaying the serial number, the current version and the positions of axes.

The display of the serial number of the module and the software confirms a functioning communication with the USB module, e. g. ncUSB – 50001. If the module has not been plugged or recognized, you will receive a fault message and *WinPC-NC* runs solely in the demonstration mode.

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

• Title bar

• Menu bar

• Bar with speed buttons

• Toolbar and display buttons

• Display area

• Status bar
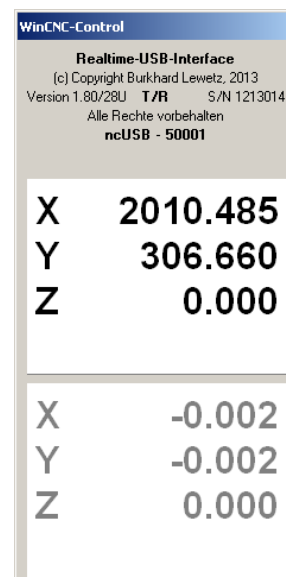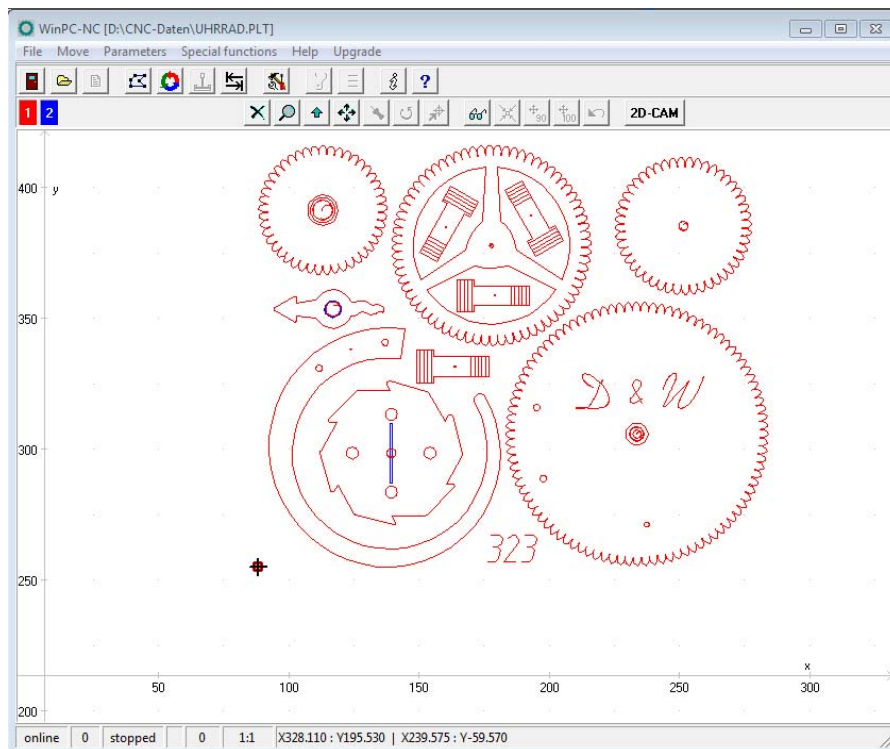
WinPC-NC screen layout

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

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

The function of the individual buttons is as follows :
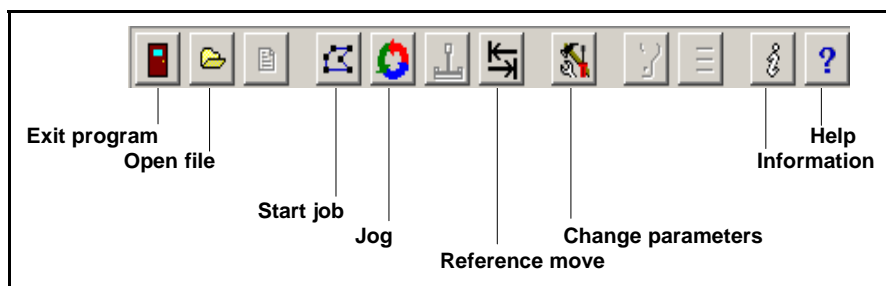
Speed button bar in *WinPC-NC*

The status bar at the bottom displays information about the operating status of the software and the machine, or else a help text about the functions which are currently active. Buttons for operating the display and the tools are located in the second button bar.

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

2.4. First setup and test moves

Executing an intial test run with the motors there is nothing else to do than connecting the machine with the PC where ***WinPC-NC Starter*** is installed.

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

2.5. *Exiting WinPC-NC*

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

3. Operating *WinPC-NC*

3.1. Graphical display of the NC file

Graphical preview of the NC files

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

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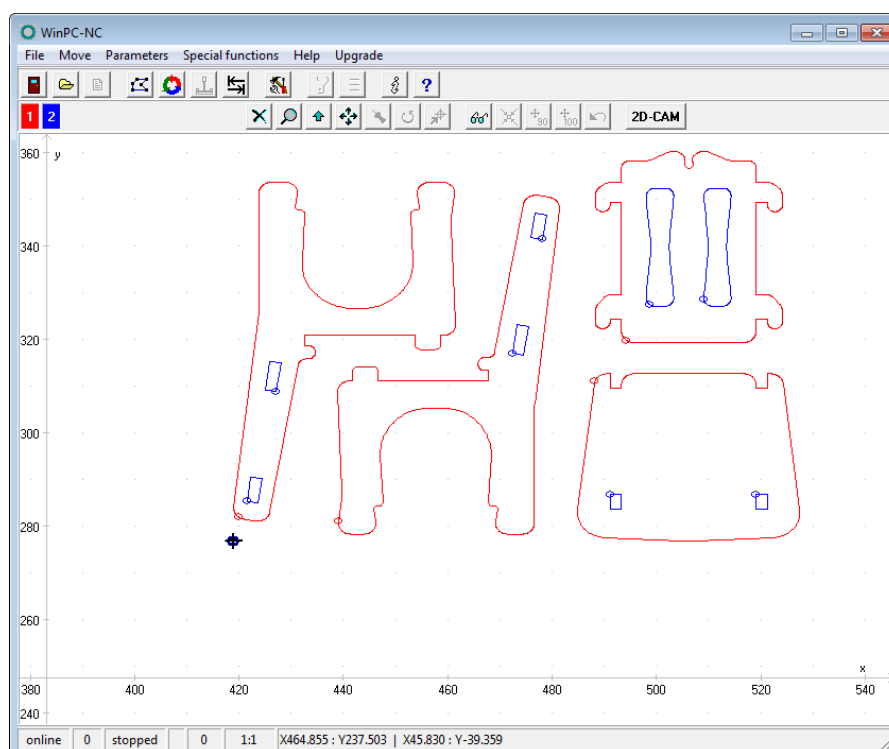
Graphical display of the NC file

The following actions are possible on the graphical display :

