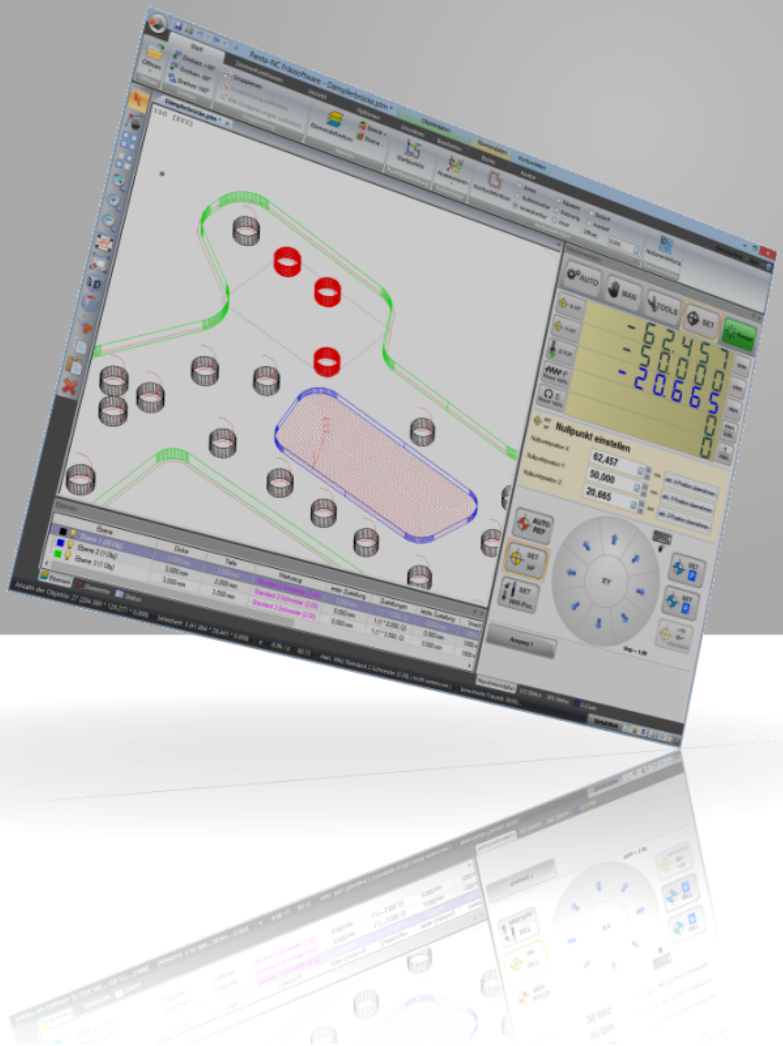
PENTA-NC

PENTA-NC User Manual



PENTA-TEC

© 2019 PENTA-TEC CNC-Automation
Lagerstrasse 1, A5082 Grödig
Mai 2019

Table Of Contents

© 2019 PENTA-TEC CNC-Automation

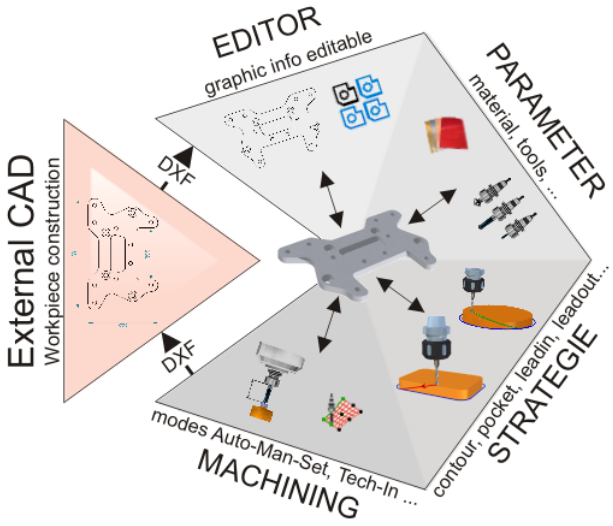
© 2019 PENTA-TEC CNC-Automation

1 Welcome To PENTA-NC

Our Solution:

PENTA-NC is a 2.5D Software offering the basic functionality of a graphic editor as well as powerful CAM-functions combined with a CNC controller.

Technology- and geometry-data are linked associatively. That means that PENTA-NC does not need a fixed sequential working order. Hence you can change both technology- or geometry-data whenever you like. These changes will be available for the machine without lengthy post processing or data transfer.



Advantages:

Optimisations are implemented directly into the production flow and can be verified on the finished component.

Whether you modify techological data, multiply geometry or change tools — you will quickly realise the possibilities and efficiency of PENTA-NC.

© 2019 PENTA-TEC CNC-Automation

2 Installation

2.1 Downloading The Latest Setup File

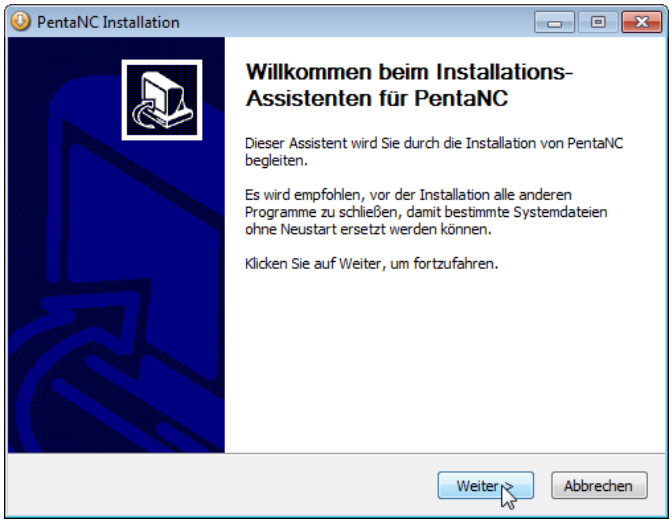
You will find the latest version of our PENTA-NC software on our homepage (http://www.penta-tec.com/download-penta-nc). The basic setup files will be specified using a numeric system as follows:

V2_ — Version number of PENTA-NC
11_ — Number increases with each implementation of an updated CAM library
6_ — Number increases with each implementation of a new EDING-CNC server
Build-1 — Number of the current build

Besides the basic setup file you will also find the latest update, specifying the build number. Additionally you can install a language pack if you require additional languages (German and English are included in the main build). The available languages will be listed next to the link.

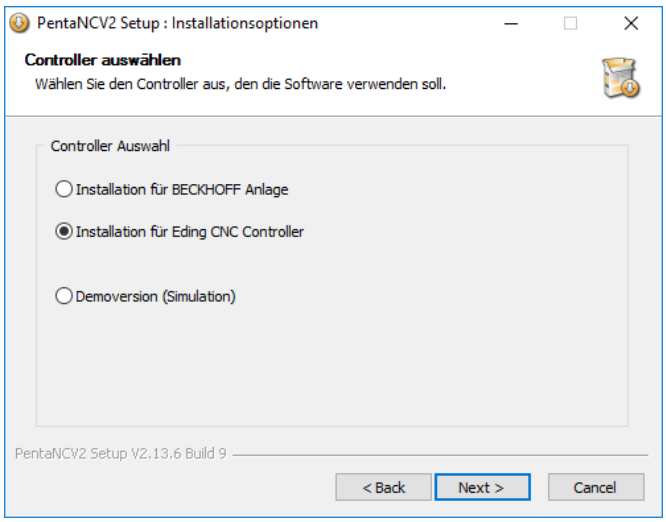
2.2 Installation

Execute the setup file which you previously downloaded.
Confirm any security queries by Windows and follow the instructions during setup. For this manual we chose the English setup option.



© 2019 PENTA-TEC CNC-Automation

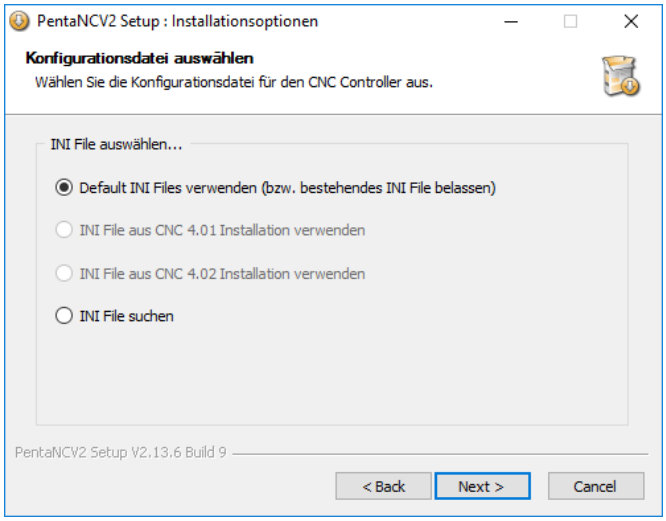
2.3 Choosing The Controller



Choose the according controller for your machine.
Alternatively, you may opt for the demo version which simulates a machine.

2.4 Choosing The CNC.INI File

EDING-CNC saves all machine-relevant details (size, axis configuration, I/O,...) in the file CNC.INI. PENTA-NC accesses this settings file in order to communicate with EDING-CNC. For this purpose, CNC.INI is saved — together with other relevant settings and configurations — in a dedicated folder (C:\ProgramData\PentaNCV2).



During installation you will have the following options regarding the inclusion of a specific CNC.INI file:

- Default INI Files verwenden (bzw. bestehendes INI File belassen)
  This is the default setting.
  If the software is being installed for the first time, a default INI file will be created. If you are simply installing a newer version of PENTA-NC, this option will keep the existing CNC.INI file. In that case you will not have to adjust all the settings anew.

© 2019 PENTA-TEC CNC-Automation

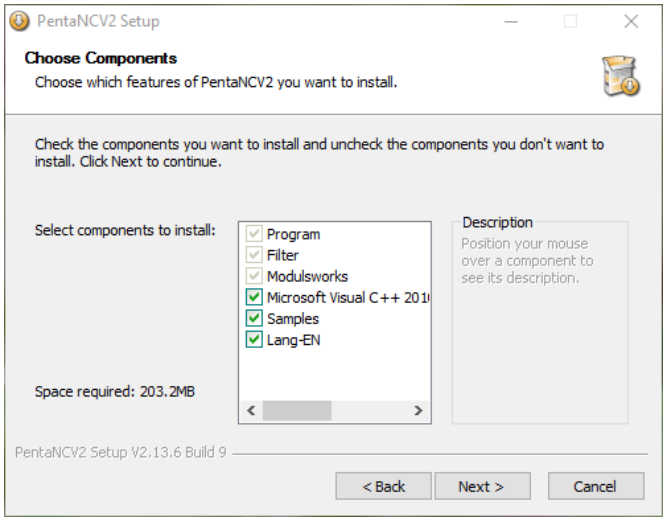
- INI File aus CNC Installation verwenden

  Use this option if you want to use a machine that has already been configured by EDING-CNC (starting from software version 4.01). This allows you to import an existing CNC.INI file from a previous EDING-CNC installation.

- INI File suchen

  If the manufacturer of your machine included a specially configured CNC.INI file you can choose this option to import that file.

2.5 Choosing Components



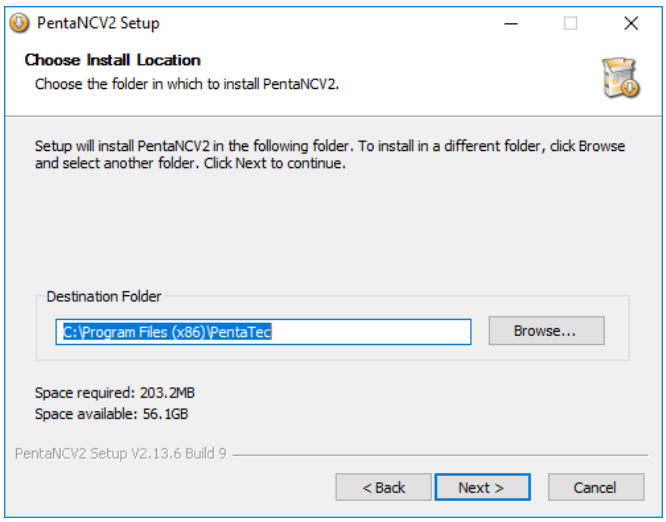
PENTA-NC requires Microsoft Visual C++ Runtime Libraries.

During your initial installation of PENTA-NC, make sure to include this option. If you simply upgrade from an older version of our software, you can uncheck the Microsoft Visual C++ Libraries. If in doubt, simply leave them checked — if they are already installed on your machine, the installation will offer to repair or upgrade your Runtime Libraries. If asked to do so, confirm this action.

You may also choose to include sample files (optional) and the English Language extension (optional, but recommended).

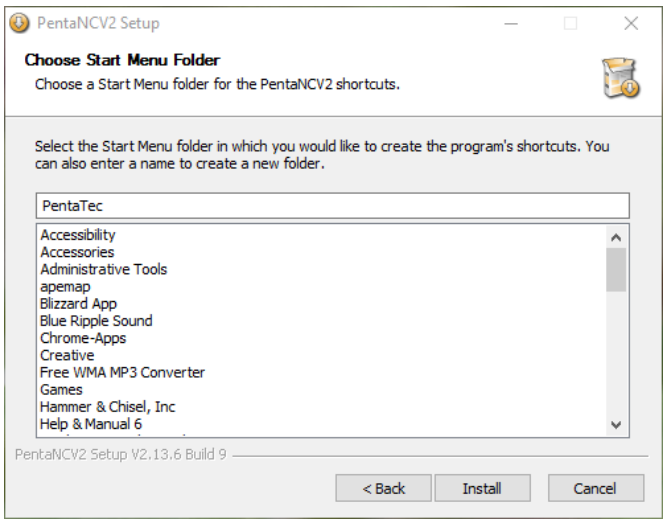
© 2019 PENTA-TEC CNC-Automation

2.6 Choosing The Target Folder



Choose the folder where you want PENTA-NC to be installed.

2.7 Choosing Start Menu Folder



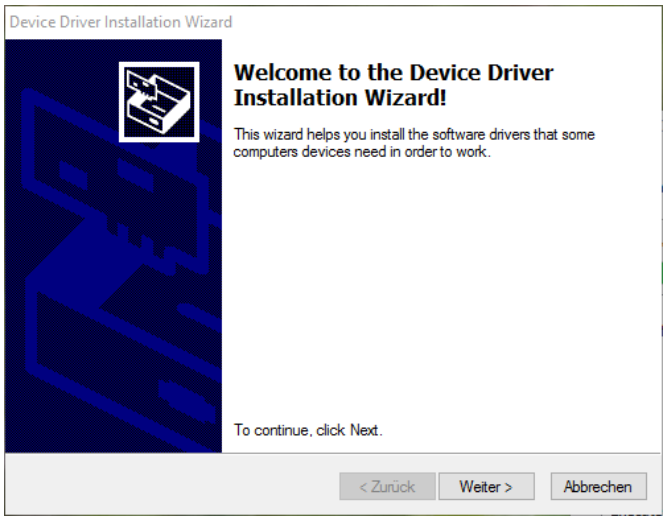
Finally choose the start menu folder.
Now that all the relevant settings have been decided, you can start the installation process.

© 2019 PENTA-TEC CNC-Automation

2.8 Installing The EDING USB-CNC Drivers

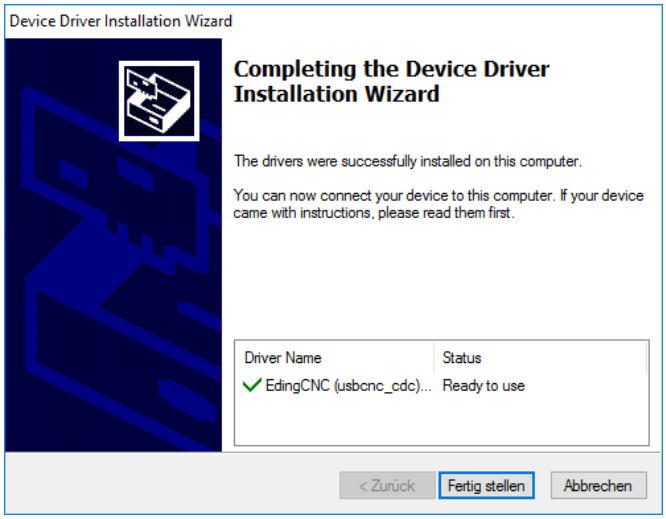
ATTENTION: *In order for the next step to work properly, it is paramount that the EDING-CNC controller is NOT connected to the computer. If it is currently connected, unplug the controller and only plug it in again AFTER the successful installation.*

After a short delay upon starting the installation of PENTA-NC, a new window will open to install the EDING USB-CNC driver.



- If you already have the EDING USB-CNC driver installed, you can cancel the installation by pressing "Abbrechen". *If in doubt, click "Weiter" to continue.*

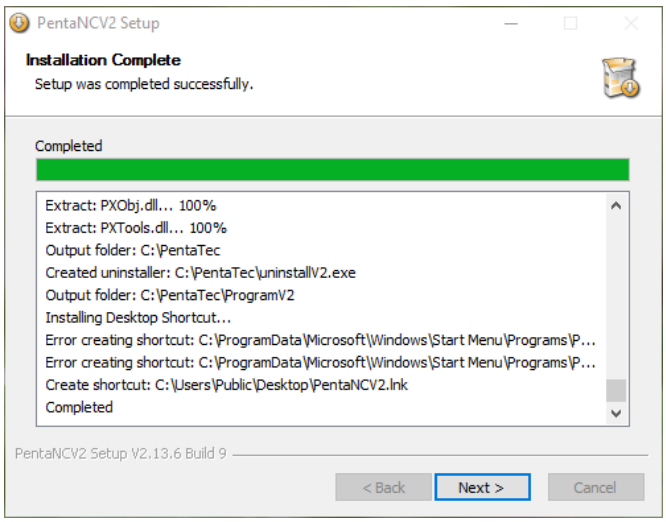
- If the EDING USB-CNC driver has not been previously installed, click "Weiter" to continue.



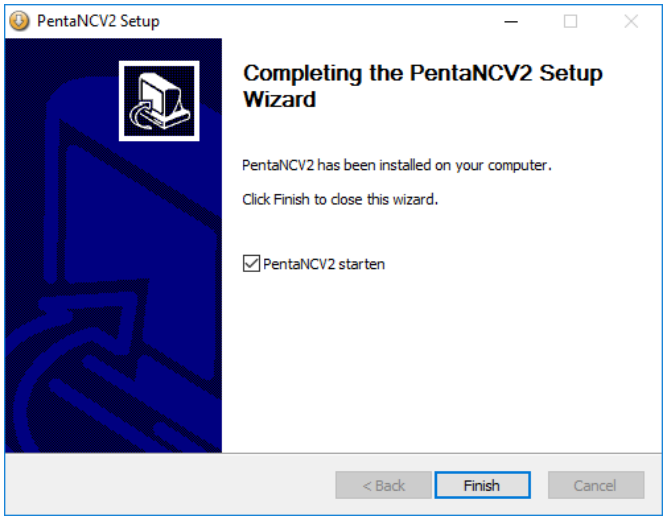
After a short installation process click on "Fertig stellen" to return to the installation of PENTA-NC.

© 2019 PENTA-TEC CNC-Automation

2.9 Finishing The Installation



PENTA-NC will finish the installation. If you included the Microsoft Visual C++ Runtime Libraries, they will be installed in the course of the process. You will need to confirm. Once the PENTA-NC installation is completed, press "Next" to continue.

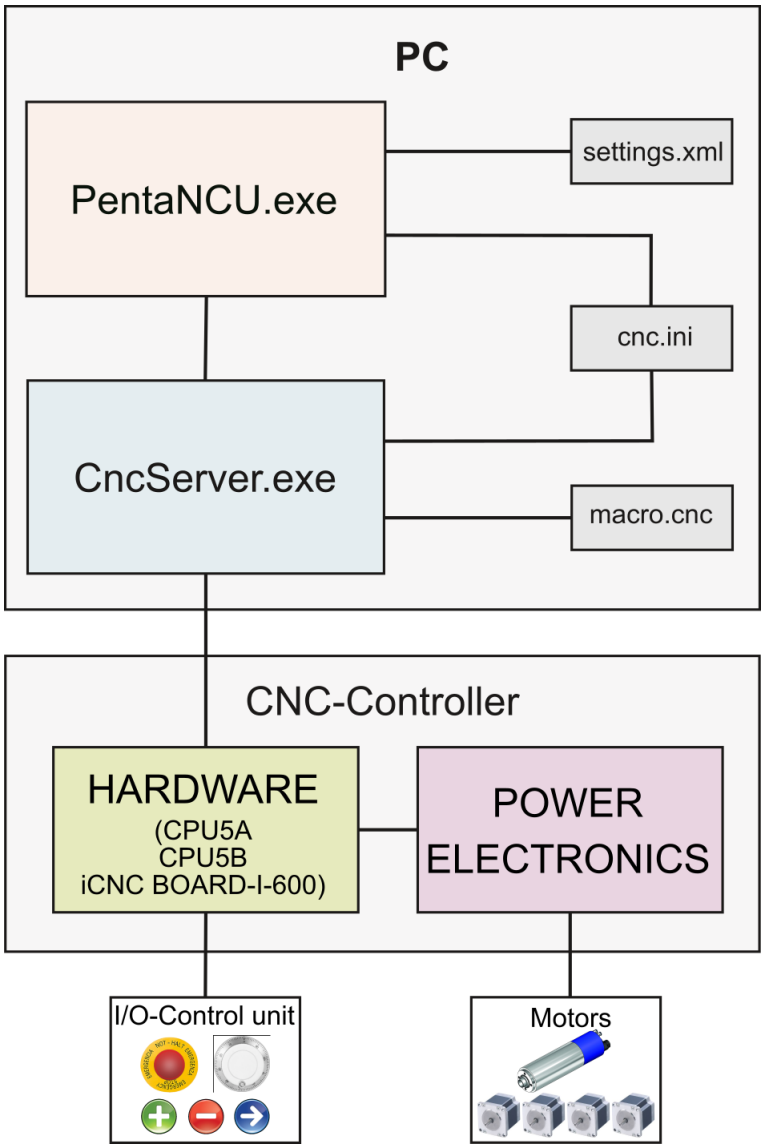


- You may now connect the EDING-CNC controller.
- If you want to install a language pack or patch PENTA-NC to a newer build, uncheck "PentaNCV2 starten" and patch after clicking on "Finish".
- Click on "Finish" to conclude the installation process.

© 2019 PENTA-TEC CNC-Automation

3 Basic Setup For EDING CNC Controllers

This diagram shows the schematic flow of communication in a CNC system with PENTA-NC and an EDING controller.
PENTA-NC doesn't communicate directly with the EDING CNC hardware but rather via the CncServer.exe process running in the background.

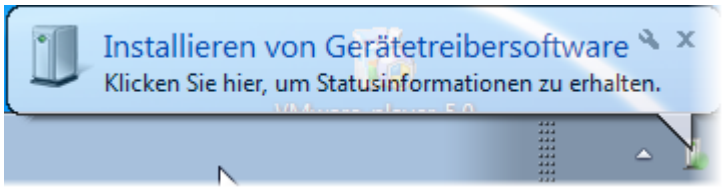


Communication between CNCServer and CNC-Controller can be achieved via USB or ethernet cable, depending on the specific controller type. The different methods to establish a connection are explained on the following pages.

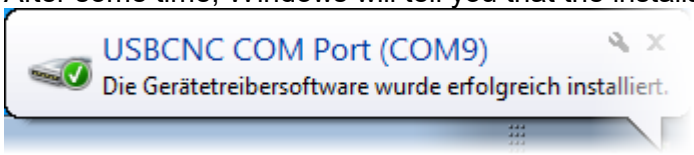
© 2019 PENTA-TEC CNC-Automation

3.1 Connecting Via USB

When you connect the USB CNC controller for the first time, Windows will automatically detect it and install the driver which was included in the PENTA-NC setup.



After some time, Windows will tell you that the installation was successful:



- The blue LED of the CNC USB port will light up to indicate that the electronics are provided with power.
- After initialising for a short time, the green LED will flash constantly on USB communication.
- If communicating via ethernet port, the yellow LED will flash instead.

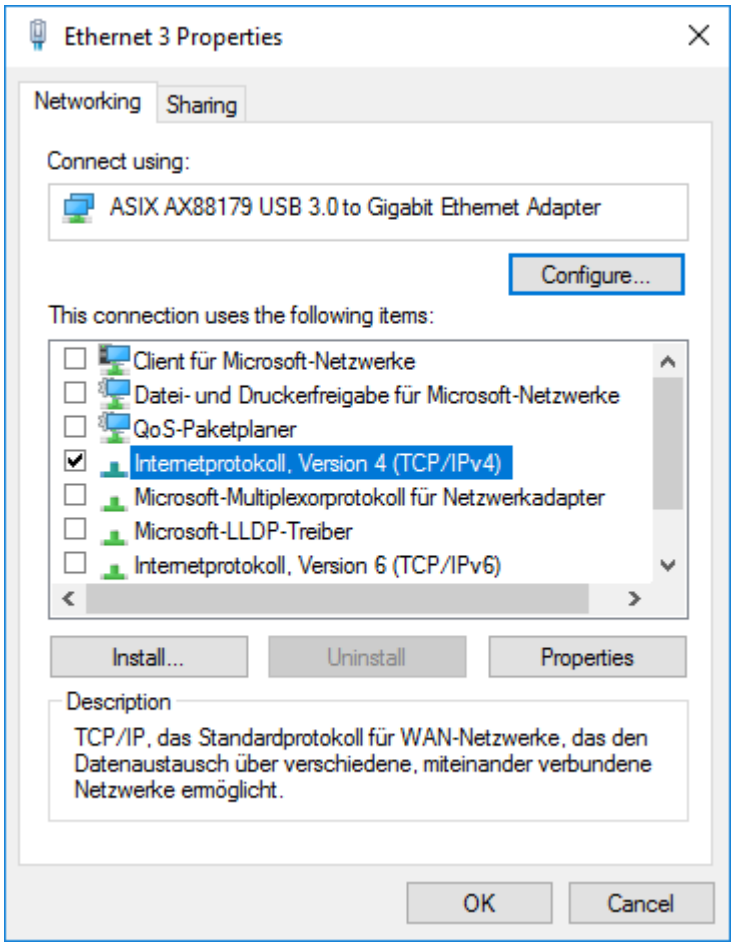
At this point, the CNC controller can be operated through PENTA-NC.

© 2019 PENTA-TEC CNC-Automation

3.2 Connecting Via Ethernet

- The connection via ethernet cable has to be direct, as opposed to using a hub or the like.
- The network interface card must support automatic crossover (most modern cards do). If it doesn't, you must use a crossover cable or adapter.
- The power supply of the controller must be properly connected and turned on.
- The USB connection between computer and controller is only used for firmware updates and should not be connected during regular operation.

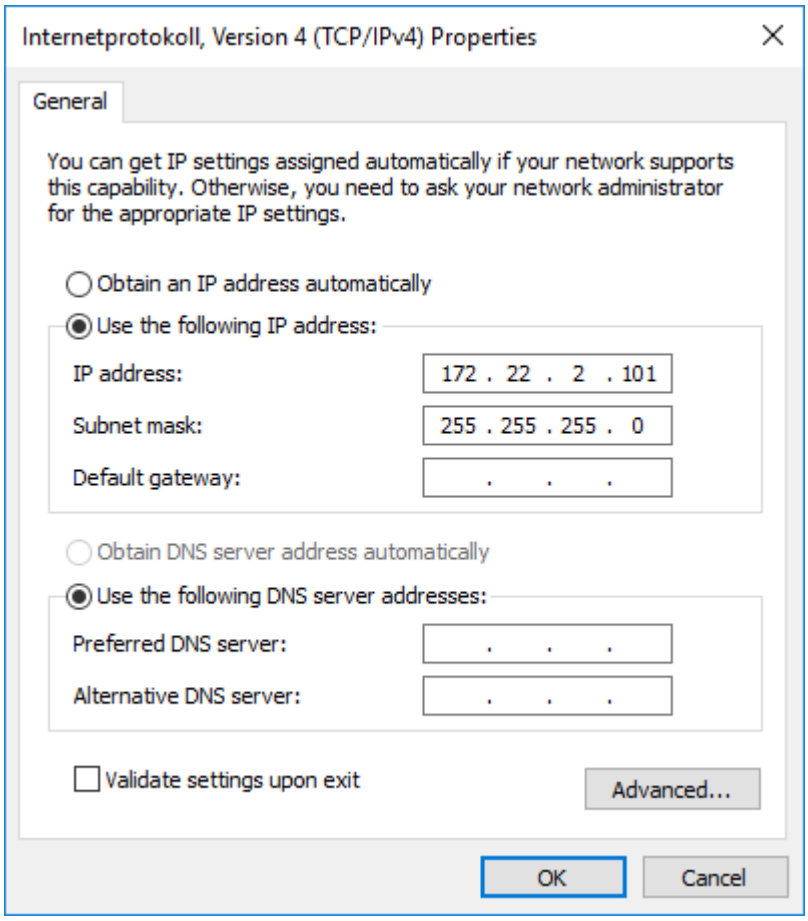
Network configuration for Windows:



Step 1:
Right-click on the corresponding network adapter and choose **Properties**.
Uncheck all connections except **Internet Protocol Version 4 (TCP/IPv4).**

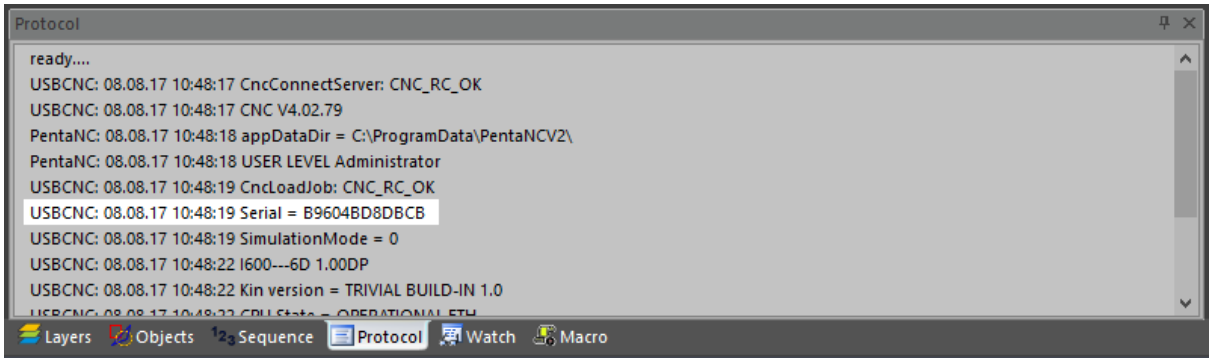
Step 2:
With **TCP/IPv4** activated, click on **Properties**.

© 2019 PENTA-TEC CNC-Automation



Step 3:
Assign the IP address 172.22.2.101 and the subnet mask 255.255.255.0 while leaving the default gateway and DNS servers empty.

Step 4:
Confirm the new settings by clicking **OK** and restart your computer.
Execute PENTA-NC.
Upon starting the program, the tab Protocol in the bottom of the window will display the serial number, firmware version and so on.

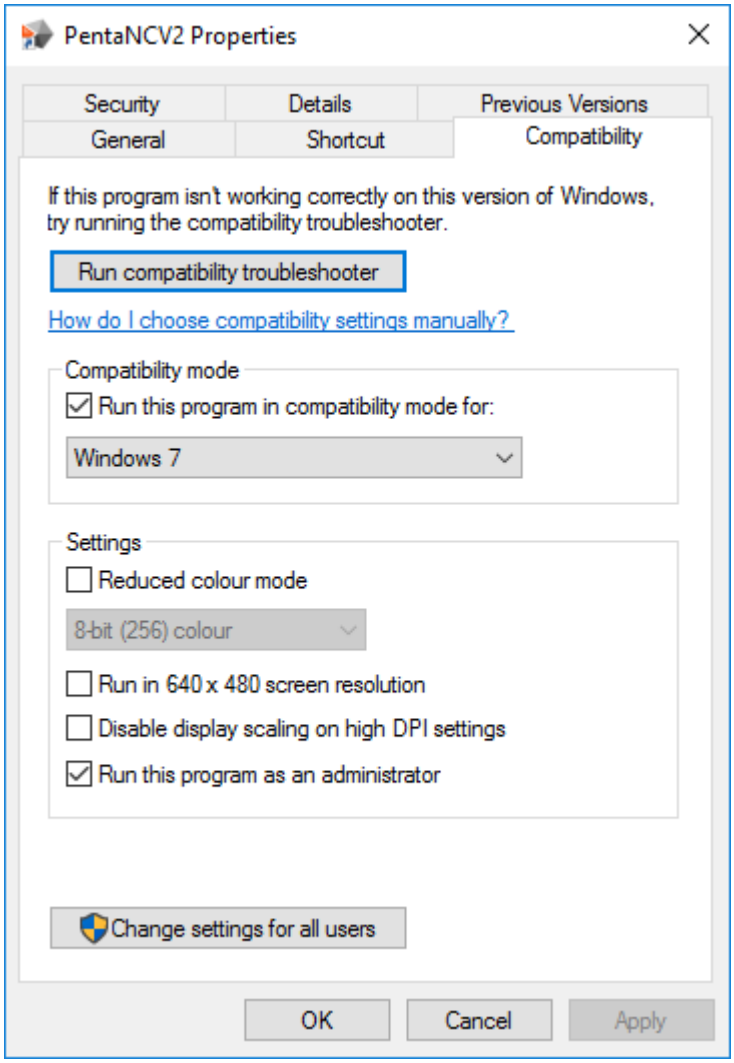


If the serial number shows 00000000, it indicates that there is no communication between computer and controller. Make sure to check the physical connection as well as the configuration.

