RTR Manual

Kevin Damen
Daan Fokker
Merijn ter Beek
Martijn Frijns
Eline Lopez-Bongertman

Version 1.1

October 11, 2011

Abstract

Thank you for purchasing a DamenCNC's RTR set. In this step by step manual you can find exactly how to operate your CNC machine. Please read this manual thorougly in order to fully understand how your RTR set works.

DamenCNC wishes you good luck and lots of fun using your CNC machine!

Chapter 1

Introduction

Thank you for purchasing a DamenCNC's RTR set. In order to operate your machine, it is highly recommended that you first read the entire manual carefully.
By doing so, you immediately get a feel for the capabilities of the RTR set. This manual is set up in different chapters, each explaining a different aspect of the interaction of the RTR set with your CNC machine.
The manual is organized as a step by step tutorial, in order to efficiently set up your system.

For any questions you might have throughout this manual we refer to the Frequently Asked Questions chapter.

Contents

1 Introduction **1**

2 Hardware Setup **4**
2.1 Introduction 5
2.2 Getting Started 5
2.3 Nomenclature 10

3 Configuring your machine **11**
3.1 Introduction 12
3.2 Getting Started 12

4 CNC Milling using USBCNC **23**
4.1 Introduction 24
4.2 Getting Started 24
4.3 Conclusion 41

5 Frequently Asked Questions **42**
5.1 Introduction 43
5.2 Frequently Asked Questions 43

6 Appendix A: Connecting an Emergency Stop **47**
6.1 Introduction 48
6.2 Hardware Connection procedure 48

7 Appendix B: Connecting a stepper motor **50**
7.1 Introduction 51
7.2 Hardware Connection procedure 51

8 Appendix C: Connecting a Home sensor **57**
8.1 Introduction 58
8.2 Hardware Connection procedure 59

9 Appendix D: Connecting a MPG Handwheel 62
9.1 Introduction . . . 63
9.2 Connecting procedure . . . 63
9.3 MPG Handwheel FAQ . . . 70

10 Appendix E: Setup of a Delta VFD and a Teknomotor 71
10.1 Introduction . . . 72
10.2 Connection procedure . . . 73

11 Appendix F: Connecting a 0-10V Spindle Speed Control 81
11.1 Introduction . . . 82
11.2 Hardware Connection procedure . . . 83
11.3 Software setup . . . 91

12 Appendix G: Connecting a Toolsetter 100
12.1 Introduction . . . 101
12.2 Hardware Connection procedure . . . 103
12.3 Testing procedure . . . 108
12.4 Using the toolsetter to zero the Z-axis . . . 109
12.5 Using the toolsetter to measure tool lengths . . . 113
12.6 Detailed information of the toolsetter's wiring scheme . . . 119

13 Appendix H: Connecting a Probe 121
13.1 Introduction . . . 122
13.2 Hardware Connection procedure . . . 124
13.3 Testing procedure . . . 129
13.4 Using the Probe . . . 130

14 Appendix I: Connecting a Brake (Relay) module 133
14.1 Introduction . . . 134
14.2 Hardware Connection procedure . . . 134
14.3 Brake Module FAQ . . . 146

15 Appendix J: Using a Relay module 147
15.1 Introduction . . . 148
15.2 Hardware setup . . . 148
15.3 Software setup . . . 150
15.4 Using external tools . . . 151

Chapter 2

Hardware Setup

2.1 Introduction

In this chapter you can find how you can set up the hardware of your machine. The outline of this chapter is as follows. First we check if the delivery is complete. Next to the delivery, we guide you through the installation of USBCNC V3. When USBCNC V3 works properly, the Emergency Stop must be attached. When the Emergency Stop works correctly one can install the stepper motors. Afterwards you can home your machine. Also the installation of the MPG Handwheel, the 0-10V servo motor spindle control, the Tool Setter, 3D touch Probe and the Brake System will be discussed.

2.2 Getting Started

Scope of delivery
In the box you received from DamenCNC you will find a document with all delivered parts.
Verify that everything you ordered is present in the delivery. It might occur that a part is not readily availabe; in that case it is denoted as a backorder. DamenCNC will send you the parts in backorder as soon as possible.
DamenCNC guarantees that all components reach you in a good working order. In the exceptional case of any damage please send us an e-mail.

USBCNC V3
DamenCNC products are designed to work with USBCNC V3, which is a program that interfaces with a PC based CNC controller. The following minimum PC requirements are as follows for USBCNC CPU V3 and V4:

- 1.4 GHz Processor
- 1024 MB RAM for XP, 4G for Windows 7
- Windows XP or Windows 7, 32 or 64 bit
- Minimum Screen resolution 1024 x 768
- USB 2.0 Card

For USBCNC V5 the previous holds, in combination with these extra requirements:

- Pentium, duo-core recommended for Ethernet
- USB-2 connection / Ethernet connection for Ethernet CPU's
- Intel 100Mbit Ethernet card for Ethernet CPU's

Windows XP and Windows 7 is proven to work fine with USBCNC V3. Windows Vista is not.

If your computer matches these requirements you can go to

http://www.edingcnc.com/index.php?pagina=7download

in order to download the latest version of USBCNC V3.

Click on the downloaded file in order to install the software. The pop up windows speak for itself. If you are running on Windows 7 you have to start the file as an Administrator. In order to do this automatically, tick this box:

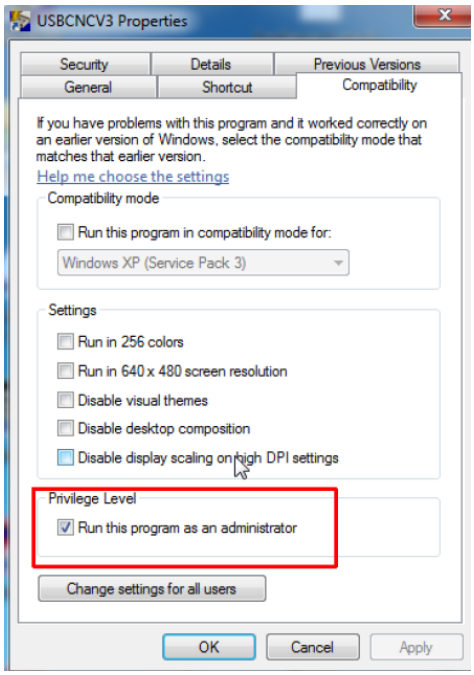


Figure 2.1: Start as Administrator for Windows 7 users

If the installation is complete, reboot your PC. As soon as you restart your PC, connect the CPU by connecting the USB or ethernet cable with your RTR set. Within approximately 10 to 60 seconds a pop up window will appear which displays that windows has found a USB COM port. Choose the option "I want to manually choose driver folder"; the drivers are located in the Folder where you installed USBCNC V3, in the folder Drivers.
Now you can launch USBCNC V3 by double clicking USBCNCV3 icon on your desktop. In order to be able to communicate with your CPU you must plug in the 220V Euro cable into your RTR set, and switch on the RTR set.



Emergency Stop
It is essential that the Emergency Stop is the first component to install. This component can shut down all moving parts at any time.
The Emergency Stop can be plugged in the back of your RTR system, in the designated slot "E-STOP". Be sure to test the functionality of the Emergency Stop by pressing the button. In the USBCNC V3 environment the red LED on the left with the caption E-STOP should transform its color to green. If you have USBCNC CPU V3 or V4 you can do this without switching on your RTR set. If you have USBCNC V5, you must switch your RTR set on. If you need any further information about the Emergency Stop please read Appendix A: Connecting an Emergency Stop.

Stepper Motors
After the Emergency Stop is installed, you can implement the stepper motors.
You can do this by plugging in the connector in the RTR system. Be sure to do this with the RTR set turned off.

Each type of motor has it's own rated current which can be found in the datasheet of the specific stepper motor.
The drivers inside DamenCNC's RTR systems are tuned conservatively; with the lowest possible current setting.
This proves to be a good start for the first time you start your machine. You can gradually update the current setting towards the maximum current setting for your respective motor. This procedure can be done inside the RTR set, using DIP switches.

Each axis has its own driver so the current needs to be adjusted for all axis. Please bear in mind that a DamenCNC Classic RTR set contains a set of HP5056 drivers, and that a DamenCNC Performance RTR set comes with HP8078 drivers.
For more information on the installation and the wiring scheme of various motors, we refer to appendix B: Connecting a stepper motor.

Homing Sensors
The Homing sensors define your working range. They should be mounted such that your system is safe to operate. Usually Homing sensors are mechanical switches or inductive sensors. DamenCNC always recommends operators to use a **NC** (normally closed) type of switch for safety reasons. As soon as an **NC** switch is accidently disconnected USBCNC V3 will give an error, wheras with an **NO** switch no error can be given.)
Be sure to test all your Home Sensors. If you open USBCNC V3 and engange each respective Home switch or sensor, on the left of USBCNC V3's environment the right red LED (Home-x, Home-y etc.) should turn green upon excitation. Test this for every axis you use.
For more information with respect to Homing sensors and the way these should be installed, as well as the wiring scheme, we refer to Appendix C: Connecting Home Sensors.

MPG Handwheel
The MPG Handwheel is a nice tool which you can use in order to be more flexible, and work more intuitively. You can use this device to zero all axes and to JOG very precisely, because of the very small displacements and the fact that you don't need to operate your keyboard. You can also Pause and Play your program using the MPG handwheel. You can install the MPG Handwheel by clicking in the connector into your RTR set in the designated slot "MPG".
For more information with respect to the MPG Handwheel, as well as the wiring scheme, we refer to Appendix D: Connecting a MPG Handwheel.

0-10V Spindle Speed control
The 0-10V Spindle Speed controller enables the operator to control a servomotor such as a Teknomotor. Be sure to use an external 220V power source as you can not use the 220V socket on your RTR set!
This is because the motor can be deactivated using the software and not using the interruption of 220V (as can be done with the RTR set). Since the installation manual is rather extensive, please read Appendix E: Connecting a 0-10V Spindle Speed control for more information.

Toolsetter
As the name of the Toolsetter already implies, this device is used in order to measure the Z-coordinate of the lower part of your tool. By doing so, Zero-ing the Z-axis can be done automatically. USBCNC V3 is capable of saving multiple tool lengths, which makes this a very handy feature. You can install the Tool Setter by clicking the connector into your RTR set in the designated slot "Probe".
For more information with respect to the Tool Setter, as well as the wiring scheme, we refer to Appendix F: Connecting a Tool Setter.

3D Touch Probe
As the name of the Probe already implies, this device is used to use your CNC machine as a measuring device. The 3D probe allows to probe in the X, Y and Z direction. For more information with respect to the 3D Touch Probe, as well as the wiring scheme, we refer to Appendix G: Connecting a 3D Touch Probe.

Brake (Relay) Module
The Brake (Relay) Module is used to prevent (heavy) Z-axis components from falling downward after e.g. a power failure. The brake module can be mounted on top of the Z-axis (double shaft) stepper motor. It automatically clamps the shaft when power is suddenly absent. There are multiple other applications which can be controlled using the relay module.
For more information with respect to the Brake System, as well as the wiring scheme, we refer to Appendix H: Connecting a Brake System.

2.3 Nomenclature

In this section DamenCNC explains frequently used abbreviations and terms which are used throughout the report. By doing so the operator is already familiar with certain terminology.

- CNC: Computer Numerical Control
- CPU: Central Processing Unit
- CS: Coordinate System; includes both the Work Coordinate System (WCS: adjustable) as the Machine Coordinate System (MCS: fixed)
- DRO: Digital Read Out. This displays the coordinates of your system in USBCNC V3
- GND: Ground
- LED: Light-Emitting Diode
- MDI: Manual Data Input
- NC: Normally Closed
- NO: Normally Open
- PCB: Printed Circuit Board
- Stylus: the part of the 3d probe that senses material
- USB: Universal Serial Bus.

Chapter 3

Configuring your machine

3.1 Introduction

This chapter is about the configuration of your machine.
There are several settings that have to be adjusted in the software before you can start using your machine.
We assume that USBCNC is already installed correctly and that it has established a connection with a CPU. If this is not the case please first read the manual of USBCNC which can be found on www.edingcnc.com.