Zoom

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

Zoom in the graphical display of the NC file

Move

The move function makes it possible to use the mouse to move a zoomed image and enables you to view another part of the NC file in greater detail. After the function has been activated, the cursor appears as a hand. With it, you can click and drag any point to move the image to where you want.
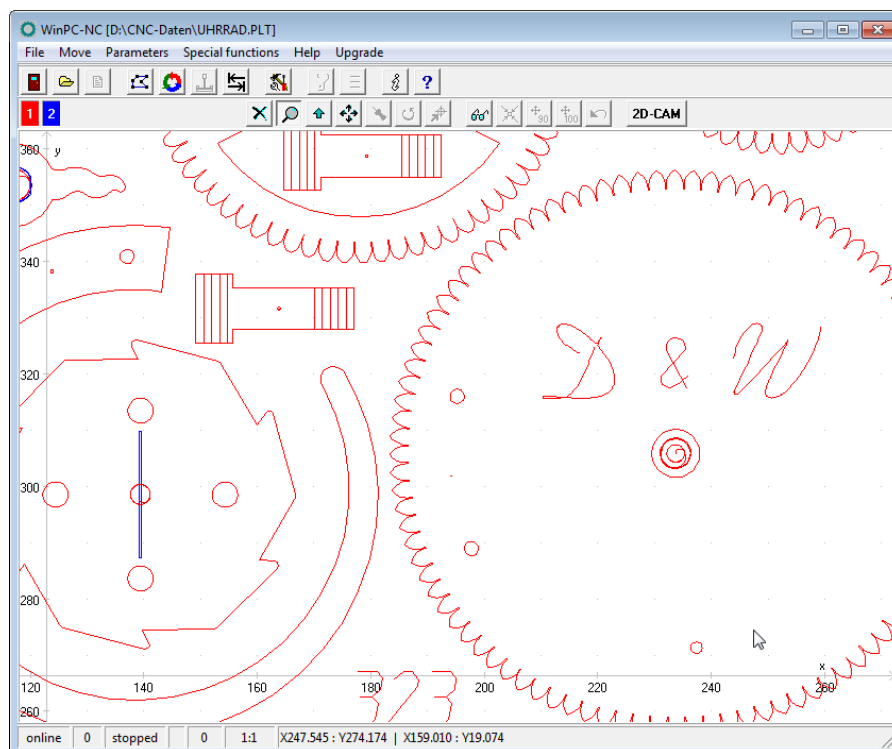
The movement track is displayed as a line while you are dragging. Once you release the mouse button, the graphic appears in the new position to where you have moved it.

Display original size

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

Define current machine position

The current machine position is represented by a small red dot in the graphic.Using the function described here, you can position the NC file in the way that the current machine position corresponds exactly to a point or a certain position in the file. The file zero point is simply recalculated internally.
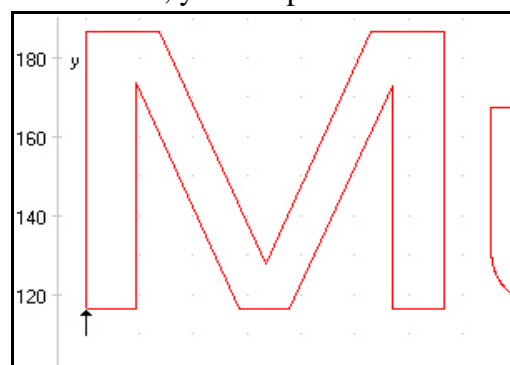
Using this method it is quite simple to position the workingpiece and the NC file with great accuracy.

Cancel function

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

Move to position

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

Selecting the display area

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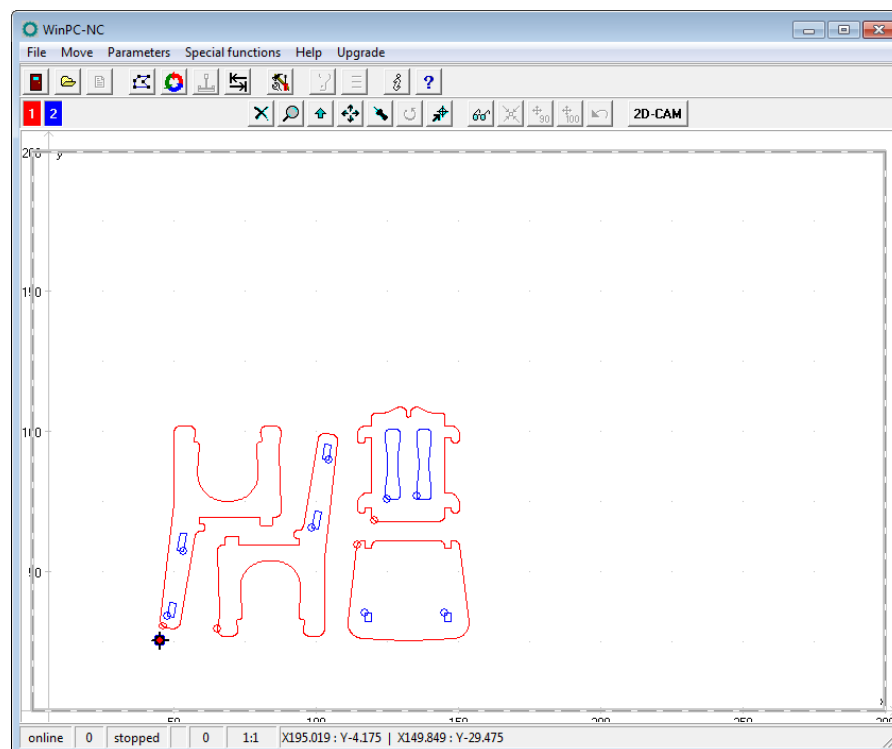
Display with machine table and working area

Set tools active/inactive

The two tools which are used in an NC file are displayed along the lefthand edge in the tool frame. They are shown in red or blue. A tool can be deactivated or activated simply by clicking the tool box. Once

tools have been switched, they immediately appear in the graphical display.
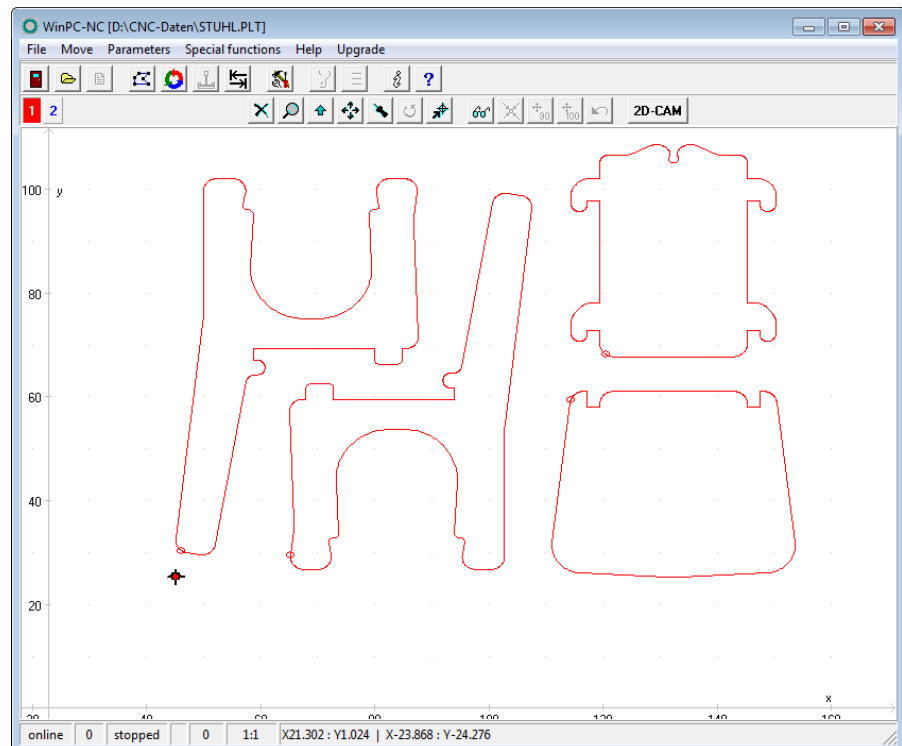
Graphical display with inactive tools
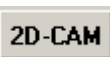
CAM functions

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

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

The main screen of WinPC-NC can easily be moved and placed next to each other. The same applies to the Window of the realtime module with the position of axis or the optionally displayed stopwatch. These positions are retained by WinPC-NC over more than one session.

3.2. Drop-down menus and function keys

Modern interface

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

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

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

Additional function keys

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

The most important function keys are :

(F1)	Activate the help system
(F2)	Load new NC file
(F3)	Start job process
(F4)	Move to XY origin
(F5)	Jog
(F8)	Start reference move
(F9)	Move to parking position
(F10)	Open pull down menu

3.3. The individual menus

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

Not all menu items are active all the time. Functions are sometimes unavailable, depending on the program status. For example, it is not possible to use the joystick if it has not been defined in the parameters.

