Changelog
WinPC-NC 3.XX

This document describes all published new functions, function extensions and bug fixes for all versions. Please scroll to the last used version and read the changelog to the latest version at the beginning of this document.

Explanation of font colour and abbreviation

S L U P F	Function/modification applies to ***WinPC-NC Starter, Light***, ***USB*** or ***Professional*** or the firmware of the axis controller
blau	New function
grün	Adaption
schwarz	Bug fix
rot	Bug fix of an critical error, Update is recommended

Burkhard Lewetz Hard- und Software Brückenstr. 7 D-88074 Meckenbeuren
www.Lewetz.de Info@Lewetz.de

Version 3.41/32 Date : 08.03.2023

Minor changes and fixes

L U P

- empty moves when using mass production eliminated
- fix at definition of protected area 3
- automatic reload of NC file only if file still is acceptable, especially when using USB sticks
- enhancement at smoothing contours

Version 3.41/29 Date : 14.11.2022

Ethernet connection as an alternative to COM port

P

For ***WinPC-NC Professional*** there is also the possibility to use a network connection with an Ethernet port as an alternative to the serial COM connection used so far. The newer CNCCON axis controllers are optionally equipped with this interface.

Minor changes and fixes

L U P

- Technology plasma, dialog somewhat better structured and cutting speed definable independently of other settings.
- loading a new profile also updates the current NC file
- spindle speed or laser power correctly output at job start
- macros only possible for the first 10 tools and no incorrect macro call for tools >10 anymore
- Correction for edge smoothing with very small vectors
- Program hangup when probing with edge finder fixed
- signal test, slider no longer switches off the laser

Version 3.41/24 Date : 23.05.2022

Length check before and after ATC

P

If an error is detected at length check before magazine release or after magazine grab there will be an error and it can be handled manually. At resume of job the length check is done again to verify the correkt mounted tool.
In a second modufication the messure tolerance can be defines different as 0,2mm which was default.

Movements at tasting plate are monitored by machine dimensions

U P

All movements now are checked against defines machine dimensions and borders and no movement will take place behind.
Z axis is lifted only by defined tool lift height

Minor changes

L U P

- faster tool selection in tool menu
- Resume of a job now uses correct Z lowering speed at first position
- Correction of wrong program display in Gcode window

Version 3.41/20 Date : 11.03.2022

Maschine selection accepts pin assignments again

P
At certain OEM versions a machine selection dialog can be activated at first startup. As a bug some pin assignments were not fransferred correctly to axes controller.

Minor changes

- tool probe will no longer move back too high
- bugfix at Start from... function
- fix at keypad from CNC-Step
- loading of next job at automatic repetition much faster

Version 3.41/18 Date : 11.02.2022

Length check after grab of new tool

P

Since two previous versions a length check couzld be enabled after grabbing a new tool via ATC to make sure the tool is fixed and mounted correctly. Now a invalid check is saved and prevents the user to resume the job without checking the tool length again

Synchronous Y axis with automatic align

P

WinPC-NC Professional was able to handle double and synchonous X axes and now also can handle a double Y axis. The alignment with help of two reference switches is same as before for X axis. A new input signal I171 RefYb was defined to assign the pin no of the RefYb switch and enables the function.
Note, to use this new function the firmware 1.62/08R or newer is needed at controller.

minor changes

U P

- bugfix at position outputs for parking and homing positions or zero point
- nc file by command line now accepts all parameters
- minimised program start also creates all windows and boxes in defined color style
- Gcode command M00 no longer needs two external start signals

Version 3.41/13 Date : 24.01.2022

Define and check critical areas

P

In parameter settings at tab monitoring it is possible to define up to 3 critical areas with XYZ coordinates and they can be checked in regular operation. If machine touches or moves into one of these areay it will be stopped and an error message appears.
In this situation you can disable the monitoring by using a new button in the upper button line and move your machine back to save positions by manual jogging.
The monitoring is switch on automatically after homing the machine or starting a new job.
To use this new function firmware 1.60/99R must be used in axes controller.

Force homing

U P

A new parameter forces the user to perform a reference or homing move before each other action. You cannot deny or skip the prompt for homing it it is necessary. This can be helpful if you use an automatic tool changer or have defined critical areas.

