

Parameter settings

In **WinPC-NC Starter** are the most parameters predefined. A few settings must be set for milling, drilling and cutting. All parameters are summarized in one window.

Tools	Depth	Plunge sp	Feedrate	Spindle speed
1	1.00 mm	5.00	5.00 mm/Sek	12000 U/Min
2	1.00 mm	5.00	5.00 mm/Sek	8000 U/Min

Start/endposition: Park position

Zero point in file: origin of co-ordinates

Language: english

Scaling factors: 1.000

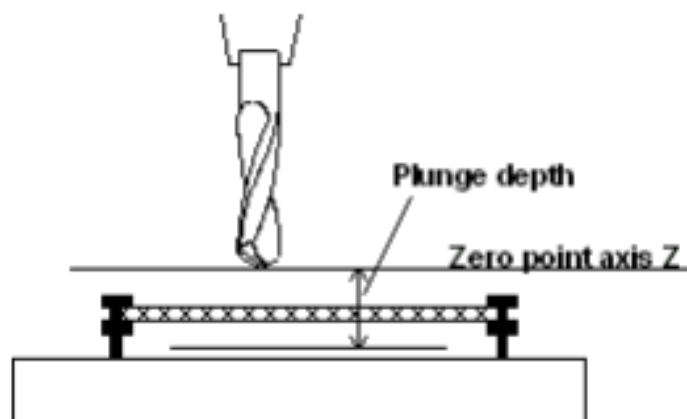
Toollift: +1.00 mm

Buttons: OK, Abort, Save...

Parameter Basic 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.

Feedrate

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.

Spindle speed

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

Start/End position

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

There are 4 possible start and end positions :

<i>Stop</i>	WinPC-NC stops at the reference position after reference movement, and at the last coordinate after each job process
<i>Zero point</i>	The machine moves to the defined zero point after reference movement and after each job process.
<i>Park Position</i>	WinPC-NC moves to the defined parked position after reference movement and after each job process.
<i>Zero point and clearance distance</i>	The machine moves to the defined zero point and lifts up the Z-axis to the defined clearance distance.

Zero point in file

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

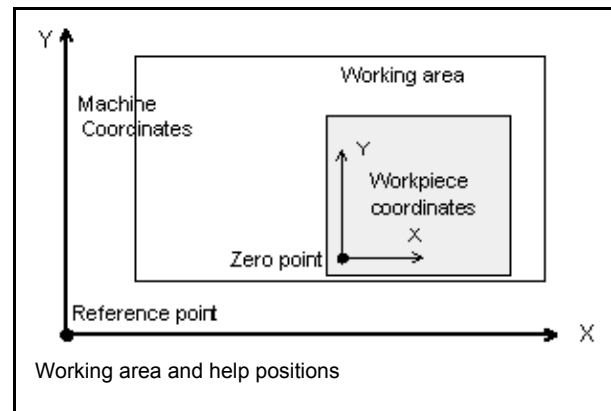
Six positions are possible:

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

*move to and
set zero point
manually*

The simplest way to define the zero point is with the JOG function. Therefore JOG to the desired position and save them as new zero point.

The position of the axis can be saved separately.



The park position can be defined at the same way.

Language

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

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

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

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

Scaling factors

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

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

Toollift

Additional distance for collision avoidance

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

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