3.3.1. FILE menu

The FILE menu includes a function used for selecting files to process and analyse them. In addition, it is also possible to exit **WinPC-NC** here.

```
Open                    F2
Exit
```

FILE-OPEN

File selection by menu

The FILE-OPEN menu item calls up an interactive function for selecting a NC file.
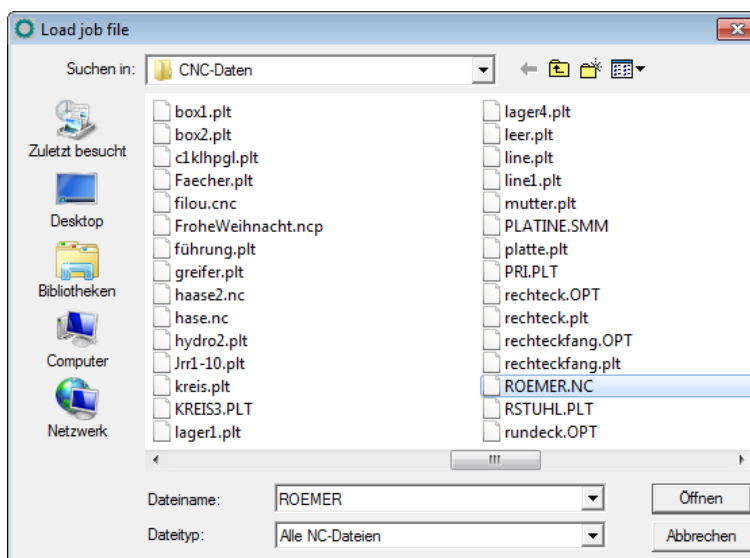
File selection dialog box

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

NC file loading with previous parameters

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

Graphical preview of an NC file

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

FILE-EXIT

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

3.3.2.   MOVE menu

Functions for moving the machine

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

Start	**F3**
Zero point XY	**F4**
Park position	**F9**
Jog	**F5**
Reference move	**F8**

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

MOVE-START

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

WinPC-NC controls the X and Y motors during move commands. The Z motor moves up and down during tool movement commands. All three axes can move at the same time in 3D files.

Progress display

During machining, a window displays the progress as a percentage. In additon the realtime file display is available where the current performed command line is marked by a cursor bar.
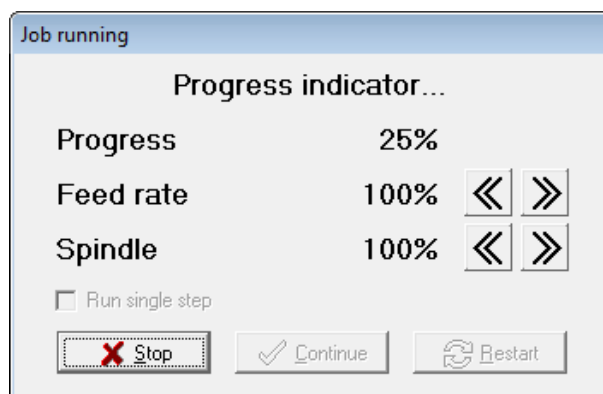
Progress displayed

Speed override

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

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

MOVE-ZERO POINT XY and PARK

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

MOVE-JOG

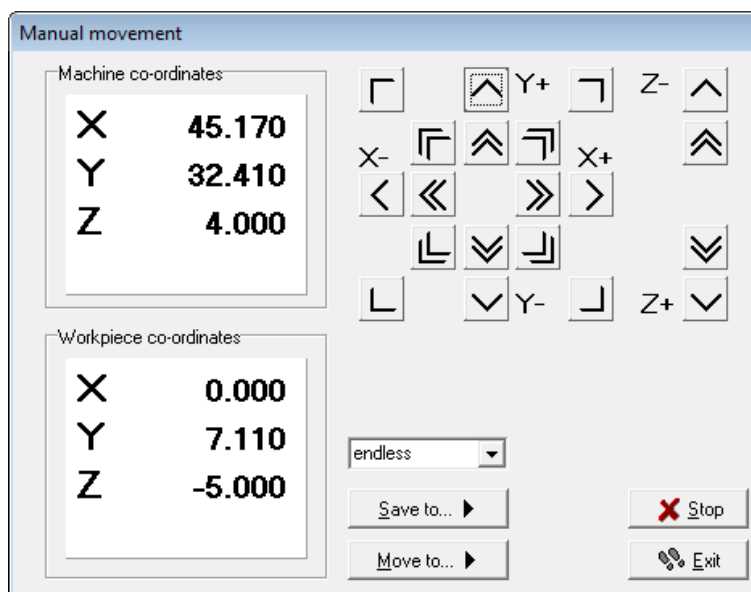The *JOG* menu item calls up the manual setup function for the machine. This function can also be accessed by pressing the (F5) function key or with the jog button.

Jog function

Exact movement with the keyboard or mouse

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

Via PC keyboard you can execute slow or fast moves. Where the latter is the case, please keep pressed the button (Ctrl).The right-hand arrow keys (1) to (9) are responsible for individual or diagonal move of the axis X and Y, the keys (+) and (-) move the Z axes. The speeds are predefined for your machine.

Display of the step counter

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

WinPC-NC distinguishes between two systems of coordinates. Firstly the machine coordinates with their origin on the reference switches and which are referred to as the reference point. Then there are the workpiece coordinates with the workpiece zero point, which is usually located in the bottom left-hand corner of the data area.
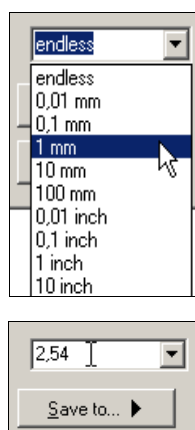
WinPC-NC can move the axes endlessly, i.e. movement continues for as long as a key remains pressed. The axis brakes when the key is released and comes to a stop without any step losses.

The second possibility involves moving discrete distances. Distances between 0.01mm and 100mm are possible, as well as inch dimensions. The curre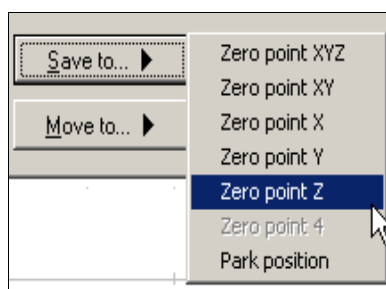ntly set distance is displayed in the window. *WinPC-NC* moves this distance in any required direction and with both possible speeds, depending on which button or key is pressed.

In addition to the predefined distances, it is possible to enter any distance in the menu line.

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

Moving to saved help points is just as staightforward. All that is required is to click the *Move to* button and   select the required help point. Then the machine moves to it.
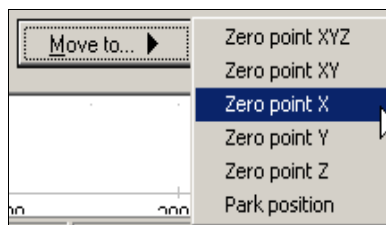
Movement in progress can be interrupted at any time by clicking the *Stop* button.

The *JOG* function can be exited by clicking the the *Exit* button.

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

Space Mouse Wireless of 3DConnexion

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

MOVE-REFERENCE

The function *REFERENCE MOVE* also called homing runs all axes to the reference switches in a defined sequence.