Minor changes

- position and state of camera window is saved and will be opened automatically if you start manual jogging

- laser speed at resuming a job is set correctly

- fixed graphical display with EPS/AI files

- resuming a laser job sets power in correct dimension

- activating a keypad now is no longer dependant from a loaded file

Version 3.41/11 Date : 03.12.2021

Tool length messurement for checking ATC or broken tool

P

If an ATC is enabled and tool length kompensation you can enable additional length measurements before releasing a used tool to chek it it is broken. The new length is compared to the known and if it is a difference more than 0,1mm an error is shown and the job will be canceled.
In addition you also can check the tool length after releasing and after grabbing an new tool to verify the correct fixing of the new tool and the correct releasing in magazine. The new checking measurements can easily be skipped to save time.

3 new output signals for In-position-status

P

There are three new output signals which can be assigned with a special pin no.
Q200 InPositionZeroXYZ
Q201 InPositionParkXYZ
Q202 InPositionRefXYZ
All outputs will be HIGH if axes XYZ are at the defined positions within a tolerance of 50µm.

Minor changes

S U P

- Elimination of a zero move in XY is only valid if no Z axis movement is programmed in same line
- Stopping a job by pendand is working again
- T0 M6 or Sp0 commands in NC files will release the last tool in magazine
- Stop input at job start and signal test function will be handled correctly
- Resume a paused job with correct speed

Version 3.41/08 Date : 30.09.2021

Some minor changes

S U P

- fixed entering of digits in input fields in manual jogging dialog without movements from cursor block at keyboard or keypad
- correction of arcs element in DXF files
- spindle speed override works now even if output pin is assigned inverted

Version 3.41/06 Date : 23.08.2021

Faster file code display

U P

The optinal nc file code display now is extremous fast and even very large nc files are displayed instantly.

New zero point during tool change no longer fials

U P

At prompt for a tool change you can switch to manual jogging and move to and save new zero points without problems.

Enhancements in plasma cutting

P

In technology plasma the defined Z zero point is used instead of the current position in former versions. It is the basis for defined ignition height and cutting height. When using the surface probe the Z zero point is measured at each contour start. The defined cutting speed is used in all situations instead of speeds in Gcode files and parameter settings. Even Z coordinates in Gcode files can be ignored during a cutting job.

Loading profiles actives included macros again

U P

As a bug the macros included in saved profiles were not activated after loading.

Inverted PWM signal was correct in spindle override commands

S U P

If an inverted PWM signal pin was defined the job override function sometimes failed or was inverted as well. This is fixed now..

Stop during tool measurement keeps Z height

U P

Job canceling during a tool measure action now keeps the Z axis in save height and will not touch down.

Controller based jobs can be started by i255 Start signal

P

When a separate saved job at axes controller and job execution without WinPC-NC is active, the job start can also be triggered by i255 START signal.

Minor small changes

S U P

- tool macros now active even with ATC usage
- changing clock signal invertion is instantly changed in controller
- several message boxes have new and clearer text
- better measurement with edge probe or tool length probe
- G53 Z0 causes WinPC-NC to lift Z axis to reference point. This command is used from Fusion360 design tool at certain posts and was not interpreted correctly by WinPC-NC in former versions.
- Start from... with laser jobs causes the laser to switch on too fast.

Version 3.40/94 Date : 08.06.2021

Minor changes and fixes

S U P

- better and more acurate realtime timer and continuous monitoring of it
- better handling of very high and totally odd axes resolutions
- G4 command form post of Fusion360 interpreted correctly
- better checking of tool height measurement
- Signaltest function possible even if machine READY signal missing

Version 3.40/92 Date : 03.05.2021

Minor changes and fixes

S U P

- Manual jogging, after Softstick function edge finding possible
- enhanced import of nc files with seldom line separators
- prompt boxes can be quit with keyboard and ENTER key
- fix in 2DCAM function at very special DXF files
- look ahead function enhanced at algorithm with factor 25-35 settings
- reference movement als correct if switch is free again until axis stop. a high precision reference movement is possible all the time
- enhanced circle commands at HPGL files

Stop at narrow arcs caused position jump

S U

When stopping a job while moving in very small vectors or arcs caused a position jump and position loss at certain situations. This is fixed now and a stop can be done at any position without lost.

