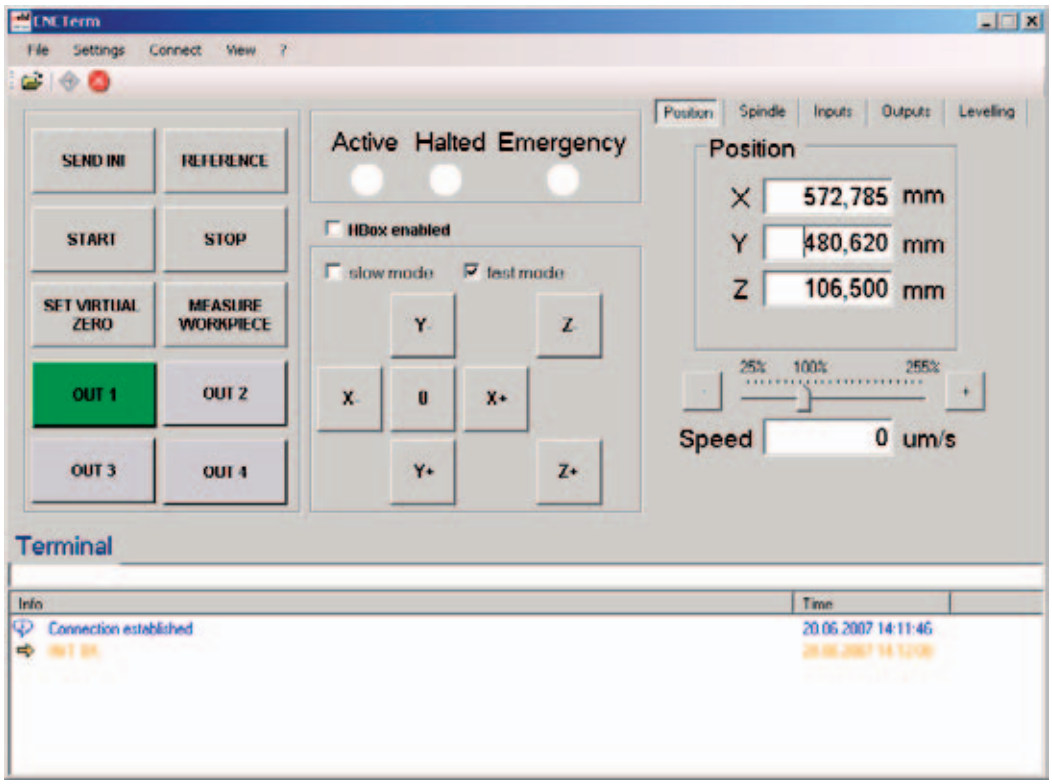
Technical Documentation

Manufacturing Software



June 2007

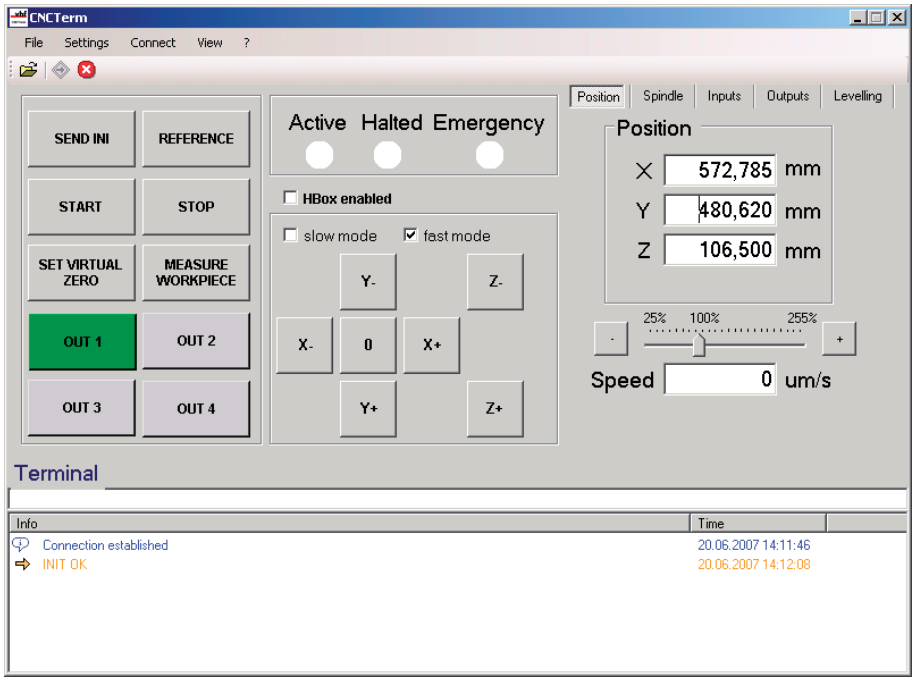
User Guide CNC Term

vhf camfacture AG ■ Lettenstraße 10 ■ 72119 Ammerbuch ■ Phone: +49 (0)70 32 970 97-0 ■ www.vhf.eu ■ info@vhf.eu

Output of Machining Data with CNC Term – Step by Step

0. Preliminary note

CNC Term is an interface between the data generated by the post processor and the machining system of vhf. All necessary setup tasks can comfortably be carried out here. Feed rate and rotational speed can be varied during the output. Furthermore, machine instructions can be entered directly via a terminal window. The installation of CNC Term requires an existing Microsoft .NET platform on systems from Windows 2000 upwards.