© 2019 PENTA-TEC CNC-Automation

3.3 Starting PENTA-NC

In order for a flawless PENTA-NC experience, make sure to adjust the program settings accordingly.
Right-click the PENTA-NC desktop shortcut and go to **Properties**. In the properties window choose the tab **Compatibility**.



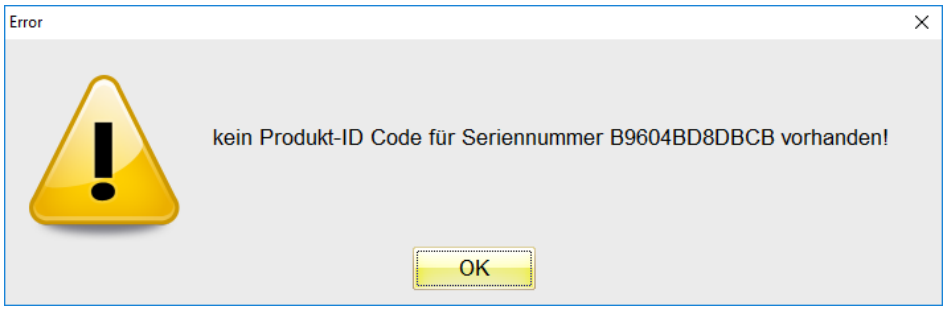
- The software was programmed under Windows 7. While it should work fine with newer versions, runtime errors may occasionally occur for some users. To avoid that, please make sure to run the program in **compatibility mode for Windows 7**.

- Due to the machine control being time critical, the program must be run in real time. In order to assure that, check the option **Run this program as an administrator**.

Confirm these changes by clicking on **OK** and then start PENTA-NC by double-clicking its desktop symbol.
You should see the PENTA-NC start screen momentarily.

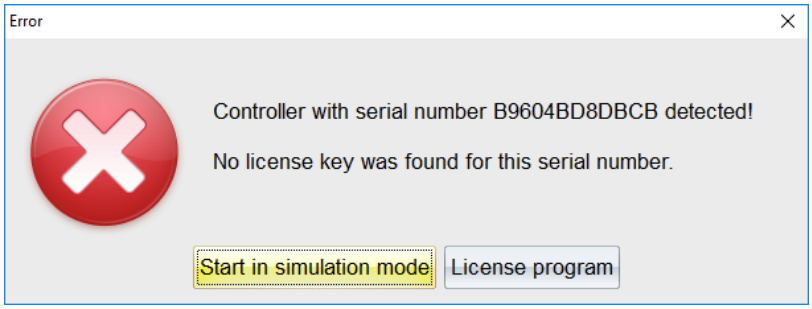
Upon your first start you will likely be reminded, that you have no valid product ID.

© 2019 PENTA-TEC CNC-Automation



Simply click on **OK** to proceed.

After that you will get the option to **Start in simulation mode** or **License the program**.



If you simply want to test the basic functionality of the software, you may proceed in simulation mode. In simulation mode, the software will not communicate with your CNC controller, but you can create milling files, material data, tool data and so on. You can also simulate the milling process on your computer.

If you wish to license the software, simply click on **License program** instead.

© 2019 PENTA-TEC CNC-Automation

3.4 Licensing PENTA-NC

If you wish to license the PENTA-NC software after choosing to use it in simulation mode, you can do so at any time by clicking on the question mark in the top right corner.

You will see a small window pop up with information about the current version of PENTA-NC as well as your controller's serial number and the license status.

In this case, the software isn't licensed. In order to get your product ID simply copy the serial number and send an e-mail to license@penta-tec.com, including the size of your machining area in square meters. Licensing fees are dependent on machine size.
If you are interested in extra modules (like engraving, restricted areas, etc.) simply state which ones you wish to purchase. We will gladly send you an overview of our ever-growing list of our modules.

After your e-mail has been processed, you will be provided with your personal product ID. Click on **Change product ID** to enter the key.

Upon successful registration of your product, the window will display that the software is now licensed.

© 2019 PENTA-TEC CNC-Automation

3.5 Basic EDING-CNC Settings

If your language isn't set to English (or your personal preference), click on **Optionen** in the top bar. Then click on **Sprache** (language) and click on the language of your choice. To be on the safe side, you may want to restart PENTA-NC to make sure all the menus will be switched to your chosen language.
For this manual, we assume you choose English.

Next you will need to configure the basic EDING-CNC settings.

Click on the PENTA-NC symbol in the top-left corner. 
Then choose **User Settings**.

Click on **Eding CNC** to access the mechanical settings for the EDING controller.

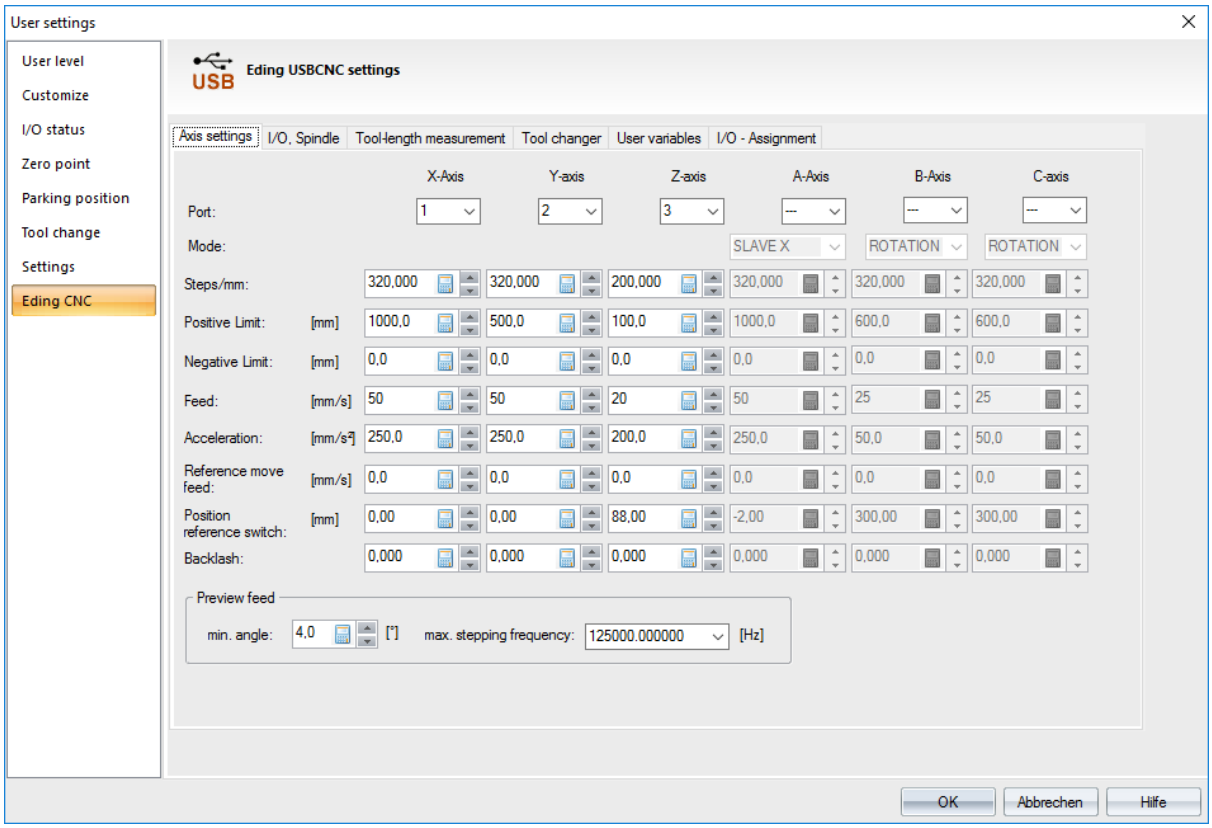
If some of the menus still appear in German, please re-start PENTA-NC.

You will find detailed info on most topics on our Penta-Tec youtube channel.

© 2019 PENTA-TEC CNC-Automation

3.5.1 Axis Settings

Under the tab **Axis settings** you will find the three main axes X, Y and Z, as well as the auxiliary axes A, B and C, all arranged in columns.
Depending on the used axis controller (three, four or six axes) the auxiliary axes will be available for different tasks.



Port:
Assign physical ports to the logical axes (X,Y,Z,...). Axes not in use should not be assigned any port.

Mode:
Allows the choice of special functions for your auxiliary axes.

- ROTATION (default)
  The axis is a rotation axis.

- SLAVE X, SLAVE Y, SLAVE Z
  The axis is defined as slave to X, Y or Z axis. Used for Gantry machines with two motors.
  ATTENTION: Tandem axes need a special procedure when referencing. This only works with two motors and two discrete reference switches.

- 4AX_MILLING
  Not yet implemented.

- 2ND_Z AXIS
  For controlling a second Z axis.

Steps/mm:

© 2019 PENTA-TEC CNC-Automation

- This parameter defines the amount of steps needed to move the axis by one millimeter.

  Example: You are using a step motor with 200 steps / revolution (1.8° per step).
  - *The stepping motor end phase is set to 1/16 micro-step mode.*
  - *The motor axis is coupled directly to a spindle with a 5 millimeter pitch.*

  In order to calculate the steps needed for a movement of one millimeter, we need to multiply the steps with the amount of micro steps and divide by the pitch:
  200 x 16 / 5 = 640 steps / mm

To reverse the direction of an axis, simply use a negative sign in front of the value (-640).

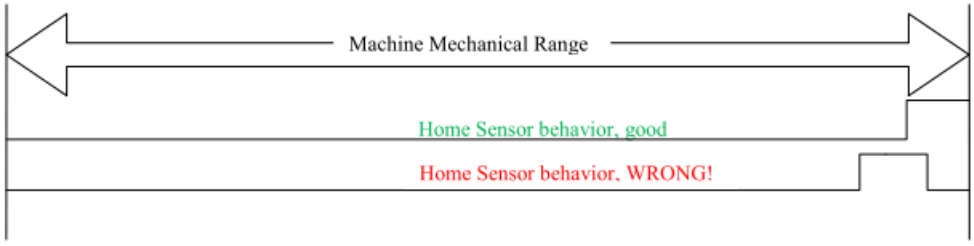
Positive Limit:
- Maximum machine position (correlates with the maximum permissible travelling distance of the corresponding axis).

Negative Limit:
- Minimum machine position (cannot be <0 because PENTA-NC does not allow negative machine coordinates).

Feed:
- Maximum permissible speed of the axis in mm/sec. All axis movements will be limited to this speed.

Acceleration:
- Maximum permissible acceleration of the axis. If you use the same value as in **Feed**, it will take one second to reach maximum velocity. Double the value to reduce acceleration time to 0.5 seconds, quadruple it to accelerate in 0.25 seconds, and so on.
  Experiment with acceleration to find your optimum settings, allowing for quick acceleration without jerking the machine too much. (Feasible values for initial tests are around 100-200 mm/sec².)

Reference move feed:
- This defines how quickly the axis moves for referencing.
- Choose a value that allows quick referencing without risking that the axis cannot stop in time to avoid a mechanical collision after reaching the reference switch.
- If the axis moves in the wrong direction, change the orientation with a negative sign in front of the value. (Keep in mind that this works only for reference feed and not the regular feed which is always positive.)
- A **value of 0** means **no automatic referencing** will be done. The axis must be **referenced manually**.
- Make sure that the axis cannot pass the switch due to high velocity or bad reference switch placement. **Once the switch gets activated, the axis must be able to stop before passing the switch entirely.**



© 2019 PENTA-TEC CNC-Automation

Position reference switch:

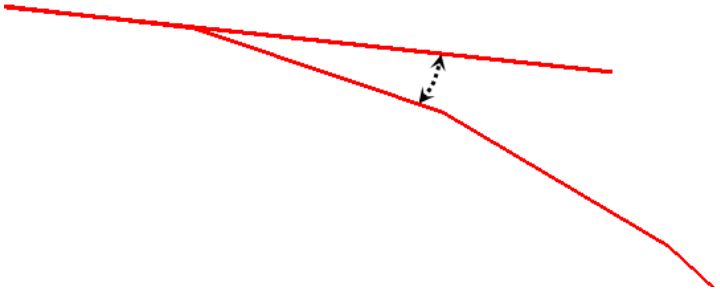
- Position at which the reference switch will be activated. It defines the machine coordinate system. The switch can be positioned at the positive or negative limit.

Backlash:

- If at all possible, a mechanical solution (for example by using a rotary ball screw spindle) should be pursued. Any software backlash solution is a last resort and will always have its flaws.

Preview feed:

- The **minimum angle** defines up to which angle the movement should not be slowed down. If the value is set too high, jerks and velocity peeks may occur. Start with a value of 3°- 6° and run a few tests to find the optimum angle setting for you.

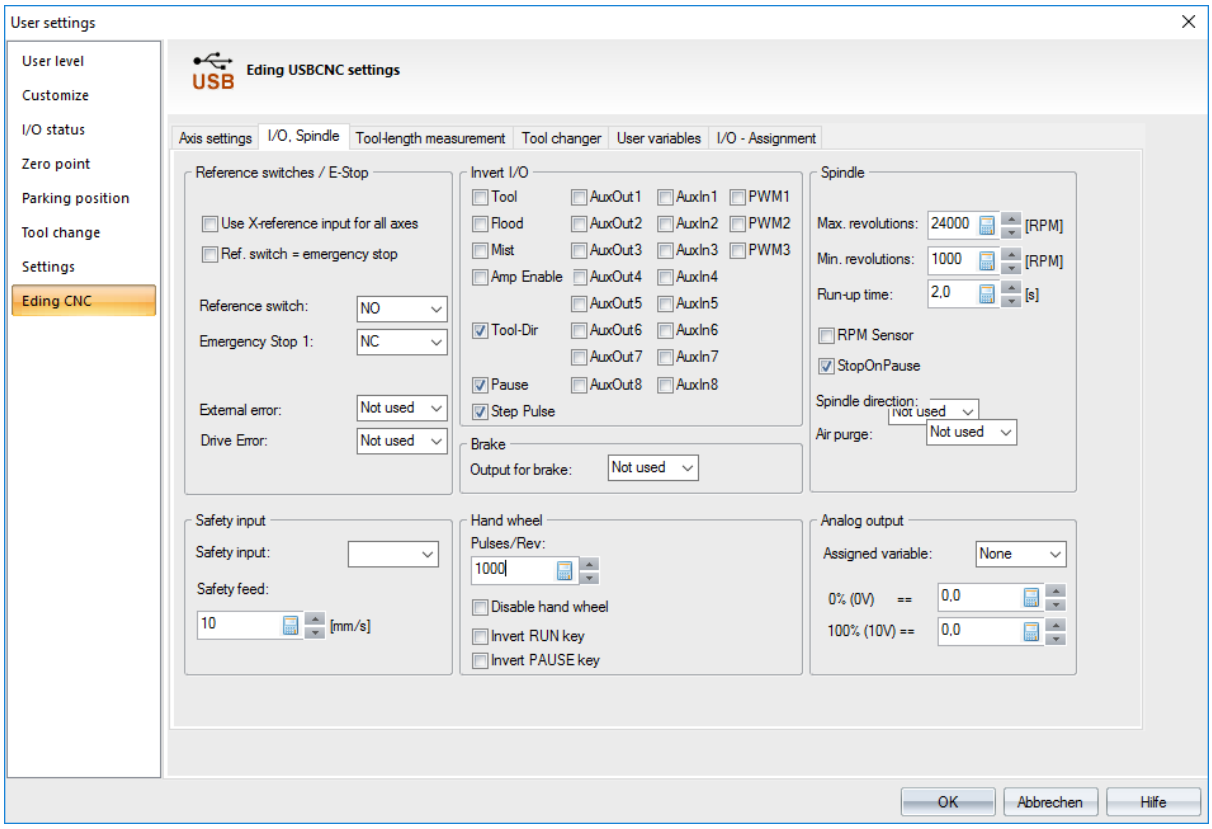


- The **max. stepping frequency** must be set in accordance to your end phase and can be found in the corresponding manual. Usually a value of 125kHz should work with most products.

© 2019 PENTA-TEC CNC-Automation

3.5.2 I/O, Spindle

Under the tab **I/O, Spindle** you can configure input, output and your spindle parameters. Input and output includes the reference switches and the hand wheel, among others. Depending on your controller, you may have considerably fewer options than in our screen shot below.



Reference switches / E-Stop:

- Use X-reference input for all axes
  If all your reference switches are in serial connection and thus assigned to only one input signal, check this box.

- Ref. switch = emergency stop
  Lets you define whether your reference switches trigger an emergency stop. This option is automatically deactivated during auto-referencing.

- Reference switch
  Choose between NO (normally open), NC (normally closed) or "not used".

- Emergency Stop 1
  Choose between NO (normally open), NC (normally closed) or "not used".

- External Error
  Allows you to use an external signal (like a thermal sensor) as error signal. Options are NO, NC and "not used".
  In case of an error, the machine will be stopped.

- Drive Error
  If your stepper end phase is able to communicate errors, you can activate this option.
  In case of an error, the machine will be stopped.

© 2019 PENTA-TEC CNC-Automation

Safety input:

- Safety input
  Lets you define an auxiliary input as safety input. When activated, the feed will be limited by the value defined under "Safety feed".
  The currently running G-code will be paused.
  Possible uses include connecting the safety input to a cover flap to minimise the risk of injuries.

- Safety feed
  Limits the feed to the value defined here, once the safety input is activated.

Invert I/O:

- Checking any of these boxes will invert the corresponding signal. This is a quick and easy way to get rid of some common problems.

- Tool
  If the spindle starts spinning when the software starts, simply check this box to invert the signal.

- Flood
  Inverts the signal for liquid cooling.

- Mist
  Inverts the signal for spray cooling.

- Amp Enable
  If the machine does not seem to function at all, the amplifiers may not be activated. Simply invert this signal.
  A good indicator for this problem is if the machine starts humming or buzzing when closing PENTA-NC and goes quiet upon starting the software.

- Tool-Dir
  Invert the direction the spindle turns. Only works if the relevant input of the frequency converter is in use.

- Pause
  Inverts the signal for pause.

- Step Pulse
  Inverts the signal for the stepper end phase. This does not change the direction, it simply changes a rising trigger to a falling trigger and vice versa.

- AuxOut / AuxIn
  Inverts the signals for the auxiliary outputs and inputs.

- PWM1, PWM2, PWM3
  These are analog outputs. PWM1 is assigned to the spindle.

© 2019 PENTA-TEC CNC-Automation

Brake:

- Lets you choose one of the auxiliary outputs to activate the brake if desired.

Hand wheel:

- Pulses/Rev
  Define the amount of pulses per revolution. Depending on the model of your hand wheel.

- Disable hand wheel
  Lets you disable the hand wheel.

- Invert RUN/PAUSE key
  Setting depends on the model of your hand wheel. When using the standard hand wheel "RCSmart", both keys need to be inverted for proper function.

Spindle:

- Max. / min. revolutions
  Define the maximum and minimum rotations per minute.

- Run-up time
  When the spindle is started, the G-code will pause for the ramp up time to avoid damage to tools, machine and work piece.

- RPM Sensor
  If you have an RPM sensor, check this box.

- StopOnPause
  If checked, the spindle stops on activating pause. Not to be confused with a stop or emergency stop.

- Spindle direction
  Depending on your controller system you may be able to change your spindle direction. You can choose an output to trigger the spindle direction.

- Air purge
  If your spindle possesses a sealing air solution, you can activate it with an auxiliary output.

Analog output:

- You can control your analog output by using a variable (IOREAL1-4) and defining the minimum and maximum values.

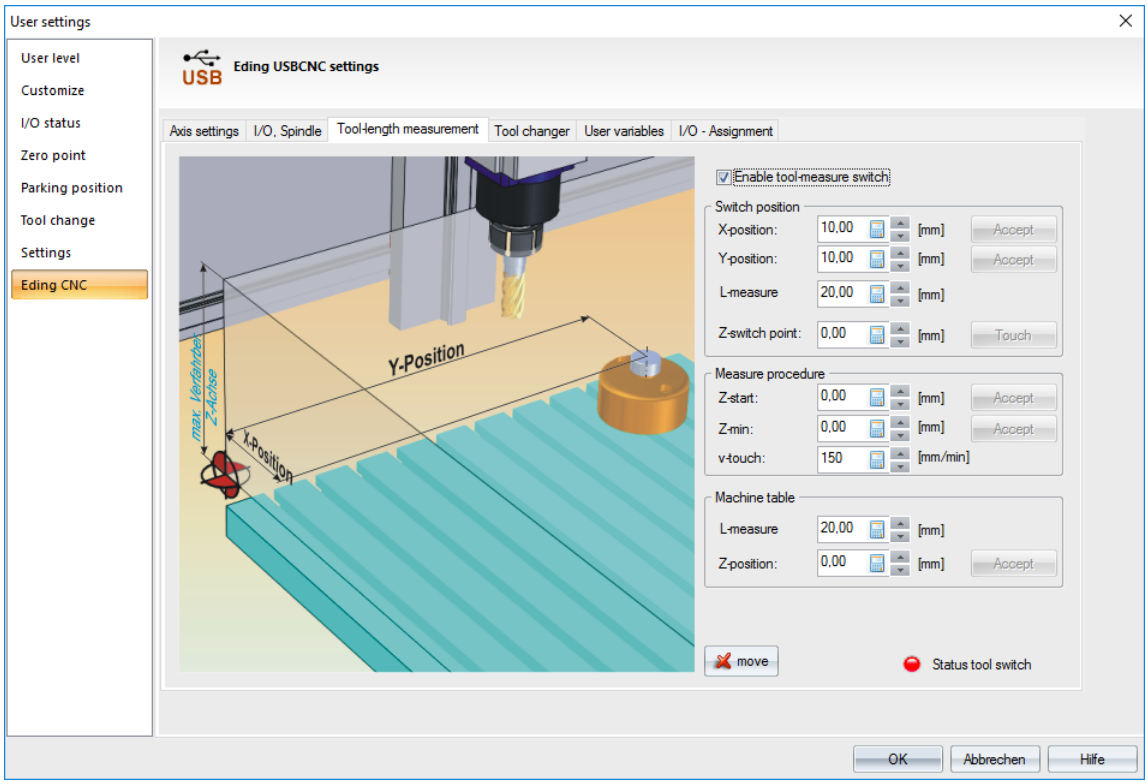
  Example:
  You want to control a pump using a pressure sensor. The sensor writes its signal into the variable IOREAL1 (choose in the pop-up menu).
  At 0 bar the pump should stand still (0% (0V) == 0), at 1.3 bar it should pump at full capacity (100% (10V) == 1.3).

  Once the sensor reaches (or exceeds) 1.3 bar, the pump will work at full capacity (signal strength 10V). Any value between 0 and 1.3 bar will be interpolated and the according signal strength will be sent to the pump.

© 2019 PENTA-TEC CNC-Automation

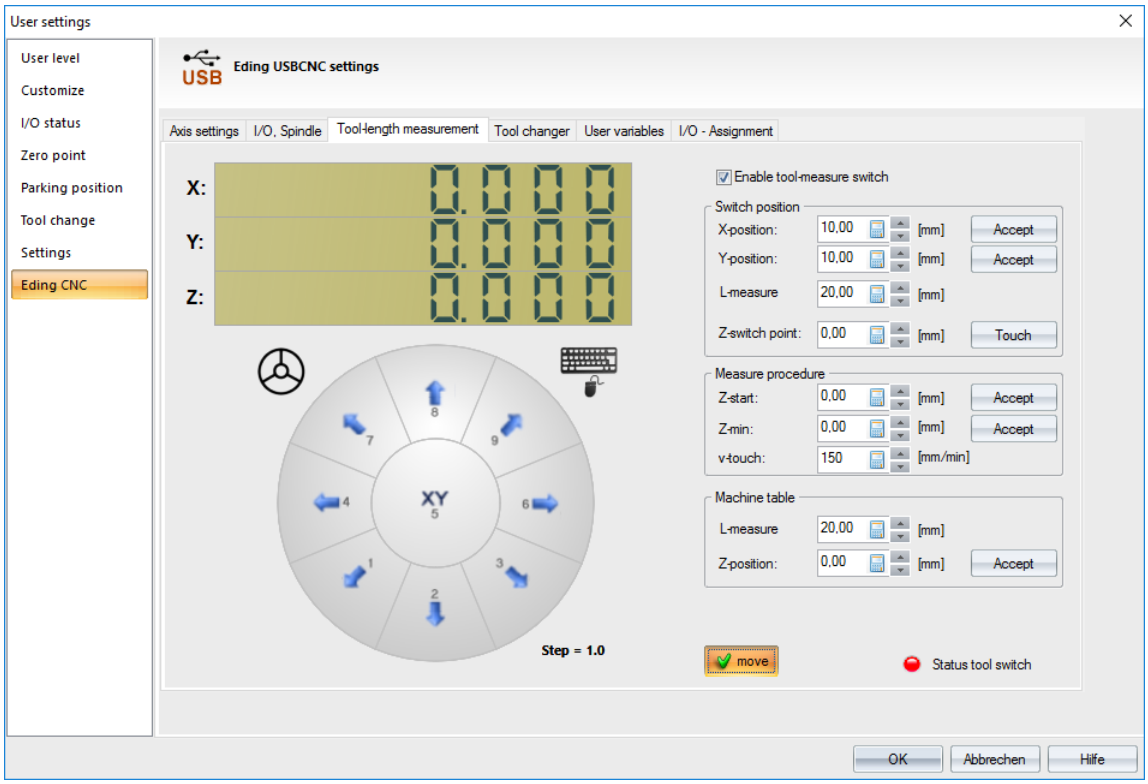
3.5.3 Tool-length Measurement

The tab **Tool-length measurement** lets you configure your tool-measure switch for automated measurement. If you do not have a tool-measure switch, simply disable the check box and ignore the tab.



At the lower right corner of the image, you will find the button **move**. Click it in order to access the configuration menu.

© 2019 PENTA-TEC CNC-Automation



Here you can move the tool to the tool-measure switch using either keyboard, mouse or hand wheel.
Once you have placed the tool directly above the switch, you can transfer the current X- and Y-coordinates by clicking on **Accept**.

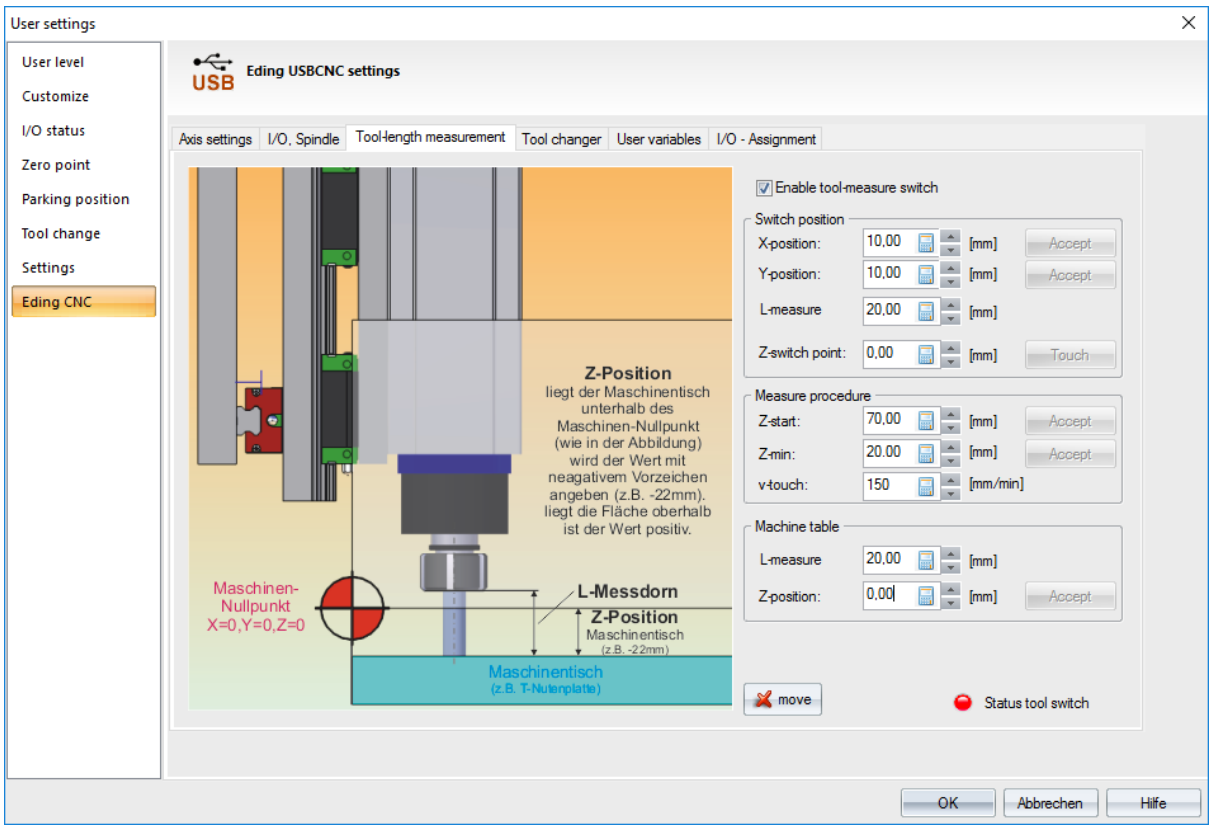
Before performing the next steps, ascertain that the measure switch is working properly. You can test this by manually triggering the switch.
Whenever it is activated, the red circle next to **Status tool switch** should light up. If that is not the case, check the connections and restart the software.

Next, jump to **Measure procedure** and choose a fitting start and end point for the measurement process.
Z-start defines the height at which automated measurement will commence. It should be high enough to allow for the longest of your tools to be measured safely. You can enter this value manually or position your spindle at the proper height and click **accept**.
Z-min defines the minimum height for automated measurement. If the switch didn't activate before reaching this position, measurement will be cancelled with an error message. You can enter this value manually or position your spindle at the proper height and click **accept**.
v-touch defines the speed used for automated measurement. Avoid speeds that could damage your tools or the switch. 150mm/min is a safe speed for most machines.

Now return to **Switch position**. In order to define the Z-coordinate of the switch, you can either use the spindle (without retainer nut) or a measuring pin. If you use a pin, measure the mounted pin and enter the length under **L-measure**.
Now you can start the measuring procedure. Simply press **Touch** next to **Z-switch point** and wait for measurement to be completed.
PENTA-NC will remind you that you need to switch to a measure bolt. Once you did that (or unscrewed the retainer nut to use the spindle for measuring) click on **OK**.
The software will automatically enter the Z-position of the switch once it is triggered.

© 2019 PENTA-TEC CNC-Automation

Machine table:
If the Z-position of your machine bed is not identical with the zero point of your machine coordinate system, you will now need to tell PENTA-NC the correct Z-position.



Info: The zero point of the machine coordinate system is the point up to which the spindle can move. If you can move all the way down without touching the machine bed, your machine coordinate system starts above the table. If it starts below the table, moving to the lower limit would result in a collision.

Enter the length of your measure pin under **L-measure**.

If you are not still in the movement section, click on **move**.