Plasma-/Oxy cutting with new macros

P

The automatic created macros when beginning a cut are redesigned and even the ArcGood monitoring and touch probe activation is running perfectly now.

Version 3.40/87 Date : 10.12.2020

Faster display of large NC files

S U P

Very big NC files are displayed much faster and optional program window is created in background

Program fault after moving to park position fixed

S U P

In very special situations a possible program fault is fixed if moving to park position or zero points is canceled shortly before reaching the end.

Checking of file permission at startup

S U P

At program start WinPC-NC checks all rights for writing and creating files in the installation folder. This is necessary for saving paramerter settings or protocols. If file writing is not allowed a message is displayed and it is recommended to restart as administrator.

Backlash compensation fixed

S U P

In position display of 4th axis the defined backlash compensation is no longer calculated twice. This was a display mistake only and did not influence axis position.

Minor fixes and enhancements...

S U P

- more commands for remote control over registry items
- fix at laser test functions if moving free distance is not defined

Version 3.40/89 Date : 08.03.2021

Minor changes and bugfixes

S U P

- operation with pendant when pressing multiple keys
- correction at digitising function
- import of PLT data with binary header
- position display after compensation of backlash

Version 3.40/82 Date : 08.09.2020

Enhancements in look ahead algorithms

U P

A complete new developed look ahead algorithm enhances dynamic speed control in arcs and circles and reduces speed according to radius. It can be selected with values of 25 to 35 at factor fpr dynamic speed control. 25 reduces most and 35 reduces speed less. It always depends to machine mechanics and moved speed to find optimised settings.
In a second step the contour smoothing factor can be set up to 5000 which will combine very small vectors to larger ones up to 5mm. This will effect very fast movements and large tool paths.

Enhancements in tangentional cutting

U P

Tangentional axes and cutting now moves smoother and faster and even the turning back to zero at end of job is eliminated. The tangentional axis is moved to new direction much faster at long vectors.

Remote control by registry commands

U P

With creating of more and detailed registry commands the remote control of WinPC-NC Professional by a special host program is more flexible and more reliable

More minor changes

S U P

- better assigne font size in different message boxes at program start
- tool length measurement at sensor plate is running correct
- backlash compensation with up to 10000 steps
- DXF import with scaling at INSERT commands
- PWM spindle speed mor accurate calculated with min and max speed settings
- macros at tool change are activated correct again
- speed reduction at diagonal moves is more accurate
- WinPC-NC Professional saves and loads PCL code in profiles as well
- ability to start more instances of WinPC-NC Professional at one computer to control more than one axes controller over different COM ports
- resume a job starts spindle after prompt now and therefore at correct time

Version 3.40/72 Date : 22.07.2020

Predefined settings updated

S U P

Minor settings of our partner's new machines are adjusted and enhanced

Pendant

U P

Keys at pendant are checked with unbouncing function to avoid short electrical impulses. Clearing all former settings and speeds if movement functions is activated again. Touching of limit switches is now handled correctly.

Cylindric engraving

U P

Orientation of 4th axis for automatic engraving a PLT or DXF on cylindric parts can be along X axis or along Y axis and can be switched in technology settings.

Technology function dispensing

P

Now also possible with Gcode files

Different minor changes...

Touch plate also handles new Z zero point for length compensation

New registry commands for remote control in background realised

Reference switch of 4th axis is no longer checked at start of homing

2D-CAM, inserts an empty move after each tool change and solved problem if single tools are deactivated

Job cancel even possible during dwell times

Enhancements at dragging knife compensation

Shortcut shift-F3 for Start-from... command

DXF files with Layer=0 are accepted with new tool

Version 3.40/68 Date : 02.06.2020

New predefined settings for new machine types

S U P

Some of our OEM machine manufacturers have created new machine types and we optimised the predefined settings which causes a much better controlling of them.

Tangentional function with digital Z movement

U P

Controlling of tangentional tool by digital signal UP/DOWN now works at correct sequence during job process.