Reference point of the machine

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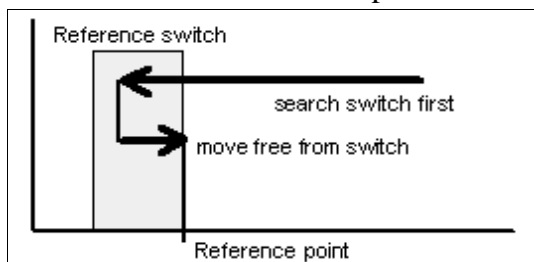
Sequence of reference movement to the reference switch

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

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

WinPC-NC Starter **cannot identify a pressed reference switch and assign it to a certain axis. If the machine touches a switch when starting the reference function, a message is show and you first must move the machine away from all switches by manual jogging.**

3.3.3. PARAMETERS menu

The PARAMETERS menu contains all the setting options for
WinPC-NC.

```
Parameter
Save
```

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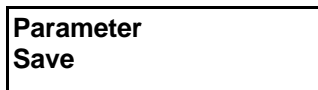Clicking the *parameters* button has the same effect. The individual
parameters are explained in detail in a subsequent chapter.

PARAMETER-SAVE

Saving parameters
specifically for a
project

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

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

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

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

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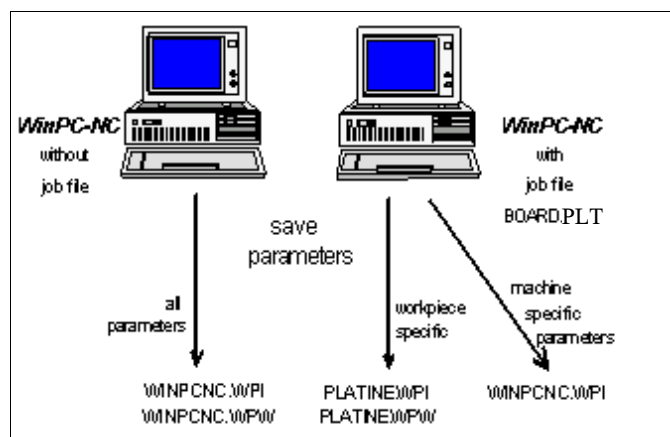
Saving parameters and tool settings

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

3.3.4. SPECIAL FUNCTIONS menu

The *SPECIAL FUNCTIONS* menu gives you access to a testing function for switches and a display of software version and serial numbers.

SPECIAL FUNCTIONS-SIGNAL TEST

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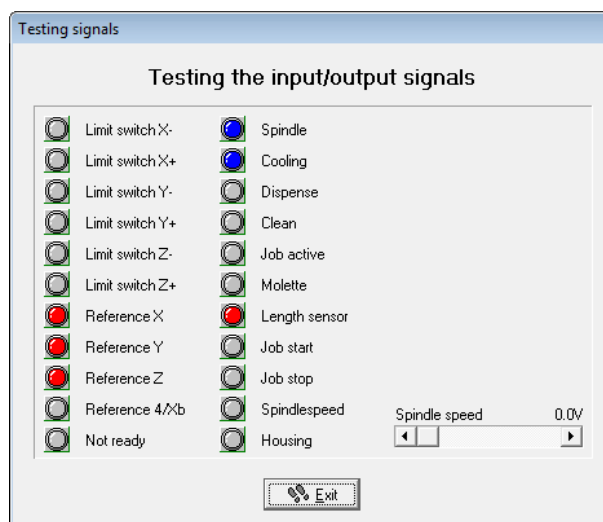
Signal test

Interactive testing of all inputs/outputs

WinPC-NC continuously checks the limit switches and reference switches on all supported axes and displays their status. Grayed out switches are not defined.

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

SPECIAL FUNCTIONS-Controller

Informationen about the program

WinPC-NC Starter can solely be used in conjunction with the special machine which must be connected by the delivered cable.
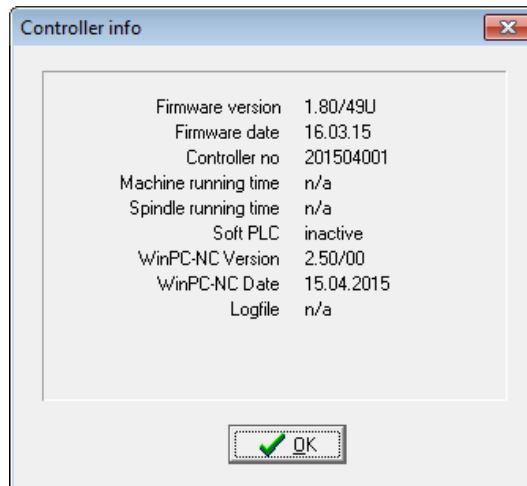
Display of the communication module version

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

3.3.5. HELP-menu

There are three items in the help menu:

Help topics...
Liability
About WinPC-NC

HELP-TOPICS

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

HELP-LIABILITY

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

HELP-ABOUT *WinPC-NC*

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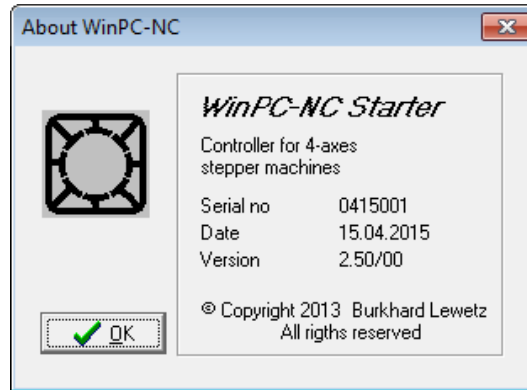
Information about the current version

3.3.6. UPGRADE menu

By presing the UPGRADE button you will get informations about a possible upgrade to the full featured version of ***WinPC-NC USB.*** You will find the mail address to contact us and in the second window your personal upgrade code, which must be indicated together with your full address details and name.
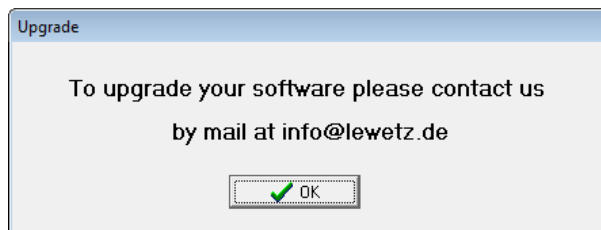
Information about the upgrade...

...and your personel upgrade code

4. 2D-CAM functions

4.1. Overview

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

– Assigning tools according to use

– Cleaning up data, deleting zero vector and double lines

– Setting output sequence according to tool number

– Optimizing empty moves

– Tool diameter offset compensation in contours

Relates to any 2D data

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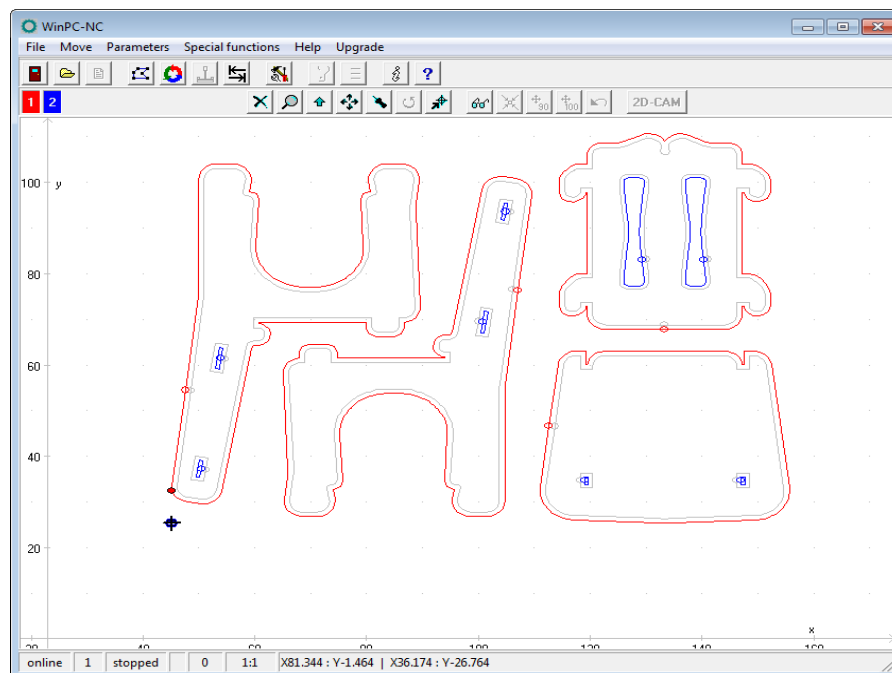
Loaded 2D job prior to CAM functions