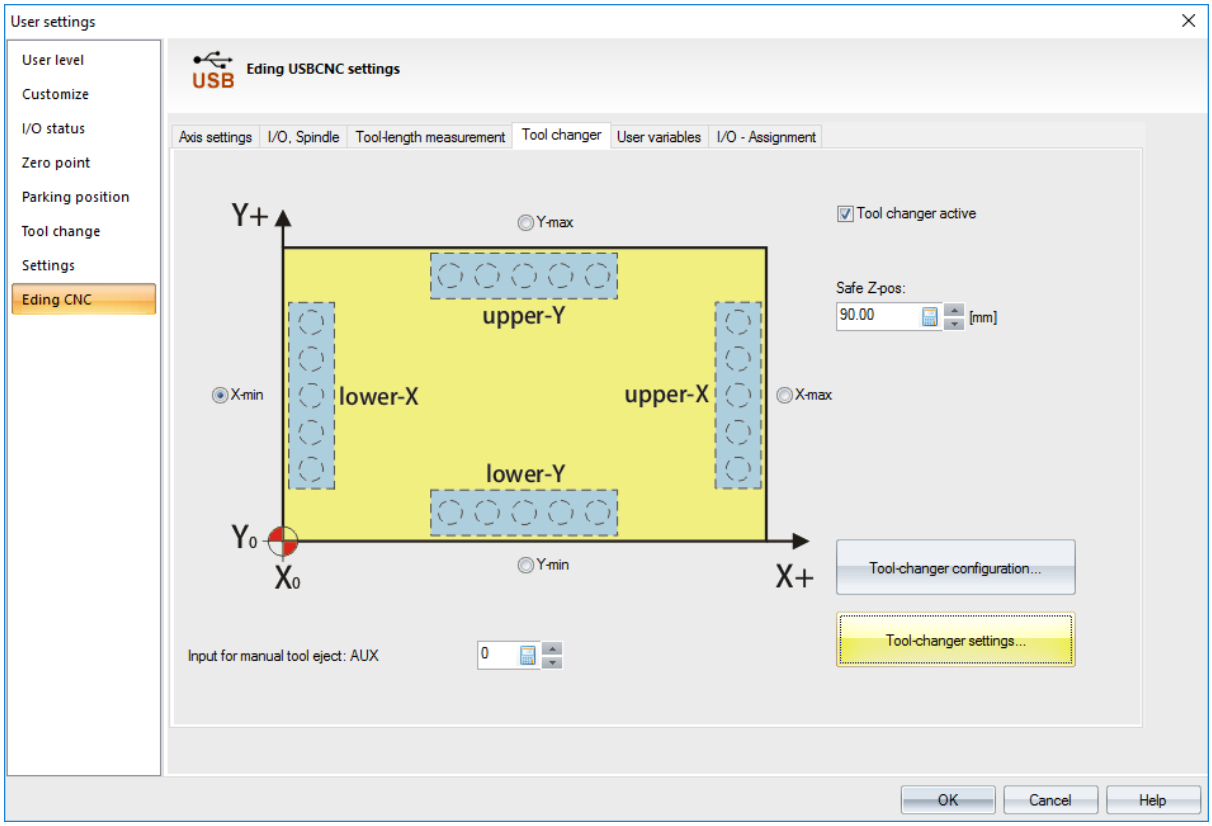
Navigate to a suitable position and then lower your measure pin. Use a sheet of paper on your machine table and slowly lower the pin until the paper cannot be moved freely underneath.
Now click on **Accept** next to **Z-position** and PENTA-NC will automatically enter the correct value. If your machine bed is below the machine coordinate system, this will be a negative value.

Once this procedure is finished, click on **OK** to save all details.

© 2019 PENTA-TEC CNC-Automation

3.5.4 Tool Changer

If your machine is equipped with an automatic tool changer, you can configure it here.



Tool changer active:
Check this box if you want to use a tool changer.

X-min, X-max, Y-min, Y-max:
Define where the tool changer is installed. Necessary for the automated tool-changing process.

Safe Z-pos:
Define from which height the whole tool-changing process shall be initiated. Make sure it is appropriate for your longest tool.

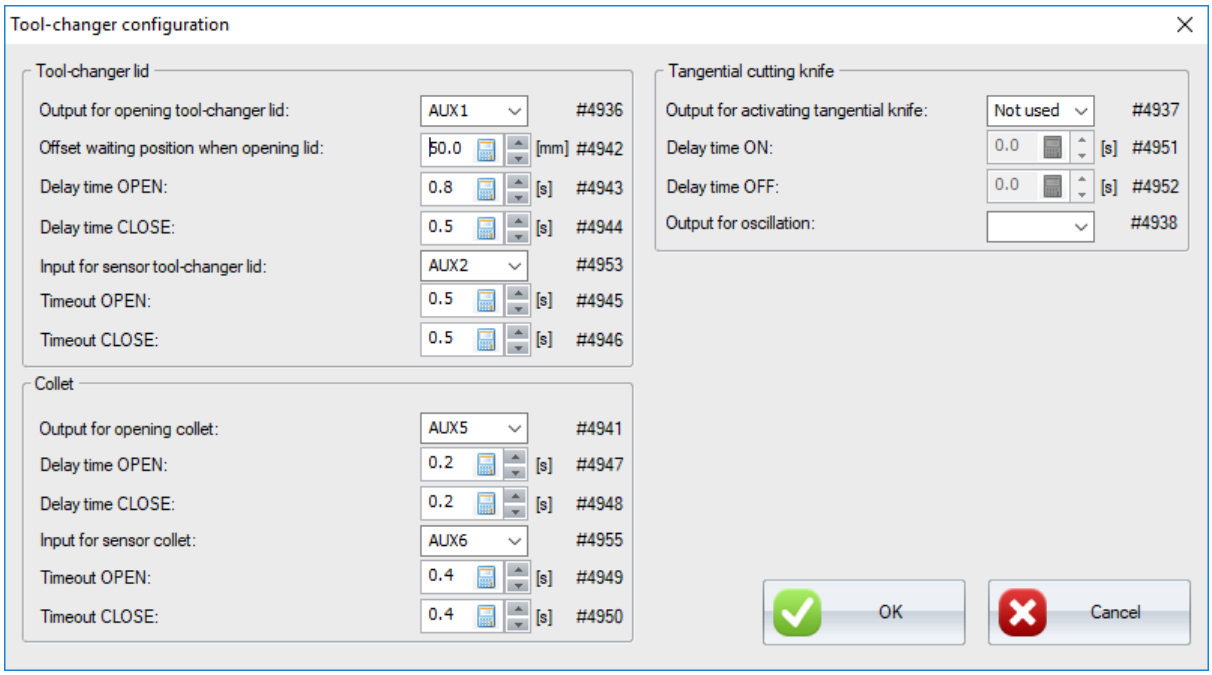
Input for manual tool eject:
Define an input for a switch that lets you open the collet in order to remove a tool manually.

Tool-changer configuration:
Configure general parameters of your tool changer (offset waiting position, delay, ...).

Tool-changer settings:
Define positions of tools in tool changer.

© 2019 PENTA-TEC CNC-Automation

3.5.4.1 Tool-changer Configuration



The tool-changer configuration allows you to define parameters for the tool-changer device.

Tool-changer lid:
Assign an AUX output for the tool-changer lid and define the offset waiting position while the lid is being opened.
If you position your tool changer at the left of the table (see **Tool Changer**), the offset position will be to the right of it. In our case, the machine head will wait at the coordinates of the tool to be dropped **plus 50mm in X**.
After a delay time of 0.8 seconds (allowing the tool lid to open) it will move to drop the current tool.

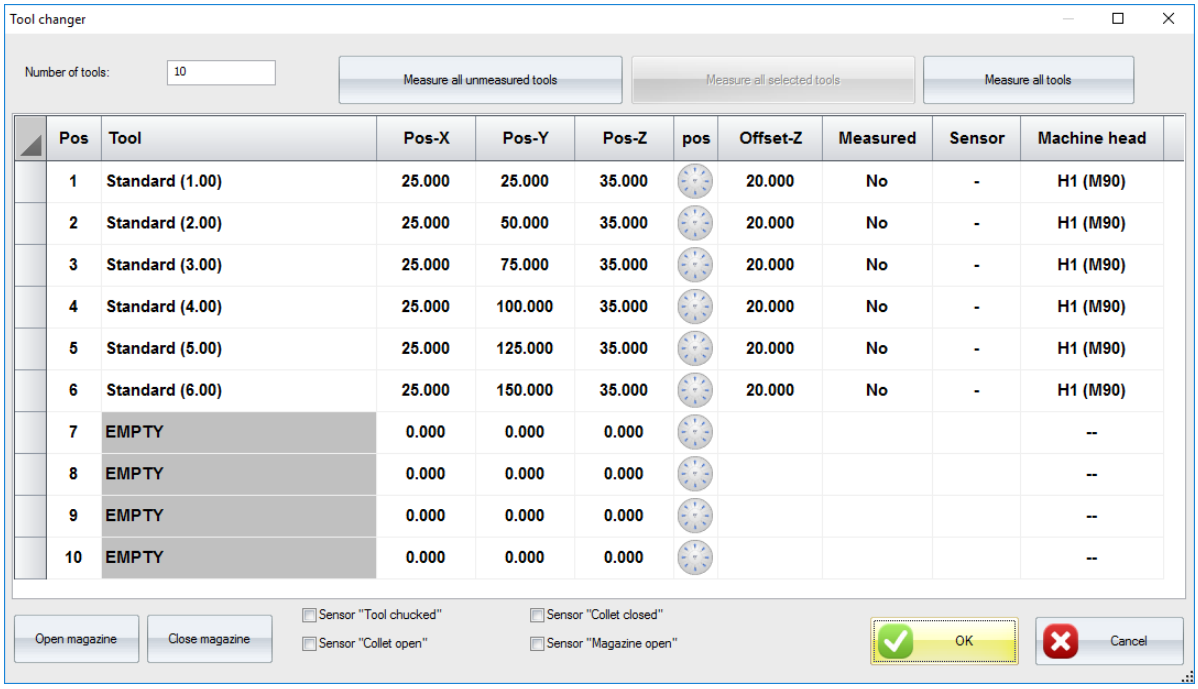
You may also assign an AUX input for the tool-changer lid sensor and define timeouts for a successful opening or closing procedure. If the sensor doesn't activate within this time frame upon the signal to open or close the lid, you will receive an error message and the tool change will be cancelled.

Collet:
Assign an AUX output for opening the collet. Define a delay between reaching the drop position and opening the collet, as well as reaching the pick-up position and closing the collet.

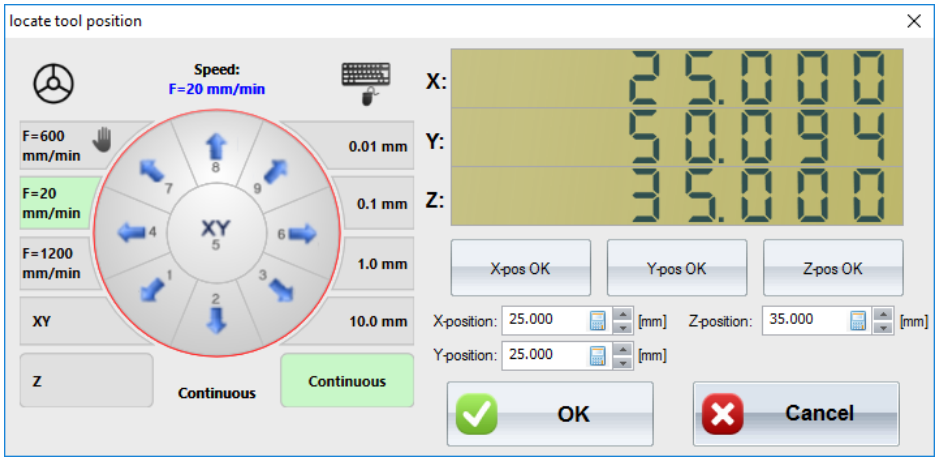
You may also assign an AUX input for a collet sensor. Define timeouts for opening and closing the collet.

© 2019 PENTA-TEC CNC-Automation

3.5.4.2 Tool-changer Settings



If the tools are highlighted red, make sure that you activated and defined the tool changer for the according tool positions in the **machine-head parameters** (see **Machine Heads**).
If everything is defined correctly, you can start entering the coordinates of each tool-changer position. Alternatively, you can teach the coordinates by clicking on the navigation pad symbols located between the **Pos-Z** and **Offset-Z** columns.



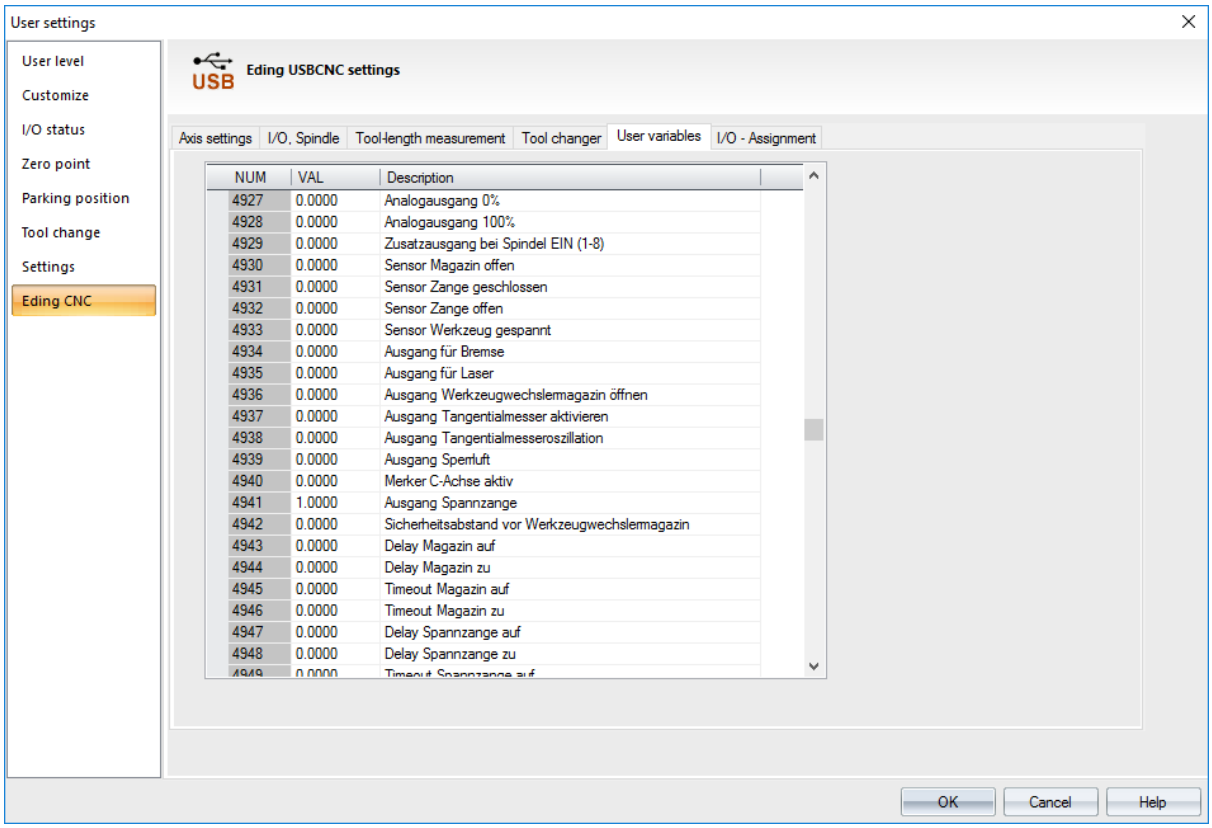
Position the machine head in the correct spot for each tool-changer position and accept the coordinates by clicking on **X-pos OK**, **Y-pos OK** and **Z-pos OK** buttons respectively. PENTA-NC will transfer the current values into the according boxes. Finally, confirm by pressing **OK**.

Do this for every position of your tool changer.

You can manually open and close the magazine from the main **tool changer** window and switch the sensors on and off.
Once you have defined all positions, confirm by clicking on **OK**. This will close the window and finalise your tool-changer positions.

© 2019 PENTA-TEC CNC-Automation

3.5.5 User Variables

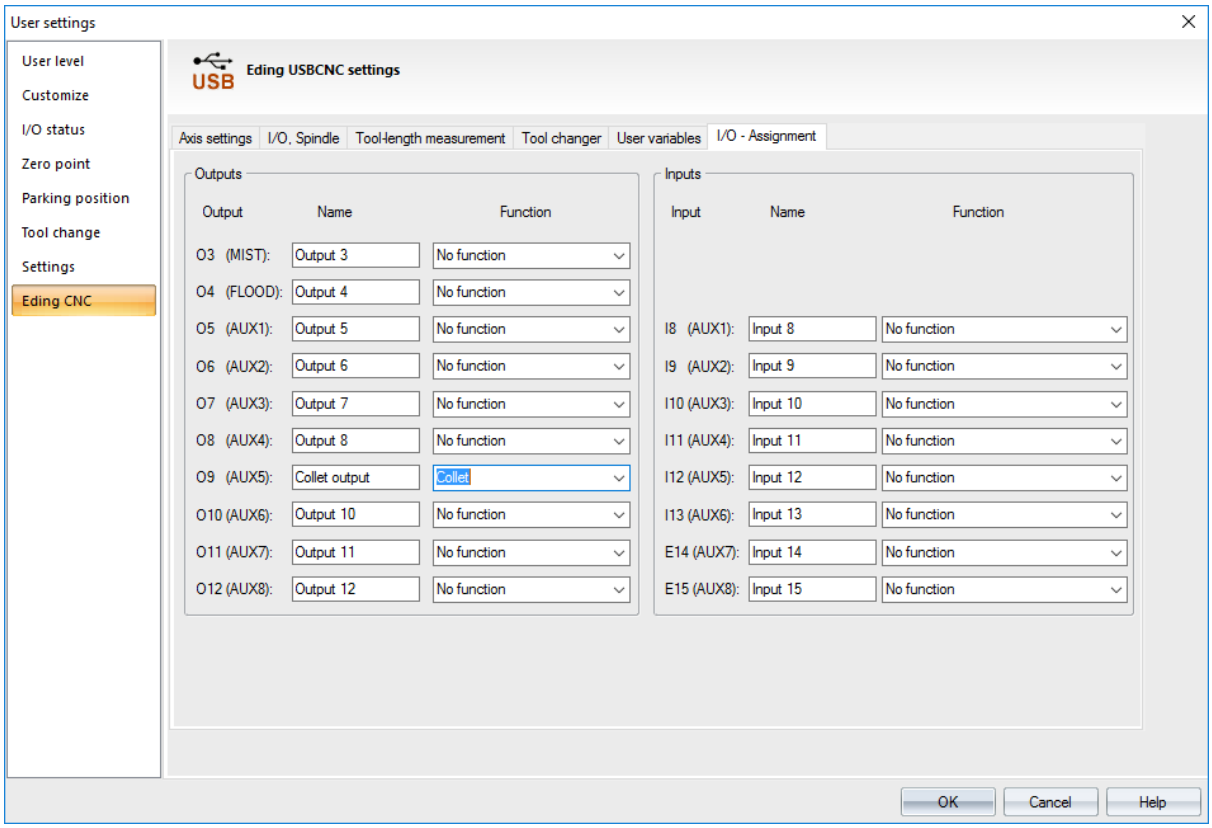


Do not change these values!

These variables are used internally for various functions and EDING macros and should not be tempered with. They are also very useful for trouble-shooting.
Unless you know exactly what you are doing, changing these values can lead to malfunction, resulting in damage to your body or machine or — in the best case — simply a non-functioning software and/or machine.

© 2019 PENTA-TEC CNC-Automation

3.5.6 I/O Assignment



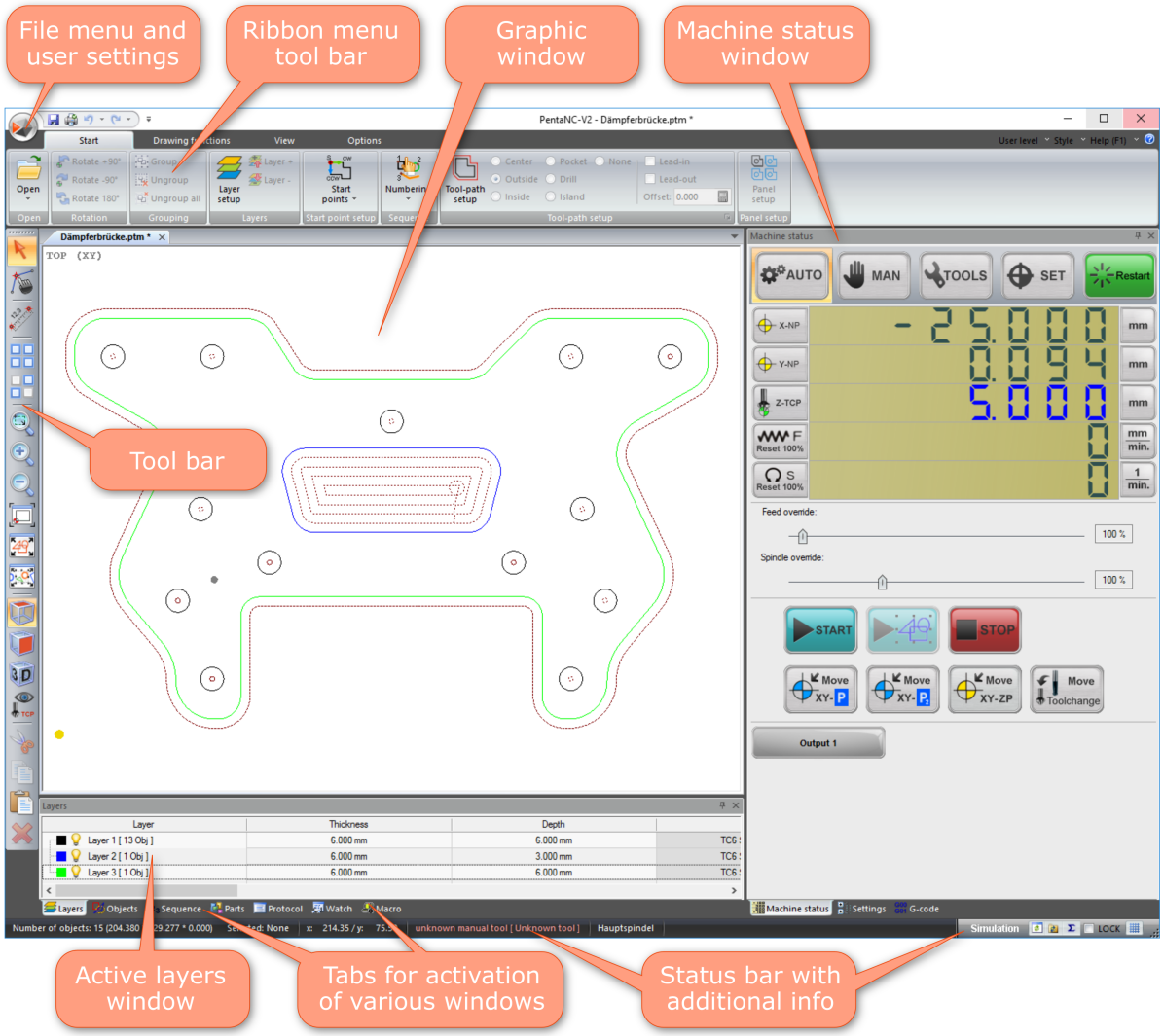
The I/O assignment lets you enter user-defined names and choose from a variety of pre-defined functions. This is useful if you have many inputs and outputs and want to see at a glance which is which.

In this instance, we defined our output AUX5 as the output signal for the collet.

© 2019 PENTA-TEC CNC-Automation

4 PENTA-NC User Interface

Upon starting PENTA-NC the first time, you will be greeted by the standard layout.

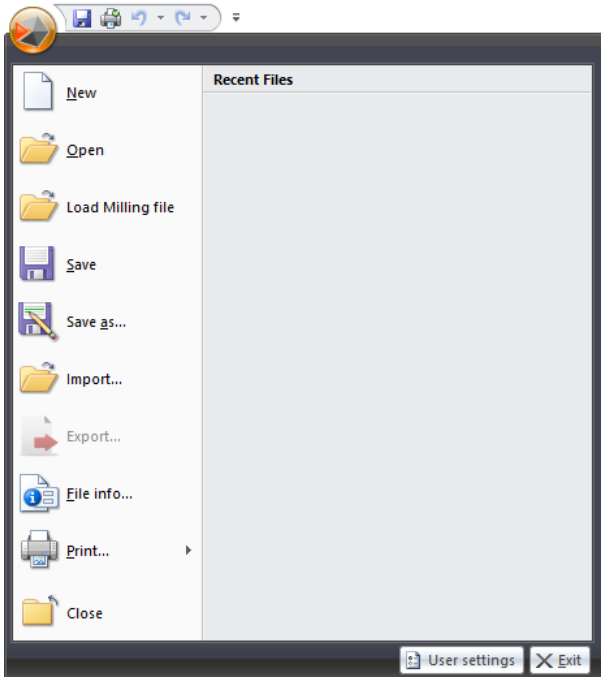


Individual screen layout

All depictions in this manual will refer to this standard layout. We recommend using the standard layout while getting acquainted with the program.

© 2019 PENTA-TEC CNC-Automation

4.1 File Menu



New	Create a new, empty file. Will close your current file.
Open	Open a file. Will close your current file. If you wish to add the file data to your currently opened file, use "Import".
Load Milling File	Open a file. This will let you restrict anyone with user level **operator** to choose a file in the folder you predefined in user settings.
Save	Save current file under existing name. Will overwrite the old file without confirmation.
Save as	Save current file and define its name.
Import	Import a file into your current file.
Export	Export your file in various different formats. Use DXF if you wish to only export graphic information.
File Info	Enter file information. This information can be displayed by pressing i in the "Open" menu.
Print	Print the contents of the graphic window.
Close	Close the current file.
User Settings	Configure program and machine parameters.

Depending on the user level, menus and layouts may vary.
For details about the definition of user levels, open the user settings.

© 2019 PENTA-TEC CNC-Automation

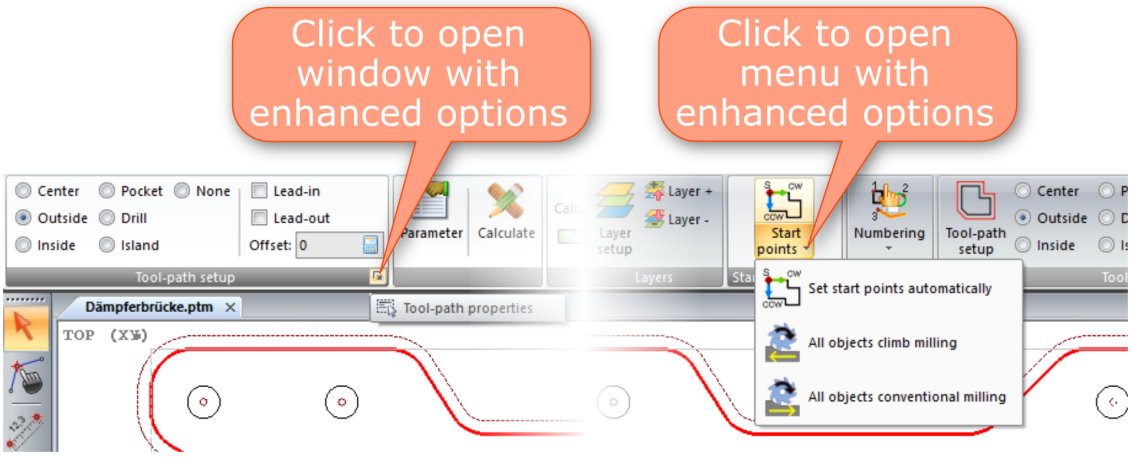
4.2 Ribbon Bar

Accessing the many functions of PENTA-NC will mostly be done via the Ribbon bar. Ribbons are structured into tabs according to group functions and tasks. The Ribbon bar is context-sensitive and will always offer functions relevant to the situation.

Currently unavailable functions will be greyed out.

Important Tools And Settings

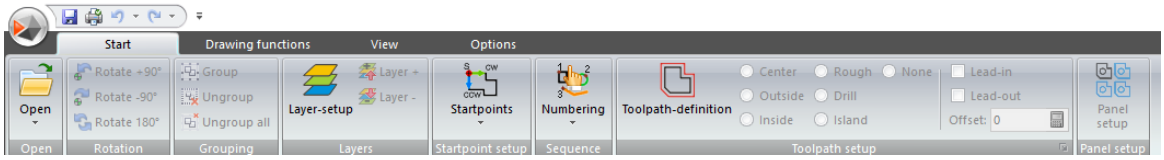
Some ribbon menus hide additional functions or settings. This will be indicated by small arrows. Click an arrow for more options.



© 2019 PENTA-TEC CNC-Automation

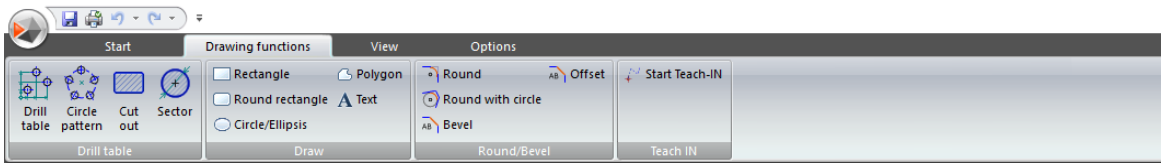
The Four Standard Ribbon Menus

Start



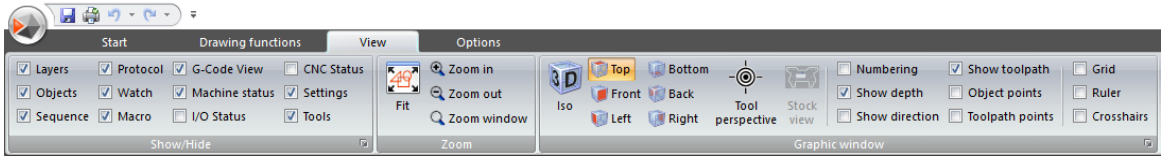
Define all relevant process parameters by going through the menu from left to right. After that you can immediately start machining.

Drawing Functions



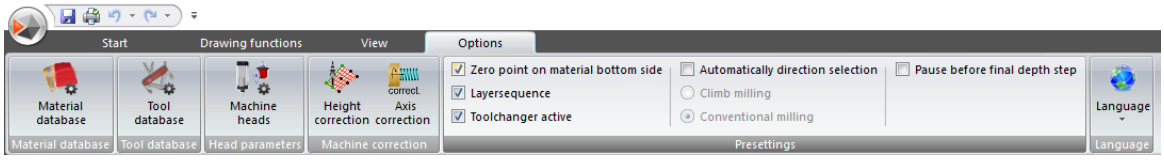
The drawing functions let you to create and/or modify simple graphic elements or text. The teach-in function allows for digitalising poly lines (e.g. by touching points on the workpiece with the tool tip).

View



You can show or hide various windows using the view functions.
For additional info in the graphic window use the check boxes in the section "Graphic window".

Options



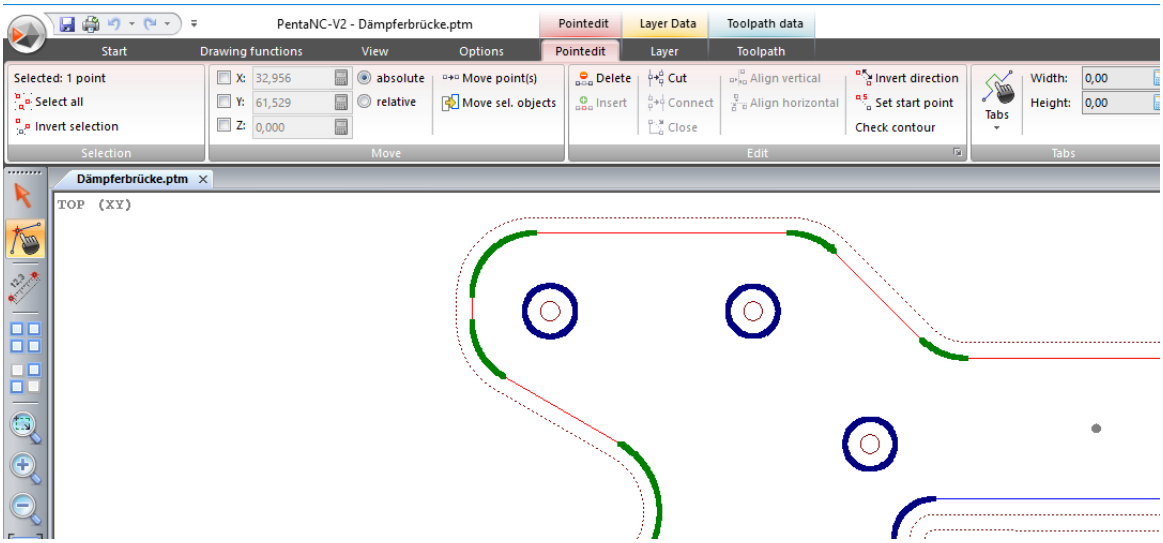
You can edit the databases for material, tools and machine heads in the options ribbon. The machine corrections (height and axis) are only available for certain controllers.