Bugfix in 2D-CAM

S U P

In some cases with imported DXF files single lines in one layers are accidentially deleted in cleanup function and therefore missing in new created file.

New function Job Saving at controller

P

For our industrial solution ***WinPC-NC Professional*** we realised a new function to permanently store a single job at the controller which can be used later on for mass production without a connected computer or the need of a running ***WinPC-NC*** host program. The stored job can be trigered by a signale digital start signal and runs the whole job in same manner than with connected ***WinPC-NC***.

New firmware 1.60/84 for axes controller

F

The new function Job Saving even needs a new and enhanced firmaware at the controller.

Version 3.40/65 Date : 26.05.2020

Gcode G83 deep drilling realised

U P

In addition to the old G81/G82 commands we realised even the G83 drilling command with rejection distance and dwell time.

Bugfixes at 2D-CAM and DXF files

S U P

At certain DXF files the cleanup function did not check and delete zero length vectors correctly and left them in the new data file.

Technology function oxy and plasma cutting

P

An new and enhanced technology function for cutting is added. The contours can be started with a moving in vector and an existing ArcGood signal is monitored and triggered. Defined cutting speed is now accepted correctly.

Minor bugfixes

- resume job now starts spindle at correct time
- profil saving with ***WinPC-NC Professional*** saves also controller data
- job canceling is also possible during dwell times
- grid in graphical display is adjusted to exact zero points
- homing direction of 4th axis
- defined zero points of G54 commands can now be saved as system or project parameters
- line numbers in program windows can be switched off
- stop watch is stopped if a job is canceled
- no moving functions while open parameter dialog
- more function for remote controlling of WinPC-NC through registry commands
- new and current firmware of axes controller now also works with very old

WinPC-NC Professional versions

- technology dispensing and very special dispensing functions run with Gcode files

Version 3.40/59 Date : 14.04.2020

Saving profiles with all current settings

U P

At moment of saving a new profile all current settings and parameters are used and not the previous saved only.

Turning data will be kept at 2DCAM functions

S L U P

At new calculation in 2DCAM functions a current turning ofdata is kept and orientation of new data is same as before

---

Sensor plate

U P

Movements and tasks using a sensor plate are corrected and organised in a new way and calculation now is much more precise than before

---

Tangential function with pneumatic Z axis

U P

If tangentional Z axis is contolled by single digital signal the superfluous moving down before turning the axis is fixed.

Diverse smaller fixes

S L U P

- checking of min/max spindle speeds and automatic corrections
- manual movement during tool change dialog is possible again
- settings of tool changer are saved correctly in profile
- ***WinPC-NC Light*** and nc100 adaptor found again
- manual movement of Z axis with keypad fixed

Version 3.40/57 Date : 20.02.2020

Faster refresh in graphical display and program code window

S L U P

At new job load and after job and after manual jogging the new graphical display is refreshed much faster and unneeded refreshes are eliminated

---

Unneeded drop down of tangentional knife eliminated

U P

With certain settings the knife was lowered without any action and lifted again to turn.

Program hangup at job start at very poor computers

S L U P

At less performed computers it could happen that ***WinPC-NC*** hangs up and could not be operated any more. This was caused by a not correct synchronised job progrss windows and is fixed now.

More less important fixes

S L U P

- protocol creation at loading DXF files
- Gcode interpreter with modal commands at dwell time command
- display error at saving XY zero point

Version 3.40/52 Date : 06.2.2020

Manuell jogging with touch monitors

Since last update of Windows10 the used messages and internal methods at touch screen driver changed and we updated our interpretation. Now manuall jogging can be used as before.

Differnt speed ups at internal processes

To meet old and very weak computers like Intel Atom processors we have enhanced lot of internal functions and created a new parameter setting *Reduce processor time*. If active the realtime graphics with red dot and update of current coordinates is reduced and slower but operation on this computers can be done as usual.

Reassignment of key codes for different keypads

Keypads of CNC-Step, GoCNC, our own bluetooth and cable keypads are interpreted correctly and can be used like before.

Version 3.40/48 Date : 22.1.2020

Parameter settings without instant correction

Modifying of parameter settings work like before, sorry for this fault.