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

4.2. Setting options
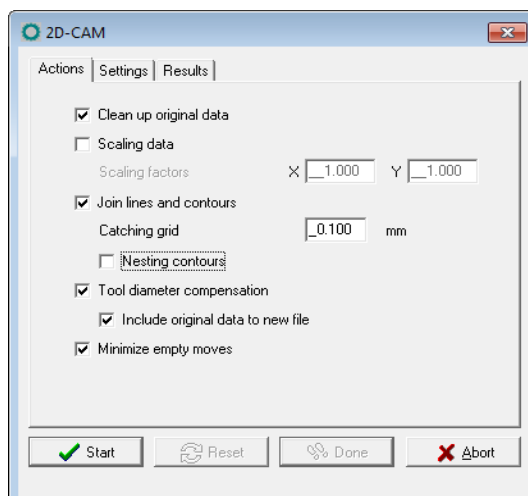
Dialog of CAM functions

Available functions and settings.

Cleaning up data

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

Scaling data

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

Searching contours/lines

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

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

By using an additional button the CAM function is induced to find and mark enclosed lines and contours, i. e. elements which are

completely surrounded by other contours. This is an essential feature for a radius correction later on.

Calculating tool diameter offsets

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

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

Optimizing empty moves

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

First of all the surrounded elements are considered and thereafter the corresponding contours. Thus it can be avoided that workpieces are machined that have previously already been completely milled.
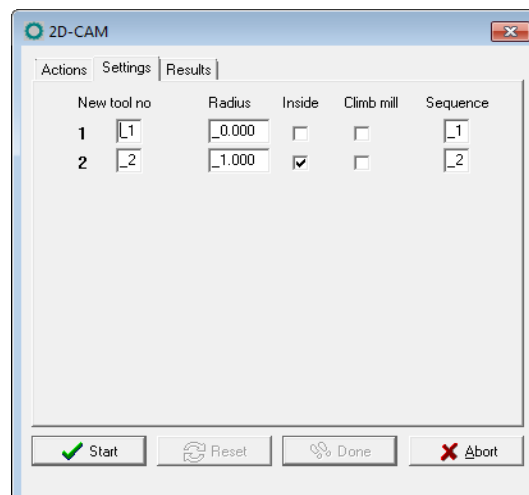
Second dialog of Cam functions

New tool number

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

Radius

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

Inside

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

Climb milling

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

Sequence of operation

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

Newly created file is immediately loaded
After completion of all required calculations and resortings **WinPC-NC** creates a new 2D file in an intern used format and gives the preceding name of the project and the ending *.OPT.

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

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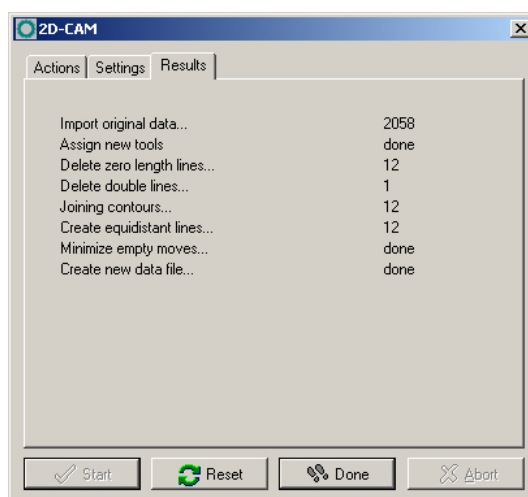
Display of results