Depending on the currently active function, the standard symbol bars may be substituted with different ribbon bars.

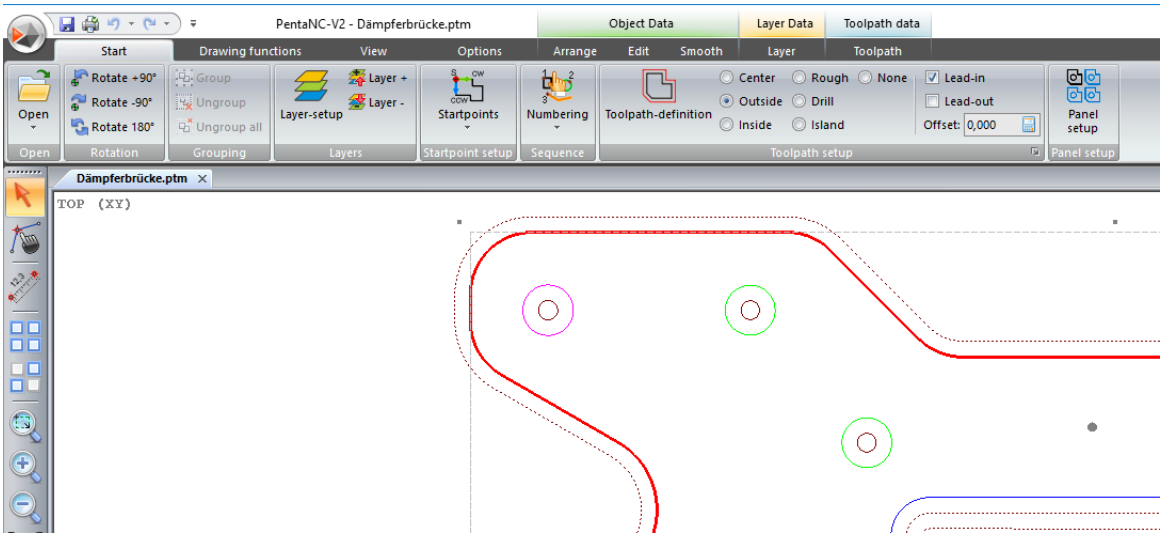
© 2019 PENTA-TEC CNC-Automation

Context-sensitive Ribbons

Choosing a specific tool in the tool bar may trigger a context-sensitive ribbon.



Selection of an object or a layer may also bring up a context-sensitive ribbon.



© 2019 PENTA-TEC CNC-Automation

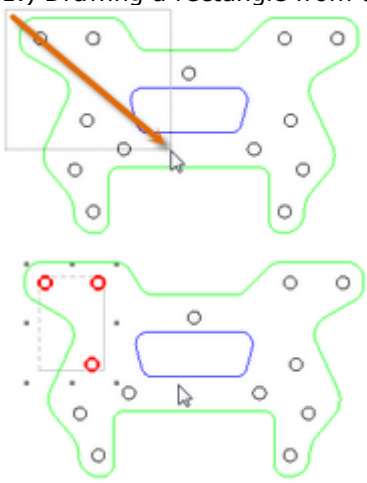
4.3 Tool Bar

The tool bar offers the most important tools to select and measure objects, edit points and change the view in the graphic window.
The default position of the tool bar is on the left-hand side of the screen. However, it can be moved to any user-defined position.

4.3.1 Selection Mode

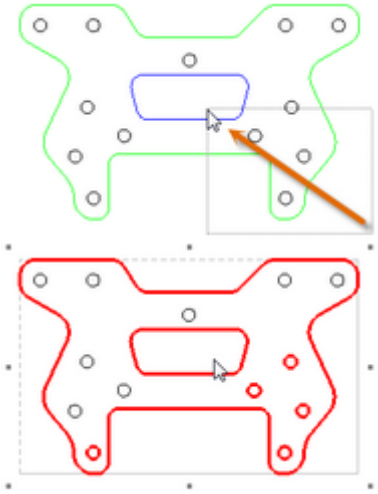
This tool lets you select one or more objects.

1.) Drawing a rectangle from top-left to bottom-right.



Only objects that are completely inside the selection rectangle will be selected.

2.) Drawing a rectangle from bottom-right to top-left.



All objects with any part of their geometry inside the selection rectangle will be selected.

© 2019 PENTA-TEC CNC-Automation

4.3.2 Point Editing

Activates the point-editing mode.
Lets you edit single points of an object.

4.3.3 Measure Mode

Measure function
Allows you to measure the distance between two points.

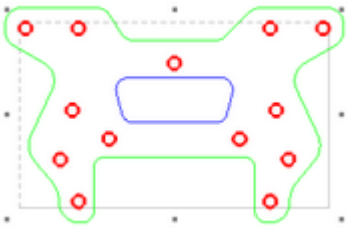
4.3.4 Select All

Select all visible objects of the current file.

4.3.5 Invert Selection

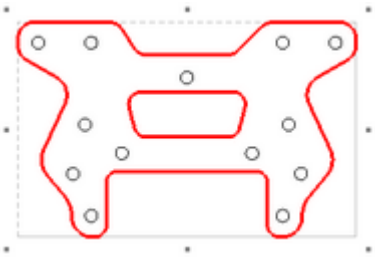
Invert an existing selection.

Example:



Drilling holes are selected, inner and outer contours are not selected.

Inverted selection:



Inner and outer contours are selected, the drilling holes are deselected.

4.3.6 Zoom Window

Draw a window around the area you want to zoom in to.

4.3.7 Zoom In

Zooms in at the currently active graphic window.

4.3.8 Zoom Out

Zooms out at the currently active graphic window.

© 2019 PENTA-TEC CNC-Automation

4.3.9 Zoom Working Area

Fits the entire working area into the graphic window.

4.3.10 Zoom All Objects (Fit All)

Fits all objects into the graphic window.

4.3.11 Zoom Selected Objects

Fits all selected objects into the graphic window.

4.3.12 VIEW Top

Activates the top view (XY-plane) in the graphic window.
You can zoom in with the mouse wheel and pan the working area by pressing the middle mouse button (mouse wheel).

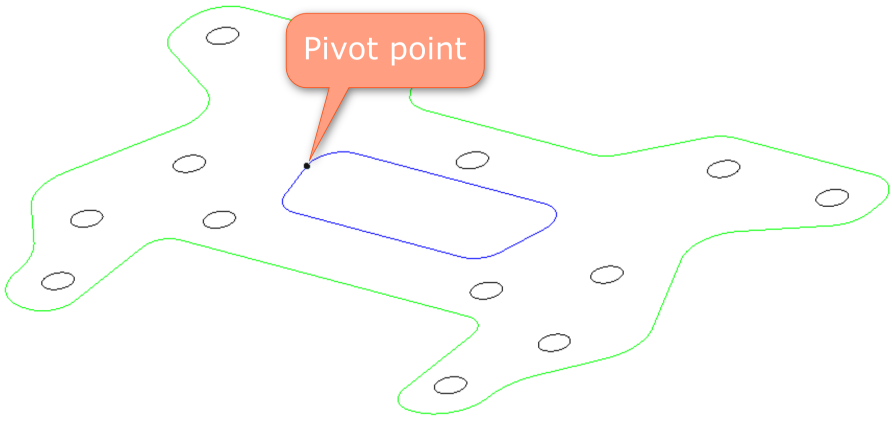
4.3.13 VIEW Front

Activates the front view (XZ-plane) in the graphic window.
You can zoom in with the mouse wheel and pan the working area by pressing the middle mouse button (mouse wheel).

4.3.14 VIEW Iso (3D View)

Activates the isometric 3D view in the graphic window.

Drag your mouse while clicking with the middle mouse button (mouse wheel) in the graphic window to rotate the view. Scrolling your mouse wheel adjusts the zoom level. To assign a new pivot point click on a point of the object with your middle mouse button (mouse wheel).



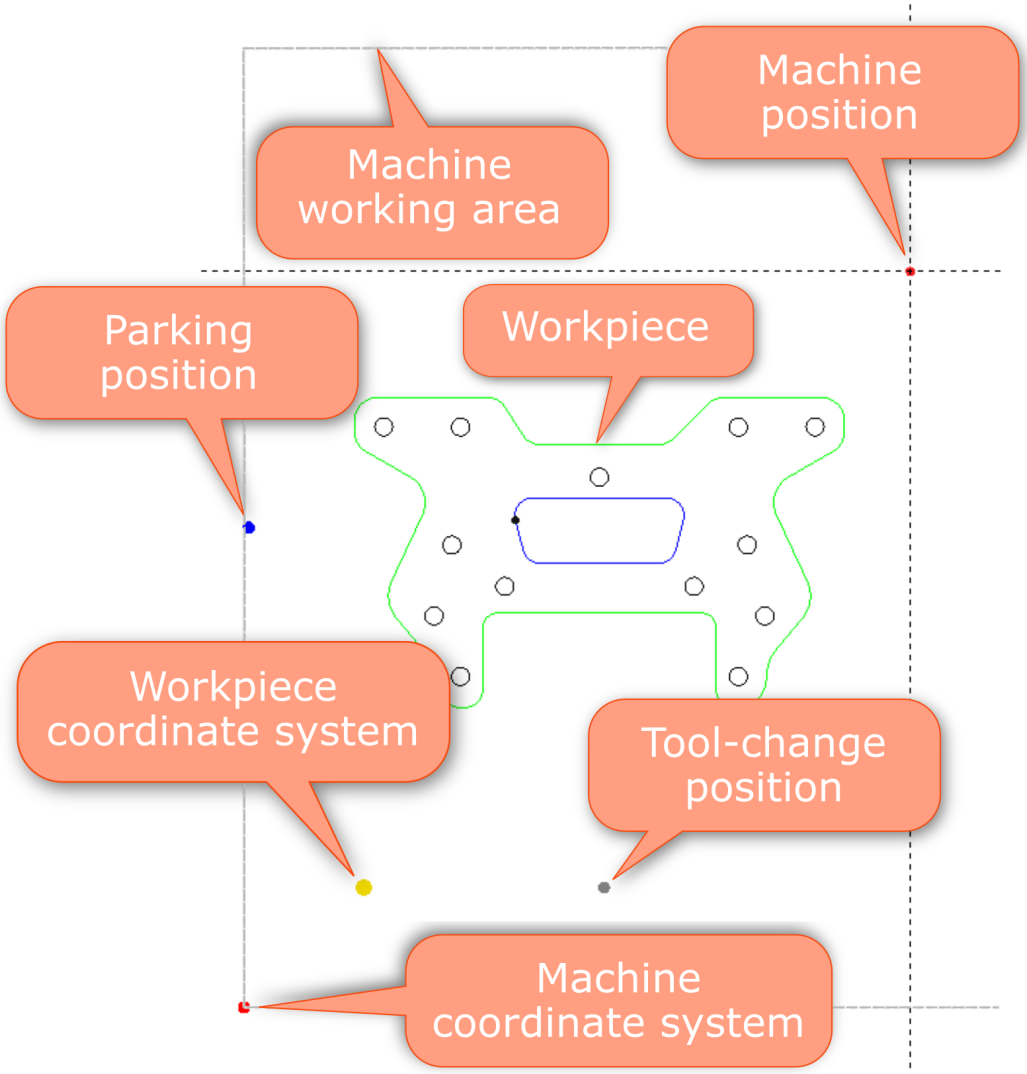
4.3.15 VIEW Tool Perspective

Activates the tool perspective.
The tool stays in the center of the graphic window and the displayed area moves with the tool.

© 2019 PENTA-TEC CNC-Automation

4.4 Graphic Window

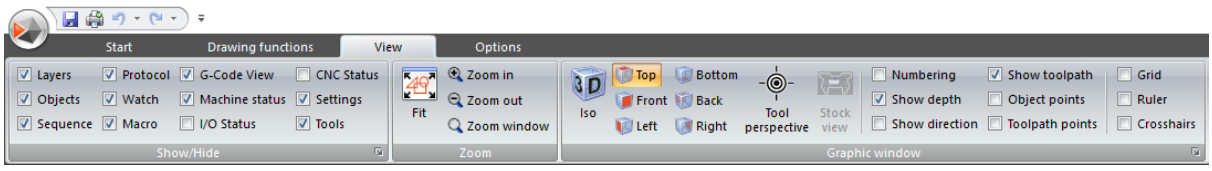
The main window below the ribbon bar offers a graphic view of the workpiece and the work area of the machine.



Editing in the graphic window is mostly done with the basic functions of the tool bar and the ribbon functions.

You can decide which details should be displayed in the graphic window by choosing "View" in the ribbon menu.

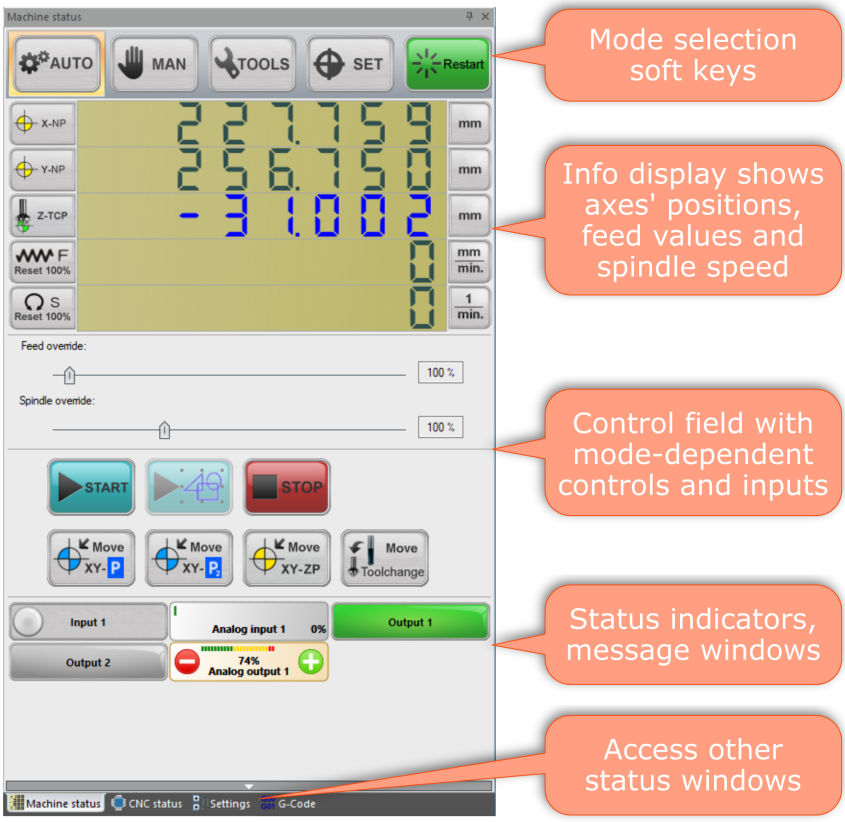
In the section **Graphic window**, located on the right-hand side, you can see all options for your personal configuration like grid and ruler.



© 2019 PENTA-TEC CNC-Automation

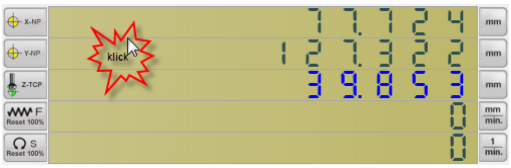
4.5 Machine Status Window

The machine status window offers all functions for operating and monitoring the machine. There are several operating modes with different information and options.

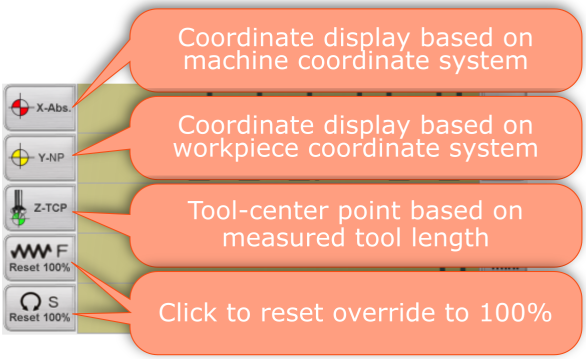


Adjusting The Info Area

You can configure the info area in the machine status window in various ways.
A click on the info area switches between big or small display, allowing for better use with different screen solutions.

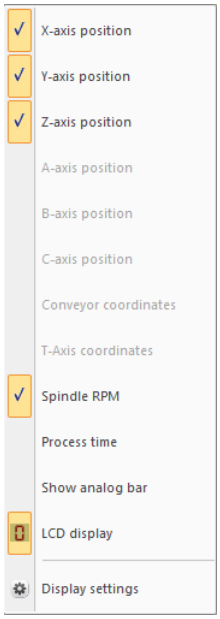


Clicking the coordinate soft-buttons lets you cycle through the various coordinate systems:

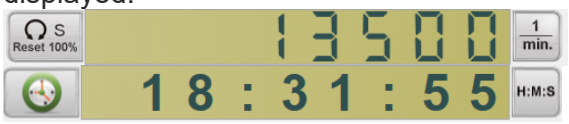


© 2019 PENTA-TEC CNC-Automation

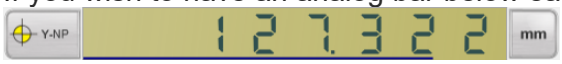
A right-button click on the display area opens a menu with additional options:



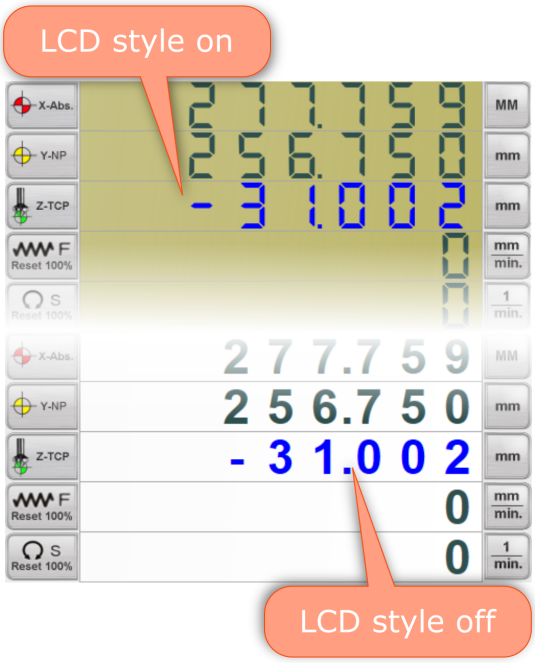
You can choose which axes you want to display in the info area.
Additionally you may choose whether you want the spindle RPM and the process time displayed.



If you wish to have an analog bar below each axis, check the according entry.



You can also deactivate the LCD-style display in the info area.

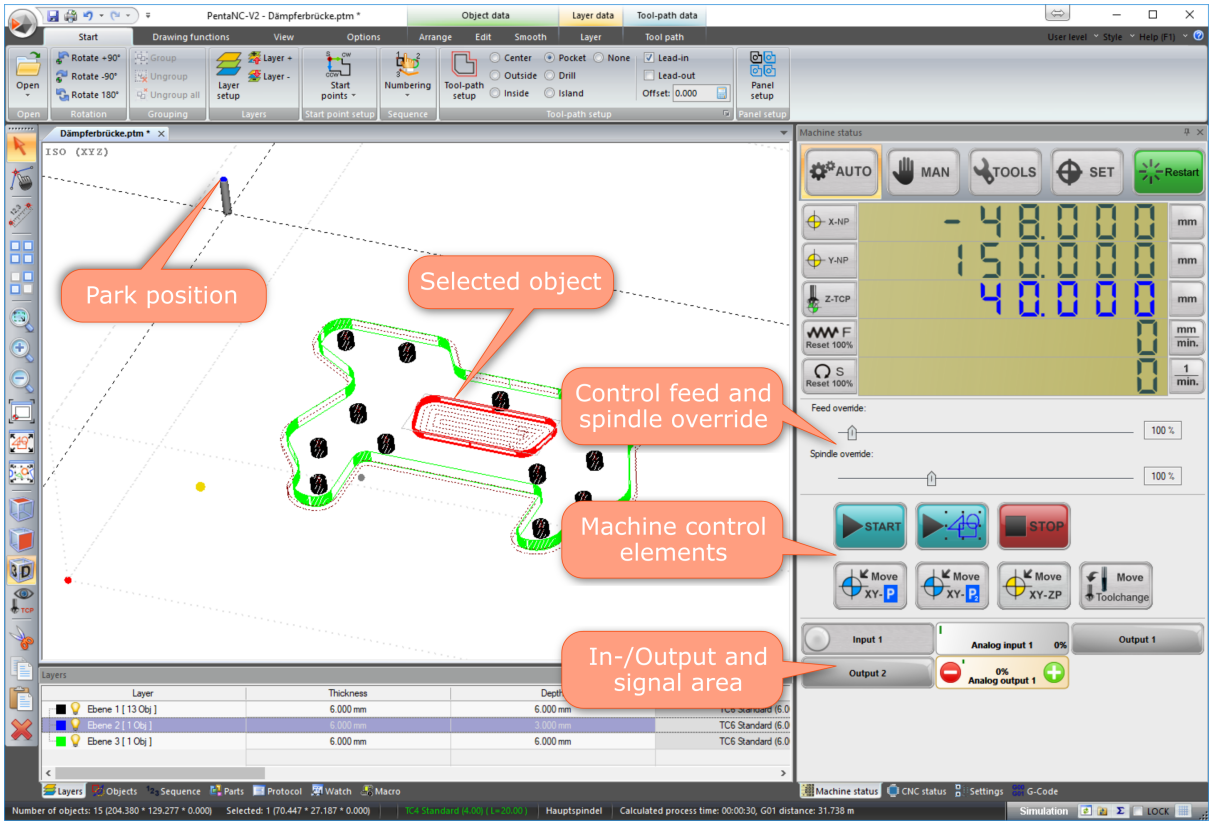


Finally, **display settings** lets you define some of the colours if you wish.

© 2019 PENTA-TEC CNC-Automation

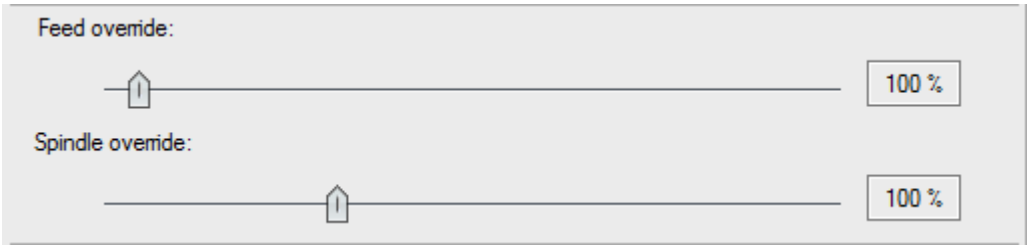
4.5.1 Automatic Mode

In AUTO mode you can start, stop and pause the machining process. You may also order the spindle to move to predefined positions.
As a distinctive feature, PENTA-NC doesn't force you into a rigid machining sequence. You may choose the order in which objects get processed or even decide to only machine selected paths.
More information can be found further down in this chapter.



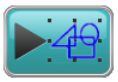
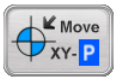
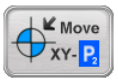
Override Control Elements

The default values of spindle revolutions and feed can be adjusted quickly by using the override elements below the info area.
Opening a new file will always reset these values to 100%.



© 2019 PENTA-TEC CNC-Automation

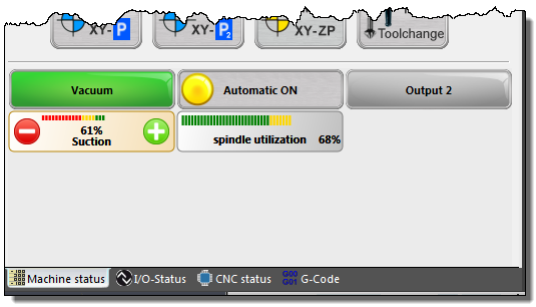
Machine Control Elements

	Start machining process for all objects of the current file. During the machining process, the button symbol changes to 
	Only selected paths will be processed. During the machining process, the button symbol changes to 
	During the machining process, the button symbol  changes to 
	Pressing this button will pause the current machining process until you commmence by pressing  or quit and cancel the process with  . During pause the Z-axis moves to the Z-parking position. The pause command can be used to perform control measurements or similar tasks for quality control.
	Continue the machining process from a paused status. The Z-axis will move to the position it had prior to the pause command and the process will resume.
	Stop the machining process. The Z-axis moves to the Z-parking position. Unlike the pause command, the stop command is used for complex procedures like changing a broken tool. After that, the process can still be resumed.
	Continue the process after a stop command has been issued. This will open a dialog window in which you can choose from which point to resume. You will be able to select the step in the G-code directly, while PENTA-NC will display the solution in the graphic window.
	Moves processing head to parking position P1. First the Z-axis moves up, then the movement in the XY-plane is executed.
	Moves processing head to parking position P2. First the Z-axis moves up, then the movement in the XY-plane is executed.
	Moves processing head to the user-defined zero point. First the Z-axis moves up to the height of parking position P1, then the movement in the XY-plane is executed.
	Moves processing head to tool-change position. First the Z-axis moves up to the height of parking position P1, then the movement in the XY-plane is executed. Finally, the Z-axis will move to the Z-coordinate of the tool-change position.

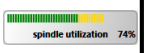
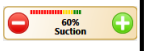
© 2019 PENTA-TEC CNC-Automation

I/O And Signal Area

In the **I/O and signal area** you can display various inputs and switch outputs on and off, or — in the case of analog outputs — be assigned varying values.
This area can be configured accordingly by an administrator depending on the specific machine.



The following fundamental control elements may be in use:

	Digital input. Shows the status of a connected digital input.
	Digital output. Clicking on this button switches the connected output on or off, unless the output is being controlled directly by the program (for example spindle or cooling).
	Analog input. Shows the analog value of an input in percent.
	Analog output. Shows the analog value of an output in percent. Can be changed by clicking + or -.

In case of an error or an abnormal machine status, an according error message will be displayed.

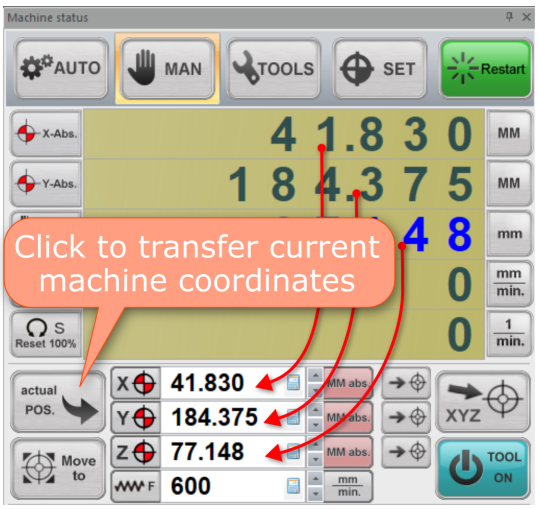
© 2019 PENTA-TEC CNC-Automation

4.5.2 Manual Mode

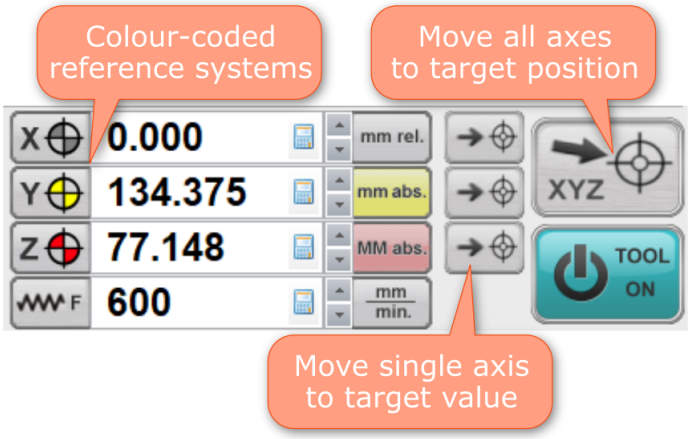
In **MAN** mode you can position the machine manually using mouse, keyboard or hand wheel. This mode can be used to set up and verify new machining data.
The window arrangement is modelled after the **AUTO** mode. There are additional functions specifically for the manual mode.

The following table details these additional functions and elements.

Numeric / Graphic Control Elements

actual POS.	 The current coordinates will be transferred to the corresponding numeric input boxes.

© 2019 PENTA-TEC CNC-Automation



Clicking the buttons X, Y or Z will switch between the reference coordinate system for each axis. Red refers to the machine zero point, yellow to the user-defined zero point and grey to the current position of the tool.

The numeric values in the input boxes define the target position in relation to the respective coordinate system.

Clicking this button will move the corresponding axis to its target position as defined in the numeric input box next to it.

The feed value of the box next to F will be used.

Use this button to position all axes at once.

The feed value of the box next to F will be used.

Switch between the input boxes for feed or spindle revolution.

Clicking this button will start the spindle, for example for manual milling.

Once the spindle is active, the button will change to 

. Click again to turn the spindle off.

Define the spindle revolutions in the numeric input box next to 

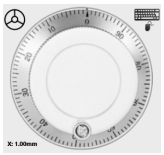
.

This allows you to manually position the machine head by simply clicking on the work area with the mouse.

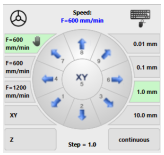
Moving The Axes With Mouse / Keyboard

If you don't have a hand wheel, you can alternatively use mouse or keyboard to manually move the machine head. There are several options at your disposal.
Keep in mind that you will need to be in the manual mode. If you haven't already, select **MAN** in the machine status window.

© 2019 PENTA-TEC CNC-Automation



If your control area displays a hand wheel, simply switch to the mouse or keyboard controlled positioning cross by clicking the keyboard symbol  next to the wheel.

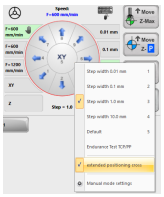


By default, the enhanced navigation pad will be displayed. You can switch to the simplified version by right-clicking on the element and deselecting **enhanced navigation pad**.

With the enhanced positioning cross you can quickly switch between various feeds that you can predefine by right-clicking on any of the three feed values and entering the desired value. A left-click then selects the preferred feed for positioning.

Below the feed you can select movement in the XY-plane or along the Z-axis. Alternatively you can left-click in the middle of the cross.

On the right side you have the option to choose incremental or continuous movement.

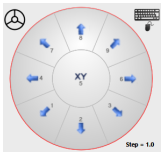


Right-clicking on the navigation pad will open a menu. Here you can choose between the increments for movement and select or deselect the enhanced navigation pad.

If you use the enhanced nav pad, you can easily access the preferred increment by using the soft buttons on the right of the cross.

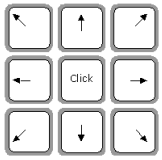
You may also enter the manual mode settings. However, you should not change the default values without good reason and knowing the exact consequences.

If you prefer the simplified navigation pad, you can switch to it by right-clicking on the control element and deselecting **enhanced navigation pad**.



If for some reason you prefer the simplified nav pad, you can switch to the simplified version by right-clicking on the control element and deselecting **enhanced navigation pad**.

You will not be able to quickly change the incremental movement or switch between three different presets of feeds, however. If you want to change increments or switch back to continuous movement, right-click on the cross and select your step width from the pop-up menu.



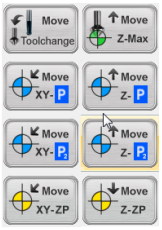
An alternative to using the mouse is the numeric pad on the keyboard.

You can use the keyboard with either enhanced or simplified positioning cross. If you use the simplified nav pad, changes to step width or a switch to continuous movement have to be conducted via the right-click pop-up menu. If you use the **enhanced navigation pad**, you can use the following short keys:

<shift> Switch between the various increments. You can see the current step width highlighted in the enhanced nav pad.
right <ctrl> Activate continuous movement. The corresponding soft button will be highlighted upon activation.