In this manual we assume you have a milling machine, with the servo motors already correctly installed. In any case, most information will also be of use when you have a lathe or any other device.

3.2 Getting Started

In order to establish a correct setup of your machine, we first explain how to configure all axes on your machine. This is done by establishing the required software. After this configuration it is explained how to verify that it moves correctly and in the right direction.
First, start your PC and engage your RTR set. When both systems run you can start USBCNC by clicking twice on your desktop icon USBCNCV3. Press "START CNC" and observe the USBCNC environment.

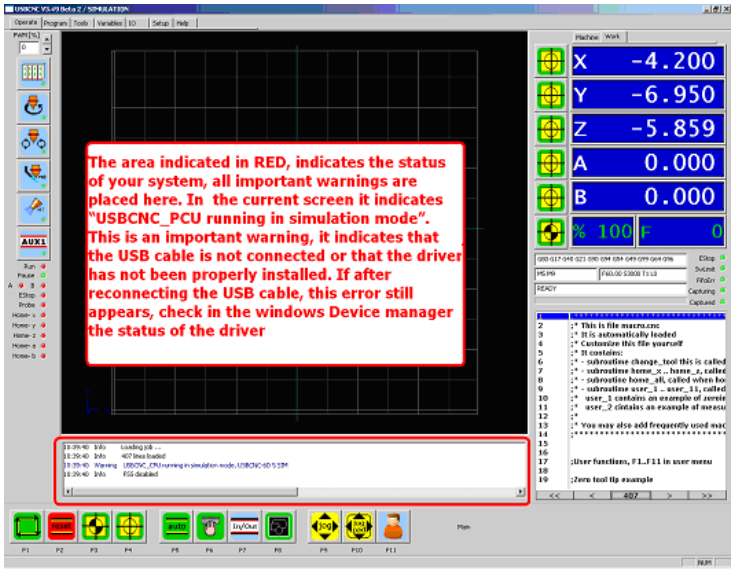


Figure 3.1: USBCNC environment

Verify that USBCNC is not in SIMULATION mode, and activate the controller by pressing reset (or F2). As soon as you activate the controller you will hear the activation sound of your steppermotor(s). Next to the activation sound you should see that the CNC controller LED turns red.



Figure 3.2: Driver LED must turn red

If your CNC controllers are turned off when you press Reset, go to the Tab Setup. The function Amp Enable should be (un)checked, according to your wishes.

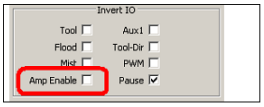


Figure 3.3: (un)Check Amp Enable if your controller does not activate at Reset

Now you should be able to JOG your system by pressing the following keyboard commands:

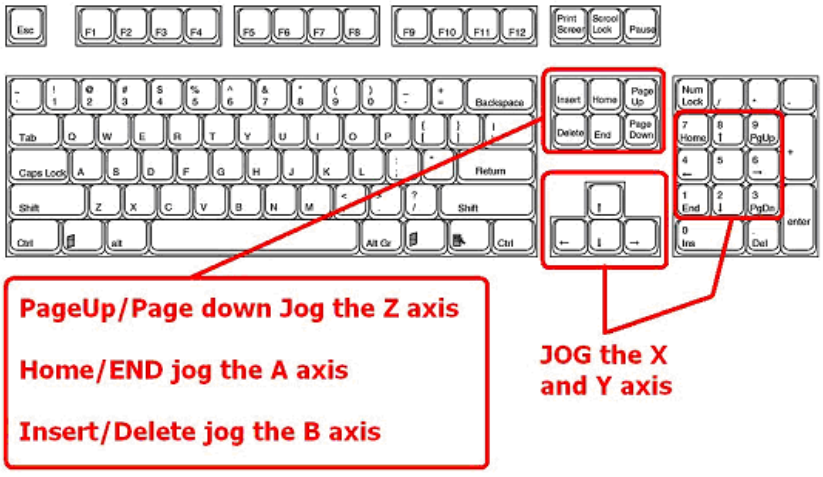


Figure 3.4: Jog your machine

If this all works properly you can move your machine to a safe position (i.e. a position where no obstacles are present) in order to establish your machine limits.
It is highly recommend to start with the Z-axis. When the Z-as is configured correctly, we can move it safely during configuring the other axis. Before starting, make sure all the sensors and one steppermotor (Z-Axis) are connected to the RTR set. In order to prevent damage, don't load expensive tools yet.
Before we can configure anyting, it is necessary to introduce USBCNC's Digital Read Out (DRO). This DRO displays the coordinates of your system. There are two coordinate sets, the Work Coordinate System (WCS) and the Machine Coordinate System (MCS). The MCS is fixed and cannot be modified. The WCS however can easily be modified, for example if you load your raw material not in the origin of your machine, and you wish to machine it without the need for calculating its position with respect to the MCS origin.

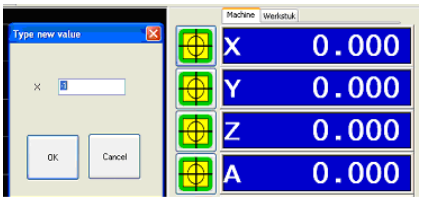


Figure 3.5: The DRO and the function for calibrating the WCS to zero

The main advantage of this function is that the WCS can be fully determined by the operator. Just fix your raw material anywhere within the reach of your machine, JOG your tool towards a convenient origin (usually below left) and load your G-Code. The G-code will automatically start from the WCS you selected. You can do this by pressing the yellow button on the left of the axis DRO, in order to calibrate the axis to zero.

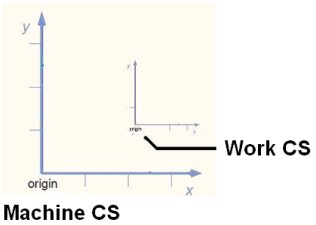


Figure 3.6: The MCS and the WCS respectively

With the DRO now known, we can start with the Z axis configuration. Before continuing any further, ensure that you are in the Operate Tab of USBCNC. Press F12 in order to come back to the Main Menu. Ensure that the DRO is active. If this is not the case, activate the DRO using USBCNC V3 settings.

Procedure

- Press the keyboard button Page Down; this should move your Z-axis down
- While pressing down, verify that the value in the DRO for the Z-axis decreases. If you started with 0.00, verify that the position is now negative

If this is both the case than you can continue with configuring your Steps / Unit. If this is not the case than observe how you can change the direction of your commands:

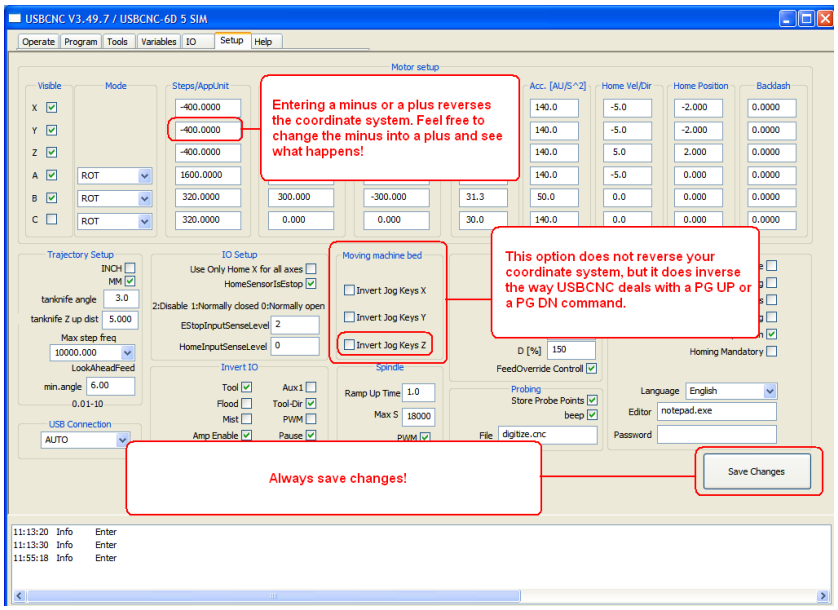


Figure 3.7: Adjusting the way USBCNC compiles your commands

Steps / Unit
The steps / unit indicate how many steps a servo motor should receive per length distance. Since this differs per system, read these instructions carefully.

- Move your system to a safe position somewhere in the middle of your reach, without any (expensive) tools
- Go to the Tab Setup; then Parameter Step/Unit.
- Roughly calculate your Steps/Unit. As an example: Usually our drivers are set a 1600 steps / revolution. If we use a pitch of 5mm, and we have a gearbox reduction of 1:2. This will result in 1600 / 5 / (1/2) = 640 Steps/Unit.
- Insert the value you have calculated for your system, and Save your result!
- Look at your DRO and note the begin position. If you don't have a DRO, mark the position with a caliper.
- Zero the value of your Z-axis (F12; F4; F3)
- Go to the MDI (Manual Data Input) menu (F12; F6)
- Insert the following text: g01 z-10 f400 and press Enter
- Verify the new position on the DRO or using your caliper. In the software the displacement equals 10mm. Verify if this is also the case for your machine
- If necessary, adjust the Steps/Unit parameter. For example, if you measured that your machine displaces 20mm instead of the required 10mm, modify the Steps/Unit to 320 Steps/Unit. Don't forget to Save!
- Again, calibrate the position to 0.00 or use your caliper
- Note the begin value of your machine
- In the MDI (F12; F6) type g01 z-50 f400 (beware! We will displace 50mm downwards, so keep your Emergency Stop within reach)
- Repeat until you have a satisfactory result. For machines using a rack and pinion or a belt, it is recommended to use a larger displacement.

Setting the Limits

The next thing to do is establishing the working domain of your machine. This is done using software limits; these are necessary for preventing your machine to move outside a given reach after the machine is Homed.

For every axis it is possible to assign both a positive and a negative limit. This can be done in multiple ways, and the method DamenCNC uses will be explained below.

All machines at DamenCNC use Zero as the positive limit for the Z-axis. The negative limit is not yet known, so we will make an educated guess. This guess is based on the distance between the two (metal) marks/ridge for the home sensor, as this is your reach of the machine. This value is now your negative limit. As an example, the Optimum BF20L has -227 as negative limit.

In a later stage it is required to verify if this software reach is actually possible for your machine. If this is not the case, you need to modify the reach to a lower value (i.e. -217 for the BF20L). You can not do this until the Homing is set.

Setting the Homing
The software limit you just assigned to your machine is only active after homing. This is logical since your machine must have a reference in order to establish its reach. This reference point can be found using the Homing functionality. In order to Home you must first have a limit switch or an inductive sensor on the end of your reach. The Home sensor of your choice must first be tested to confirm its functionality. Do this by pressing the button or in the case of an sensor, hold a piece of metal in front of it. The red LED on the USBCNC environment must turn green.

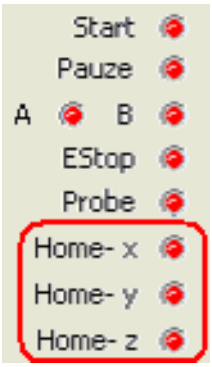


Figure 3.8: Verify that your Home sensors work correctly; red LED must turn green

If this works it is necessary to let USBCNC know what kind of sensors you use. Go to the Tab Setup, and look up the function HomeInput senseLevel. If you have Normally Closed (NC) switches, enter a "1". If you have a Normally Open (NO) switch, enter a zero.
In the Tab Setup in the above box you can find the box Home Vel/Dir. In this box you can assign with what speed and direction (plus or minus) your machine should home. The DamenCNC method requires a positive number since we Home upwards. As a begin value, choose +4. Adjacent to Home Vel/Dir you can find Home Position. In this box you can enter the value where you want to home to. It is recommended not to use 0, but instead use +1. This offset of 1mm is due to the hysteris in every system; it prevents your system from an unwanted trigger of your sensor.

Home testing
Now you can test your Home functionality. Move the Z-axis to somewhere in the middle of your reach. Keep your emergency stop (or the Pause button) within reach. Go to the Operate tab, press F12; F3; F3. The machine will now move upwards (verify that this is the case!). Does the machine stop at your Home sensor? It is recommended to trigger your switch manually the first time.