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

Functions as required and in any possible combination

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

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

4.3. Example 1

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

1. We are loading the chair in HPGL format and obtain the following picture.
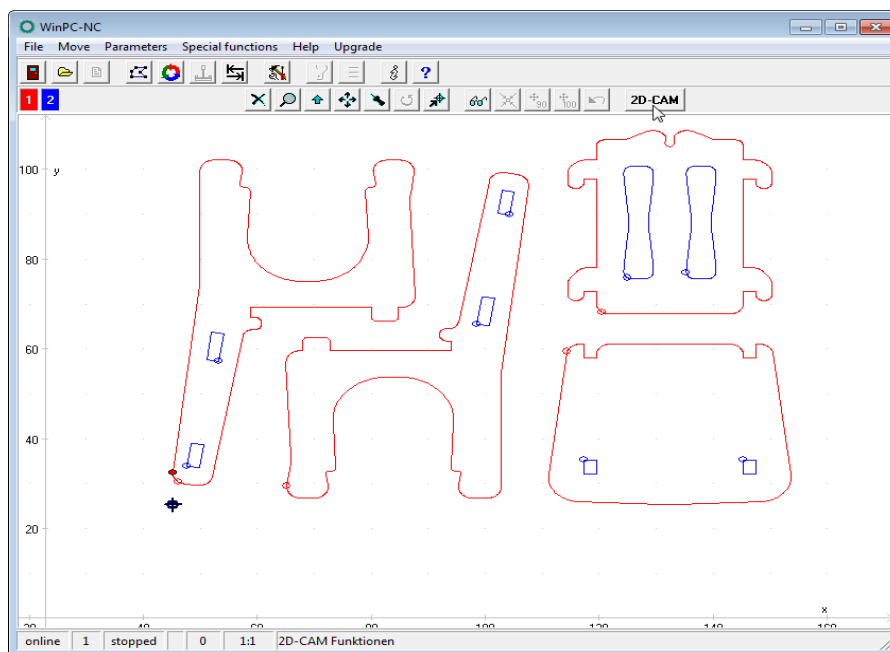
2. The contours consist of two colours and are to be milled with two tools. The inside contours are offset to the inside with a 1,6 mm milling tool and the outside contours with a 2mm milling tool. Settings are as follows :
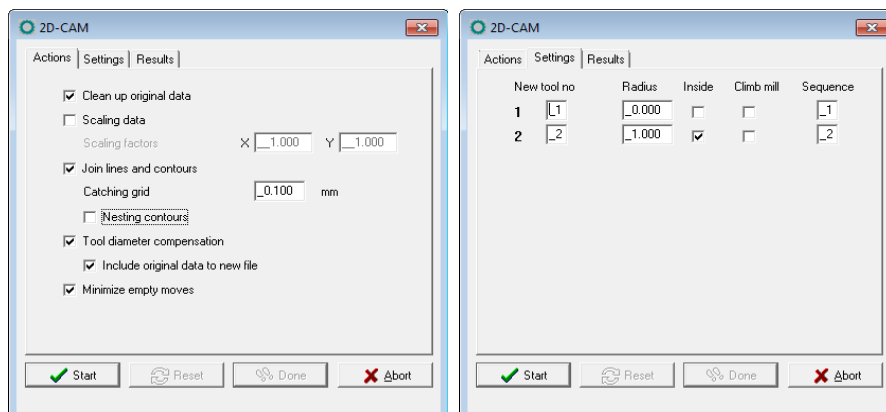
3. We just define the values for tool 1+2, start a search for contours, calculate radii compensation and optimize empty move and the result is...
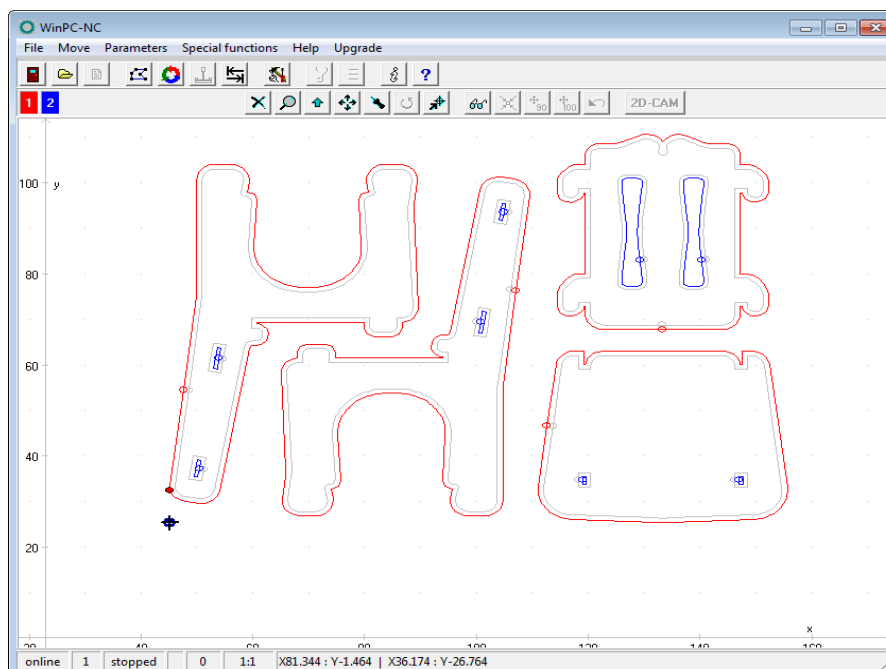
4. If we want to mill workpieces by solely one tool and do not intend to carry out a tool change, calculation has to be renewed. The function is reset, change tool assignment from tool 1+2 to the new tool 1 and start a new calculation for all contours with radius 1 mm.
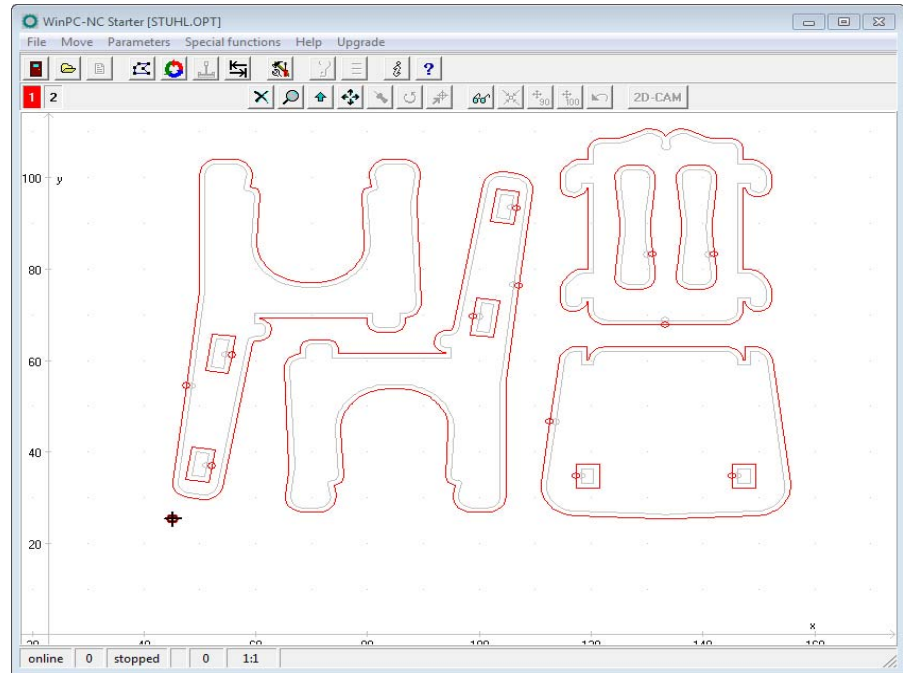
5. However, the offset of the recalculated path of the inside contour is inexact and all found contours are corrected towards to the outside. Now we still activate the button for the automtic search of surrounded contours and start calculation for the last time.
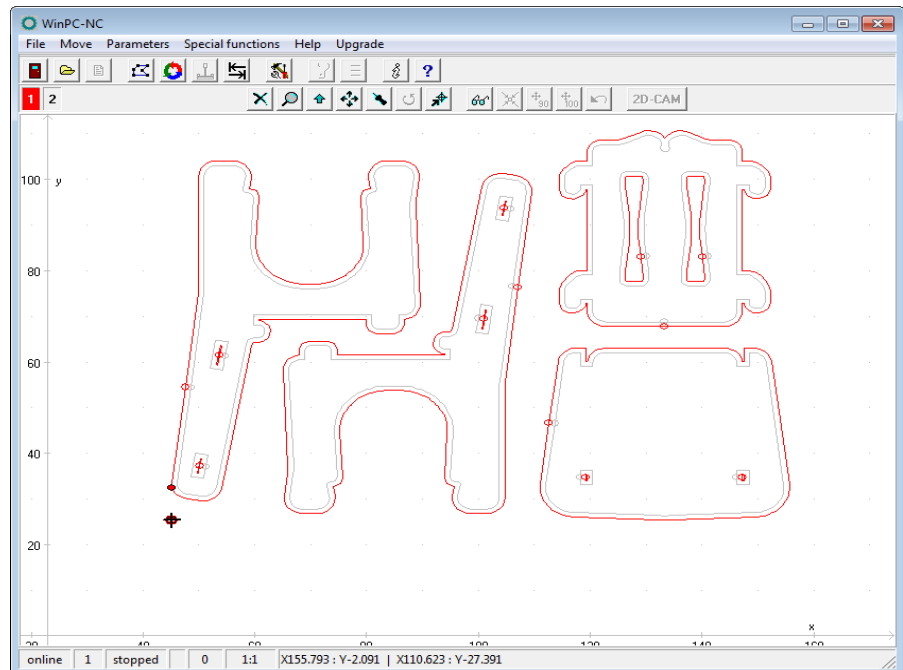
6. Now, the result fulfills our requirements and expectations, sequences are correct and all workpieces can be machined with our 2mm milling tool.