© 2019 PENTA-TEC CNC-Automation

Moving To Predefined Positions

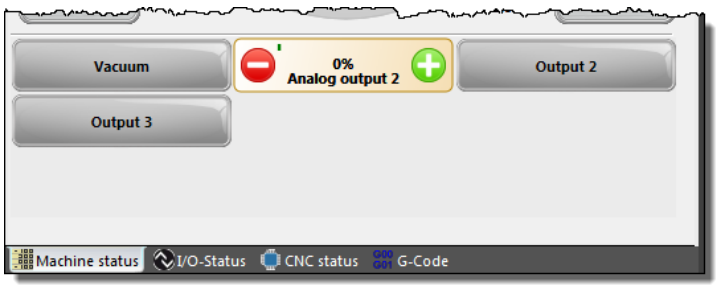


The two button rows left (XY) and right (Z) of the navigation pad are used to move the head to predefined positions.

For details about the movement procedure please refer to the table in the section regarding the **AUTO** mode.

I/O And Signal Area

The function and design of the **I/O and signal area** in **MAN** mode corresponds to the **AUTO** mode. However, the layout of the various elements may differ.



© 2019 PENTA-TEC CNC-Automation

4.5.3 Set Machine Points

The two previous chapters (**AUTO** and **MAN** mode) already mentioned predefined positions, such as parking and tool-change positions, and how to move the machine head automatically to those coordinates.

The **SET** mode allows you to define and configure these positions.

Functions

	When clicking on AUTO REF, the machine will start an automatic homing sequence. If configured correctly, the axes will move towards the reference switches one by one. Upon reaching them, they will be positioned at a predefined starting point. *Machines with an absolute-encoder system will not start a homing process. They will instead simply move the axes to a predefined starting point.*
	Activates the position settings for the workpiece zero point.
	Activates the position settings for the tool-change position.
	Activates the position settings for parking position 1.
	Activates the position settings for parking position 2.
	Restores the workpiece related coordinate system to the default value.

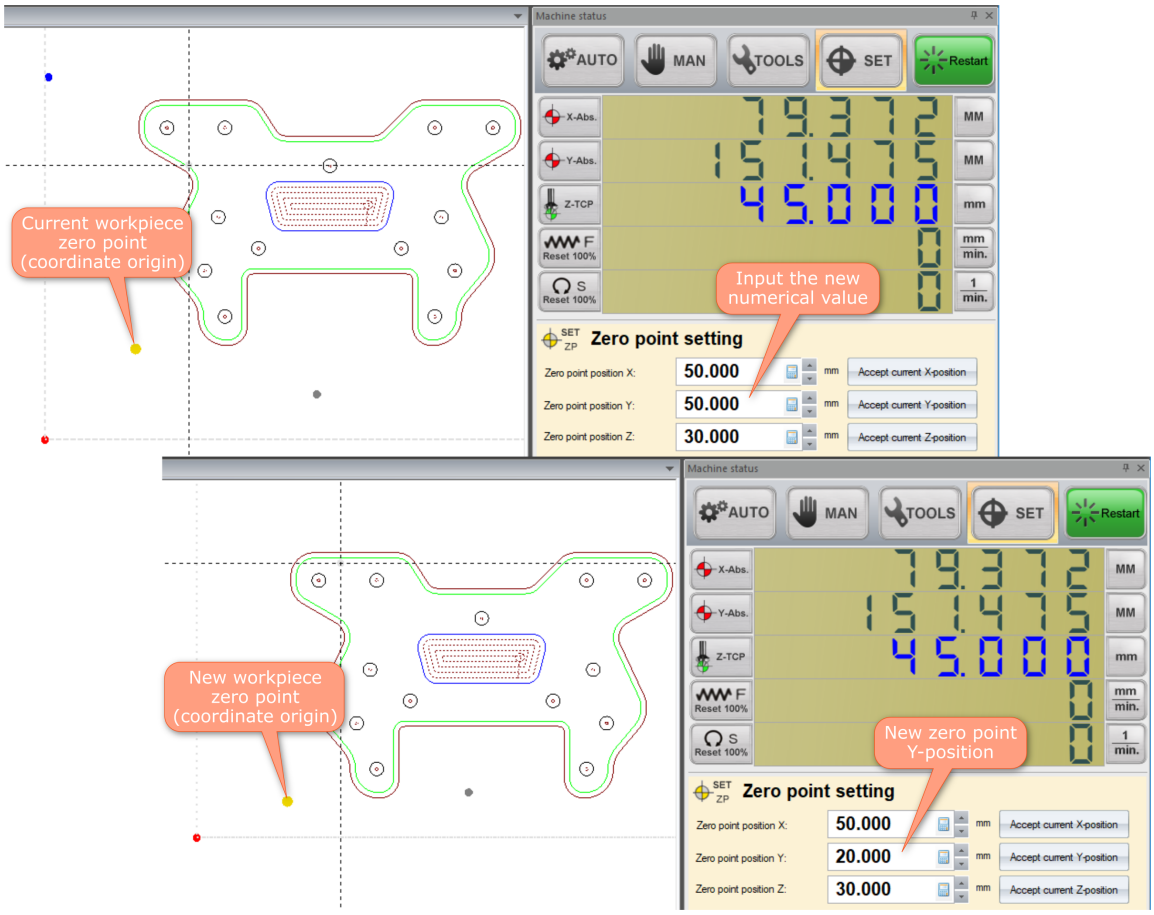
Position Settings Window

All the position settings use the same fundamental functionality. As an example, we will look at how to define the coordinate system for the workpiece by setting the zero point.

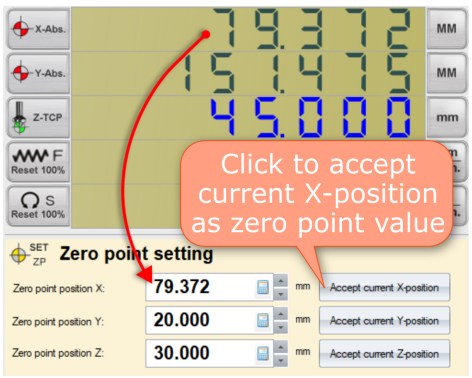
You have two options to set the zero point:

© 2019 PENTA-TEC CNC-Automation

1. **Enter a numerical value:**
   You may simply enter the numerical value of any given coordinate (X,Y,Z) directly into the boxes. The values will be set after a short delay or upon pressing <Enter> and the graphic display will be updated accordingly.



2. **Adopt the actual position of the machine head:**
   Move the machine head manually to the desired position. Subsequently, click on **accept current position** for the according axes.
   PENTA-NC will automatically store the absolute coordinates in the corresponding fields.



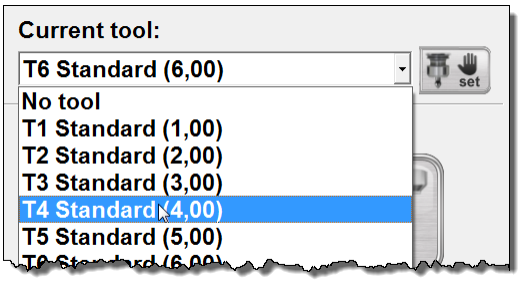
© 2019 PENTA-TEC CNC-Automation

4.5.4 Tools

This area allows you to manage all tools and tool data.
Here you can order the machine to change tools, measure them, and so on.
Additionally you can add and configure tools and the tool changer if you possess one.
If you do not have a tool changer, some of the functions below may not be shown.

Position / Function select:

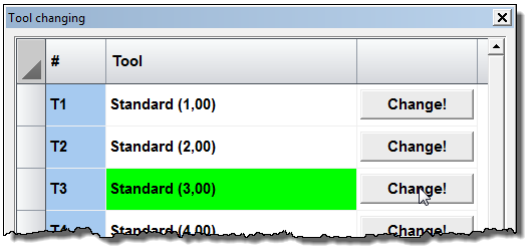
Manual tool choice



Clicking this button will open a drop-down menu.

Simply select which of the listed tools is currently active in your machine.

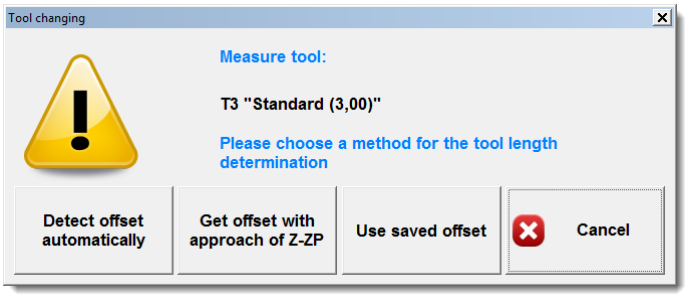
Replace tool



Use this function to replace a tool with a new one, for example if it has become dull.

The current tool is highlighted in green (or red, if it hasn't been measured yet). Click on **change** next to the tool to start the tool-changing process.

Measure tool



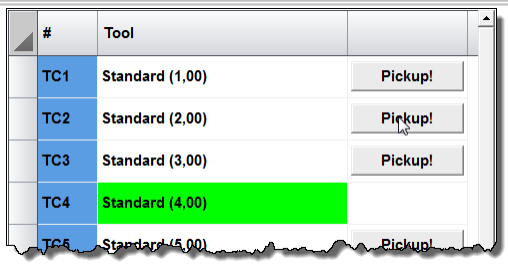
This function lets you measure a tool.

If you are using a tool with unknown length, it will have to be measured first in order to guarantee the correct machining depth.

There are several measurement methods. The method may be chosen from a query window or can be predefined in the settings. For more details see the section **tool library**.

Pick tool

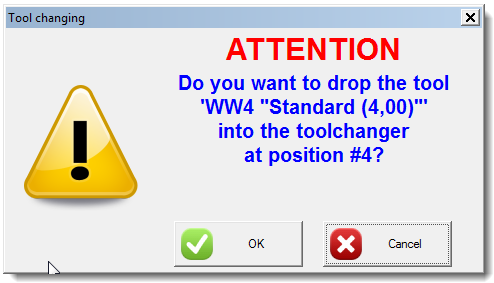
© 2019 PENTA-TEC CNC-Automation



Only available in combination with a tool changer. Allows you to pick up a tool automatically.

Click on **pickup** next to the tool you want to pick up in order to start the tool-changing process- The current tool will be placed in the tool changer and the new tool will be picked up.

Drop tool

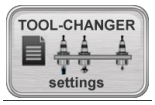


Places the current tool in the tool changer.

No new tool will be picked up. A pop-up window will ask you to confirm whether the current tool shall be placed in the corresponding tool-changer position. Make sure the position is vacant and confirm.

Tool-library settings

Opens a window with further options for editing tool settings and configurations.
For more details see the section **tool library**.



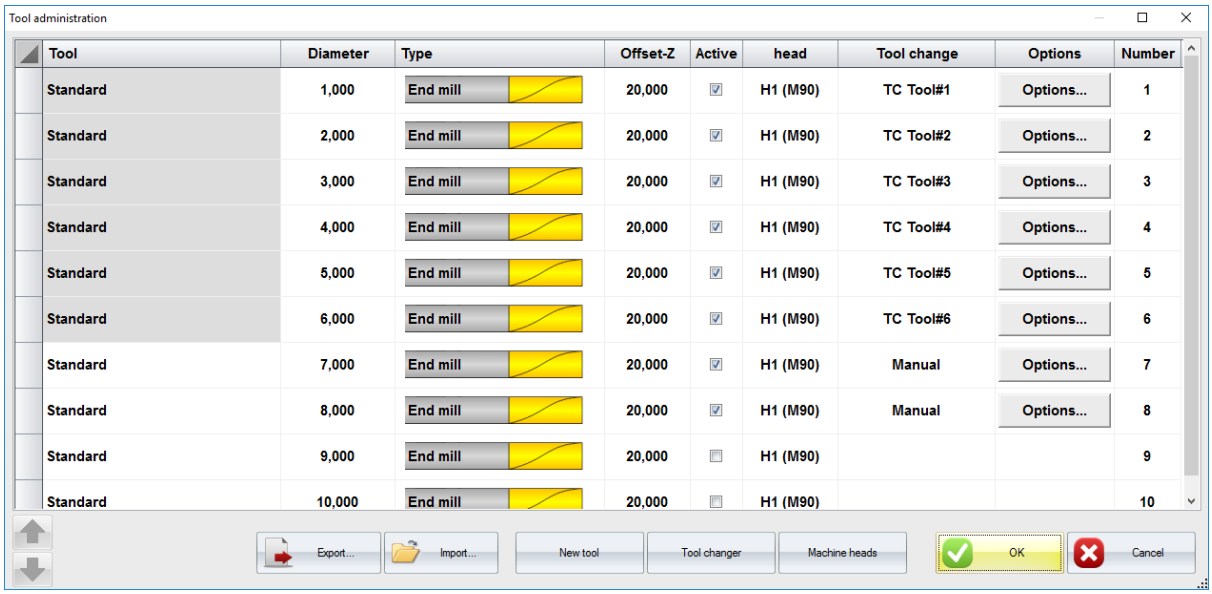
Tool-changer settings

Opens a window with further options for editing tool-changer settings.
For more details see the section **tool changer**.

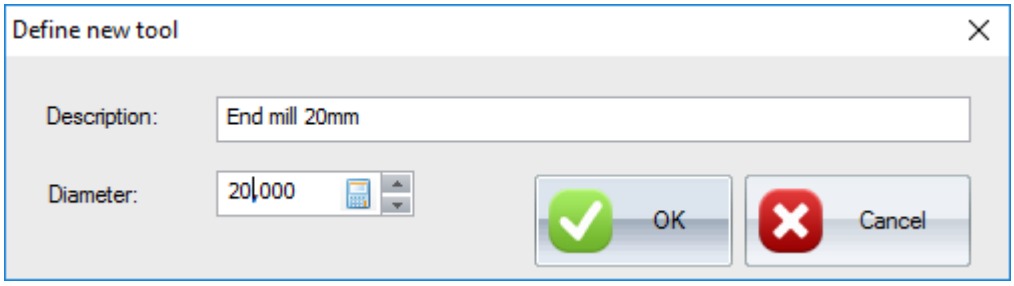
© 2019 PENTA-TEC CNC-Automation

4.6 Tool Library

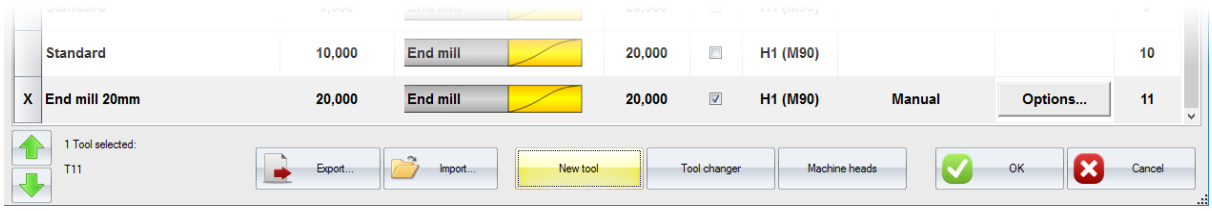
If you open PENTA-NC for the first time, you will need to define the tools of your machine. Go to the machine status window, activate the soft button **TOOLS** and click on **tool-library settings**.



You will find a list of standard end mill tools listed in the window. If you want to delete tools, right click on an entry and choose **delete**.
You can edit existing tools by pressing **options** in the corresponding row (only available for tools that are set to **active** in the column next to **Offset-Z**).
You may also create new tools by clicking on **new tool** in the bottom. When creating a new tool, you will be asked for its name and diameter.



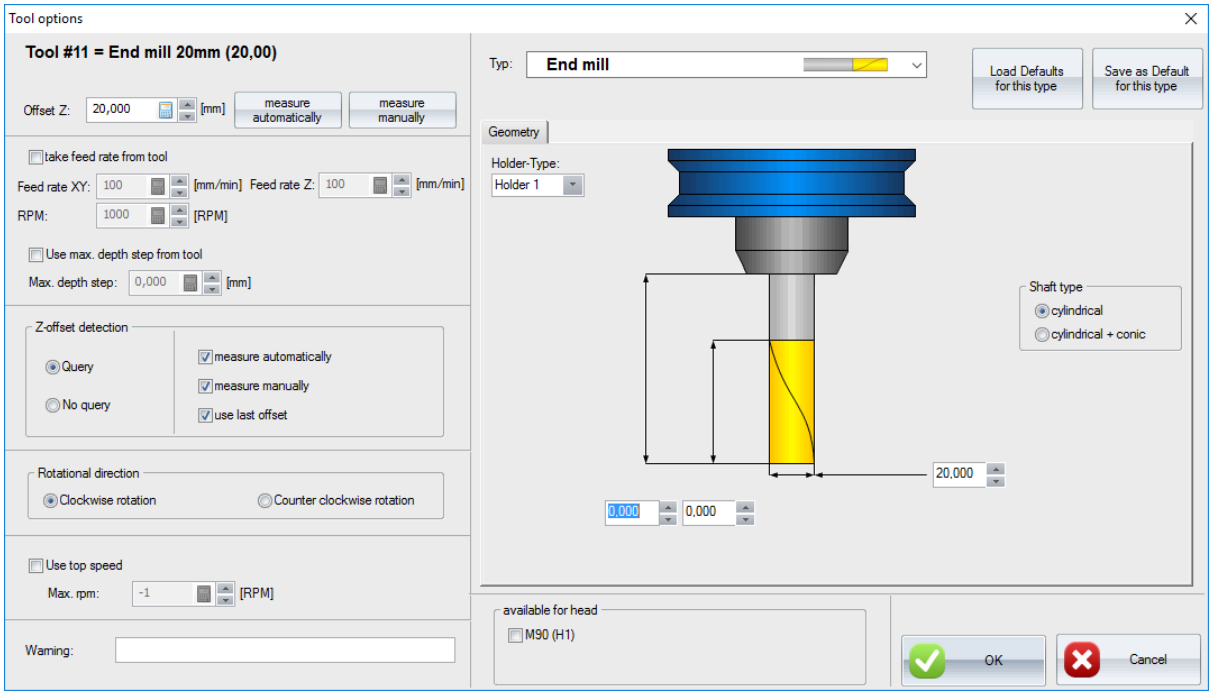
Upon entering and confirming it will be listed in the tool library and can be accessed by clicking **options** in order to define it further.



When you click on **options**, a new window will open. Here you can configure the tool in

© 2019 PENTA-TEC CNC-Automation

detail.

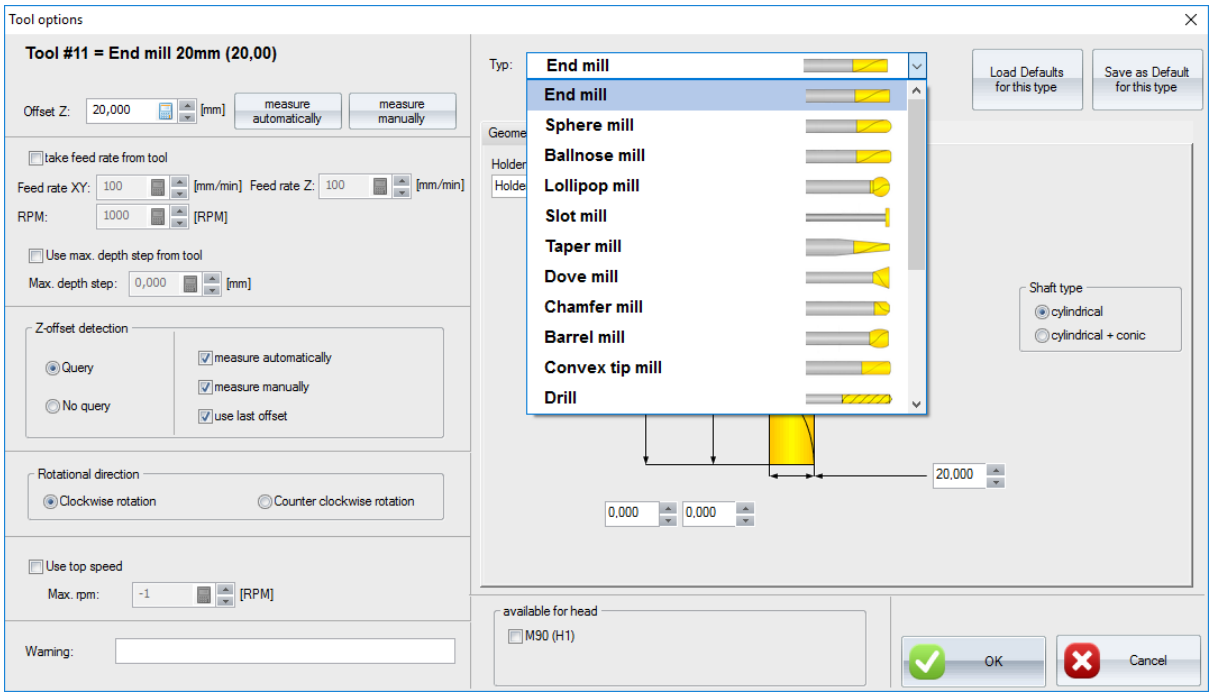


You can enter a specific offset value, measure the tool automatically (with a tool measurement switch) or manually by carefully positioning it at the previously defined zero position in the Z-axis and then adopt the Z-value.
If you want to override the general feed for the machine, you can choose **take feed from tool** and enter your preferred values. Note that if you use a higher feed here, the software will internally revert to the machine limits.
You can also choose to define the maximum cut depth for your tool.

Next you can decide whether you want PENTA-NC to ask which method of Z-offset detection should be used when switching to this tool or whether you'd prefer it to use one method without further query.
The section **rotational direction** allows you choose between clockwise or counter-clockwise rotation. You can also define the maximum revolutions per minute for this specific tool.

If you wish, you can enter an additional individual warning message that will pop up and require manual confirmation whenever you change to this particular tool.
Under **type** you can choose between various tool types.

© 2019 PENTA-TEC CNC-Automation



After selecting a specific type, the graphic display will change accordingly. Choose the shaft type (if applicable) and enter all other geometrical data.

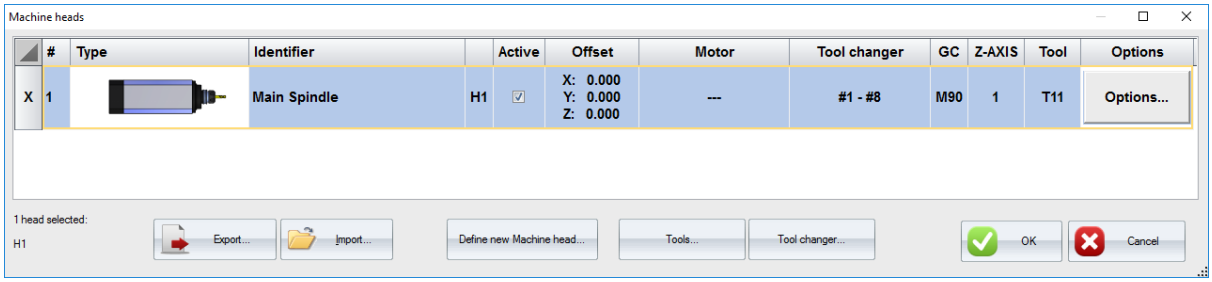
Finally, choose for which machine head this tool is available. If you have only one machine head, make sure to check it nonetheless in order to avoid possible conflicts.

After you entered all the data, click on **OK** to confirm all the changes. This will return you to the tool-library window. Click **OK** again to close the window, confirming all the changes to existing and new tools.

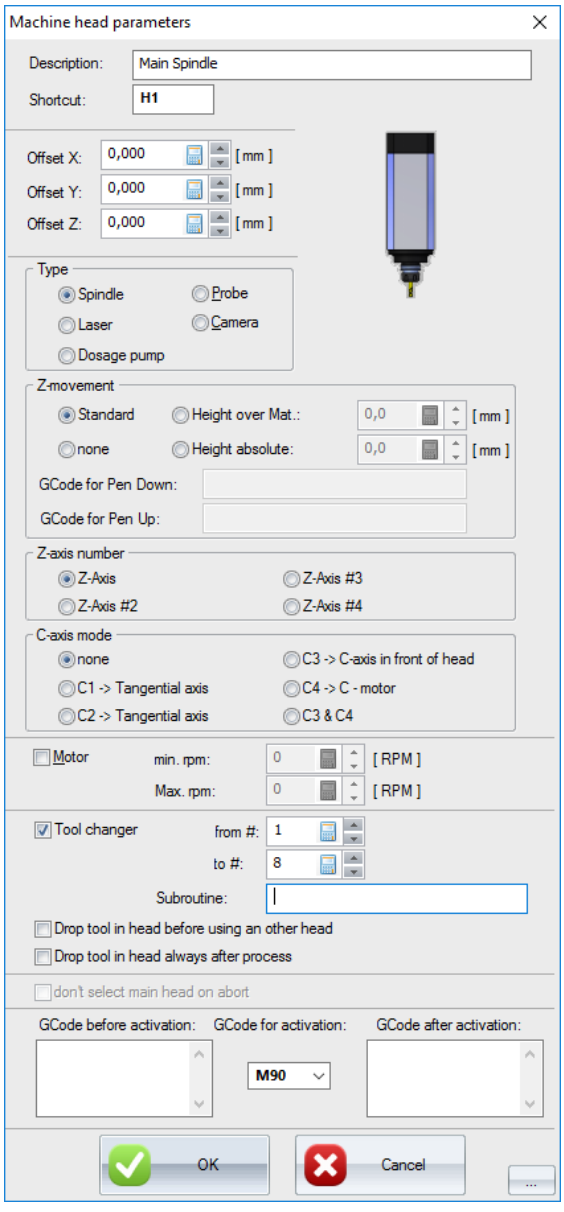
© 2019 PENTA-TEC CNC-Automation

4.7 Machine Heads

In the **tool library** (see previous section) you can access the **machine heads** by pressing the according button.
PENTA-NC will open the machine head library.



You can either define a new machine head by pressing the button in the bottom or use **options** to set the parameters for an existing machine head.
This will open the **machine head parameters** window.



Enter the machine head's name under **description** and its shortcut right below.

If the machine head has an offset, you can enter the values for each axis.

Next you can choose the **type** (spindle, laser, ...) and the **Z-movement** of your machine head.

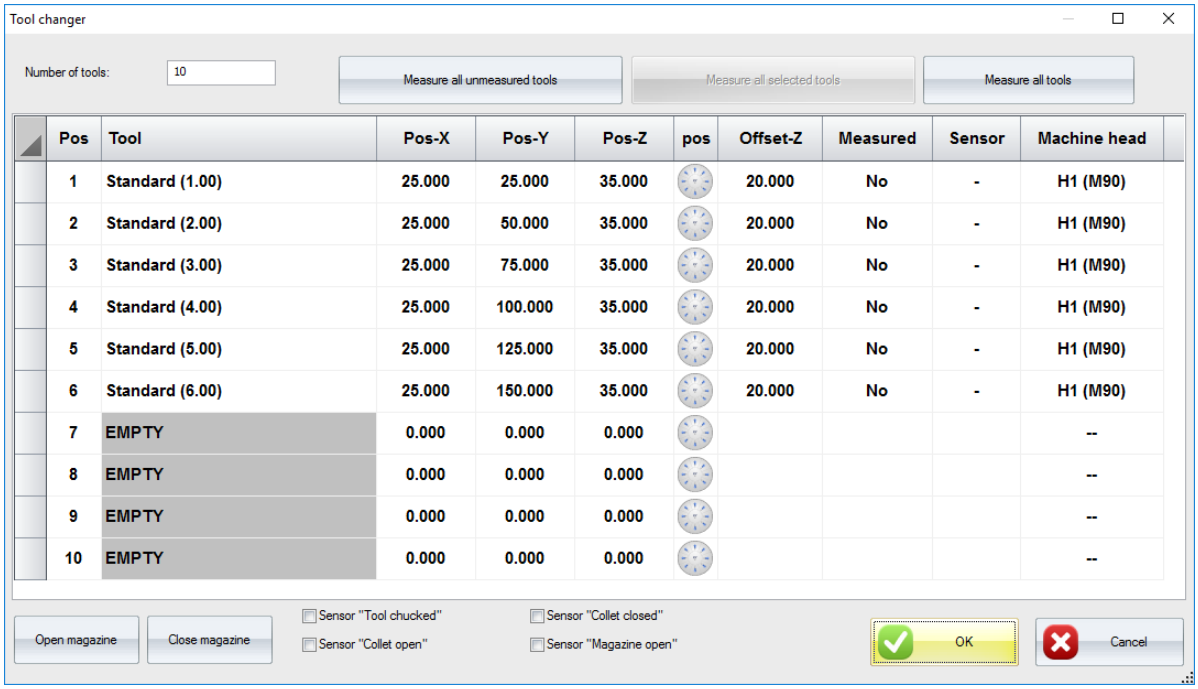
Choose the **Z-axis number** and the **C-axis mode**. If you wish, you can also define the **minimum and maximum RPM** for each machine head separately.

If you want to use a **tool changer** (see next section), you will need to define which positions the machine head should access.

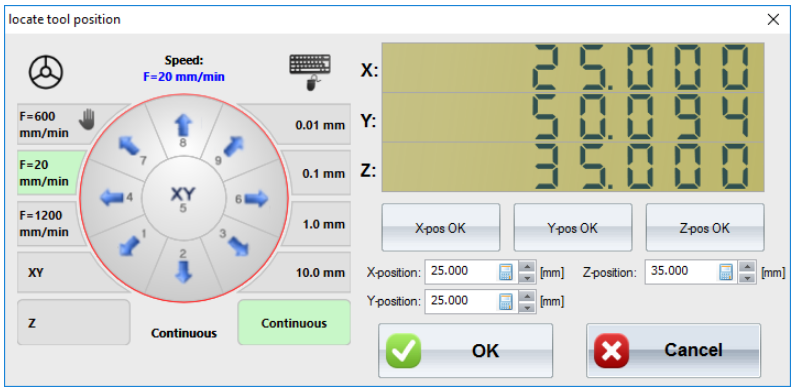
© 2019 PENTA-TEC CNC-Automation

4.8 Tool Changer

If you have a tool changer, open the configuration window by clicking on the **tool-changer settings** soft button in the **TOOLS** mode of the machine status window.



If the tools are highlighted red, make sure that you activated and defined the correct tool-changer positions in the **machine head parameters** (see previous section).
If everything is defined correctly, you can start entering the coordinates of each tool-changer position. Alternatively, you can teach the positions by clicking on the navigation pad symbols located between the **Pos-Z** and **Offset-Z** columns.



Position the machine head in the correct spot for each tool-changer position and accept the coordinates by clicking on **X-Pos OK**, **Y-Pos OK** and **Z-Pos OK** buttons respectively. PENTA-NC will transfer the current values into the according boxes. Finally, confirm by pressing **OK**.

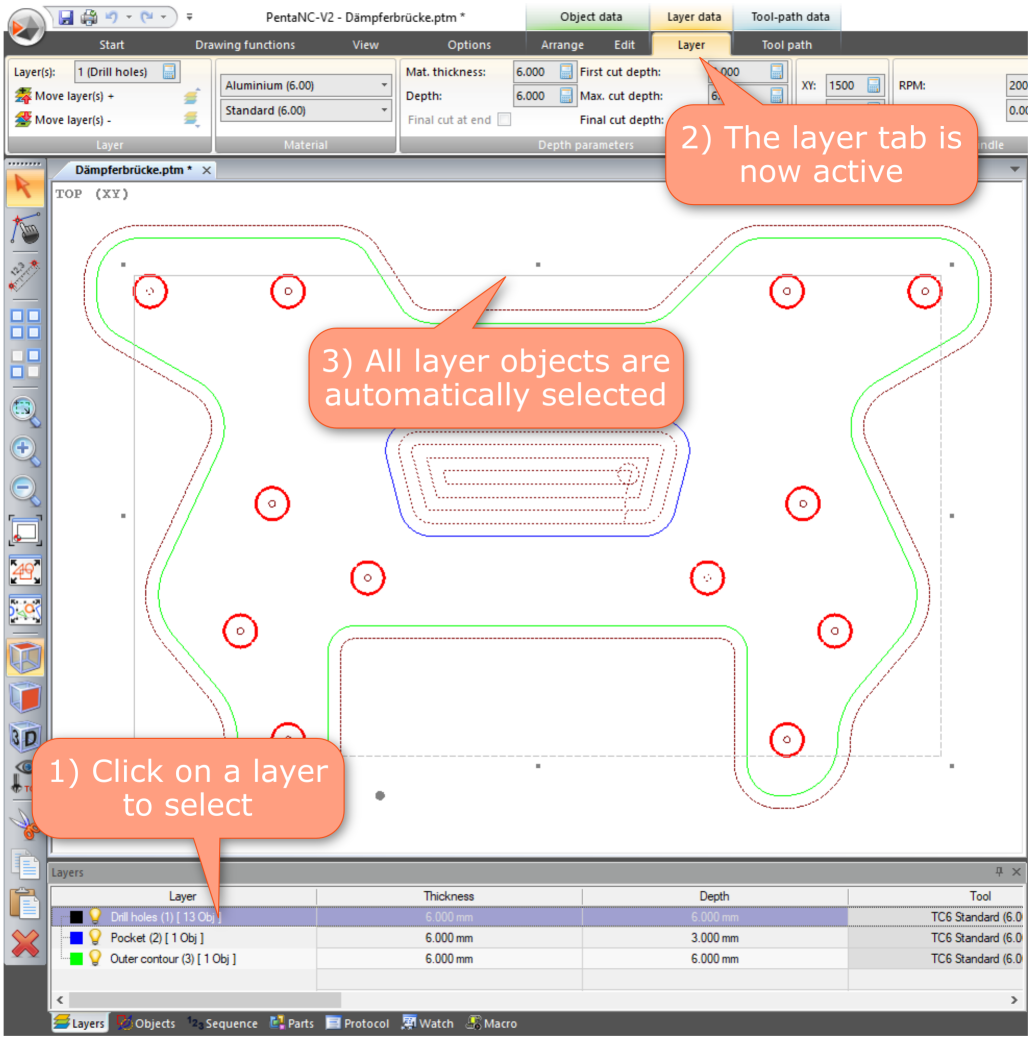
Do this for every position of your tool changer.