Repeat procedure for the X and Y axis
You can repeat this procedure for both the X and the Y axis. Bear the coordinate systems in mind! If you fail to do so you might end up with a machine machining mirror images of a product.
The coordinate systems look as follows:

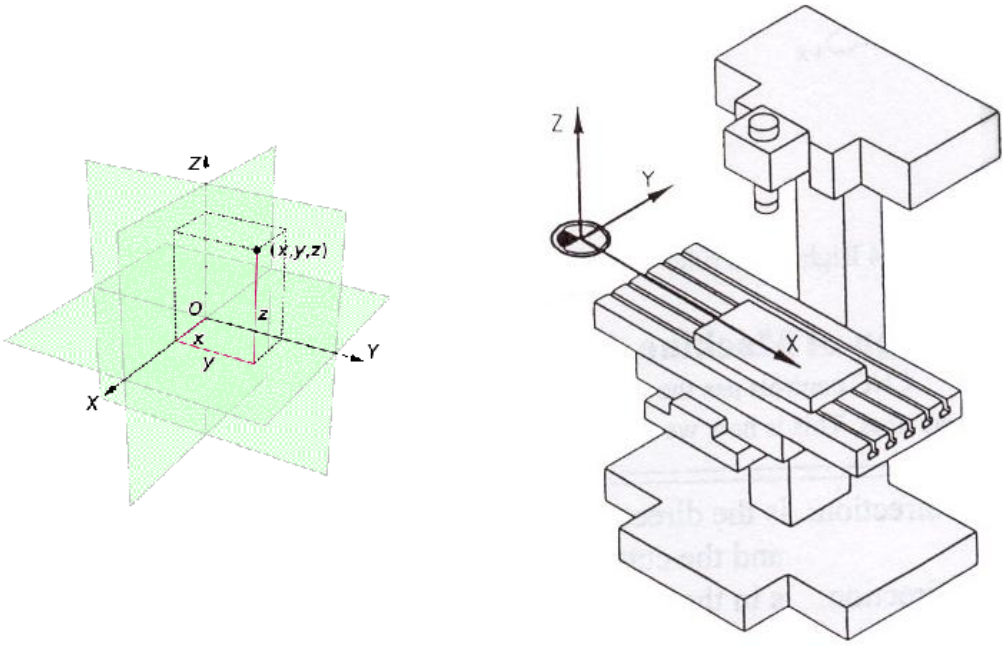


Figure 3.9: A normal coordinate system (left) and a mill coordinate system (right)

Note
Always look at the movement of the tool with respect to the material; and not to the table or your portal!

Checking the X axis

- Move the Z-axis downwards using the Page Down button or using the JOG menu (F12; F10)
- Zero your X-axis reference (F12; F4; F1)
- Go to the MDI (F12; F6) and enter the following text: g01 x25 f300 and press Enter
- The coordinate system is set correctly if the tool moves to the right with respect to your product (in the case of a conventional mill: the table moves to the left as seen by the operator)
- If the direction is wrong, implement a minus sign in the Tab Setup (Steps/Unit)
- Go back to the Operate tab and press the keyboard button arrow to right; this should move your tool with respect to the product to the right (in the case of a conventional mill: the table moves to the left as seen by the operator)
- If this is not the case, then adjust this using the Inverse Jog setting in the Tab Setup

Checking the Y axis

- Move the Z-axis downwards using the Page Down button or using the JOG menu (F12; F10)
- Zero your Y-axis reference (F12; F4; F1)
- Go to the MDI (F12; F6) and enter the following text: g01 y25 f300 and press Enter
- The coordinate system is set correctly if the tool moves in the positive y-axis direction with respect to your product (in the case of a conventional mill: away from the operator and hence the table must come towards the operator)
- If the direction is wrong, implement a minus sign in the Tab Setup (Steps/Unit)

- Go back to the Operate tab and press the keyboard button arrow up; the tool must move in the positive y-axis direction with respect to your product (in the case of a conventional mill: away from the operator and hence the table must come towards the operator)
- If this is not the case, then adjust this using the Inverse Jog setting in the Tab Setup

Setting the limits
The limits are set using the following procedure:

- Let the machine search in the negative direction (i.e. -6 in the Home Vel/Dir menu).
- Test your switches and verify that the response is indicated in the USBCNC environment (the right red LED must turn green, check if you did not confuse the X axis with the Y axis)
- Go to the Homing menu (F12; F3; F1 (X) or F2 (Y))
- Visually check if the Homing is performed correctly
- Test if the limits are correct by jogging your machine
- If necessary, adjust the reach of your machine

Homing sequence
The Homing procedure can be set such that all three axes Home after eachother. Verify that this works by pressing F12; F3; F11. If you wish to adjust the sequence in which your machine is Homed, please go to the macro.cnc file and manually adjust the sequence there (F12; F5; F4).

E-stop after Homing
The function 'Home is Estop after homing' is a nice feature which you can tag in the Setup tab. If the machine encounters the Home sensor after its Home procedure (something that should not be possible) then an instant emergency stop is returned. If you wish to use this function it is demanded that you use the same type of switches for every axis (i.e. all Normally Open (NO) or Normally Closed (NC)).
If you use this function you might receive an error message. This is because the X, Y, and Z Home sensor are stationary different from the Home sensor for axis A and B. If this is the case, than you need to hotwire the GND of the "faulty" sensor such that in the USBCNC environment all colors of the Home function are either green or red. The system can not deal two different signals with this feature.



Figure 3.10: E-stop after homing demands the same colors in stationary mode

Chapter 4

CNC Milling using USBCNC



4.1 Introduction

This section is intended to enable you to make the first steps with your DamenCNC RTR system. We assume that the entire setup and configuration of your system has been completed with succes. If this is not the case please first read Chapter 3: Configuring your machine on Configuration and setup of your machine. In this manual we assume you have a milling machine, but most of the information will also be of use when you have a lathe or any other device.

4.2 Getting Started

In order to efficiently get started with your system, follow the following steps:

1. Start your PC
2. Turn on the RTR set. This can be done by switching the green switch on RTR sets which are manufactured after january 2010. If you have an older version of the RTR set, a red switch can be found on the backside of the RTR set.
3. Start the program USBCNC V3 by double clicking the USBCNC V3 icon on your computer.



Figure 4.1: Double click on this icon

4. Click the "START CNC" button in the startup screen



Figure 4.2: Click the START CNC button

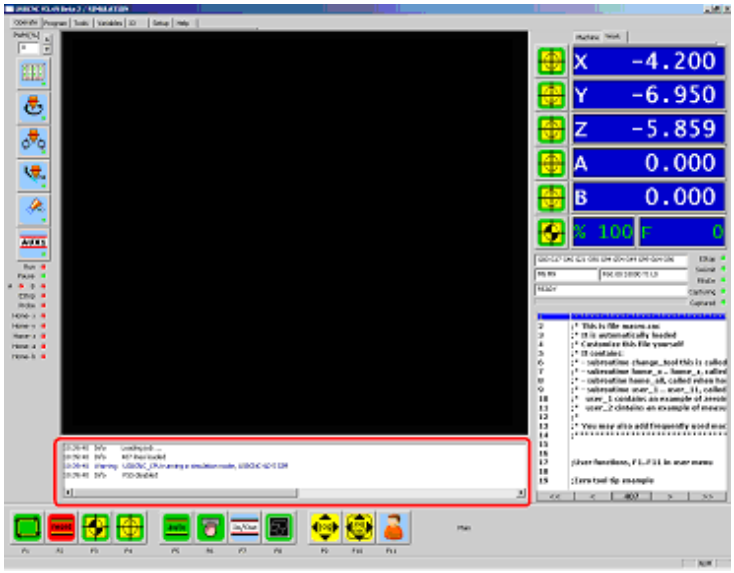


Figure 4.3: Operating environment of USBCNC

5. Observe the Operating environment of USBCNC

   The area which is indicated by a red box indicates the status of your system. In this area the most important warnings are adressed. If you look closely to the picture shown you can see that the system indicates "USBCNC CPU running in simulation mode". This is an important warning, as it indicates that the USB cable is not (properly) connected or that the drivers has nog been properly installed. If this indication remains after the USB cable has been connected, check in your Windows OS Device manager the current status of the driver.

6. Activate the controller

   When the system is started for the first time USBCNC disables the CNC controller. To activate the CNC Controller press the RESET button in the lower left side of USBCNC.

   In the USBCNC operating environment's left top corner you should see that the red LED turns RED, which is an indication that the drivers and the CNC controllers are enabled.

Hints: In case your CNC controllers is turned off when pressing RESET, there is an easy fix. Go to the TAB SETUP. The Amp Enable box then needs to be checked or unchecked if it was already checked.

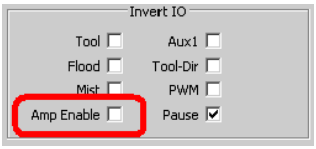


Figure 4.4: click AMP Enable in case controllers are turned off at RESET

Also, in the lower left corner of the USBCNC operating environment's tab of SETUP you can see a section which is labeled as Invert IO. Which this section you can inverse the working of the Input or Output.

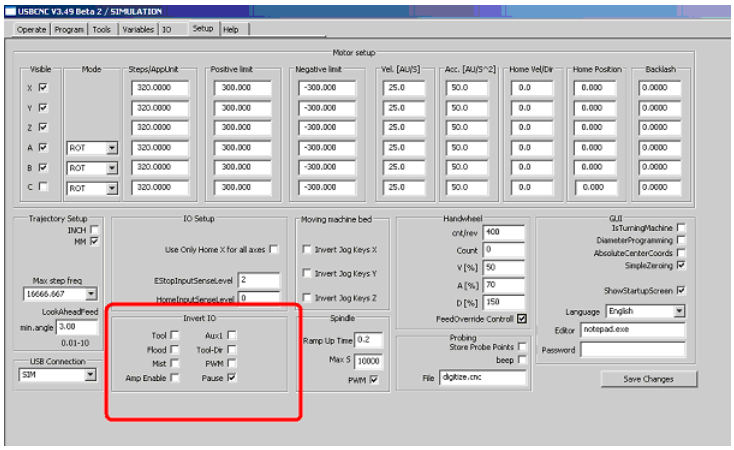


Figure 4.5: Inverse the Input and Output properties

7. Homing the machine

Before you can start using your machine needs to be homed (homing is an operation in which the machine calibrates itself). In the previous part of the manual you have already configured the machine such that the homing is done properly.

Automatic Homing:
In order to use the automatic homing function, press the Home button as indicated in the image. We assume homing is correctly set up. If this is not the case, do not press this button and go back to the previous chapter)

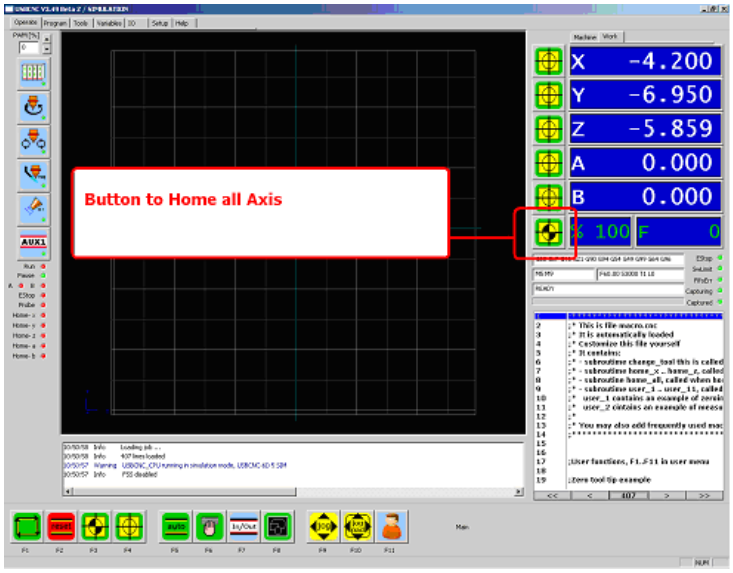


Figure 4.6: Automatic homing function

8. Calibrating the Work Coordinate System

   There are two coordinate systems. First, there is the Work Coordinate System (WCS). Next to the WCS there is the Machine Coordinate System (MCS). The MCS values are fixed for a given system. The WCS can be modified. The WCS is used most in practice.
   Now that the machine has been homed, you can see that the Machine Coordinate System values are set zero. This can be seen in the blue boxes on the right of the operating environment.