Opening of settings dialog causes delay

Problem fixed and dialog can be opened as usual

Macro commands for waiting with new message

At active macro waiting commands there is a new message in status line and you will recognise why the job is paused and which signal is waited for.

Gcode modal commands G02/G03

Sometimes G02/G03 commands are not set modal and following commands are not interpreted correctly. This is fixed now.

2DCAM enhanced

Better smoothing of contours in CAM functions.

Macro for tools are invoked at tool selection dialog as well

Until now defined tool macros are running in job mode only. Now they are invoked in tool selection dialog as well and immetiately if a new tool is selected.

Only used tools editable

Unused tools from a loaded file are disabled and cannot be modified in settings dialog. This will minimise incorrect parameter settings.

Memory problem fixed

When loading different nc files subsequently sometimes the used memory was not released completely and causes a full memory error. This is fixed in new version.

Version 3.40/43 Date : 16.12.2019

Import of EPS/AI enhanced

Better recognition and identify of EPS/AI files.

Macro command SetOutput fixed

Macro command now can be used several times with different output numbers

Version 3.40/41 Date : 01.12.2019

Laser test function with new fucus diameter setting

U P

To avoid overlapping laser lines you can define the focus diameter as a new parameter setting in test functions.

Grey scale engraving enhanced

U P

Better grey scale engraving with 8 bit laser power control and much more than 16 possible gray scales as before.

Finished drag knife function

S L U P

Drag knife offsets even with very small vectors and contours.

New parameter to control JobEnd macro

U P

Selecting the JobEnd macro even if a job is canceled.

Version 3.40/37 Date : 13.11.2019

Check for file access at startup

S L U P

WinPC-NC needs writing access in installation folder to save settings. This is checked at startup and when missing an message will be shown.

Correction of PWM control at lasers

U P

PWM signal is now created much more acurate and creates better grayscale engravings with different lasers.

Version 3.40/33 Date : 30.9.2019

CAM functions : Move in vector

S L U P

In combination with searing closed contours you can define lead in vectors to the contour from inside or outside. This prevents ugly marks at starting end ending the contours.

CAM functions : Drag knife compensation

S L U P

With new function WinPC-NC can create and compensate a small knife offset which is typical for drag knifes. All moves at angles and edges are a little longer and an arc is created to turn into new direction. The result is much more accurat cutting of foil contours.

CAM functions : Better sorting and empty move optimization

S L U P

With a new checkbox in dialog WinPC-NC will completely finish all parts first befor starting with a new part in job. This minimizes job time and helps in handling parts.

Enhancements in laser control and calibration

U P

- defined maximum power is used in all functions
- some corrections in calibrating functions
- use of correct power values after pausing in grayscale calibration

---

Enhancements for oxy and fuel cutting

P

- surface finding function
- monitoring of digital input lines for moving up/down during job and for manual jogging the Z axis
- controlling of PROMA height control system with inputs for moving and ArcGood and ArcFault signal

---

Minor fixes

S L U P

- leaving parameter field with ENTER key does not clear current tool
- completed dutch language block
- missing tool no in Gcode file like T M6 is handled correctly

---

Version 3.40/29 Date : 12.8.2019

Correction at Laser test functions

U P
Grayscaletest starts with correct speed. Switching on/off at laser controls PWM signal as well. This is important for lasers which are controlled by PWM only and have no extra digital on/off input.

At very strong lasers the maximum load can be limited at xx%. This is important if machine cannot move fast enough to create any grayscale because of too powerful lasewr beam.

Laser on/off at commands PU/PD and G0/G1

U P
Laser will be switched correctly at all DXF or PLT or OPT files.

Correction on different input/output info lines

U P
In different parameter tabs the displayed output lines and assigned pins are displayd correctly again.

Version 3.40/27 Date : 18.7.2019

Rehoming check available for all *WinPC-NC USB* softwares

With this version the function rehoming check is available for all ***WinPC-NC USB*** variants and licenses. Please note that the function can display big deviations, if the machine moves to fast to the reference switches and crashes against the mechanical machine limits.
If the function displays big deviations that are not plausible, please reduce the Homing speed, search parameter.

Bugfix New macro command Set offset XZY

Bugfix during the execution of this macro command.