4.4. Example 2

1. We are loading a more complex DXF-file and intend to mill all contours with a 1 mm milling tool and place the contours independently.
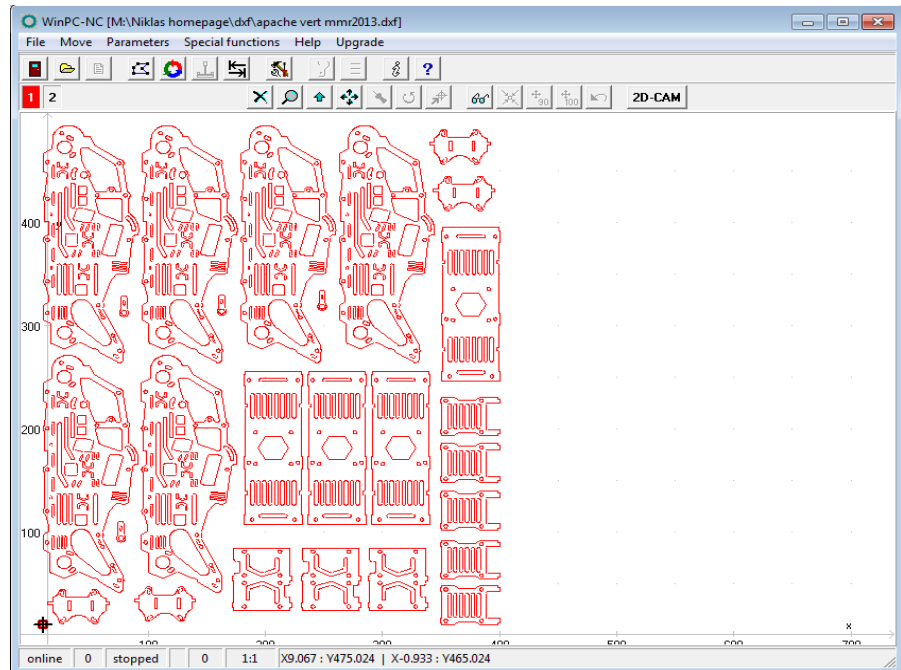
2. Settings in both dialogues are as follows.
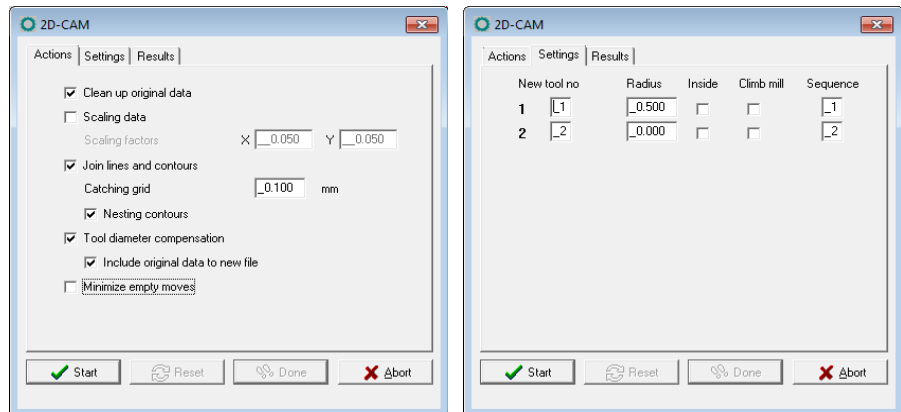
3. The result seems to be quite perfect and the surrounded contours as well as the offset paths to the inside and outside are cleary displayed. However, after zooming, it can be noticed that some contours have not been correctly recognized and calculated.
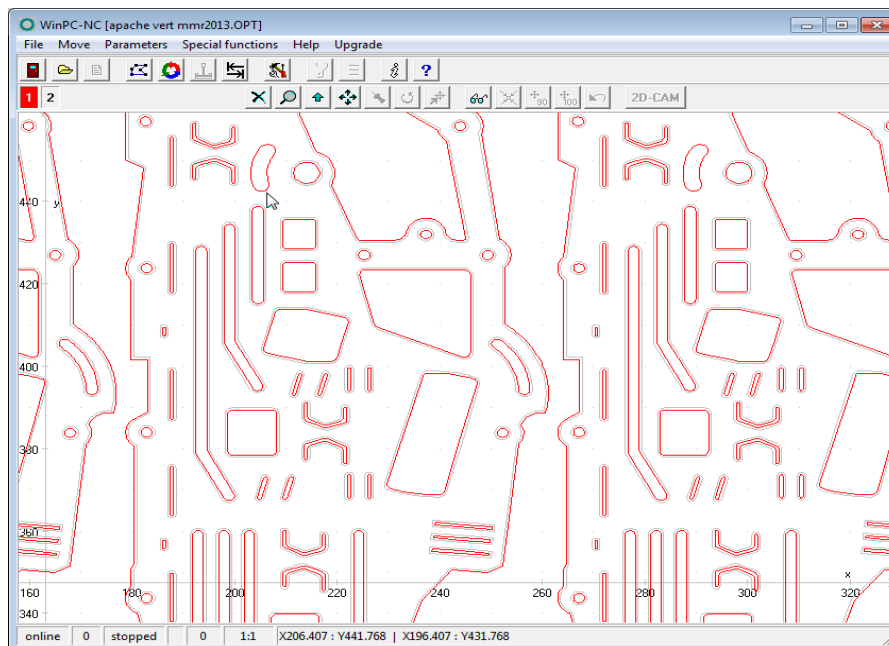
4. After zooming again, the incorrectness can be clarified. There is a spacing between two successive lines and it is bigger than the set catching grid.
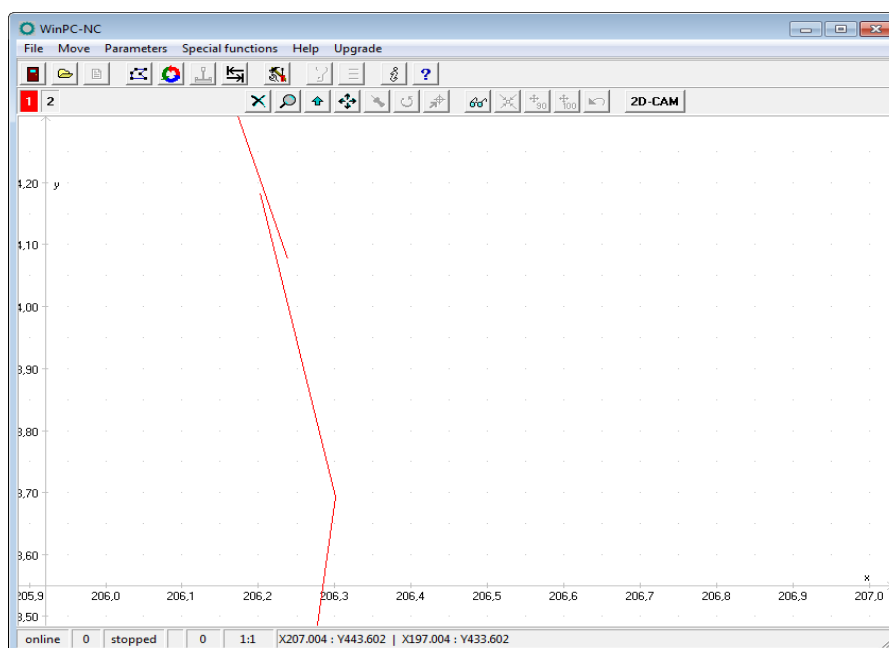
5. We extend the catching grid up to 0,2mm and start calculation again. Now we get a perfect result and milling can be started..
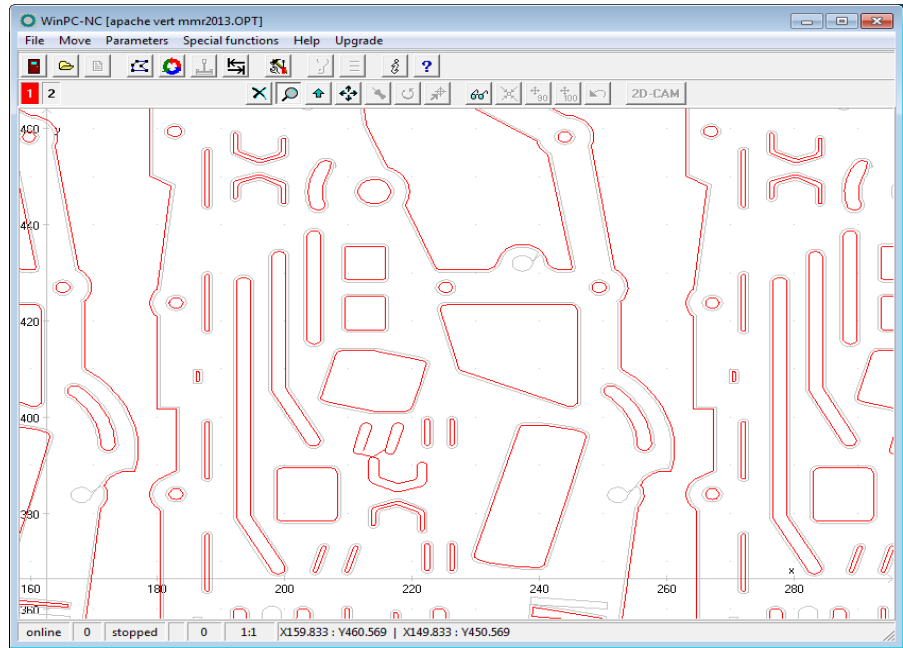
5. Parameter settings