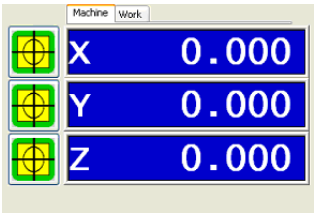


Figure 4.7: Calibrating the Work Coordinate System

Note: The values don't have the be zero, they correspond to the value which have been given for the Home Position in the SETUP tab. So they need to correspond these values.

Also, it is possible that the values don't exactly match to the Home position but differ a small amount (0.5mm for example). This contingency depends on how steep the acceleration/decelleration ramps are (set) on your machine.

9. Moving the machine with the JOG buttons

   When you are machining a part, this will not always be from the Home position. With the JOG buttons of your computers keyboard or with the Pendant it is possible to move the machine to a desired position. Your machine responds to the following input from your keyboard:

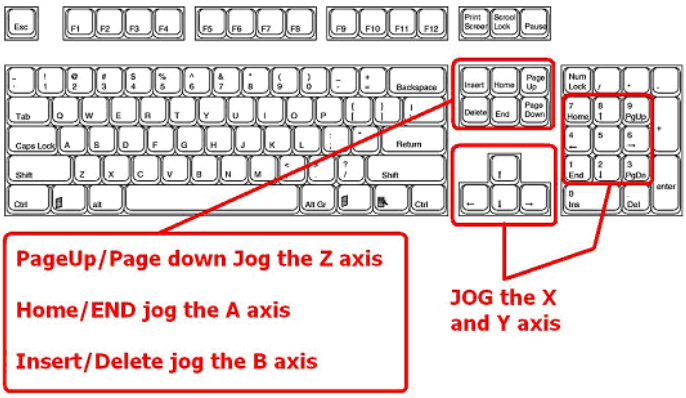


Figure 4.8: Keyboard inputs for JOG menu

DamenCNC advises that you give this feature a try. When you press the arrow-up(8) and arrow-left(6) button, the machine should be moving in a diagonal manner (assuming NumLock is not active).
You can speed up the JOGging process by first holding the Control button. The translation of the machine will increase even further if you hold the Shift key.

When you JOG, you should notice two things. In the upper right section of the screen on the Digital Read Out (DRO) box you can see the current XYZ position of the machine in machine CS. On the left in the image you can see the working area of you machine and also which

movement has just been made. The green line visualizes the movement that has been made by your CNC system.

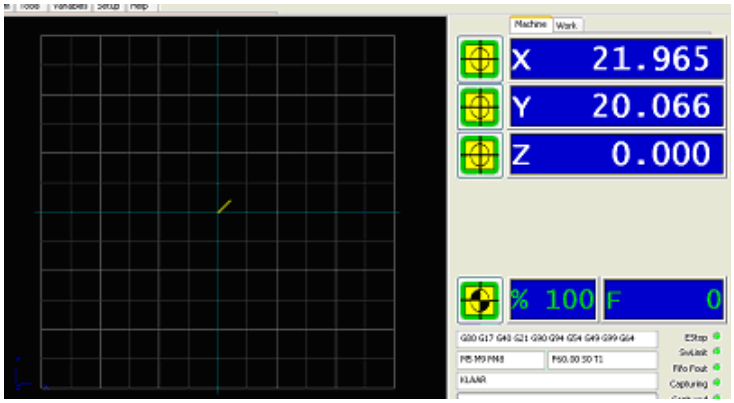
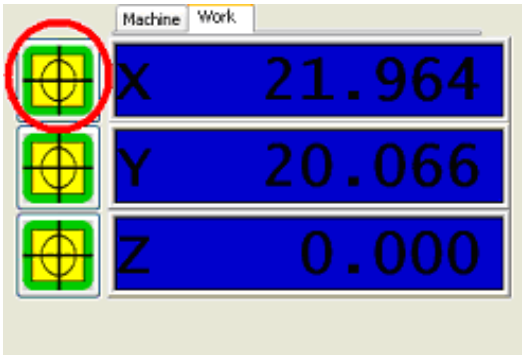


Figure 4.9: Visualization of the CNC System movements and the DRO

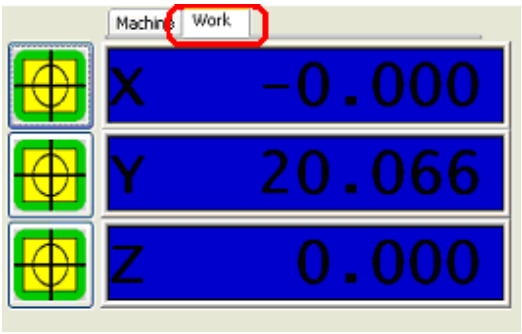
G-Code

Now let's assume you wish to make the position that you have JOGged to the new zero position of the work CS. The zero position of the Work CS is the position from which a G-code files is executed. So if in your G-code you send the machine to "g1 x0 y0" it would move to the current (Home) position.

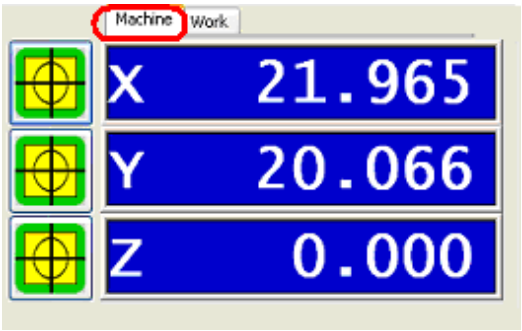
- Press on the tab Work
- Press the button circled in red



- By doing so, the work CS is set to zero
- It works equally for the Y and Z axis



- When you go back to the Machine CS tab, you can see that the coordinates have not changed
- We can conclude that pressing the Zero buttons only influence the Work CS
- The machine CS is used to keep track of the machine position within its boundaries, and can only be influenced in the SETUP tab; not in the operate TAB



Importing files

Now let's look back on what we have done so far. First, the machine has been homed. Then we have set the Work CS zero point. The machine is now ready to execute a CNC program. There are 2 options for loading a drawing.

- Loading a .cnc, .nc or .txt file
- Loading a .dxf or HPGL file, which USBCNC can convert to G-code

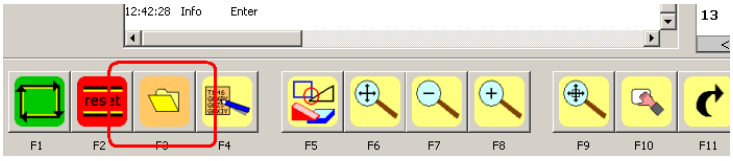
These two options will be elaborated on the next coming pages.

1. Loading a .cnc, .nc or .txt file

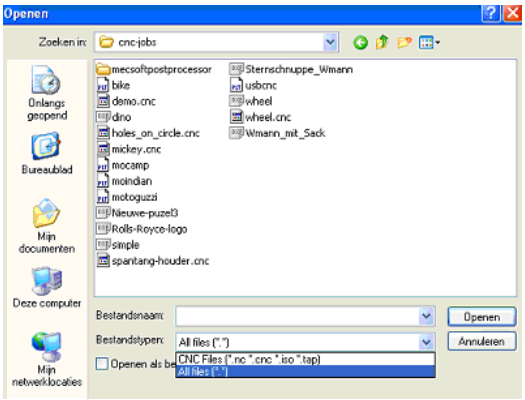
In order to load a .cnc, .nc or .txt file click on this button, or press F8:



Next, click on the following button or press F3:



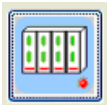
Select the file you wish to import from your PC:



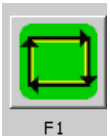
Most G-code files have an .nc .nc .sio or .tap extension. Some programs create .txt files which contain G-code, if you wish to load these type of files, set the file type to All files.
When you have selected your file, USBCNC visualizes the contours of the product in the black square. USBCNC shows the contours such that it becomes clear where in your system range the product will be made. By doing so you can visually check if the zero point has been set correctly, and if the G-code of your product is correct.

The next thing you should always do before you start your use your machine is to check if the drivers are enabled. This can be done by

verifying that the red LED next in the top left corner is active. If this is not the case, press the RESET button.



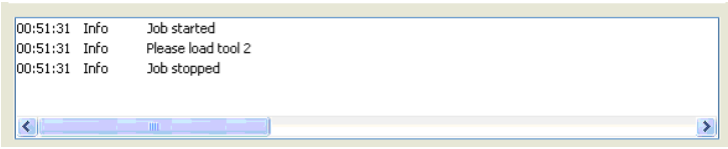
Now it is time to start the machining. By pressing the green button on the bottom left corner (or press F1) your G-code will get executed.



When USBCNC is executing a file the Start button will turn into a Pause button. When the pauze button (F1) is pressed it pauzes the machine motion. When one presses the Start button (F1) again it will continue its motion.

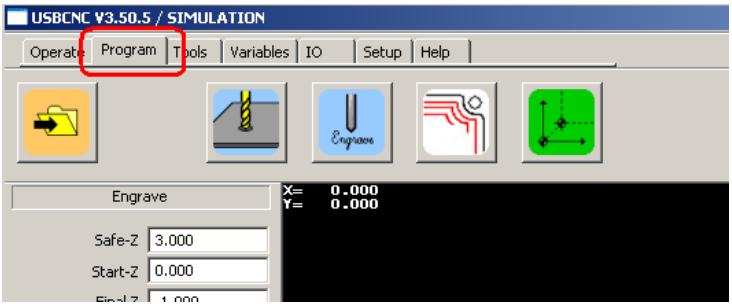


Note: Sometimes the execution will be stopped directly, and you will be prompted to Load a tool. This phenomena is a safety check to see if the correct tool is loaded. If this is the case, press the Start button (F1) again.
USBCNC does this safety measure when it encounters an m6 code in the G-code, which is used for toolchanges.

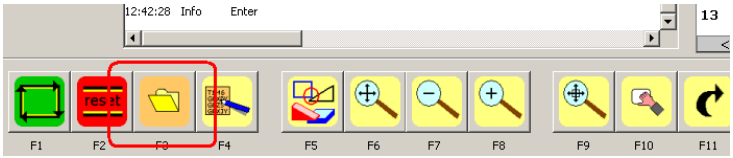


2. Loading a .dxf or HPGL file, which USBCNC can convert to G-code

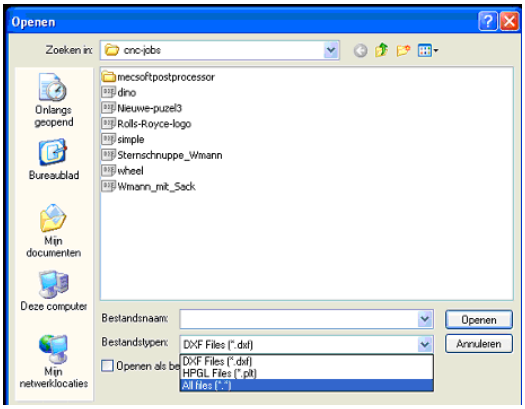
In the previous section we have loaded, and executed a G-code file. USBCNC also has a small (two dimensional) CAM import module that allows DXF files to be loaded. We will now show how you can import a .dxf or an HPGL file.
Up until now the operations required adjustments in the Operate TAB in the USBCNC environment. In order to import a .dxf or HPGL file, click on the "Program" TAB.



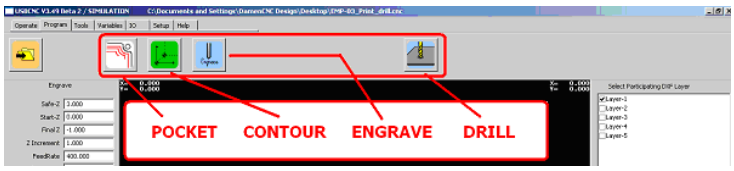
Load your .dxf or HPGL file using the Open button on the top left side.