Version 3.40/26 Date : 11.7.2019

Bugfix parking after job

L U P

Sometimes an error occured during long lasting movements to parking or zero position at end of job

---

New output signal for running homing movement

L U P

A new output signal Q252 homing active is created to signalize a running homing process. This signal is needed to control our new adapters for parallel or slave axes and the new automatic adjustment funktion during homing.

Forgotten test lines added

S L U P

At two new functions we forgot to ranslate the corresponding text lines and added it in this version. Sorry.

Version 3.40/25 Date : 9.7.2019

Bugfix Camera

S L U P

An active camera window could not moved correctly

Incorrect feedrate speed from Gcode files

U P

With very special parameter settings feedrate F commands from Gcode files were not moved and speed settings from tools were used instead.

Lost communication during long moves to help points

S L U P

During very long moves to zero or parking position the communication was lost sometimes.

Minor fixes

S L U P

- Selective profiles are loaded correctly even if saved with older versions
- Zero point setting with surface block is saved correctly

Version 3.40/20 Date : 29.5.19

Different minor fixes

S L U P

Fix at hangup when moving to zero point or parking position after homing
subsequent loading of profiles caused an error in Professional
Activation of sensor probe macro at plasma cutting in Proffesional
Laser engraving and Start from... function caused lot of PWM signals
Better adjustment of messages in job progress window

Invalid parameters without error message

S L U P

Parameters outside valid area will now be corrected silent and without show error message

---

Technology Laser and more activated at one job

U P

Until now only one technology at one time could be active. Now Laser+Circular axis or Laser+Tangentional axis can be activated and used at same time.

Version 3.40/16 Date : 23.5.19

Bugfix at start of *WinPC-NC* and different function

S L U P

Certain definitions of spindle speed items caused a program hangup.

Version 3.40/15 Date : 17.05.2019

Enhancements with hand wheel

S L U P

No unexpected jumps when starting movement in step mode. New settings to move to park position with button Goto0s and tool lift height when setting Z zero point.

Various minor fixes

S L U P

Endless reset is accepted even at diagonal movements
Tool length measurement at 30 tools shows now correct tool number
Canceling of homing also cancels parking and measurement
Buttons of manual movement dialog are show correctly
Switching of spindle and cooling with hand wheel creats correct spindle speed

---

Fixes with touch probe

U P

Touching and calcultaion of angle is enhanced and prevents any inaccuracy.

Definition of tool settings at laser technology

U P

By using a new checkbox setting in laser technology dialog you can select speeds and power settings from tool parameter instead of fixed laser speed.

Version 3.40/10 Date : 08.05.2019

Program hangup at canceling jobs fixed

U

In certain situations there happend a totally program hangup after canceling a job. This is fixed now.

Version 3.40/08 Date : 06.05.2019

Running times and data for machine maintenance are now editable

U P

The running times and machine data are editable now. This allows to enter the previous machine data to the new version 3.40 and keep all information up to date.

Running times and data for machine maintenance according to pin assignment

U P

The Running times and data for machine maintenance for spindle, cooling and dispensing/laser are only incremented for assigned output pins of the respective function.

Pilot laser can be activated by hand wheel HR-10

U P

The pilot laser of the laser functions can be activated with the ***hand wheel HR-10*** now. To activate the pilot laser the outputs for spindle and cooling must be set to active.

The angle for turning the graphics is saved as nc file parameter

U P

The measured angle from the touching plate is now saved as nc file parameter. This allows to continue the job with the same angle later. The angle is set to 0° during restart of ***WinPC-NC*** or during opening a new file without own turning angle.

Version 3.40/05 Date : 26.04.2019

Selective Profiles

S L U P

Additionally to the normal profile functions ***WinPC-NC*** offers the selective profiles since version 3.40. With that new selective profiles it is possible to load and save only a selective part of the parameters. This new function leads to a much more easier configuration of new additional accesories for your CNC machine in the future.
More information and a detailled explanation of the selective profiles can be found on our website soon.

Turning graphics by degree

S L U P

Graphic display and file can be turned by any angle in limits of -25 up to +25 degreens to justify a non parallel fixed part. It can be used by help of a touching plate or moving to two different points in graphics or by entering the value as parameter.