1. Generation of machining data via post processor

Safeguard that you have correctly placed the workpiece origin in your CAM programme. It has to be approached later. Continue to mind that the appropriate tool numbers and tool parameters are correct. Then generate a file with the milling data with the post processor of your CAM programme.

Most often several tools are intended for processing the workpiece. If your system is equipped with an automatic tool change, you can create a single file which includes all steps and the corresponding tool change commands. If you change your tools manually, you should make a separate file for each step of processing.

Mind that the post processor generates the data in a format which is compatible with the controllers CNC 550/580/800/950/980 of vhf.

2. Communication between CNC Term and the controller

Start CNC Term and click in the menu *"Connect"* on *"Start"* (or click on the corresponding symbol in the icon bar). On successful connection with the controller you receive the confirmation *"Connection established"* in the terminal window. In case you get an error message, please check whether all components are switched on, are correctly connected by cable with each other and whether all interface settings in CNC Term are correct (menu *"Settings"*, item *"Interface"*).

3. Sending the initialisation file

a) You either received one or several .ini files by us. If you have got several initialisation files, please safeguard to load the file which is designated for your intended application. You load the file in question via the menu *"Settings"*, item *"InitFile"*. After restarting CNC Term the file which was loaded latest is reloaded by default.

b) Send the file which was loaded just now by clicking on the button *"SEND INI"*. The processing of the file by the controller may take a few seconds. The correct processing of the file is confirmed in the terminal window by the message *"INIT OK"*. The machine carries out a reference move and stops at the machine origin.

4. Loading the machining data

Load the file which should be processed by clicking in the menu *"File"* on *"Open"*, respectively on the corresponding symbol in the icon bar on the very left.

5. Clamping a tool

In order to approach the workpiece origin, a tool has to be clamped in the spindle. If your system has no automatic tool change, you manually clamp the tool which is used for the next machining step.

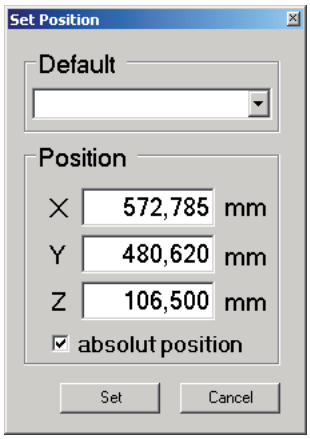
If your system has an automatic tool change, fetch the tool in question by entering the corresponding command into the *"terminal"* line. Enter (without inverted commas) the expression "Tx;" whereas x stands for the number of the designated tool. The semicolon directly activates the command. To fetch, for example, the first tool, the entry would be "T1;". Using the command "T0;" you return the tool.

6. Approaching the workpiece origin

Approach the workpiece origin which you have determined in the CAM software. For this purpose you have got various possibilities:

a) Moving via control buttons in CNC Term: By clicking the corresponding buttons for x, y and z, you move the system in positive or negative direction. If no speed option is set, the system moves 1 mm per click. If the option *"slow mode"* is set, the system moves 0.1 mm per click. If the option *"fast mode"* is set, the system moves with maximum positioning speed and that as long as the button is pressed. Hence be very careful in this mode. By clicking on the field *"0"*, the system moves in fast motion back to the current zero point and stands still (also see point 7 – Setting of the virtual zero point).

b) Direct entry of values: Click under "Position" on one of the fields which show the system position in x, y or z direction. You then get an input window with which you can approach exactly your designated position. Here you may also use the position memory management, in which you can, under "Default", save and name different starting positions which you use more frequently.

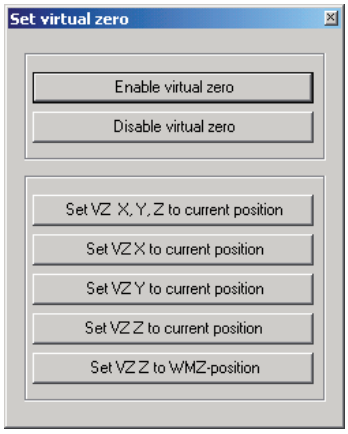


If the box for "absolute position" is ticked, the machine origin is considered as starting point and not a potentially set virtual zero point (also see point 7). Hence be careful approaching a position when a virtual zero point is set and this box is not ticked. All entered values are added to the virtual zero point here and are not calculated from the machine origin. The entry of oversized values may particularly result in letting down the z axis too far and in damaging the clamping area, workpiece, tool and spindle.

c) Moving via control panel: If your system is equipped with the vhf control panel, you may approach the designated starting position with it. For this purpose the option *"Hbox enabled"* has to be set in the main window. The respective current position is shown in CNC Term and may e.g. also be directly taken into the position memory.