Once you have defined all positions, confirm by clicking on **OK**. This will close the window and finalise your tool-changer positions.

© 2019 PENTA-TEC CNC-Automation

4.9 Layer Setup

PENTA-NC uses layers to manage graphical and technological parameters.
Each layer has its specific technical information in regards to the machining process, such as the used tool, feed, RPM, depth per cut, ...
Therefore, it makes sense to use different layers for different steps in your machining process.



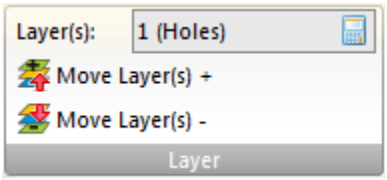
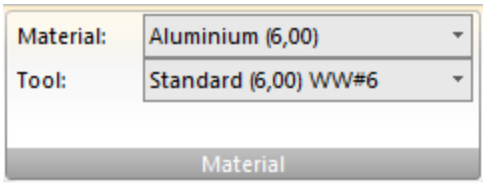
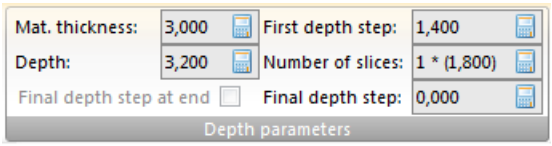
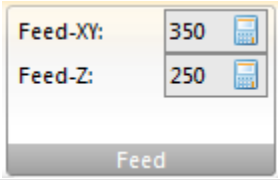
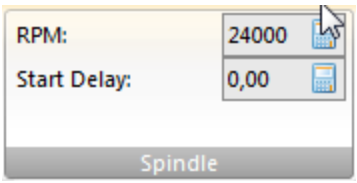
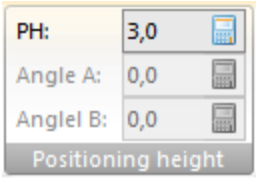
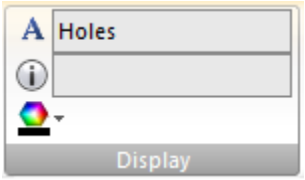
In the sample file above, we have defined three layers:

- A layer with all the drill holes
- A layer with the pocket
- A layer with the outline

We can define technological parameters for each layer.

© 2019 PENTA-TEC CNC-Automation

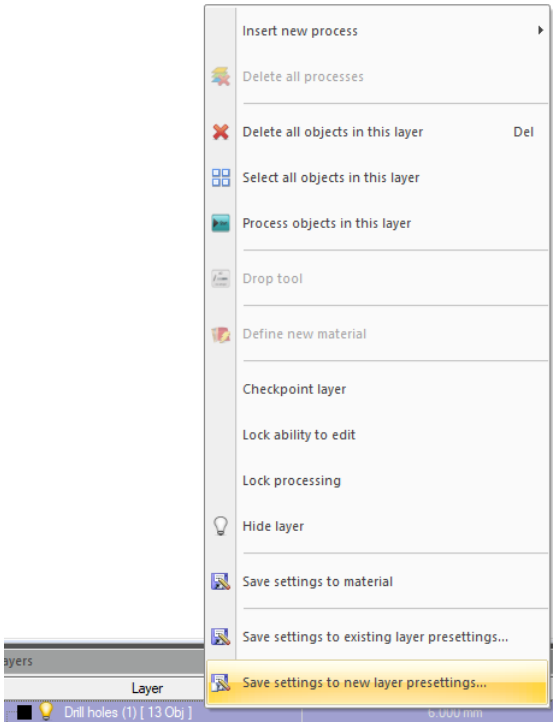
The Layer-Specific Technological Parameters

Layer		Each click on the button moves the selected layer up. Each click on the button moves the selected layer down.
Material		When choosing a material from the material data bank, all parameters of this particular layer will be automatically filled out accordingly (including the choice of tool). You can manually change any given parameter. If you change the tool to one with a different diameter, the new tool path will be calculated immediately.
Depth Parameter		In order to cut through the material, the cut depth must be equal to or greater than the material thickness. You can also enter the maximum cut depth as well as the first and the final step.
Feed Parameter		Here you can define the feed for the XY-plane as well as the Z-axis. This feed can be varied between 0% and 300% with the override sliders in automatic mode.
Spindle Parameter		RPM: Define the rotations per minute. Start Delay: Define the amount of delay of the machining process after switching on the spindle.
Positioning Height		PH: Define the positioning height above the material for safe positioning in fast motion. Angle A/B: Define the angle of the layer plane for index machining.
Display		Choose name and colour of any layer. You can also enter a short description for a layer.

© 2019 PENTA-TEC CNC-Automation

The Layer Context Menu:

Right-clicking on a layer will open a context menu. Depending on your version and additional modules, it may vary slightly.

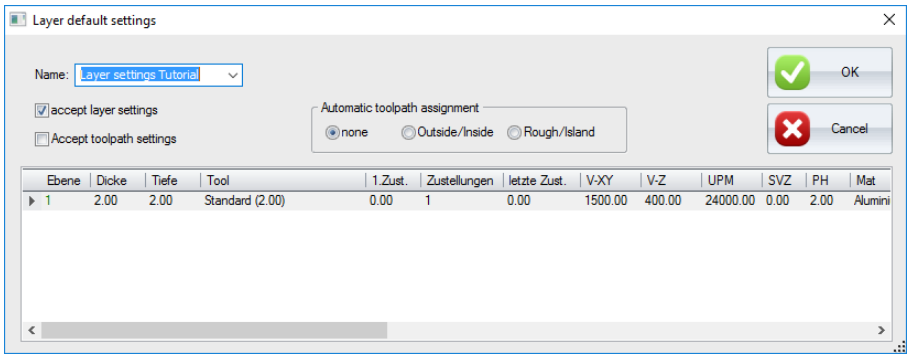


Most options are self-explaining (**delete, select or process objects** as well as **lock edit** or **processing** and **hide layer**).

You may also **save settings to material** if you have made changes in your layer that you would like to make the default machining process for this particular material.

If you want to automate your future projects even further, you can also decide to save the layer settings.

Save settings to new layer presettings will open up a new window.



Enter a name for your new layer settings and decide whether you want to also accept the tool-path settings. This option will only be available if all the layer's tool paths have the same definition.

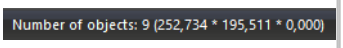
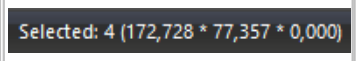
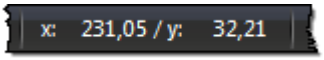
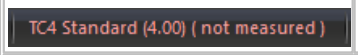
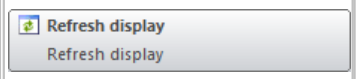
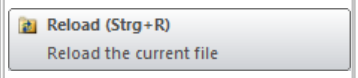
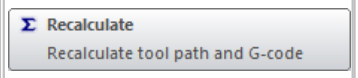
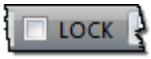
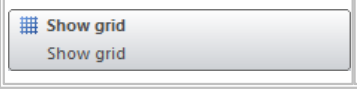
- If you want to add another layer to an existing presettings file, choose **save settings to layer presettings** instead and choose the preferred existing setting in the new window.

© 2019 PENTA-TEC CNC-Automation

4.10 Status Bar

The status bar displays various important information in regards to the current machining file and program status.

For your convenience, the sections are arrange next to each other in the status bar in the bottom and described in the table below:

	Shows the total amount of all objects of the current file as well as their area.
	Shows the amount of currently selected objects and their area.
	Shows the current cursor position coordinates (in regards to workpiece zero point).
	Shows the current tool and its status.
	Shows whether PENTA-NC runs in simulation mode or is connected to a CNC controller. Clicking here will toggle between modes.
	Refreshes the graphic window and—if applicable—deletes tool paths that have already been processed.
	Reloads the current file. If any changes have been made, a pop-up window will let you decide whether to save those changes prior to reloading.
	Tool paths and G-code of all selected objects will be recalculated.
	When working with complex data files, a recalculation of tool paths and G-code may use up a lot of resources. In order to avoid recalculation with every step during the data preparation, you can check the **LOCK** box. You will however have to deselect it again before processing the data.
	Shows or hides grid lines in the graphic window.

© 2019 PENTA-TEC CNC-Automation

5 Interaction Between Machine and PENTA-NC

It is crucial to understand how the software and the machine interact and how the actual positions of the machine are displayed in PENTA-NC.

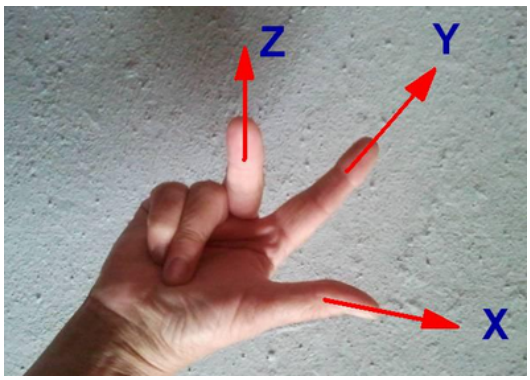
Read the following chapters thoroughly to ensure an optimum experience with PENTA-NC.
A basic understanding of the communication between software and hardware are vital for efficient work flow.

It can also prevent damage to your body, machine, tools and workpiece.

© 2019 PENTA-TEC CNC-Automation

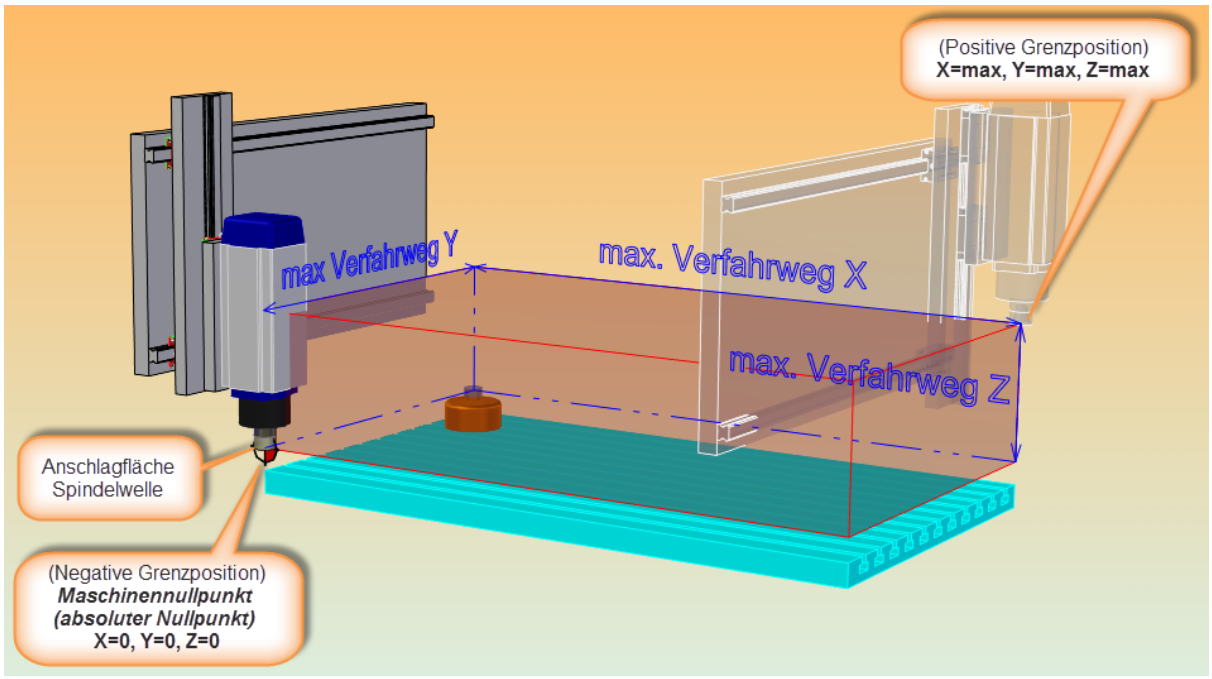
5.1 Working Area

Generally speaking, a CNC machine will work with a Cartesian coordinate system with its main axes aligned in XYZ.
The directions of the axes are defined by the **corkscrew rule** (or **right-hand rule**).



The origin of the coordinate system for the working area will usually be placed in the lower left (coordinates 0/0/0).
The maximum coordinate values thus equal and define the maximum travelling distance of each main axis.

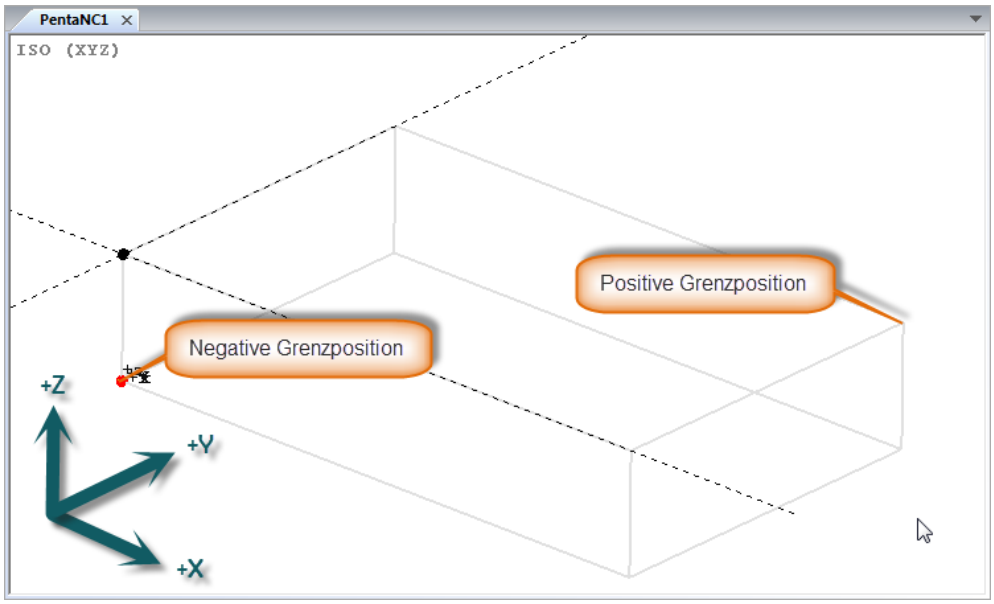
At a milling machine you will most commonly find the following layout:
From the operator's point of view the zero point of the machine coordinate system will be the bottom left corner with the lowest possible Z-position of the machine head (without tools). Originating here, all other points and movements will be measured (all with positive values).



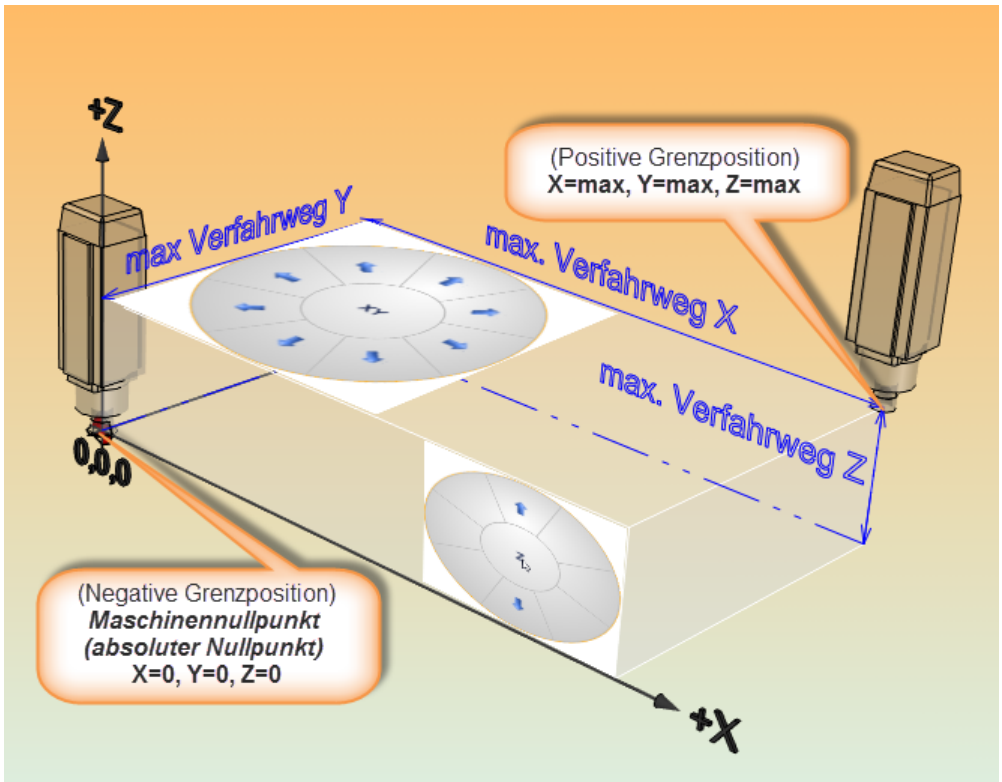
Within these limits, the machine head can be moved to any user-defined position.

© 2019 PENTA-TEC CNC-Automation

Correspondingly, the screen display will show the following:



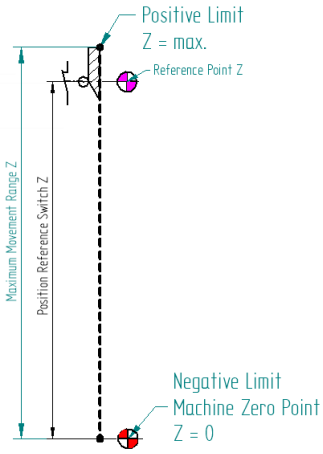
We strongly advise that the machine coordinate system be positioned in such manner, that the origin point (0/0/0) is in the lower left corner from the operator's point of view!
This ensures that the machine will move in the desired correction when using the software navigation pad.



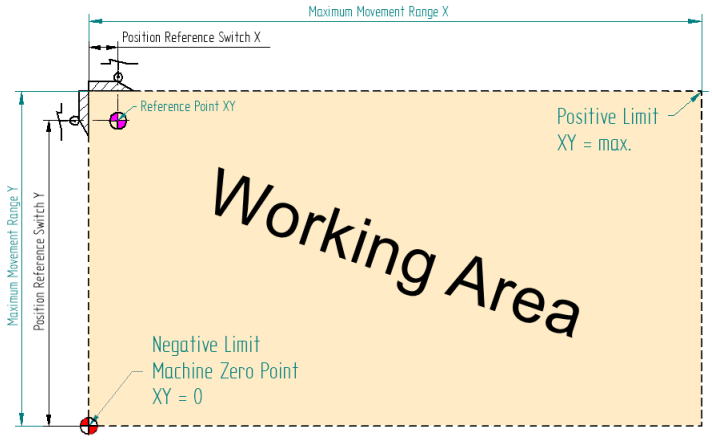
© 2019 PENTA-TEC CNC-Automation

5.2 Referencing And Reference Point / Machine Zero Point

Upon turning on the controller, the positions of the axes are unknown to machine and software (except for axes with absolute position encoder).
Hence, the axes have to be referenced first. In order to do this, the machine will move each axis towards its reference switch until it triggers, allowing for calibration.
The position of the reference switches should be positioned in such a way that referencing will not lead to unwanted collisions (for example between chucked tool and machine bed).
For that reason most Z-axis reference switches will be located at the upper limit.



The same general principle applies to the X- and the Y-axis.
The schematic picture below shows a machine with its X-axis reference switch at X-min, and the Y-axis reference switch at Y-max position.



Most switches allow for overtravel (as depicted above). After the initial contact the switch remains active, allowing the machine to stop without passing the switch entirely. This means that the switches will have to be placed a bit inside of the mechanical limits of the machine (which has to be taken into consideration when configuring the machine area).

It is paramount that the reference switches stay activated until the actual mechanical limit of each axis.
Otherwise the machine could potentially look for the reference switch after already passing it, resulting in damage to the machine and its components.

Reference point versus machine zero point:

© 2019 PENTA-TEC CNC-Automation

Commonly, the reference point and the machine zero point are used interchangeably.

This is, however, not entirely correct, because the machine zero point is calculated using the reference point but is not necessarily identical.

If the reference switch is mounted **at the negative limit of an axis**, the machine zero point is identical with the reference point (i.e. the point where the switch gets activated) unless you specify a value different from 0 in the **axis settings** under **position reference switch**. For example, using a value of -1mm would result in the machine zero point being 1mm away from the reference switch.

If the reference switch is mounted **at the positive limit of an axis**, the machine zero point results from the point where the reference switch gets activated minus the distance from the actual axis limit.

Example:
Our Y-axis has a travel distance of 750mm, the position of the reference switch is defined at 737.00mm.

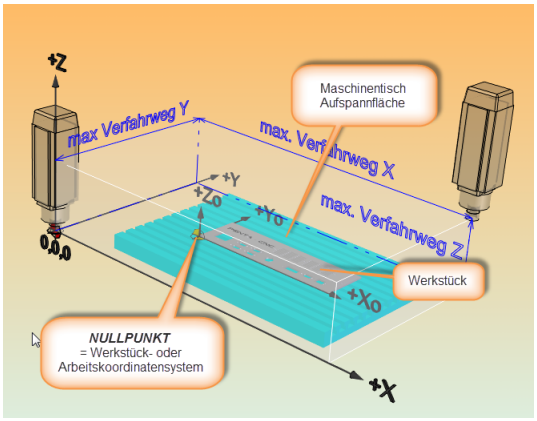
Procedure when referencing:

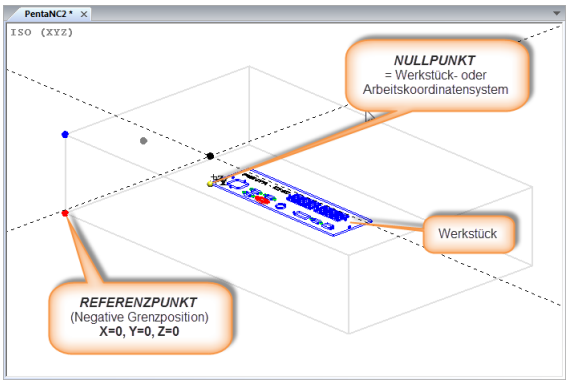
- *The Y-axis travels in positive direction towards the reference switch until the switch activates.*
- *The Y-axis comes to a stop while still activating the switch.*
- *The Y-axis reverses slowly until the switch opens (deactivates) and stops.*
- *The distance the axis needs to come to a complete stop after the switch is released is the braking distance. In our case this distance is 0.5mm.*
- *The calibrated position after referencing the Y-axis is now 737.00 - 0.5 = 736.50mm.*
- *In order to reach the machine zero point in Y, the Y-axis must move -736.50mm.*

© 2019 PENTA-TEC CNC-Automation

5.3 The Zero Point As Work Coordinate System

In practice, the reference point of a workpiece (for example one of its corners or the center of a circular object) will rarely—if ever—be identical with the machine zero point.
Therefore the user may define their own **work coordinate system** (i.e. **zero point**) which can be placed anywhere in the machine's coordinate system.





Corresponding display in PENTA-NC

XY-Position of the zero point:

The zero point should be located in such a way that it has a sensible position in regards to the workpiece.
A common position would be the bottom left corner or the center of a circular object.

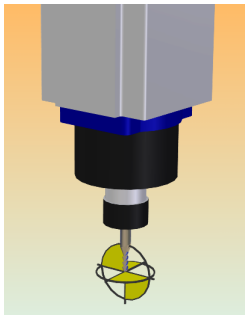
Z-Position of the zero point:

The Z-coordinate of the zero point will usually refer to the height of the plane on which the workpiece rests (machine bed, vaccum fastening device, ...).
If you prefer to use the surface of your workpiece as referencing position, you can choose to do so by clicking on the corresponding soft key.

More details in Setting Z-Zero Point.

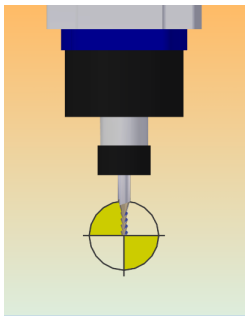
© 2019 PENTA-TEC CNC-Automation

5.4 Z-Zero Point And Tool Length



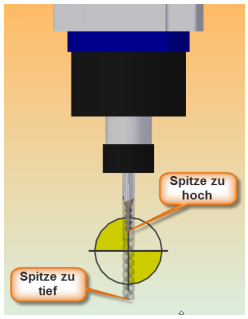
Every XY-positioning command to an NC-controller will usually refer to the central axis of the milling spindle.

In practice this means that when the machine head is positioned at X=0 / Y=0 the center of the tool (top of the drill, etc.) will be at position X=0 / Y=0.



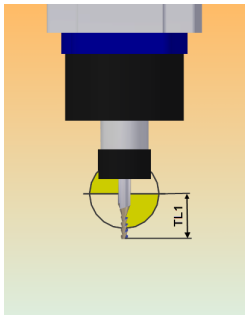
For Z-positioning the software must always take the current tool's length into consideration.

The tool center point (TCP) of the current tool will be at position Z=0 if you used it to define the zero point.



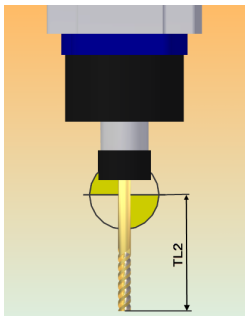
If you change tools, the tool tip may either be too low (longer tool) or too high (shorter tool).

In order to ensure that the TCP is always at the correct height, PENTA-NC stores an offset value for each tool.



For this to work, you must either enter each tool's length manually or determine it by gently touching the Z-zero point and transferring the value. Alternatively, you can automate the process by using a tool-length measure switch. For more details refer to the section Tool Library.

Once PENTA-NC knows the offset value of a tool, it can be positioned precisely.

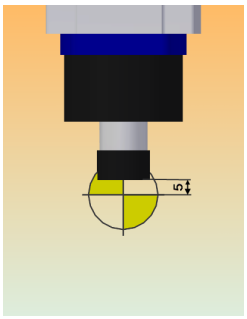


Once the correct offset values are saved for each tool necessary for milling your workpiece, the software will automatically adjust the height of the machine head accordingly.

It is paramount to ensure that each tool has its correct offset value!

© 2019 PENTA-TEC CNC-Automation

5.5 Setting Z-Zero Point



For systems with manual tool changing it is best to choose the collet (or collet nut) as reference surface. Alternatively, you may use a tool with known length (measured while already chucked).

Accordingly, for systems with automatic tool changer, you may want to use the empty tool holder's bottom surface.

Approach the table (or preferred reference plane) and upon touching, set the Z-zero point.

Setting the zero point:

In order to define your work coordinate system, switch to **SET** mode in the machine status window. Make sure that **Zero point setting** is active (you will see it right below the info area with the coordinates).
If it isn't, click on the soft button **SET ZERO-Position**.
After that, move your machine head to your preferred zero point.
Commonly you will use a tool with known length in order to determine X, Y and Z in one go.

Determining X and Y is rather easy (simply click on **Accept current X-position**, respectively **Y-position** once you reach the desired position), the trickier part is to correctly determine Z without damaging tool or machinery.
Carefully approach the table (or material) with your tool. Place a piece of paper underneath and proceed to lower the machine head slowly. Once you cannot move the paper freely underneath the tool, click **Accept current Z-position** and lift your machine head again.

You have now successfully configured your zero point.

Make sure to understand the difference (and correlation) between the machine coordinate system and your work coordinate system! You can find an explanation in The Zero Point As Work Coordinate System as well as on our Penta-Tec youtube channel.

© 2019 PENTA-TEC CNC-Automation

6 From Data To Workpiece

This section will show you how to load/import and edit your 2D CAD data in PENTA-NC.

You will also find detailed information on our Penta-Tec youtube channel.

6.1 File Import

Click on the PENTA-NC symbol in the top-left corner. In the menu choose **Open** to open your CAD file. This will open your file in a new graphic window in PENTA-NC.

Alternatively you can choose **Import** if you already have an open document and wish to import your file. This can be useful if you wish to place several of your files into one PENTA-NC file.

Choose your file and your preferred settings. For general purposes you can simply leave the default values as they are.
Once you import or load your file, the settings remain in the import filter.

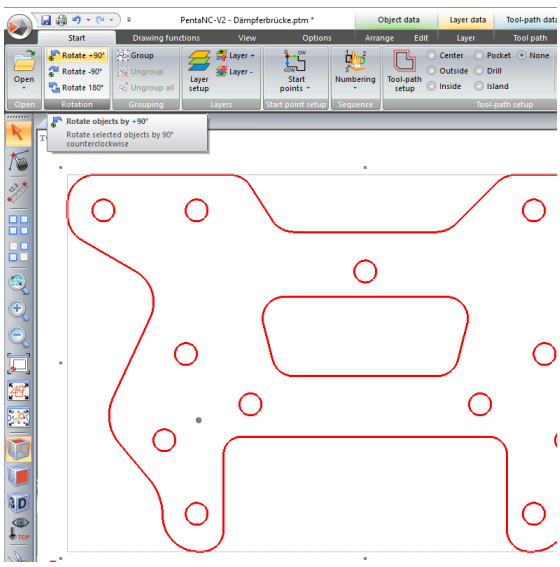
6.2 Preparing 2D Data

After importing the desired file(s), you can start to prepare your data for milling.
Until you have acquired a certain intimacy with PENTA-NC it makes sense to work through the **start** ribbon bar from left to right.

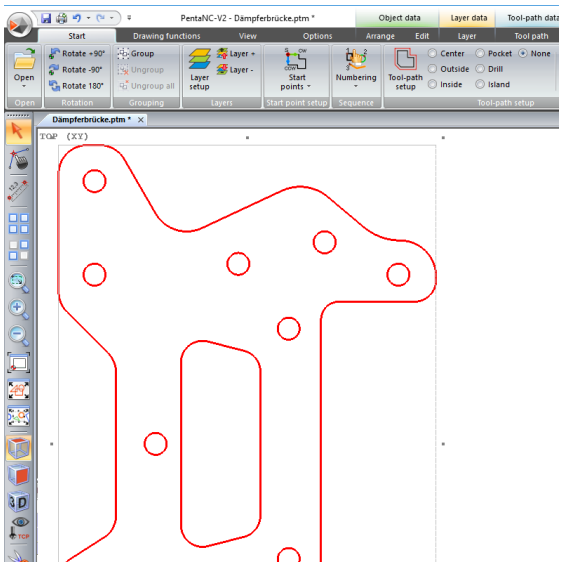
Before doing so, make sure you have already set your zero point as described in Setting Z-Zero Point.

Rotate:

You can rotate the object(s) as needed to arrange them in the working area.