Maximum of 30 tools

U P

A maximum of 30 tools can be used instead of limited 10 tools upt o now. It must be enabled in settings for compatibility reasons

---

Running times and data for machine maintenance

U P

Different running times and counters are managed as information and to estimate machine maintenance tasks like machine running time, spindle running time, switching spindle on/off and more. All data can be shown as controller information

---

Correction of key assignment at certain keypads

S L U P

Wrong key assignment at certain 3rd party keypads is fixed

Ignore 0/0 movement

U P

Movement to X=0/Y=0 at job start and end is detected correctly and can be eliminated on demand

---

G5x commands in Gcodes

U P

Switching different zero points with G5x commands did not accept a modal moving command like G0 to move to new calculated position.

New firmware at axes controller at WinPC-NC Professional

P F

By use of this new version an update of firmware at axes controller is needed. Please use new firmware 1.60/75r which is included in update as well as info sheet.

Version 3.02/07 Date: 14.03.2019

Window position for more then one computer screen

S, L, U, P
WinPC-NC remembers the window positions for the next software start.

Numerik keypad and keypad from other manufacturers

L, U, P
Adaption of function keys for keypads. The selection of the keypad can be done in the parameter settings basic settings-interface-keypad.

Version 3.02/06 Date: 04.03.2019

Machine speed during first DIN/GCode job

U, P

Sometimes ***WinPC-NC*** did not select the right machine speed during the first DIN/GCode job. Now ***WinPC-NC*** uses the right speed selections.

Correction for minimum laser power

U, P

Correction for minimum laser power in the technology functions of ***WinPC-NC***.

Version 3.02/05 Date: 29.01.2019

Progress bar with decimal places

U, P

With this version job files with at least 10000 program lines will be displayed with two decimal places in the progress bar. The function „Move from“ accepts decimal places to continue jobs more accurate.

Version 3.02/03 Date: 23.01.2019

Adaption for Heiz keypad

L,U, P
The orientation of buttons in the JOG window will not change automatic during the usage of Heiz keypad.

Version: 3.02/02 Date: 21.01.2019

G92 command for 3D printing

U, P

In gCode files created from CURA the spending of filament is resetted by G92 command from time to time. ***WinPC-NC*** will now accept and interpret this command in correct manner.

Version: 3.02 /01 Date: 18.01.2019

Enhanced DXF import

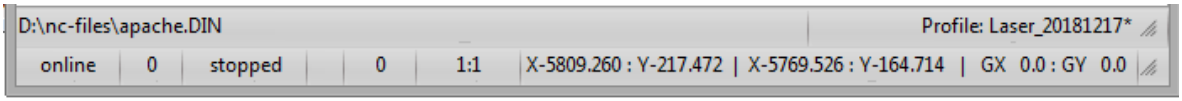
S, L, U, P

From this version on you can import much more DXF files and with more complex elements. We have realized the import of SPLINES, nested blocks and LWPOLYLINE elements.

Display of loaded profile and loaded work file

L, U, P

The loaded file with file path is now displayed in the bottom of the program window next to the name of the loaded profile. The profile default names the last saved parameter settings. A * symbolize not saved changes compared to the loaded profile.



Adjust a minimum spindle speed

L, U, P

WinPC-NC supports speed control of spindles with integrated minimum speed. A new parameter is added to the settings Basic Settings-Spindle to configurate the a minimum spindle speed.
The default value of this prarameter is 0 and should not be changed, if the frequency converter calculates a linear interpolation.

Security check to avoid pressing move buttons by mistake

L, U, P

Under Parameters-Misc-Display/Operation a new checkbox „Movebutton with confirmation“ is added. If this checkbox is activated, ***WinPC-NC*** shows a additional security window when clicking on move buttons. With this parameter the activation of „move to zero“, „move to parking place“ and „Homing“ by mistake should be prevent.
The already implemented parameter „Exit with confirmation“ has the same function for the exit button.

New functions for Laser

L, U, P

WinPC-NC offers new functions to control a cutting- or engraving-laser.

CAM functions in special function menu

S, L, U, P

The CAM functions for 2D data can be found in the menü under Special Functions-2D-CAM.