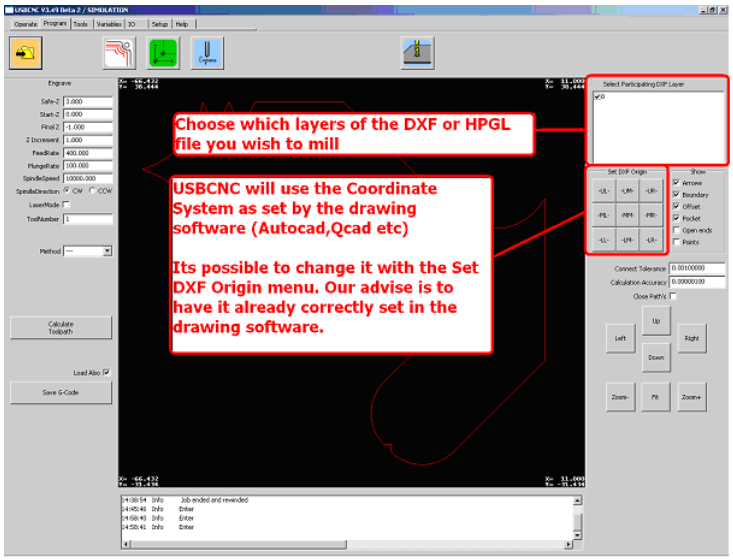
Be sure to change the File type to All files, in order to view both .dxf and HPGL files.



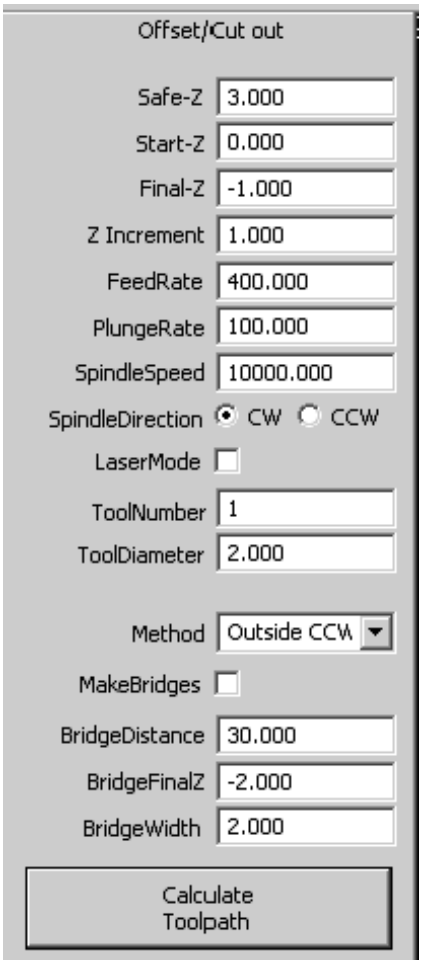
In the Program TAB you can select four different operations you can perform with a .dxf or a HPGL file after it has been loaded. These operations include pocketing, contouring, engraving and drilling.



With the file now imported, the following screen will show up. On the top right side you can select the layers of your file you wish to use for your product.



On the left side of the Operating menu you can see a list of parameters. These parameters are explained in the following section.



- Safe Z
  Safe Z is the height the Tool retracts to when making fast moves(G0) in the XY plane. It needs to be set higher than any items used to clamp the fixture in order to avoid collisions.

- Start Z
  Start Z is the start value of the Z plane. DamenCNC recommends to leave this parameter ZERO.

- Final Z
  Final Z is he final depth at which the tool needs to machine.

- Z-Increment
  Z-Increment is the maximum increment the mill is allowed to go in the Z direction.

- Feedrate
  Feedrate is the feedrate in the XY plane. The velocity is in milimeters/minute.

- Plungerate
  Plungerate is the velocity in the Z direction into the material, also indicated in milimeters/minute.

- Spindle Direction
  Spindle Direction indicates the direction of rotation of the milling motor. This is almost always clock-wise.

- Lasermode
  Lasermode is used for PLASMA or LASER cutters.

- ToolNumber
  ToolNumber is used since it is possible to organize a database with tool length and diameters.

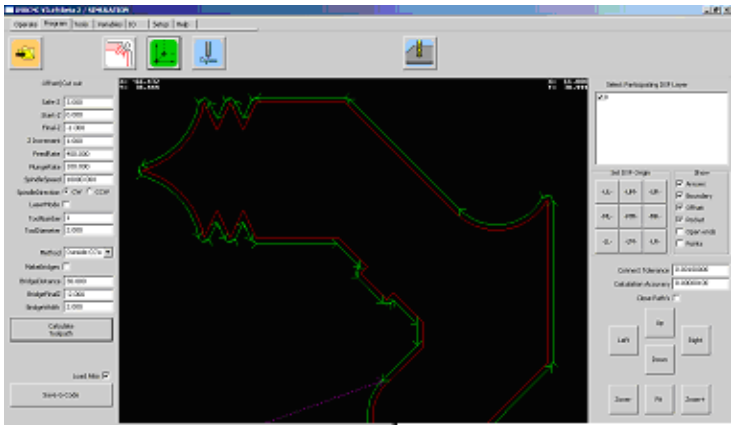
- ToolDiameter
  This is the tool diameter with which the program needs to calculate. Its very important that this parameter is correctly set. The program will use this value to calculate the offset of the toolpath.

- Method
  Here the method of milling can be choosen, if its outside or inside offset and the direction in which the path is taken (CW or CCW).

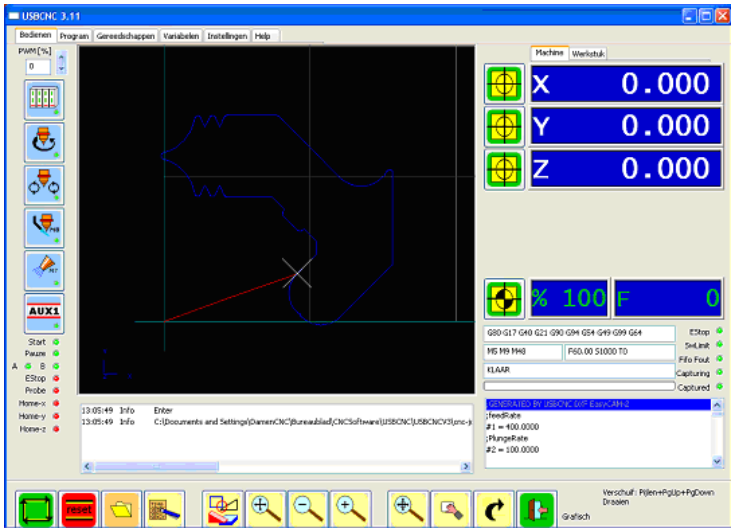
- MakeBridges
  With this function the software will leave small sections of material intact. Its used for small delicate parts, to provent them coming loose during milling. If they do they can be damaged by the mill. Its a very nice feature for milling model Aeroplanes or advertising texts.

Calculating the Toolpath

After all these parameters are set, press the "**Calculate Toolpath**" button. When the PC is done calculating, the toolpath is shown in the visualizer. From this visualization you can visually check whether all settings are correct. You can select different options and change parameters in order to see the effect on the toolpath.



If the tool path is correct, click on the "**Save G-code**" button. USBCNC will upload the G-code to the main menu, where the G-code is visualizer in the working range of your system. Again, visually check whether all settings are correct.



Please note that the machine is most likely still in it's Home position. Do not forget to set the workpiece CS before executing the files!

Milling from another position than the Home position

It is of course possible to start milling from a position which differs from the Home position. It could be that somewhere in the range of your machine you have placed a piece of material you wish to CNC mill. What you need to do is place the workpiece zero point on this piece of material. Often the lower left corner of the material is set as zero point. To achieve this, JOG to the zero position of your choice and press the Zero buttons for XYZ.

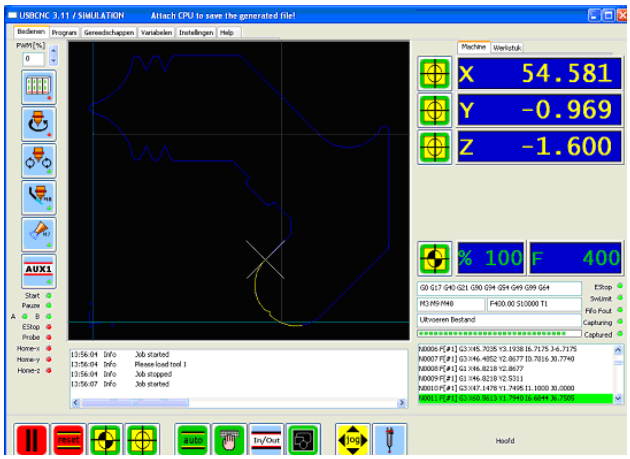


Note: Any position on the material can be used as a zero point. The most important is that the zero point in USBCNC corresponds to that of the DXF or HPGL file that you have loaded!

From this point onwards you can continue with machining by pressing the Start button (or press F1)



When the machine is executing the G-code, you can monitor it's progress in the visualizer. The movement of the system is shown as a green line. In the Digital Read Out (DRO) in the top right corner you can track down the system's position.



Making changes while executing

There is a limited amount of operations you can perform when you are executing a program. These modifications will be adressed in this section.

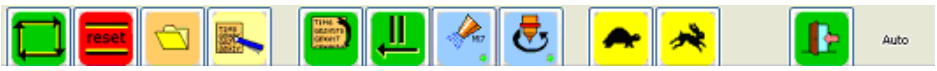
You can always **Pause** the machining at any time, by pressing the red Pause button (or press F1):



If you press on the **Auto** button, an extra set of options available during operation appears.



These extra options can be seen in this row of icons which will be explained in this section:



We start with the **Feed rate** button, with which you can alter the velocity of the translating system. By pressing the turtle, the system will slow down. By clicking on the rabbit, the system will enhance it's speed.



The **Set line** button is present in order to let the G-code start at an other line in the G-code than the first sentence. In order to use this function, first open the G-code file. Click on the the line at which you want to start your program. Then click on the Set line button:



An other option during execution is the **Return to resume position**.



This button is used when you have paused the CNC program and have moved the machine to change a tool for example. If pressing start to resume the program you will get an error "Warning X not at Resume position" Pressing this button the machine will move to its pause position. After this procedure the program can continue when you press the start button (F1).

As a final modification there is the **edit G-code** button. When you press this button the G-code file that is loaded in the PC's memory can be edited. On the latest version of USBCNC the file that is changed will automatically be loaded.



4.3 Conclusion

In this chapter the basic functionalities of USBCNC are adressed. DamenCNC advises to read the section of your interest several times before you start with machining.
If there are still any doubts or questions regarding the operation of USBCNC in combination with a DamenCNC RTR set, carefully read the Frequently Asked Questions in the next chapter.
In the exceptional case that also the FAQ does not answer your questions please contact me at kpdamen@damencnc.com.

Chapter 5

Frequently Asked Questions

5.1 Introduction

In this chapter the most common frequently asked questions are adressed and explained. Be sure to carefully read all the instructions in the case you recognize a question you have.

5.2 Frequently Asked Questions

In this section the questions we receive most are explained in detail.

How to start a program from halfway?
It can always occur that you need to stop your work, and you need to continue your work some other time. This can be done as follows. The first thing you should do is to browse to the Auto menu. You do this by first clicking on the Auto button: From this menu, click on the Set line button:

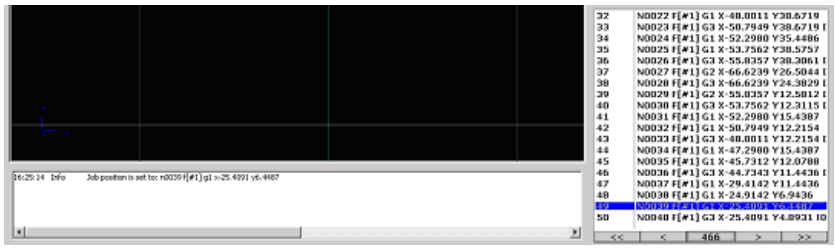




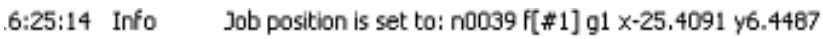
Using this button, you can select from which sentence of the G-code you wish to continue your machining.
There are two options how you can continue from this point.

Option 1

In the lower right side if the screen, you can scroll through the loaded G-code program. The line which is marked blue is the active line.
If you press F12, F5, F5; the Set line function will set this blue active line the actual position.