Rotating objects

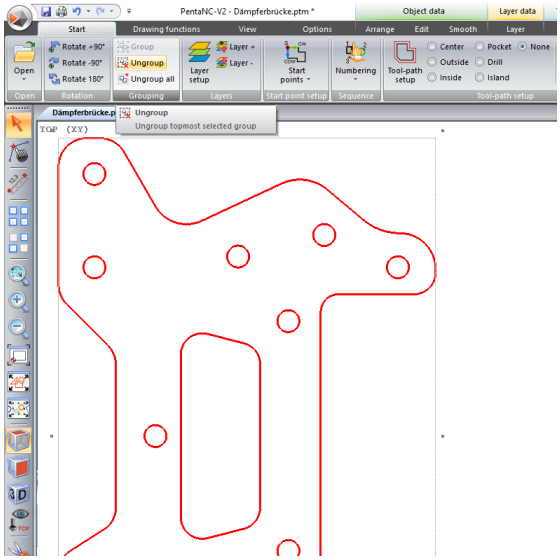


Object rotated 90°

© 2019 PENTA-TEC CNC-Automation

Ungroup:

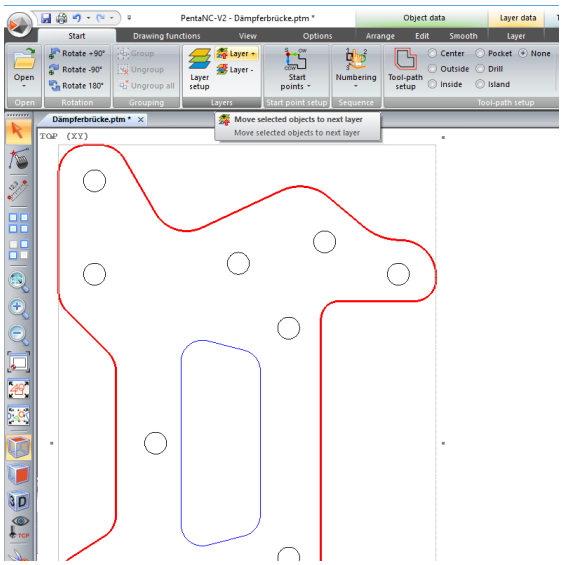
If you imported your objects (as opposed to opening or loading them), they should be grouped. You can now ungroup them to edit each object individually.
Simply choose the objects and click on the symbol **Ungroup**.



Ungrouping objects

Move objects onto different layers:

Now you can move various objects onto different layers. This allows for different parameters for machining (depth, feed, tool, ...) as these are stored in the layer settings.

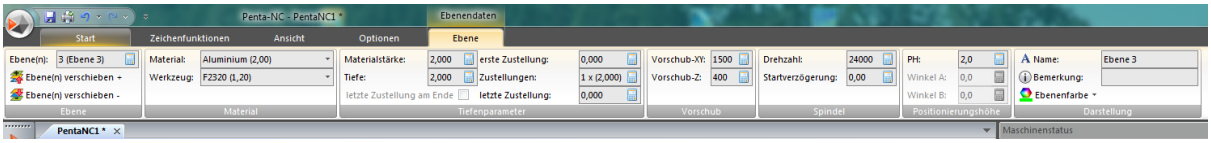


Moving elements to another layer

© 2019 PENTA-TEC CNC-Automation

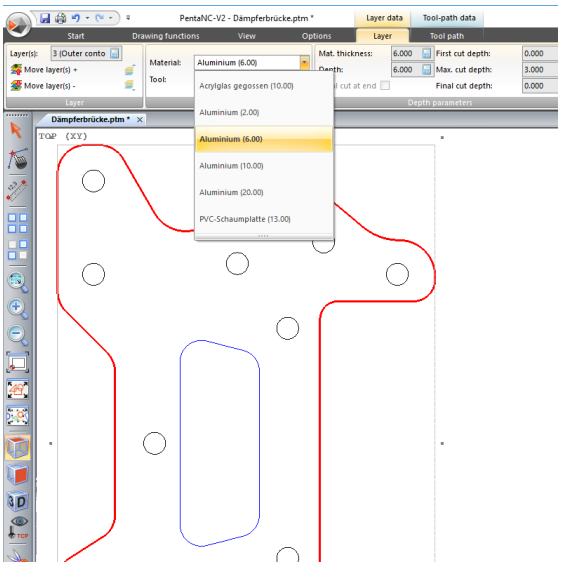
Set machining parameters:

By clicking on the soft button **Layer Setup** you can now access the layer definition.
Here you can configure all the parameters relevant for milling the layer elements. This includes material, thickness, tool selection, feed and depth, spindle RPM, and many more settings. You may also change the colour that PENTA-NC uses for displaying the layer in the graphic window.



Layer Setup

Depending on the parameter, you may set the details via a drop-down menu (material, tools) or by entering the according data (feed, RPM).
It makes sense to enter a meaningful name for your layer, especially if you use many different layers.



Choosing material

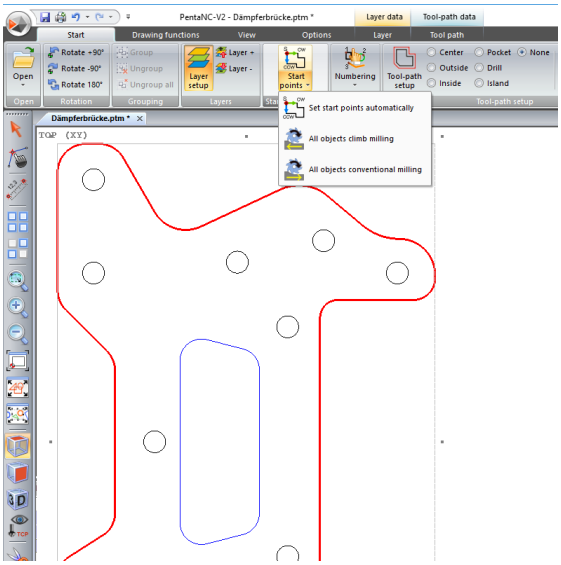
Start point and direction of milling:

The next step lets you define starting points and decide between climb milling and conventional milling.
By smart positioning of the starting points of each path, you can minimise the idle travel between milling.

After activating the soft button **Start points** you will notice that the mouse cursor changes design. You can now simply left-click on a path to define where the milling should start. A right-click on such a starting point will change the milling direction.
Click on the soft button again to stop assigning starting points.

Alternatively, click on the small triangle in the lower part of the **Start points** soft button to access the automated assignment of starting points. Click on **Set start points automatically** to access a menu that lets you choose which rules to apply.

© 2019 PENTA-TEC CNC-Automation

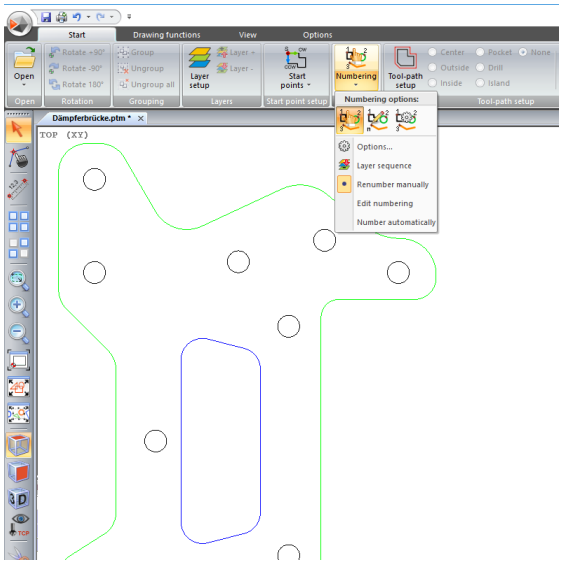


Assigning start points manually or automatically

Defining order of processing:

Next you can define in which order the paths shall be processed.
Click on the soft button **Numbering**. You will notice the mouse cursor changes design. Clicking on paths will number them successively. A right click on a path with an existing number of order will lower that number by one with each click.
After having clicked on all objects, the mouse cursor will revert to an arrow.

Alternatively, click on the small triangle in the lower part of the **Numbering** soft button to access automated assignment options. You will also find the option to manually renumber paths, which allows you to simply edit the existing order instead of having to go through each object one by one.
The pop-up info for each option is self-explanatory.



Numbering options

© 2019 PENTA-TEC CNC-Automation

Configuring tool offset for tool paths:

In order to tell PENTA-NC (and in order your machine) how to interpret the various paths, you need to define them in the next step.

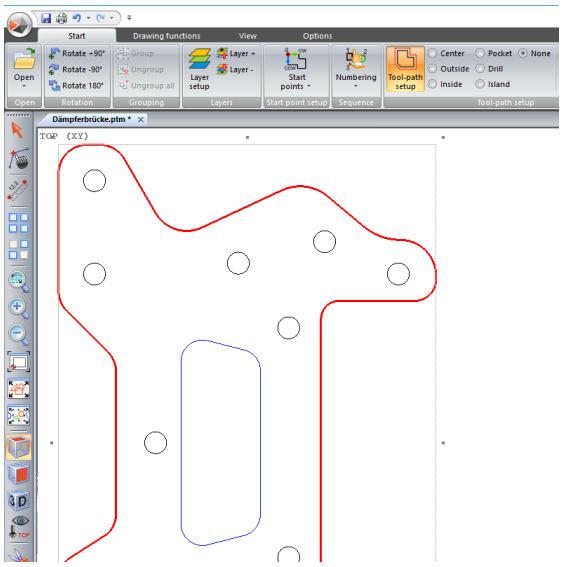
Click on the soft button **Tool-path setup** and notice that the mouse cursor now has the contour symbol next to the arrow. Now click on a path and define it next to the highlighted **Tool-path setup** button.
Additionally to choosing between various path options (outside, inside, pocket, ...) you can decide whether to use a lead-in and/or lead-out as well as an additional offset.

For example: If you define a path as *Outside*, you will see a dotted line outside of the path. This is where the center of the tool will actually move in order to make sure the cutting edges of your milling tool are positioned right on the path.

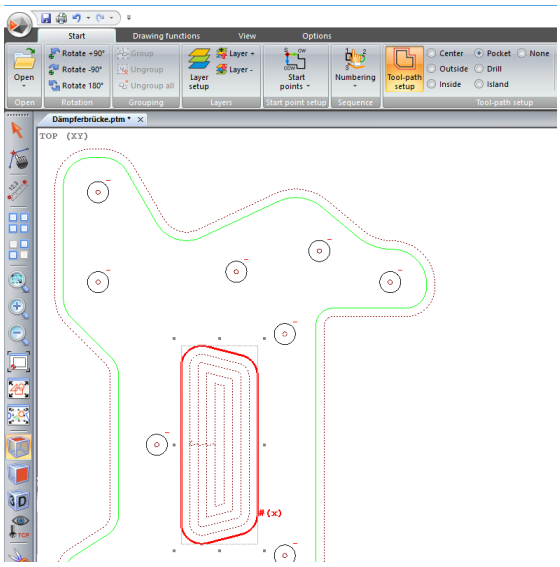
Changing path definitions (including lead-in, lead-out and offset) will take immediate effect and the tool path will be displayed as dotted line in the graphic window.

Once you are done assigning tool paths, simply click on **Tool-path setup** again.

Alternatively, you can always define or change a tool path for any path(s) you have currently selected.



Defining tool paths



Assigned tool paths (dotted lines)

© 2019 PENTA-TEC CNC-Automation

Arranging the milling object:

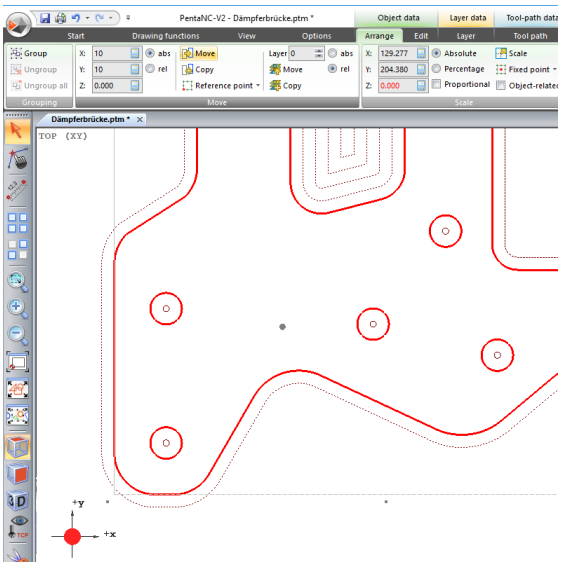
Make sure to place the object over the working area as well as the material you wish to process.
Select all elements by pressing Ctrl+A or clicking on the **Select All** button in the tool bar.

Next you click on the menu **Arrange** in the ribbon bar. This will allow you access to various modifications, including move, copy, rotate, mirror, scale, and so on.

Options in the ribbon bar to arrange objects

Let's assume you want to arrange your object so that its bottom-left corner is 10mm from the zero point (both in X and Y).
With all elements still selected, click on **abs** in the **Move** section of the ribbon. The values you will enter now become absolute coordinates (as opposed to relative distances).
Enter 10 in the boxes next to **X** and **Y**. Finally click on **Move**.

If you prefer a different reference point than the bottom-left corner of your object (for example for a circular object), simply click on **Reference point** and choose your preference in the pop-up menu.



Moving objects to absolute coordinates

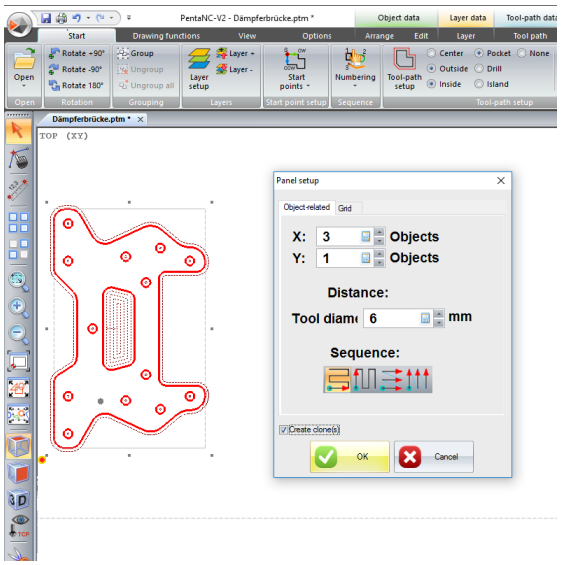
© 2019 PENTA-TEC CNC-Automation

Multiple milling of one object:

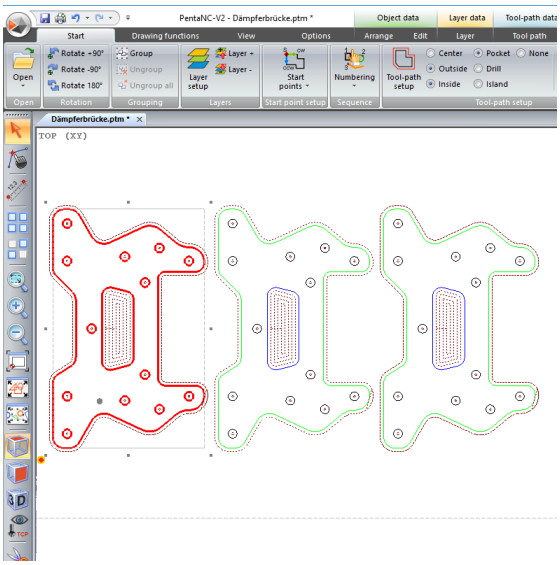
If you want to mill an object more than once in one process, you can set up a panel and duplicate the object. Simply click on **Panel setup**.

Once in the setup window, enter the amount of objects to be placed in X- respectively Y-direction. Keep in mind that the original object already counts as 1.
You can either define the arrangement of the new objects in the tab **Object-related** or **Grid**.

Use object-related panels to let PENTA-NC place the objects as closely together as your tool allows. The grid tab allows you the freedom to place the objects as you like.
If you check the box **Create clone(s)**, any changes to the original object will also occur in its clones. Also, the milling file will be smaller.



Creating clones in the panel setup



The created clones

Start milling:

After this preparation, save your file and switch to **AUTO** mode in the machine status window.

PENTA-NC will start milling your object(s) once you click on **START**.

© 2019 PENTA-TEC CNC-Automation

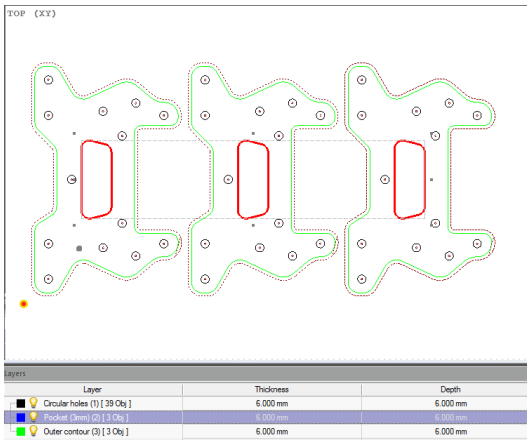
6.3 Preparing 2.5D Pockets

PENTA-NC allows you to define various milling depths for different layers. In order to make pockets in an object, you will therefore need to move the according elements onto a different layer.

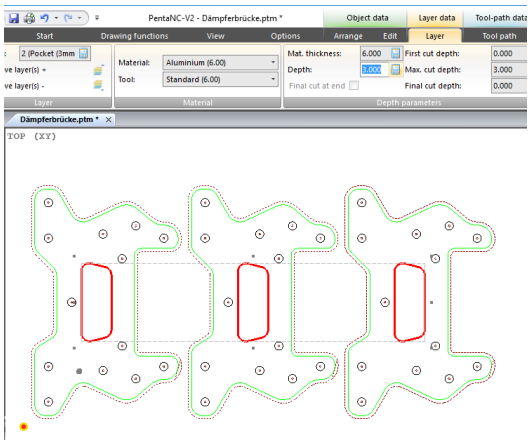
It is important that you understand the meaning and importance of layers in PENTA-NC if you want to make the best use of it. You will find more details in the chapter Layer Setup or on our Penta-Tec youtube channel.

Once you have placed all desired objects onto a new layer, you can define up to which depth the layer should me milled. Make sure to activate the correct layer.

PENTA-NC will highlight all objects on a selected layer to help avoid confusion.



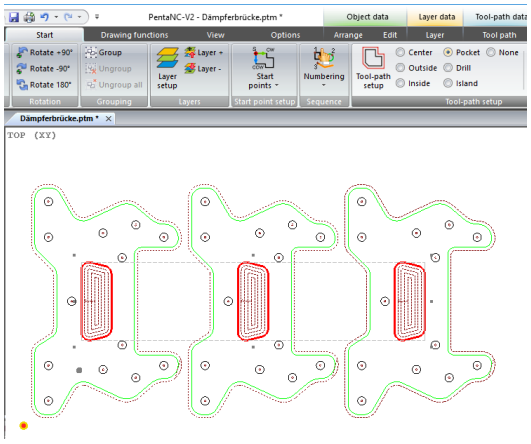
Choosing layer for pocket milling



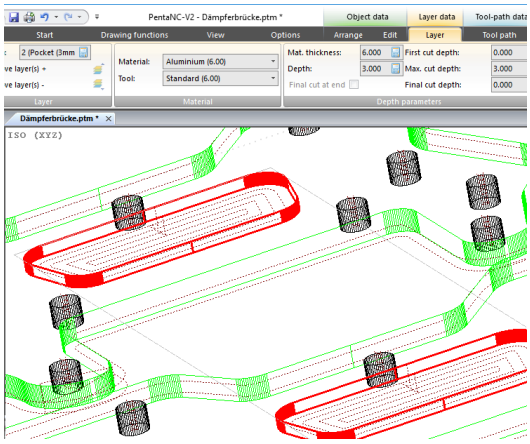
Setting the parameters in the layer setup (depth to 3mm)

After making sure all the parameters are set correctly, you must still define the objects as pockets. Do so by activating the tab **Start** in the ribbon bar and then choosing **Pocket** in the Tool-path setup.

PENTA-NC will immediately start calculating the required tool paths and you will see them inside the objects as dotted lines.



Defining tool path as pocket



Isometric view (notice that the pockets are 3mm deep while the material is 6mm)

© 2019 PENTA-TEC CNC-Automation

6.4 Drill Holes

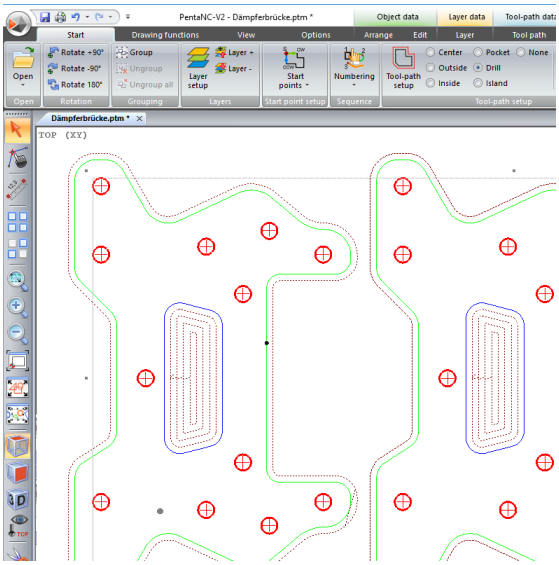
If you need circular holes in your object, you don't necessarily need to mill them. If you have the according tool, you can choose to drill them.

For example, for holes with a diameter of 6mm we simply choose a 6mm milling tool in the layer setup. We want to drill through the entire material, so we keep the depth equal to the material thickness.
These settings affect the entire layer.

Now we choose the actual elements on the layer we want to drill. With the selection active, we click on **Start** and then **Drill** in the **Tool-path setup**. All the selected elements should now have a cross in their centers, marking them as drill holes.

Important:
You do not necessarily have to move elements you want to drill to a different layer *if* the layer parameters (tool, thickness, depth, feed, ...) are the same as for the rest of the elements on that layer. A layer can have different tool-path definitions!
However, for work-flow related reasons, it will make sense to put all the elements with the exact same procedures on their own layer and name it accordingly. This will make it much easier to manipulate certain parameters if the need arises.
It will also allow you to easily create layer presettings for milling objects that are similar to the current one.

Therefore it is wise to make it a habit to use different layers for objects with different tool-path settings.



Defining objects as drill holes

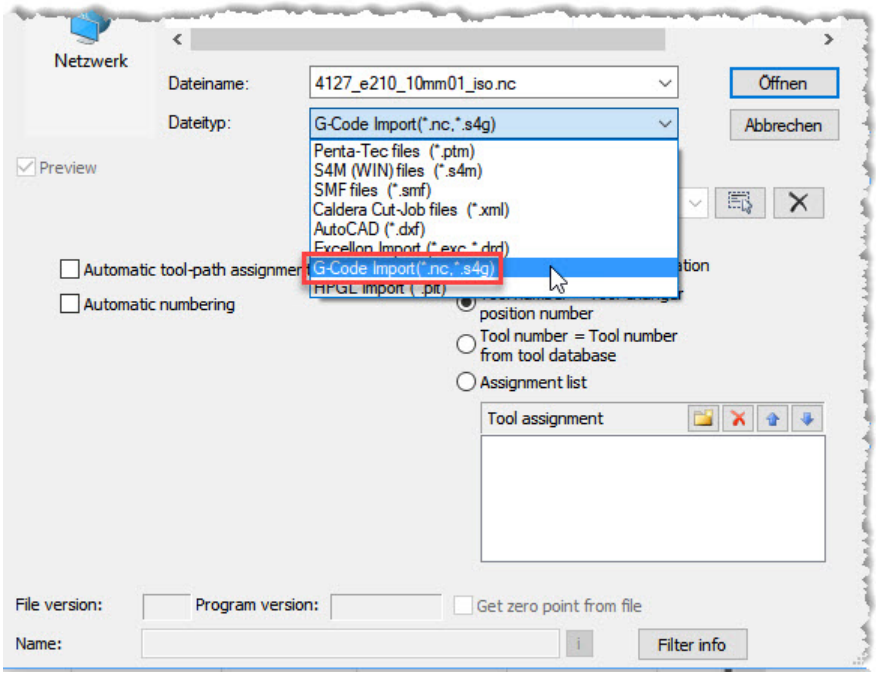
© 2019 PENTA-TEC CNC-Automation

7 Import from G-Code (DIN/ISO-Code) Files from 3rd party programs

If an external CAM system is used (for example, because you want to edit, for example, 3D objects) you can also import this data in PENTA-NC and work with this data to control your machine.

To do this, select the filter <G-Code Import> from the import filter list.

The file extension must be either in the form * .NC or * .S4G.



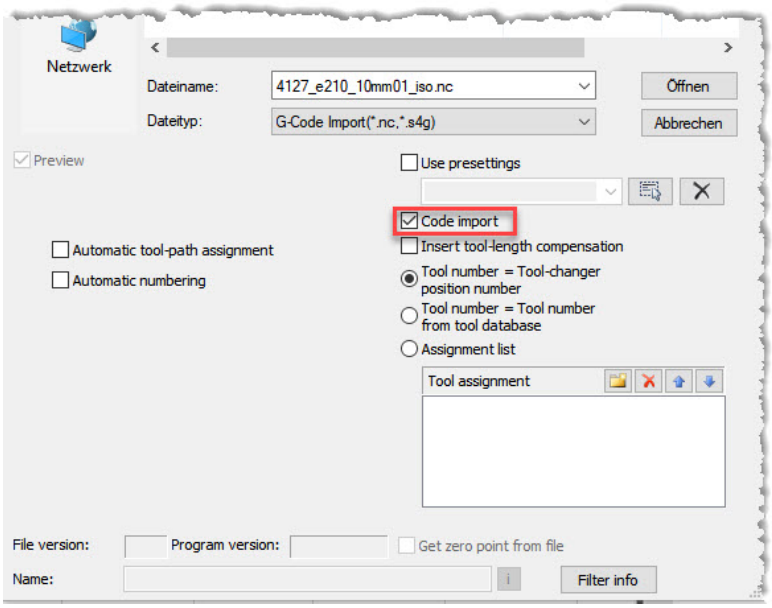
Important: Please note that there are two types of G-code import:

- Code Import ->The G-Code is forwarded almost unchanged to the machine control, as it was issued by the post processor.
- Geometry Import -> Here the graphic information is extracted from the G-Code and further processed as "normal" geometry objects.

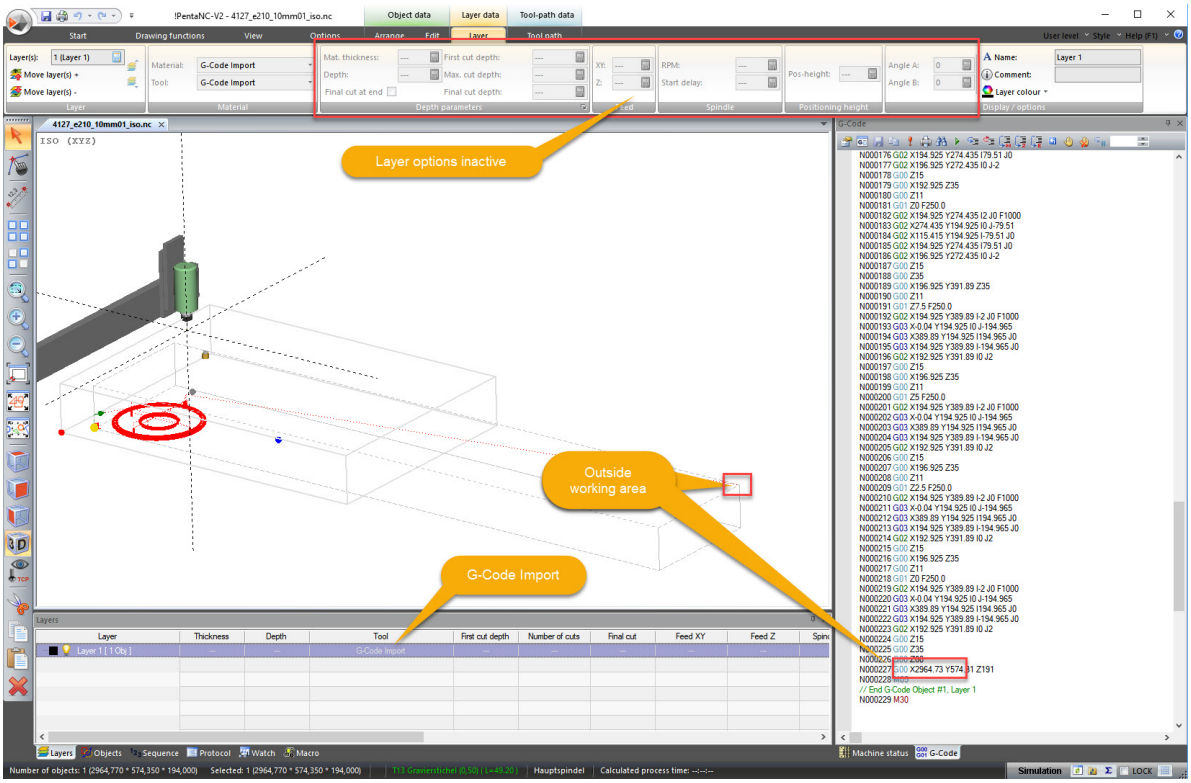
The difference between the two methods is explained in more detail below.

© 2019 PENTA-TEC CNC-Automation

Code Import G-Code:



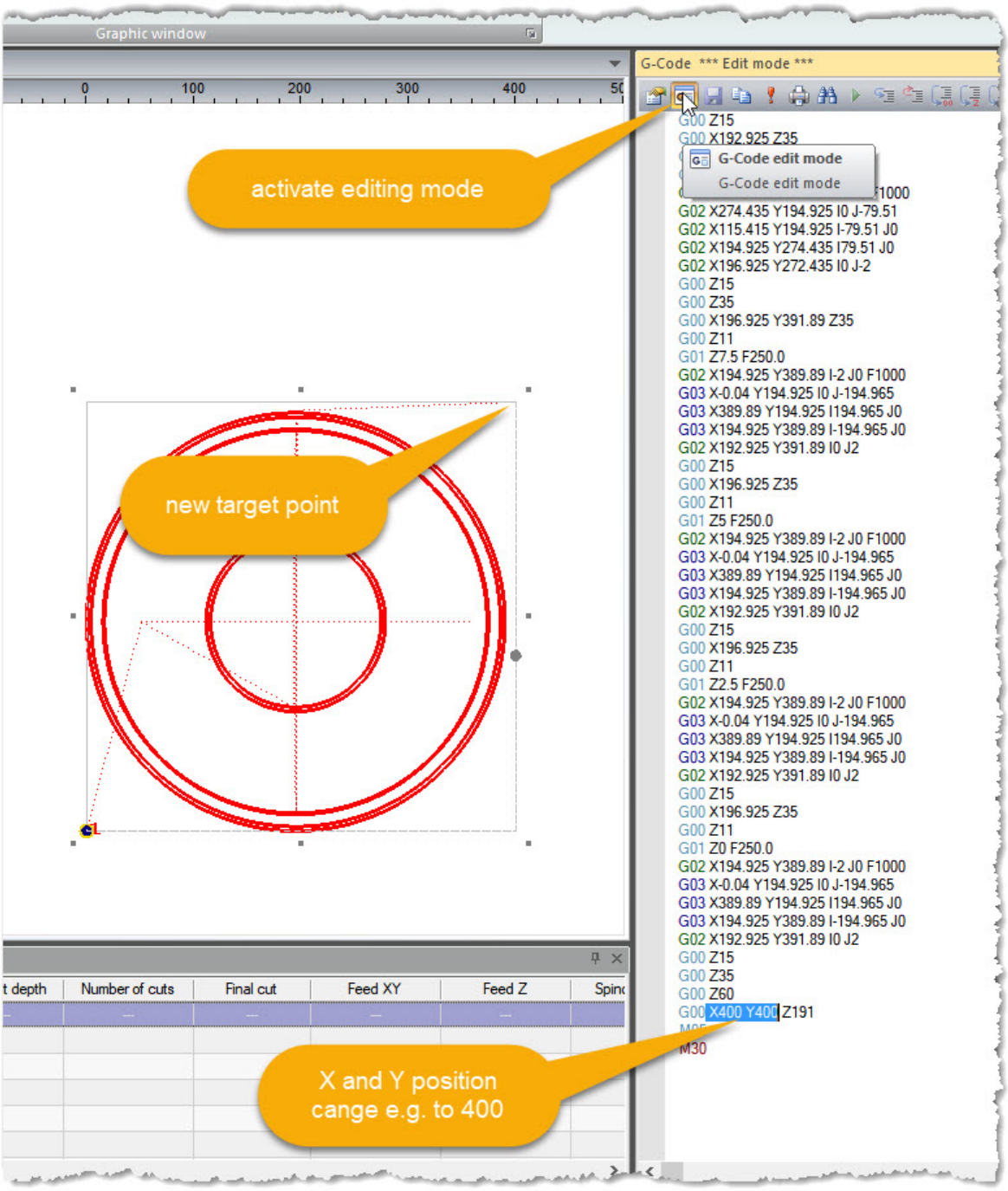
In most cases, the G-Code is created completely in the CAM system and contains all the information about the tools used, feed values, spindle speed etc. If the checkbox <Code Import> is activated, the G-Code will be adopted unchanged and forwarded directly to the controller.