All settings in one dialog

Parameter settings

Plunge depth

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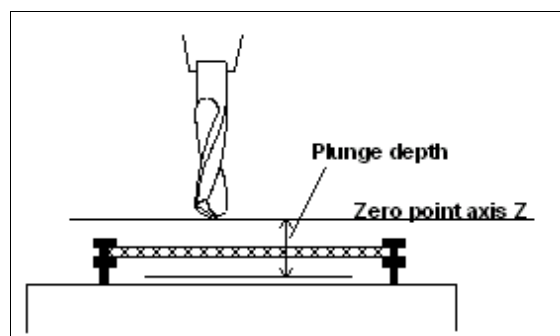
Plunge depth measured from the zero point of the Z-axis

Plunge speed

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

Advance speed

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

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

Start and end position

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

There are 4 possible start and end positions :

Stop	*WinPC-NC* stops at the reference position after reference movement, and at the last coordinate after each job process
Zero point	The machine moves to the defined zero point after reference movement and after each job process.
Park position	*WinPC-NC* moves to the defined parked position after reference movement and after each job process.
Zero point + toollift	*WinPC-NC* moves to the defined XY zero point and lifts the tool to its safety clearance

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

Zero point in file

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

Six positions are possible :

Bottom left	The zero point is at the smallest X and Y-axis coordinates in the file, normally at the bottom left edge. Mostly used with HPGL files.
Origin of Coordinates	The zero point is at the coordinate origin, i.e. where the CAD programs places it for the output. N This setting is to be recommended if several files are being used on the same workpiece, e.g. routing and drilling a board or when using G code files.
Center	The zero point is in the middle of the workpiece, i.e. exactly in the center of the coordinate dimensions in the X and Y-axis directions. This setting is useful for processing round workpieces, e.g. plates.
Bottom right	The zero point is positioned at the highest X and smallest Y coordinate of the file.
Middle right	The zero point is positioned at the highest X coordinate and exactly between the smallest and highest Y coordinate.
Top left	The zero point is positioned at the smallest X and the highest Y coordinate of the file.

Language

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

According to the status of April 2015 eightteen most important languages are available.

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

Zero point

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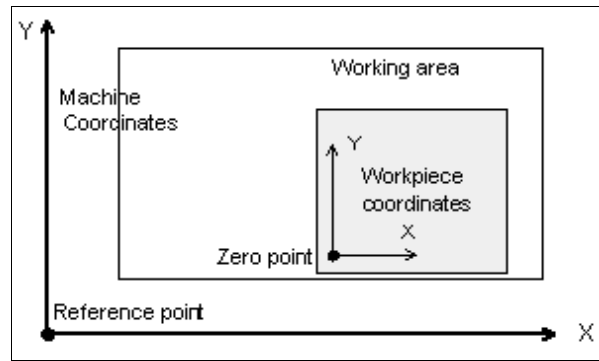
Working area and help positions

Zero point as origin of the workpiece coordinates

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

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

Park position

Defined position for breaks

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

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

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

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

Scaling factor

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

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

Tool lift

Additional clearance for avoiding collisions

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

6. Additional Information

6.1. Interpreters

At present, interpreters are implemented in **WinPC-NC Starter** for two NC data formats. In full featured vesion of **WinPC-NC USB** additional file formats are imported and even complex 3D files in G codes.

- HPGL
- drilling data
- Postscript (vector information in EPS and AI files)
- DXF (2D)

6.1.1. HPGL interpreter

Plotter data with the HPGL Interpreter

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

Drawings from...

- AutoCAD® and AutoSketch®
- Eagle®
- HCAM®
- CADdy®

Windows HPGL drivers

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

- CorelDRAW®
- Designer®

...and other Windows drawing programs use for preparing their data and storing data in a file. Alternatively, it is also possible to use a driver for following plotters: *HP7475, HP Color Pro* or *HP Draftmaster*.

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

PA [*x,y*[,*x,y*]];	Move to absolute position(s)
PR [*x,y*[,*x,y*]];	Move to relative position(s)
PU [*x,y*[,*x,y*]];	Lift and move tool
PD [*x,y*[,*x,y*]];	Lower and move tool
SP *x*;	Select tool
IP *x,y*,[,*x,y*];	Define scaling points
SC *x,y,x,y*;	Define scaling factors
CI *r*[,*a*];	Draw complete circle
AA *x,y,a*[,*c*];	Absolute arc
AR *x,y,a*[,*c*];	Relative arc

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

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

The following non-standard expansions can also be used :

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

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

- There must always be a pair of coordinates x,y
- All commands with coordinates must have no, one or more coordinate pairs
- Every command must be followed by a semicolon (;) or a new command
- With PA and PR, the subsequent commands are also set to absolute or relative positioning
- The last command should lift the tool (PU;)
- units in HPGL are 1/40 mm length

HPGL example

PU;	*Start by lifting tool*
PA1000,1000;	*Then move to 1000/1000*
SP1;	*Select tool 1*
PD;	*Lower tool*
PA1000,2000;	*Move square*
PA2000,2000;	
PA2000,1000;	
PA1000,1000;	
PU;	*Lift tool*

6.1.2. Drilling data interpreter

Various drilling data formats

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

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

Drilling data sntax

The syntax is as follows:

% oder %%
[X*val*][Y*val*][T*x*]
.....
M30

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

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

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

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

Sample of drilling data in format 2

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

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

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

6.1.3. Postscript interpreter EPS and AI

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

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

6.1.4. DXF import

WinPC-NC can import files in DXF format but will accept included 2D informations only. DXF is a common used exchange format for drawings and technical constructions and can be created by nearly all drawing programs.

Before creating a DXF file for **WinPC-NC** please ommit polylines and splines and other complex data elements and unchain all existing groups to single elements.

6.2. Error messages

Error messages are displayed in a small window in the middle of the screen during operations in **WinPC-NC**.Messages in a gray box do not indicate there is a fault, but only provide information.

The following error messages may occur :

- **Limit switch reached**

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

- **Machine not ready**

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

- **HPGL syntax error**

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

- **Connection canceled**

    The connection to the DLL communications module is interrupted. This can happen when other window programs or drives interfere with the realtime tasks.

- **Incorrect parameter**

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

- **Stop signal detected**

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

- **Go to reference position first**

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

- **Error xx from controller**

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

- **Data cannot be displayed**

    The NC data for the current file cannot be displayed graphically. Either they do not match the selected format or they do not contain any tool actions. Another cause for failure may be that the format identification is switched off or that an incorrect data format has been adjusted.

    WinPC-NC Starter can display data formats like HPGL, PLT, EPS/AI, DXF (2D) and drilling formats. It cannot import Gcode files.

- **Tool not yet defined**

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

- **No tool file found**

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

- **Machine is at limit switch**

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

- **Command rejected by controller**

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

- **Maschine not ready**

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

- **USB communication failed / Timeout**

    Communcation of *WinPC-NC* interferes with the realtime-DLL. Please make sure your machine and USB hardware is connected with the original cable and there is no USB hub between. Disable all power saving functions for computer and especially from the USB port in your hardware manager of Windows.

- **Protection hood not closed**

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

- **File with invalid format**

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

- **Help file not found**

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

- **Workpiece exceeds working area**

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

- **Next position outside working area**

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

- **Unknown command**

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

- **Cannot create file**

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

- **No park position defined**

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

- **Required tools not defined**

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

- **Tool moves in too far**

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

- **Maschine positioned on the reference switch**

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

- **Cannot create file**

    WinPC-NC tried unsuccessfully to install a file. Possible causes are insufficient memory capacity or missing allowance for access in the current register.

- **Out of memory**

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

- **Incorrect realtime-DLL**

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

- **Error in speed**

    The defined speed does not match the corresponding machine parameters.

6.3. **Special versions of *WinPC-NC***

Special
applications
on request

The authors are always grateful for opinions and suggestions.

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

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

Here are some examples:

- Dispensing plants for glass inserts and reconstructed motif glasses

- Laser welding plants and laser marking devices

- Plasma cutting plants and oxy cutters

- Laboratory applications for complex measuring tasks

- and there are many other applications besides

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