If you did this correctly, USBCNC will generate a message which looks somewhat like this:

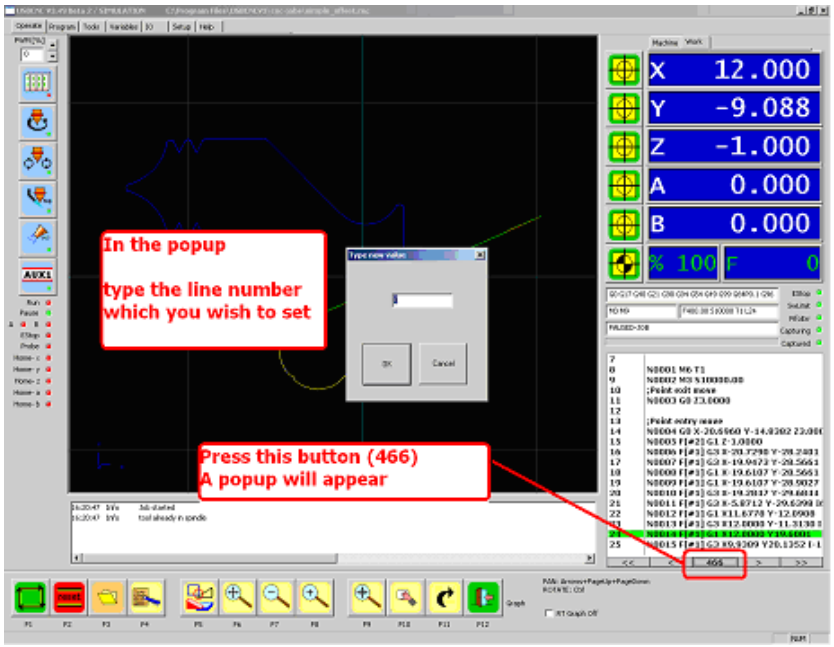


If this is the case, you can press the Start button in order to continue your program.



Option 2

The second option to continue your work halfways is as follows. If you have remembered the line number on which you have stopped the CNC program, you can set the program such that it will continue working from this line.



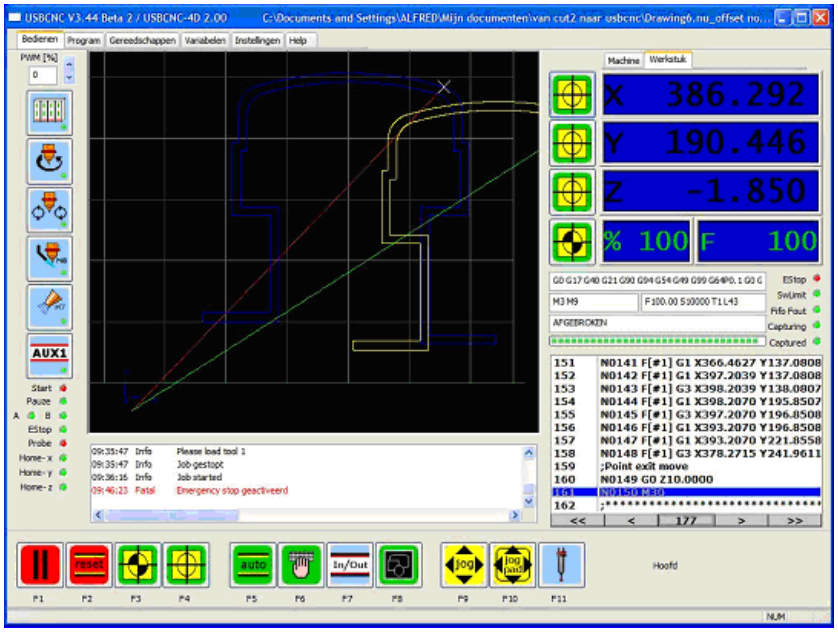
If you have entered the line number in the prompt, press F5. After pressing F5, press the Start button.



Note:
Starting from a line is a nice feature, but be very careful! USBCNC will execute the code exactly as its written! DamenCNC recommends to start on a line which contains a Z axis movement. Otherwise it could happen that the machine will be contouring above or below the Z value you would like.

The graphic display does not match with what the machine is actually milling

This question can best be illustrated using the following picture:



This problem can occur as a result of loading a .nc file, while the zeropoint has been changed.
USBCNC does not automatically update the graphic window. The reason for this is that with large files (i.e. programs containing more than 100.000 lines) there is a rather long processing time. There are however multiple easy remedies available to this problem:

- Reload the .nc file; type F12, F5, F4
- In the graphical menu menu, refresh the screen; type F12, F8, F11
- Change the working order; first correctly set the zeropoint, then load the .nc file

Chapter 6

Appendix A: Connecting an Emergency Stop



6.1 Introduction

In this chapter we explain how you can connect an emergency stop. An emergency stop enables you to shut down your CNC machine with a single push on a large emergency-stop button.

6.2 Hardware Connection procedure

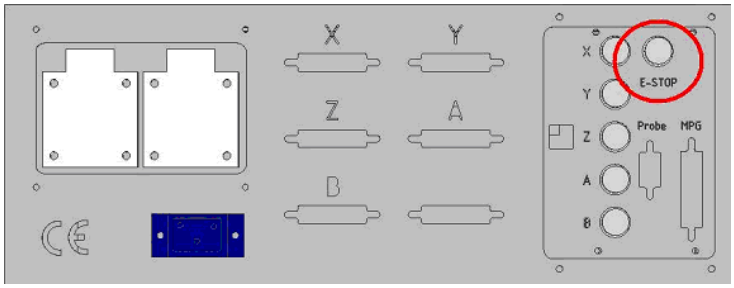
There are two options for connection of the Emergency Stop push button.

1. 1. You purchased a plug and play Emergency Stop on www.damencnc.com
2. 2. You purchased your Emergency Stop elsewhere

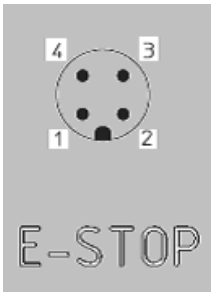
First DamenCNC's plug and play Emergency Stop is explained.

1. You purchased a plug and play Emergency Stop on www.damencnc.com

Connect the Emergency Stop by plugging the 4 pole connector which is attached to the emergency stop, in the bus of the RTR system as shown on the figure below.



2. You purchased a Emergency Stop elsewhere
The E-STOP bus on the connector side of your DamenCNC RTR set has 4 connections. These connections look as follows:



The wiring scheme for this Female connector is as follows:

- Pin 1: NC (Not connected)
- Pin 2: NC (Not connected)
- Pin 3: GND
- Pin 4: E-STOP Signal

The IPS Cable Assembly from DamenCNC allows you to connect your own Emergency Button to the DamenCNC RTR set. The wiring scheme of this IPS Cable Assembly is as follows:



- Pin 1: Brown
- Pin 2: White
- Pin 3: Blue
- Pin 4: Black

When the E-STOP signal is short-circuited with the GND (thus connecting Pin 3 and 4), USBCNC reads an E-STOP.

Chapter 7

Appendix B: Connecting a stepper motor



7.1 Introduction

In this appendix we explain how to connect a stepper motor to your DamenCNC RTR set. This appendix is built up as follows:

- **Hardware Connection procedure (both DamenCNC's stepper motors as well as other stepper motors)**
- **Setting the correct motor current**
- **Testing your stepper motors**

7.2 Hardware Connection procedure

There are two options for the stepper motors your system has.

1. You have purchased your steppermotor on www.damencnc.com
2. You have purchased your steppermotor elsewhere
3. You have purchased a DCNC-Driver/Motor Interfacecard

First DamenCNC's own steppermotors are explained.

1. You purchased your steppermotor on www.damencnc.com
You can now connect the motor by plugging in the 5 pole connector which is attached to the motor, in to the bus of the RTR system. Each bus has a name above the bus which indicates the respective axis.

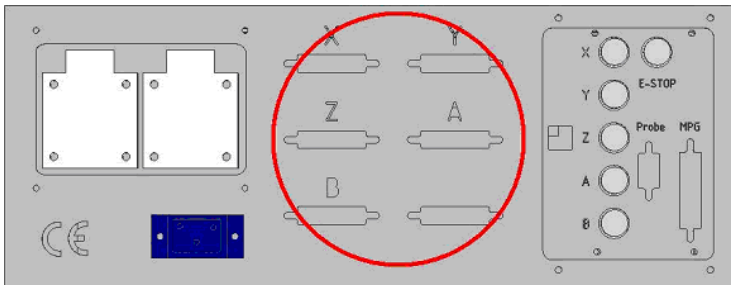


Figure 7.1: Plug in the motor in the respective bus

Note: Make sure only to connect or disconnect a motor when the power of the RTR is switched off!

2. You purchased your steppermotor elsewhere

The drivers which are located inside the RTR system can be schematically visualized as follows:

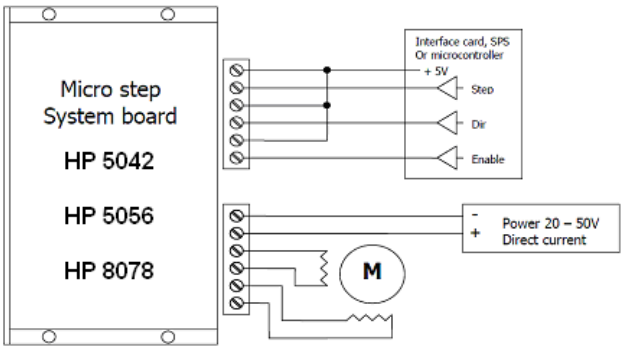


Figure 7.2: The lower four cables should be attached to the steppermotor

The four wires of the coils which control the motor are connected to the 5-pole axis bus on the RTR. When viewed from the outside of the RTR Controller it looks as follows:

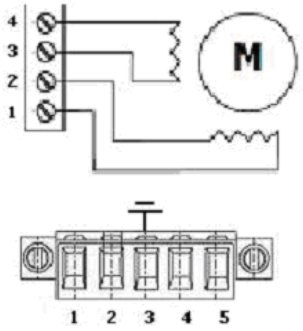


Figure 7.3: Terminal 1 and 2 are for the first coil, 3 is the GND, 4 and 5 are for the second coil

As can be seen from the figure, cable number 1 and 2 are for coil one.
The centre cable (cable number 3) is connected to the Ground of the RTR system; the shielding of the motor cable must be attached to this terminal. Be sure to connect the shielding of the cable only to this terminal; i.e. do not connect the shielding of the cable to the stepper motor itself. Connecting the shielding on both sides can result in EMC errors, this is due to the ground loop effect.
Cable number 4 and 5 should be connected to the second coil.

3. You have purchased a DCNC-Driver/Motor Interfacecard

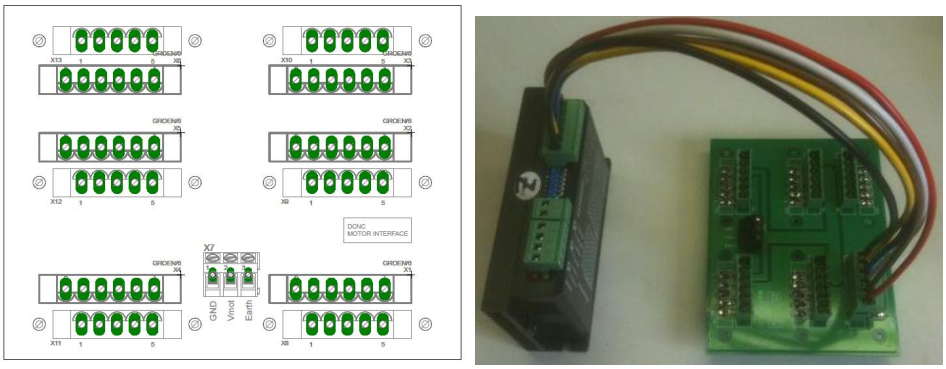


Figure 7.4: DCNC-Driver/Motor Interfacecard and a 5056 Driver

The CNC-Driver/Motor Interfacecard is the PCB that is integrated in the RTR system. We also sell it on the webshop as a seperate item. Therefore we provide some information on how to connect it when you are not using our RTR system.