7. Setting the virtual zero point

The zero point which is set in the CAM programme corresponds with the virtual zero point in CNC Term. After having approached the designated workpiece origin, as indicated above, click on *"Set Virtual Zero"* and you get various possibilities to define the virtual zero point.



a) If you click on "Set VZ x, y, z to current position" the current values for x, y and z of the main window are set as new virtual zero point (VZ). If you return to the main window, all these values have now become 0.

b) Depending on your preferences, you can also set the virtual zero point (VZ) for each axis (x, y, z) individually using the 3 corresponding buttons below.

c) The last button "Set VZ to WMZ position" sets the virtual zero point (VZ) in z direction to the value determined by the automatic z adjustment (WMZ = Workpiece Measurement Z) taking account of the thickness of the small plate (also see point 8).

   If the value determined by the automatic z adjustment should be set as virtual zero point in z direction, please activate in the menu "Settings" the item "VNZ after WMZ" (Virtual Null Z after Workpiece Measurement Z) with a tick. Each time you carry out an automatic z adjustment, the virtual zero point in z direction is then set newly.

d) A currently set virtual zero point can be disabled by clicking on *"Disable virtual zero"*. Afterwards the absolute values of the machine origin are displayed in the main window. By *"Enable virtual zero"* the virtual zero point can be activated again. By clicking on *"Reference"* in the main window, the machine executes a reference move and remains at the absolute machine origin. In doing so, a virtual zero point is deleted.

8. Automatic z adjustment

The automatic z adjustment unit serves for measuring absolutely exactly the workpiece origin in z direction – thus the workpiece surface – within seconds. For this purpose the workpiece itself has no longer to be scratched:

a) Move in x and y direction a few centimetres over the workpiece so that you have sufficient room to apply the adjustment unit safely to the workpiece. The levelling procedure can, but needn't, be carried out at the workpiece origin.

b) Now place the adjustment unit that way between workpiece and tool that in any case the round measuring point of metal is touched by the descending tool. Not before this is safeguarded, click on the button *"Measure Workpiece"*.

c) Then the z axis starts to descend in slow motion until the contact via measuring point of metal is made. The z axis stops immediately and moves upwards a bit to release the adjustment unit. You may cancel the levelling procedure by clicking on *"Stop"* in the main window.

d) You now have to remove the adjustment unit. The system moves again to the earlier set workpiece origin and has saved the just determined height of the workpiece surface as new z position. The thickness of the adjustment unit is considered automatically.

Warning

The z axis stops only, if – via the measuring point of metal and the tool tip – an electric contact is made. If the measuring point is missed, the axis moves on and adjustment unit, tool, spindle and workpiece may be damaged. Additionally, there is a risk of injury if you intervene manually during the levelling procedure. Hence proceed out ouf the danger area of the tool after having placed the adjustment unit correctly.

As for the measurement, an electric contact must be achieved, you are only allowed to use tools of conductive materials (metal). Do not use diamond tools (MCD and PCD) and coated tools with the automatic z adjustment.

9. Switching accessory items

Accessory items which are necessary for processing the current workpiece (e.g. collet chuck, cooling/spraying unit, dust extraction, etc.) are switched on and off by clicking on the according button (*"Out1"* to *"Out4"*). The numbering corresponds to the order of outputs on the CNC controller. Safeguard that all necessary accessory items are switched on before you start the processing.

10. Operating the spindle

For test purposes you can, under *"Spindle"*, switch on the spindle manually and set the designated rotational speed.

11. Starting the output

The system moves to the starting position and starts the output. Meanwhile the status bar shows green light *("Active")*. If a machine halt is caused (e.g. by interfering in an area secured by light barriers, by pressing the halt button on the control panel or after pressing the *"Stop"* button in CNC Term) this is indicated by the yellow *"Halted"* signal. The output can be continued after a machine halt.

After pressing the emergency stop button, the red status bar is illuminated *("Emergency")*. Before the output may be continued, the system has to be newly initialised.

You can adjust feed rate as well as rotational speed of the spindle during the processing. The adjustment is executed percental to the values saved in the output file. Just move the corresponding control to the designated value. The adjustment of speed is immediately carried out during the processing.

Warning

Only adjust these values in very small steps! If you work with wrong values here, this may lead to worse machining results. In awkward cases (e.g. considerably too high feed rate) the workpiece may loosen or the tool may break. In such cases there is a risk of injury for persons standing nearby.