The entire file is stored as a single G-Code object in a layer. However, most of the parameters for this layer are locked because these settings are taken directly from the G code.
In the example above, you can see that the last positioning command would cause positioning outside of the machine's work area. To correct such errors, the G-code can also be edited later.

© 2019 PENTA-TEC CNC-Automation

Editing the G-Code:

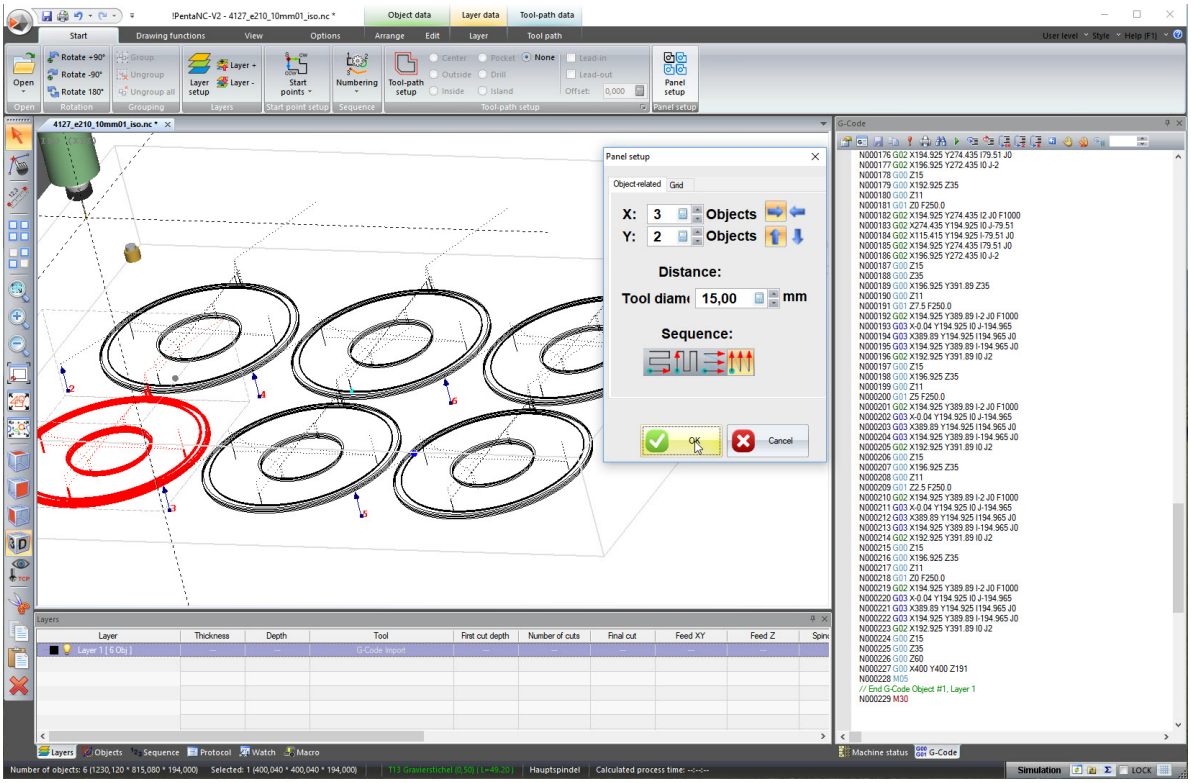


Any change to a positioning command will also be displayed graphically, and you will immediately see the effect of the change.

© 2019 PENTA-TEC CNC-Automation

Further processing of G-Code objects:

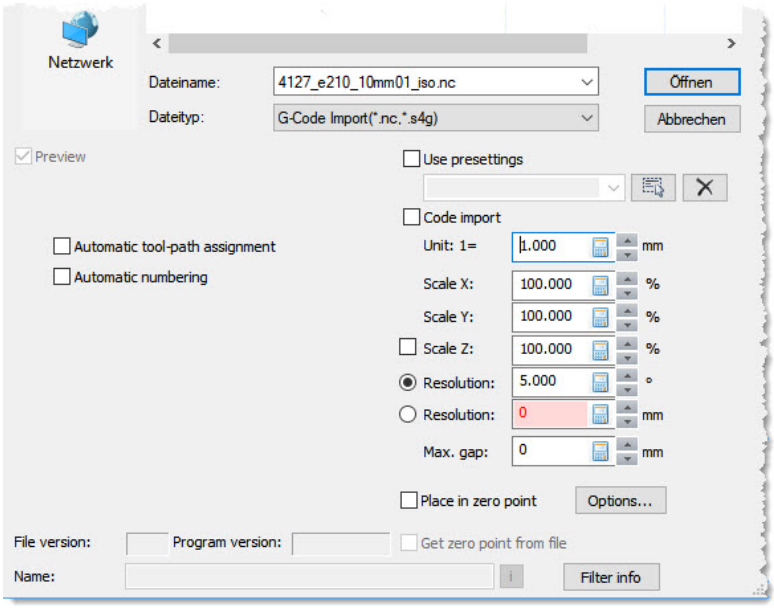
The big advantage of PENTA-NC is that G-code objects can be moved and duplicated, and so on as well as like normal milling objects.
Thus, e.g. From the previously individually imported and reworked G-Code a complete multiple cut are produced.



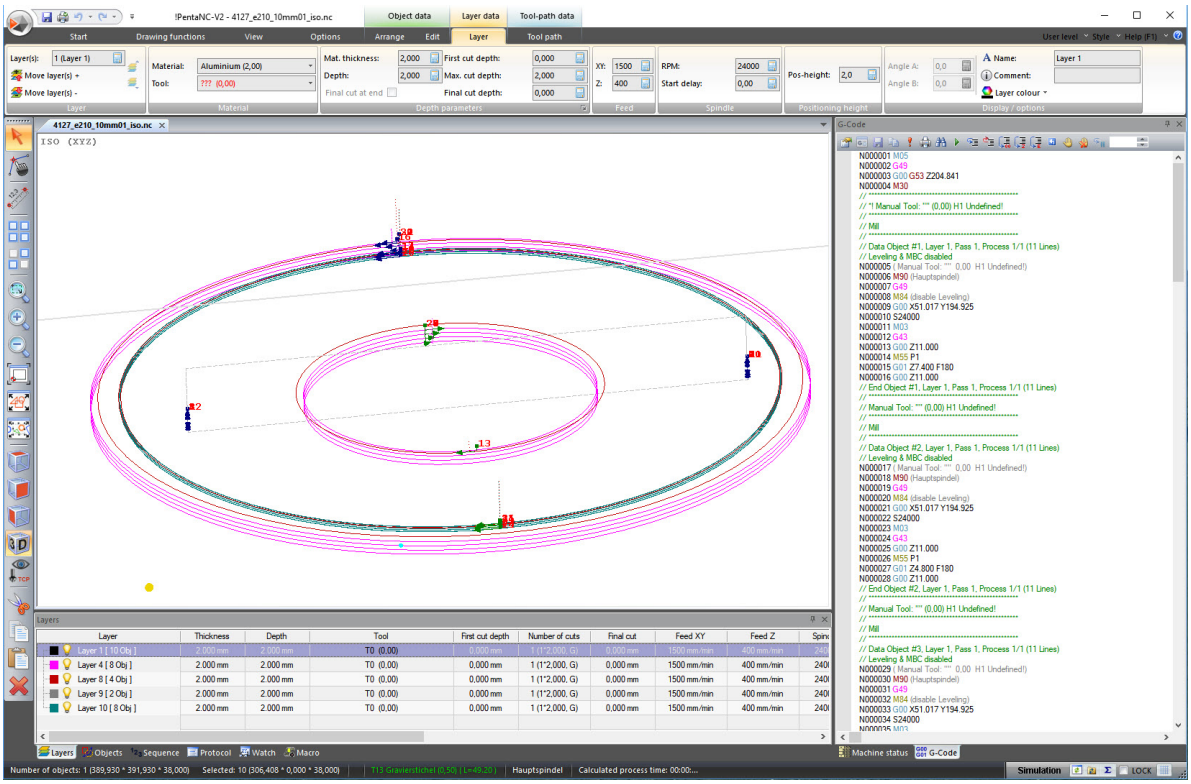
© 2019 PENTA-TEC CNC-Automation

Geometry import of G-code:

In some cases it may be useful to use only the graphic information from the G-code. The further processing should then be done as usual by assigning tool, feed rate, speed, etc. by the layer settings.



In this case, the tick in the checkbox <Code Import> is taken out. The G-code is then analyzed during the import and only the geometric information is used to create "normal" PENTA-NC geometry objects.



If several tools are used in a file, one layer is created for each tool and the associated objects are assigned to this layer. The machining parameters are taken from the respective layer definition.

© 2019 PENTA-TEC CNC-Automation

7.1 Information about the G-Code file structure

G-code data are basically standardized by DIN66025 / ISO6983. With the standardization, however, only the structure of the program code and some basic commands have been standardized.
Most manufacturers have also incorporated their own NC commands into their controllers, resulting in a number of "control dialects".
When the CNC program is output, the code is translated by a post processor in syntax understandable to the target controller.
PENTA-NC only use a subset of the entire DIN / ISO standard for the G-Code import filter. When customizing the post processor of your CAM system, make sure that only the commands listed in the table below are used. This is the only way to ensure that programs are correctly interpreted by the respective controller.

Command overview PENTA-NC DIN-ISO import filter:

Befehl	Parameter	Beschreibung
(		Indicates the beginning of a comment
)		Indicates the end of a comment
%		Indicates the program start
F	nnnnn	Definition of feed in mm / min. - before or in the line of the positioning command
G00	X Y Z	Positioning at rapid traverse speed to position X, Y, Z
G01	X Y Z	Position with working feed from the act. Pos. To position X, Y, Z Feed F must be set
G02	X Y I J	Circular interpolation CW / X Y = target position, I J = specification of the center point relative to the actual position
G03	X Y I J	Circular interpolation CCW / X Y = target position, I J = specification of the center point relative to the actual position
G04 F	H	H = dwell time in Seconds
G43		Tool length compensation ON - (is deleted by tool change). The tool length compensation is only active with the following positioning command.
G49		Tool length compensation OFF
G53		Absolute positioning ON (non-modal is therefore always used in combination with G0 or G1)
G70		All following coordinates in inches
G71		All following coordinates in millimetrs (default)
S	nnnnnn	Definition of the spindle speed in rpm (is set by M03 / M04)
M03		Spindle ON counter clockwise rotation (S must be set)
M04		Spindle ON clockwise rotation (S must be set)
M05		Spindle stop
M06	nn	Tool change to tool with the number nn When changing tools, the length compensation G43 is deleted and must be reseted

© 2019 PENTA-TEC CNC-Automation

Befehl	Parameter	Beschreibung
M30		end of program
M74		Cooling ON
M75		Cooling OFF
M76		Suction ON
M77		Suction OFF
M78		Vacuum ON
M79		Vacuum OFF
M81		Machine bed leveling ON
M80		Machine bed leveling OFF

Example of the structure of a G-code file:

To read a G-code file through the PENTA-NC software, the structure of the file must comply with certain rules.
The example file below contains 3 squares. Between the first and second square a tool change is proceeded. The transition between square two and three takes place without tool change

Befehl	Beschreibung
(Penta-Tec NC1)	Comment is not interpreted and must be in round brackets
%	Defines the program start
G71	Sets mm as a unit
(Start Data Object #1, Layer 1, Pass 1)	Comment is not interpreted and must be in round brackets and own line
(Tool #2 HM-2mm)	Comment is not interpreted and must be in round brackets and own line
M06 02	Toolchange to tool #2
G00 X0.000 Y40.000	Rapid travel to the X, Y position of the first object
S15000	Set the spindle speed to 15,000rpm
M03	Spindle ON clockwise direction
G43	Tool length compensation on
G00 Z10.000	Rapid travel to the approach hight
G01 Z1.000 F400	Positioning from X0 Y40 Z10 to X0 Y40 Z1 with a feed value of 400mm / min
G01 Y0.000 F500	Positioning from X0 Y40 Z1 to X0 Y0 Z1 with a feed value of 500mm / min
G01 X40.000	... processing object 1
G01 Y40.000	... processing object 1
G01 X0.000	... processing object 1
G00 Z10.000	Rapid travel to the approach hight
(End Data Object #1, Layer 1, Pass 1)	Comment ...
(Start Data Object #2, Layer 1,	Comment ...

© 2019 PENTA-TEC CNC-Automation

Pass 1)	
(Tool 3 HM-6mm)	Comment ...
M06 03	Toolchange to tool #3
G00 X50.000 Y40.000	Rapid travel to the X, Y position of object #2
S10000	Set the spindle speed to 10,000rpm
M03	Spindle ON clockwise direction
G43	Tool length compensation on
G00 Z10.000	Rapid travel to the approach hight
G01 Z1.000 F400	... processing object 2
G01 Y0.000 F500	... processing object 2
G01 X90.000	...
G01 Y40.000	...
G01 X50.000	...
G00 Z10.000	Rapid travel to the approach hight
(End Data Object #2, Layer 1, Pass 1)	Comment ...
(Start Data Object #3, Layer 1, Pass 1)	Comment ...
G00 X100.000 Y40.000	Rapid travel to the X, Y position of object #3
G00 Z10.000	Rapid travel to the approach hight
G01 Z1.000 F400	... processing object 3
G01 Y0.000 F500	...
G01 X140.000	...
G01 Y40.000	...
G01 X100.000	...
G00 Z10.000	Rapid travel to the approach hight
(End Data Object #3, Layer 1, Pass 1)	Comment ...
M05	Spindle STOP
G49	Tool length compensation OFF
G00 G53 Z300	Rapid travel to absolute position Z300
G00 G53 X500 Y400	Rapid travel to the absolute position X500; Y400
M30	End of programm

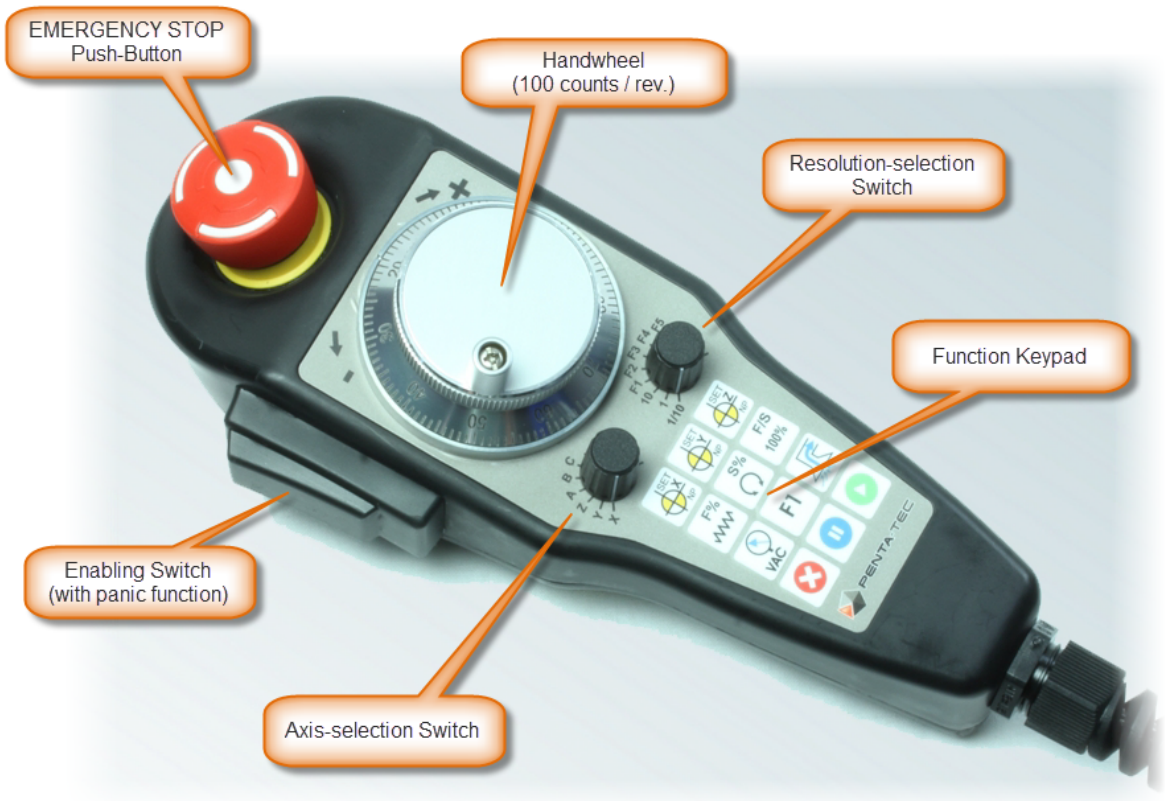
© 2019 PENTA-TEC CNC-Automation

© 2019 PENTA-TEC CNC-Automation

8 Manual Control Unit RCS01 (BECKHOFF-System)

The flexible connection of the control unit allows for manual control from the optimum vantage point at all times. This makes it much easier to set up the machine and monitor any processes.

The control elements are as follows:



Control element	Function	Remarks
E-STOP Palm button	Stops the machine in an emergency situation.	The details of the emergency shutdown procedure are dependent on the danger potential of any given machine. The required performance level and security structure is to be determined by the machine manufacturer, based on a risk analysis.
Enabling button	Certain functions can only be accessed when the enabling button is activated.	Functions in need of an activated enabling button will be designated with ⚠ in our list below. The panic function means that the button has to be kept inside a sweet spot. Releasing or pushing too hard (for example when panicking) will both deactivate the button.
Hand wheel	The hand wheel generates 100 pulses per rotation.	The distance an axis moves per pulse is determined by the resolution switch.

© 2019 PENTA-TEC CNC-Automation

Control element	Function	Remarks
Resolution switch	Defines the resolution of hand wheel and axis	• 1/10mm / hand wheel rotation = 1/1000mm / scale line • 1mm / hand wheel rotation = 1/100mm / scale line • 10mm / hand wheel rotation = 1/10mm / scale line • F1 to F5 = feed value (1 = low ... 5 = fast)
Axis selection	Choose an axis	• X, Y, Z, main axes (linear) • A, B, C, rotation axes (parallel to X, Y, Z)
Function keypad	Allows the execution of various special functions	Depending on the current operational state of the machine (G-code running, manual mode, etc.) different functions will be available. For details consult the functions keypad table below.

Functions keypad

Control element	Enabling button	Description
	⚠ active	The active axis will be moved by turning the hand wheel with activated enabling button. Clockwise -> axis moves in positive direction Counter clockwise -> axis moves in negative direction *When feed or spindle override functions are active (see below), the respective override values will be changed instead of moving the axis!*
		**Set zero point X** • Press the button -> LED flashes slowly • Keep the button pressed until LED flashes quickly • The current position is now saved as X-zero
		**Set zero point Y** • Press the button -> LED flashes slowly • Keep the button pressed until LED flashes quickly • The current position is now saved as Y-zero
		**Set zero point Z** • Press the button -> LED flashes slowly • Keep the button pressed until LED flashes quickly • The current position is now saved as Z-zero
	⚠ active	**Feed override:** (only valid when running G-code) Press the button -> LED on Change the feed override with the hand wheel. Allowed values: 0% to 300%

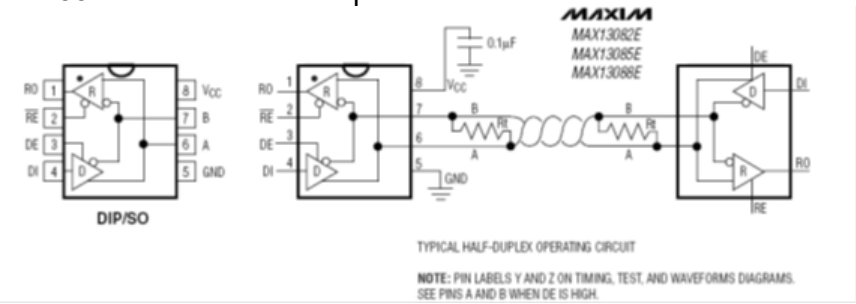
© 2019 PENTA-TEC CNC-Automation

Control element	Enabling button	Description
	⚠ active	**Spindle override:** (only valid when running G-code) Press the button -> LED on Change the spindle RPM override with the hand wheel. Allowed values: 10% to 300%
	⚠ active	**Reset override values:** • Push the button to reset feed and/or spindle override to 100%.
		**Vacuum ON / OFF** (May be a different function at the machine manufacturer's discretion.)
		Function defined by machine manufacturer.
		**Extraction ON / OFF** (May be a different function at the machine manufacturer's discretion.)
		**Abort process** (Only if a program is being executed)
		**Pause process** (Only if a program is being executed)
	⚠ active	**Start process**

© 2019 PENTA-TEC CNC-Automation

8.1 Structure and Hardware

- Master (controller) ⇔ Slave (RCS01)
- RS485 bidirectional half-duplex:



- Extended ESD protection for RS-485/RS-422 I/O pins ±15kV human body model
- True fail-safe receiver while maintaining EIA/TIA-485 compatibility
- Hot-swap input structures on DE and RE
- Termination: B with 1k to +5V, A with 1k to mass R_t=150R

RS485 port settings:

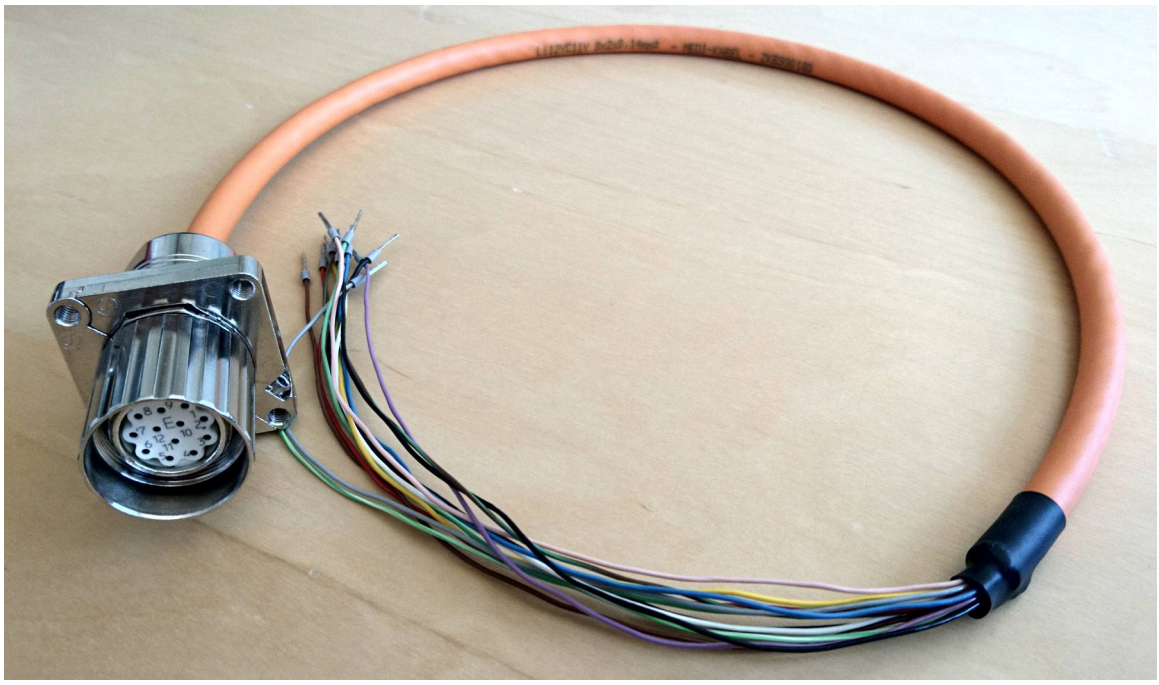
19,2kBit (standard setting), 8 data bits, no parity, 1 stop bit, without flow control

Connector pinout:

M23 circular connector, 12-pin (EPIC 44420037 + 73002716).
The wire colours in the table correspond the colours at the loose end of the pre-fabricated connector.

Pin	Function	Wire number	Colour
2	24V+ DC supply	1	white
3	24V- DC supply / RS485 GND	2	brown
1	RS485 B	3	green
4	RS485 A	4	yellow
5	E-STOP button, circuit 1, contact 1	5	grey
6	E-STOP button, circuit 1, contact 2	6	pink
7	E-STOP button, circuit 2, contact 1	7	blue
8	E-STOP button, circuit 2, contact 2	8	red
9	Enabling button, circuit 1, contact 1	9	black
10	Enabling button, circuit 1, contact 2	10	purple
11	Enabling button, circuit 2, contact 1	11	white/green
12	Enabling button, circuit 2, contact 2	12	brown/green

© 2019 PENTA-TEC CNC-Automation



© 2019 PENTA-TEC CNC-Automation

9 Manual Control Unit RCSmart

When using Eding CNC controllers, you may opt to use the manual control unit **RCSmart**. It is a simplified variation of the **RCS01** control unit for Beckhoff. Be aware that this unit does not have an enabling button (and thus no panic function).

The control elements are as follows:

Control element	Function	Remarks
E-STOP Palm button	Stops the machine in an emergency situation.	The details of the emergency shutdown procedure are dependent on the danger potential of any given machine. The required performance level and security structure is to be determined by the machine manufacturer, based on a risk analysis.
Hand wheel	The hand wheel generates 100 pulses per rotation.	The distance an axis moves per pulse is determined by the multifunctional **Pause** button (see table: Functions Pause button).
Run button 	Multifunctional button	The **Run** button has multiple functions. For details see table: Functions Run button.
Pause button 	Multifunctional button	The **Pause** button has multiple functions. For details see table: Functions Pause button.

© 2019 PENTA-TEC CNC-Automation

Functions Run button

Press	Function	Remarks
short	Switch modes	Every short push of the Run button flicks through the modes. The sequence of the endless loop is **AUTO > MAN > SET**.
long in **AUTO**	Start processing	Holding down the button in **AUTO** mode starts the milling of the current file.
long in **MAN**	No function	
long in **SET**	Zero position	Holding down the button in **SET** mode sets the present position as zero coordinate of the currently active axis.

Functions Pause button

Push	Function	Remarks
short	Switch through axes	Every short push of the Pause button flicks through the available axes. The sequence of the endless loop is **X > Y > Z**. If axes **A**, **B** and **C** are activated, they will follow the main axes in this order. After the final active axis the next push will activate the navigation pad (use keyboard or mouse) before starting with the loop again.
long during milling	Pause function	Holding down the Pause button while milling will put the machine in pause.
long in MAN or SET	Change resolution	Holding down the Pause button in **MAN** or **SET** mode will change the resolution of the axis movement per hand-wheel revolution by a factor of 10. The sequence of the endless loop is 1 > 0.1 > 0.01. The factor stays active even if you switch to a different axis.

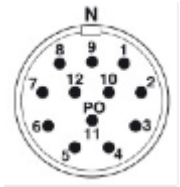
© 2019 PENTA-TEC CNC-Automation

9.1 Structure and Hardware

Connector pinout:

M23 circular connector, 12-pin (EPIC 44420037 + 73002716).

The wire colours in the table correspond the colours at the loose end of the pre-fabricated connector.



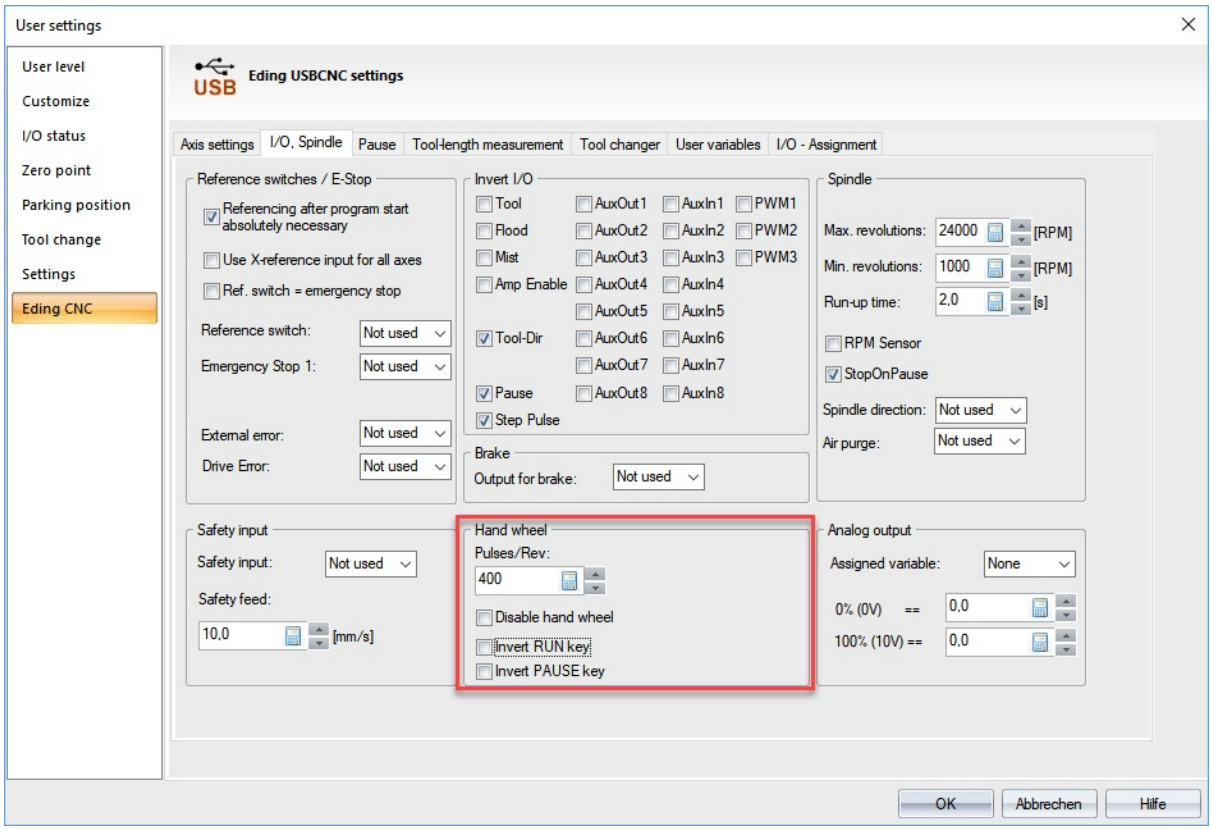
Pin	Function	Wire number	Colour
	Pendant (J13)		
3	Hand wheel track A	1	green
11	Power supply 5V+	2	white
12	Power supply GND	3	brown
4	Hand wheel track A/	4	yellow
5	Hand wheel track B	5	grey
6	Hand wheel track B/	6	pink
1	IN-RUN	7	red
2	IN-PAUSE	8	blue
	Safety (J8)		
3	K1-IN	9	purple
4	K1-OUT	10	black
5	K2-IN	11	green/white
6	K2-OUT	12	green/brown

© 2019 PENTA-TEC CNC-Automation

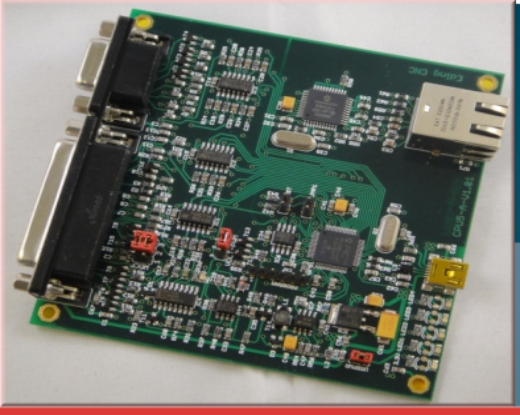
9.2 Settings in PENTA-NC

To ensure that the handwheel works correctly with the PENTA-NC software, the following settings must be made in the menu
<User Settings -> Editing CNC - IO, Spindle>..

© 2019 PENTA-TEC CNC-Automation



© 2019 PENTA-TEC CNC-Automation



PENTA-TEC

©PENTA-TEC CNC-Automation.e.U.
Lagerstrasse 1, A5082 Grödig
Tel.: ++43 6246 73642
e-mail: office@penta-tec.com
www.penta-tec.com