- How to connect the powersupply
  On the DCNC-Driver/Motor Interfacecard you can see a black terminal block which as 3 connections. On the PCB there is a text describing which connection goes where. From left to right GND,Vmot and Earth. The GND is connected to the Ground of your DC powersupply, often also indicated as V-. The Vmot is connected to the V+ terminal of your DC powersupply. And the Earth is connected to the Earth of the powersupply. If the the powersupply does not have an earth connection, connect it to the housing or enclosre in which you are assembling your CNC controller. Please keep in mind that the stepperdrivers can have different maximum voltages. For example the 5056 has a maximum of 50V and the 8078 has a maximum of 80V. The voltage that you connect to Vmot will go to the stepperdrivers, so make sure it does not exceed the maximum for the drive you are using!

- How to connect the stepperdriver
  To make the connection between the interface and the stepperdriver we sell premade cables in the webshop. If you have purchased a premade cable. All you need to do is plug it into the 6 pole terminal block of the driver and the interface. Please keep in mind that the pin outs

on stepperdrivers will differ. For example the pinouts of the 5056 and the 8078 are different! When using the incorrect cable, damage to the system can occur!
If you wish to make your own cable, please see Table 7.1 for the correct pin numbering

Terminal NR	**Description**	**Cable Color**	**Connection on Stepperdriver**
1	Gnd	Black	Ground
2	Vmot	Yellow	Vmot or V+
3	A+	Blue	Coil A+
4	A-	White	Coil A-
5	B+	Brown	Coil B+
6	B-	Red	Coil B-

Table 7.1: Pin numbering of 6 pole terminal on DCNC-Driver/Motor Interfacecard

- How to connect the steppermotor
  The connection to the steppermotor is the same as has been described in Figure 7.3, which is on the previous page.

How to set the correct current on the stepperdrive
Each type of motor has it's own maximum rated current which can be found in the datasheet of the specific stepper motor. Each axis(XYZAB) has its own driver so the current needs to be adjusted for each individual axis(driver). Its not a problem to use a different driver and motor on each axis. Just be careful when plugging them in and out that you dont interchange them, since an incorrect current setting can damage the motor and driver.
The current can be adjusted with the dipswitches found on the stepper-drivers, inside the RTR system. The drivers have a table with information on how to set the dipswitches printed on the side. You can also find this information by looking at the datasheets of the stepperdriver, we have this on our website for each driver.

DamenCNC recommends that you start from the lowest current setting on the stepperdrive. This is to prevent any damage to motor and drive should anything be incorrectly connected. Step by step the current should be increased, to get more performance from the steppermotor. The testing procedure that we recommend is indicated in the next section.
Please bear in mind that a DamenCNC Classic RTR set contains a set of HP5056 drivers, and that a DamenCNC Performance RTR set comes with HP8078 drivers.

Test Procedure StepperMotors and Current setting
To test your stepper motors.

- Connect the cable to the steppermotor and connector as instructed above
- The motor is not yet connected to the mechanics of the machine
- Power off the RTR system!
- Plug the connector into the controller, whichever axis you wish to test (XYZAB)
- Power on your RTR system
- Start the USBCNC software
- Using the arrow keys and PG UP and PG DN on your keyboard you can individually control each stepper motor.

- How is the temperature of the steppermotor? It should stay cool (less than 60 degrees celsius), if the motors gets very hot, the current is set too high
- When the motor is not connect to a device its not uncommon to have some vibration
- If the motor stays cool and there are no excessive vibrations, connect the motor to your machine
- Test run your system, if there is no excessive heat of vibrations power off the RTR set and increase the current to the next step in the table
- Repeat the above procedure several times, at a certain point, the performance (speed and acceleration) will not increase any further, but the motor does get hot, at this point you need to go back one current setting in the table
- It is very common for a motor to be running at at its best at a much lower current setting than the maximum allowed according to the datasheet
- If the rotational inertia of the system is much lower than the inertia of the motor, vibrations will occur when the current is set to a high value. To prevent vibrations in low inertia systems, decrease the current setting.

Chapter 8

Appendix C: Connecting a Home sensor



8.1 Introduction

In this appendix we explain how you can connect a Home Switch/Sensor. A Home switch or sensor is necessary in order to prevent damage to your CNC system, by preventing the machine from translating outside a specified range. It also enables your CNC system to calibrate it's position. This appendix is built up as follows:

- **Hardware Connection procedure (both DamenCNC's sensors as well as other switches)**
- **Testing procedure**

8.2 Hardware Connection procedure

There are two options for the Home switch or sensor your system has.

1. You purchased plug and play Inductive Proximity Switches on www.damencnc.com
2. Your CNC system already has switches and these need to be connected to the DamenCNC RTR set

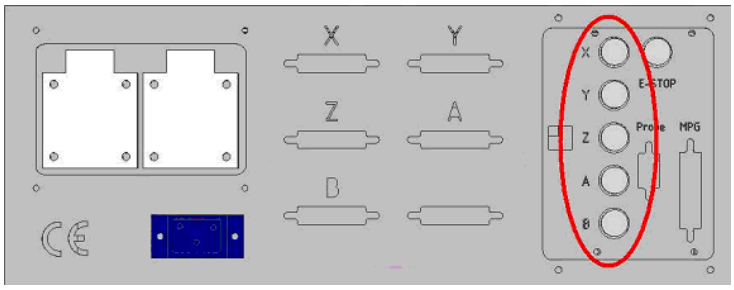
First DamenCNC's plug and play Inductive Proximity Switches are explained.

1. You purchased plug and play Inductive Proximity Switches on www.damencnc.com

The inductive sensor has a 4 pole connector attached to it's cable. The sensor thus consists of these components: Connect the inductive sensor by



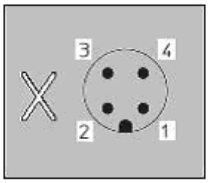
plugging in the 4 pole connector in the bus of the RTR system. Each contact has a name next to the bus which indicates the axis, as shown in the following picture:



Note: If you are using USBCNC to control a lathe there is a special version of the USBCNC CPU v4.0 which allows an inductive sensor to measure the RPM of the main spindle.

2. My CNC system has it's own Home switches or sensors
In the table we have provided the pin numbering you need to make the connections. Connecting your own sensor/switch is not difficult, but take your time making the connection. If you connect the sensors wrong, you risk breaking an optocoupler which opto-isolates the DamenCNC RTR set from the USBCNC software.

If you look at your DamenCNC RTR set you see a Female connector (in this case for the X-axis):



The wiring scheme of the Female connector is as follows:

- Pin 1: +12V (Output)
- Pin 2: NC (Not connected)
- Pin 3: GND
- Pin 4: Axis-signal

Consequently, if you look at the Male cable connector you see the following:



The wiring scheme of the Male connector is as follows:

- Pin 1: Brown
- Pin 2: White
- Pin 3: Blue
- Pin 4: Black

As soon as the +12V and the Axis-signal get short-circuited USBCNC reads a home-stop.
In order to connect a mechanical switch to this system, it should switch between Pin 1 and 4. If you do this, DamenCNC always recommends using a **NC** (normally closed) type of switch for safety reasons. As soon as an **NC** switch is accidently disconnected USBCNC will give an error, wheras with an **NO** switch no error can be given.)

The IPS Cable Assembly from our website can also be used to connect different types of sensors/devices, as long as the wiring scheme is followed carefully.

Testing the Home Sensors
If you open USBCNC and click on each respective Home switch or sensor, on the left of USBCNC's environment the right red LED (Home-x, Home-y etc) should turn green upon excitation. Test this for every axis you use.

14 Appendix D: MPG Handwheel Pendant

The MPG Handwheel Pendant is a very useful add-on, designed to improve the user friendliness of any CNC machine using USBCNC.

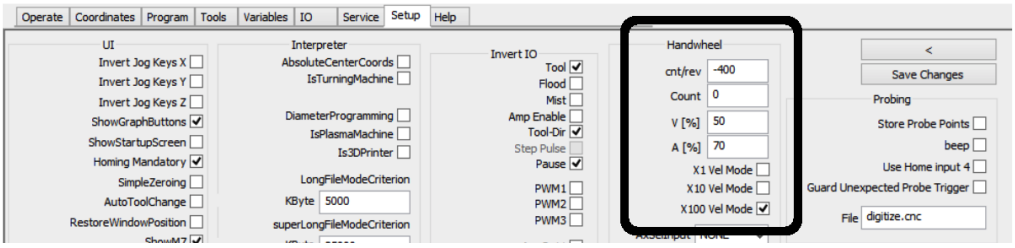


The MPG Handwheel Pendant has four main functionalities:

- Control and move machine axes individually
- Machine Feed Over Ride during operation
- Start and Pause your current job
- Zero-ing of individual axes

The installation procedure of the Handwheel can be found on the next page.

There are 5 settings which need to be set prior to using the MPG Handwheel. These settings can be found on the second page of 's Setup Tab.



Cnt/rev
Cnt/rev sets how many counts one complete revolution of the handwheel is. It is possible to invert the functionality of the Handwheel by placing a minus sign in front of the value. The cnt/rev is set to 400 counts by default.

Count
Count shows the current count position of the handwheel.
Count can be used to figure out how many counts/rev your handwheel has. When you start at zero counts, and perform a full 360 degree rotation of the handwheel the count value should read 400 for DamenCNC handwheels.

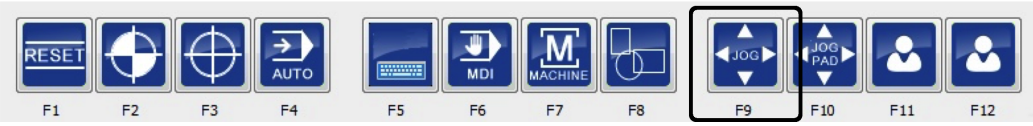
V[%]
V[%] is the maximum velocity in percentages (as is set in Setup tab) for each axis that will be used during moving your machine using the handwheel. This value is by default set to 50%, allowing the Handwheel to be useful for accurate determining the Zero position of individual axes.

A[%]
A[%] determines the percentage of the maximum acceleration (as is set in the Setup tab) of an axis that will be used during a jog with the handwheel.

X1...X100 Vel Mode
When the user selects any Velocity Mode, during jogging your machine will immediately stop when you stop rotating the handwheel.
The absolute position of the Handwheel will not be maintained if any velocity mode is selected.
If no velocity mode is selected, during jogging your axis may not immediately stop if the handwheel rotation stops. Therefore it is safe practise to to leave this setting just as is.

Control and move machine axes individually

The Handwheel allows the user to move the machine axes within 's functionality called JOG. The JOG menu can be found by pressing F9 in the main menu or click the JOG tab:



Now it is possible to control and move your machine using the Up / Down, Left / Right, Pg Up / Pg Dn keys on your keyboard. The Handwheel mode is activated by select any of the velocity modes by pressing F9, F10 or F11:



Here the velocity mode which is selected by pressing F9 is most sensitive, where each Handwheel rotational increment typically results in a 0.01mm movement. The F10 velocity mode typically yields a 0.1mm movement, and F11 usually results in 1mm displacement for each click. Press the right button **XYZAB** to switch to an other axis.

Machine Feed Over Ride during operation

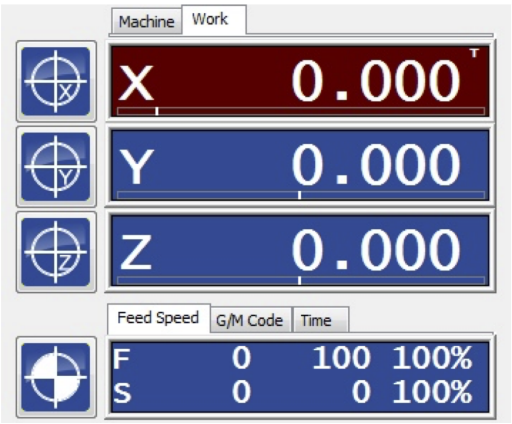
During a CNC job the machine feed rate can be reduced from 100% towards 0%, or increased up to 300% by rotating the Handwheel. This feature is useful when you are performing a prototype CNC job, or entered an either too high or too low machine feed rate.

Start and Pause your current job

The two push buttons on the MPG Handwheel can be used to pause and (re-)start your job during operation.

Zero-ing of individual axes

Use the Handwheel to zero individual axes within the Work Coordinate System (which differs from the Machine Coordinate System) by entering the JOG menu described on the previous page. Jog towards the location which you want to set to zero. When the machine has moved to the correct position, press the Zero (left) button. Note that the value on 's Digital Read Out (DRO) is now set to zero:

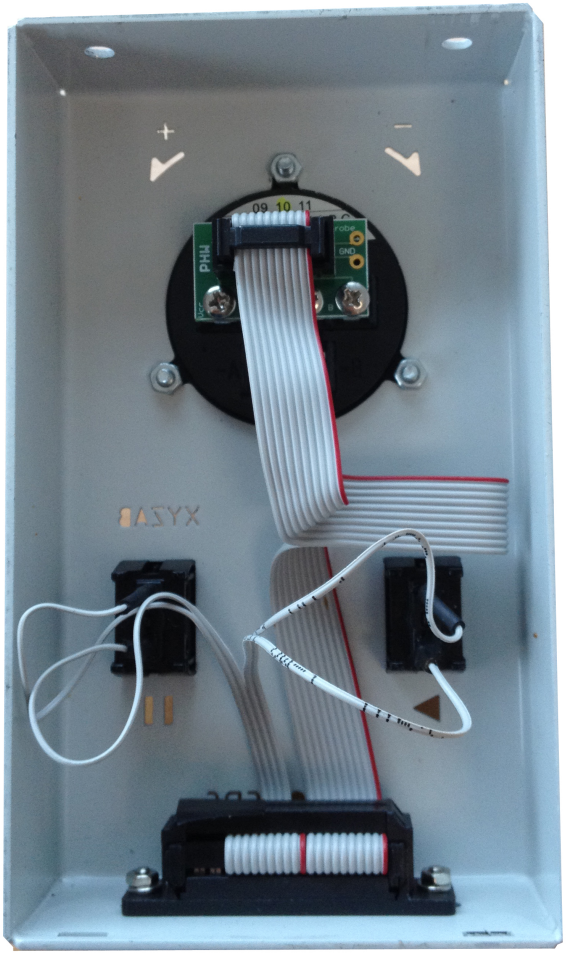


Press the right button **XYZAB** to switch to another axis if you want to zero another axis as well, and repeat the process.

Electrical Pin-out 25P SUB-D and Ribbon cable

Note that the first cable of the Ribbon cable is red, which corresponds to Pin 1 of the 25P-SUB-D Connector.
When you mount the Ribbon cable you can use this conversion table:

Pin Number 25P SUB-D	Function	Comment	Cable Number Flat Ribbon cable
1	Not connected		1 - RED
2	HANDWHEEL -A	Encoder Signal A	3
3	HANDWHEEL -B	Encoder Signal B	5
4	Not connected		
5	+5VDC USB	Power from CPU	9
6	Not connected		
7	RUN (Zero axis)	Input on CPU	13
8	PAUSE (Change axis)	Input on CPU	15
9	Not connected		
10	Not connected		
11	Not connected		
12	Not connected		
13	Not connected		
14	Not connected		
15	Not connected		
16	Not connected		
17	GND PHW	Ground PHW	8
18	Not connected		
19	Not connected		
20	GND RUN	Ground RUN	14
21	GND PAUSE	Ground PAUSE	16
22	Not connected		
23	Not connected		
24	Not connected		
25	Not connected		


+
09 10 11
PHW
GND

Frequently asked questions regarding MPG Handwheel

Q: When I press the Zero button, it does not zero but place a value?

A: When you have entered a tool diameter in the Tool Table and that tool is loaded, will compensate for half the diameter when pressing the ZERO button. The area which we have circled in red, is where displays the values it has stored in memory. So if in your G-code you would not specify a Feed rate (f) when programming a G1 code, would use the value of 60.00mm / min stored in memory. The same is true for the tool, if not written in your program, will assume it can use the tool stored in memory, in this case tool 1. If you don't want to compensate for the Tool Diameter, go to the Tab Tools, and change the Tool Diameter of the tool to 0.00, and save your settings.

Q: When I rotate the MPG Handwheel clockwise (+ direction), my coordinates (or Feed Over Ride) decreases in value, whereas I expect them to increase?

A: To fix this problem, go to the Tab Setup. There is a box designated for the MPG Handwheel. In this box you see the subbox cnt/rev. If you add an minus to the value you see there (i.e. -400 instead of 400) or vice verse, the MPG Handwheel will reverse its counting. Don't forget to save your settings!

◇

Chapter 10

Appendix E: Setup of a Delta VFD and a Teknomotor

10.1 Introduction

This section describes the setup of a Delta Variable Frequency Drive (VFD), in combination with a Teknomotor.

The Delta VFD is a device which can interface with USBCNC or stand-alone; allowing the user to control an electrospindle such as a Teknomotor.

Teknomotors are known for their high reliability, high performance and low maintenance.

This documentation is also included in the scope of delivery for a Delta VFD; this documentation can be used as a quickstart manual.

Since a VFD contains multiple capacitors, always be very careful! After unplugging the VFD device be sure to wait several minutes to ensure the VFD is discharged!

There are three Delta VFD types: VFD-S, VFD-E and VFD-L. Therefore small deviations in figures might be present, but the connection procedure is the same.

The setup procedure if the Delta VFD and a Teknomotor is as follows:

1. Connect a 220V / 380V euro cable
2. Connect a brake resistor
3. Program the VFD
4. Remove power and wait several minutes
5. Connect the Teknomotor
6. Testing procedure
7. FAQ

10.2 Connection procedure

1. Connect a 220V / 380V euro cable

Strip a 220V / 380V euro cable and attach fork terminals on the wires. It is essential that the used 220V / 380V plug is grounded. Remove the Delta VFD plastic protection plate on the lower side of the VFD.

In case a 220V euro cable is used, the terminals must be mounted as can be seen in the figure 10.1 (observe the wiring diagram on the left).

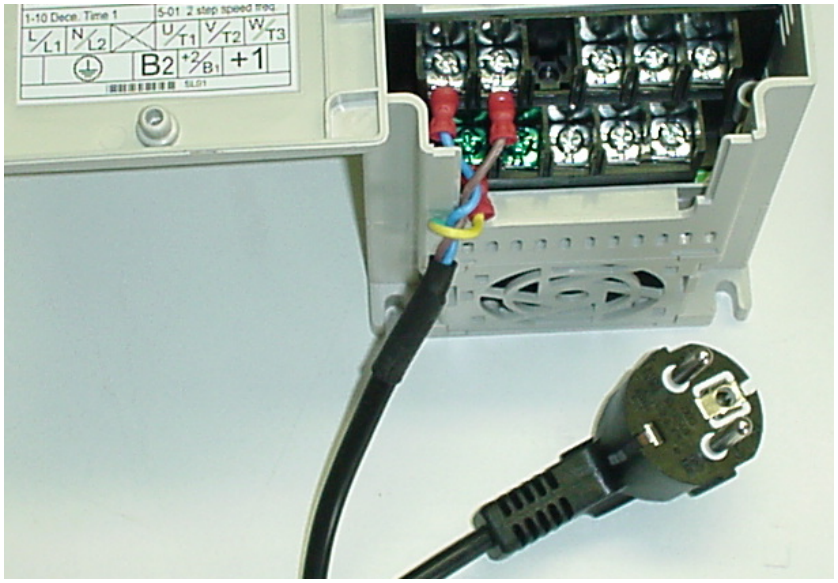


Figure 10.1: Mount the 220V blue, brown and yellow/green terminal cable on L/L1, N/L2 and GND (earth logo) respectively

This wiring diagram holds for any version of a 220V Delta VFD.

2. Connect a brake resistor

A brake resistor is a device which allows the Delta VFD to dump rotational energy into. This is necessary when quick variations in angular velocity are required.
If you do not mount this brake resistor and your Delta VFD reports an over-voltage error message during operation. In this case you need a brake resistor.
As a rule of thumb, if you have a 18.000 RPM HF Spindle Teknomotor and do not have strict requirements with respect to acceleration and deceleration (i.e. if 3 seconds run up time is enough) you do not need a brake resistor.

A 24.000 Electrospindle Teknomotor will require a brake resistor.

DamenCNC offers two types of brake resistors:

- ERF 150W J101
- ERF 150W J401

The first brake resistor (ERF 150W J101) is suitable for a 220V Delta VFD.
The latter brake resistor (ERF 150W J401) is suitable for a 380V Delta VFD.

These brake resistors can be installed by attaching fork terminals to the ends of the two wires. Be sure to perform some extra stripping of the sleeve in order to firmly connect the fork terminals).
These fork terminals should then be mounted on port **B2** and port **+2/B1** (it does not matter which wire goes onto which port) as shown in figure 10.2:

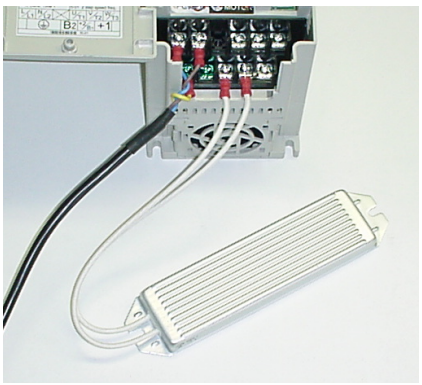


Figure 10.2: Wiring diagram for a brake resistor

3. Program the Delta VFD

The Delta VFD operating parameters which are set in the Delta factory need to be changed in order to control a Teknomotor.

In table 11.1 the user can find the parameters which need adjustments; aftwerards we explain how you can change each parameter.

Note:
The maximum operating frequency parameter depends upon the used Teknomotor model: an 18.000 RPM HF spindle Teknomotor has a maximum operating frequency of 300Hz and a 24.000RPM Electrospindle Teknomotor has a maximum operating frequency of 400Hz.

You should always use the Delta VFD in an environment where your 220V socket is **NOT** attached to a group which supplies power to any important electronical (data) devices; the capacitors in a VFD might trigger your RCD (Dutch: aardlekschakelaar)!

In addition, **NEVER** use the 220V socket from your RTR set!

Instead, use a seperate wall socket.

Parameter	Factory	18.000 RPM	24.000 RPM	Function
2-00	D0	D3	D3	Potentiometer speed control
1-00	D60	D300	D400	Maximum operating frequency
1-01	D60	D300	D400	Maximum frequency
1-09	D10	D3	D3	Acceleration time in seconds
1-10	D10	D3	D3	Deceleration time in seconds

Table 10.1: Parameters which need to be changed for the Delta VFD

Setting a parameter

In this example it is explained how you can change the first parameter 2-00; all other parameters can be set in the same way.

1. Apply power by inserting the 220V / 380V plug in a socket which is **NOT** attached to a group which supplies power to any important electronical (data) devices or your RTR set; the capacitors in a VFD might trigger your RCD (Dutch: aardlekschakelaar)!
2. Press **Prog data** button
3. Using the **Up** and **Down** buttons browse and select **2**
4. Press **Prog data** button
5. The display should now read 2-00
6. For the first parameter you do not need to change the values after the **2-**, press **Prog data** button again
7. The factory setting for the speed control is **D0**.
   Using the **Up** and **Down** buttons browse and select **D3**
8. Press **Prog data** button
9. The display should now display **END**, which indicates that this parameter is programmed.

All other respective parameters can be set by repeating step 2-9.

After this programming, verify that if you press the **RUN** button, and rotate the potentiometer to its max value, the digits display the correct maximum frequency (300 for a 18.000 RPM HF spindle Teknomotor, or 400 for a 24.000 RPM Electrospindle Teknomotor).

4. Remove power and wait several minutes

The Delta VFD capacitors are still charged when you disconnect the 220V / 380V euro cable. Be very patient and wait several minutes until the Delta VFD is discharged.
Serious injury and damage to your components can occur!

5. Connecting the Teknomotor

The Teknomotor can be used in either **Star** or **Triangle** configuration.

If you use a monophase (220V) DamenCNC's Delta VFD in combination with a Teknomotor, the Teknomotor should be configured in a **Triangle** configuration as we have a 1 phase 220V input, and a 3 phase 220V output.

The Teknomotors are factory set in a **Star** configuration, which means that it is required to adjust the small metal connection plates such that the connection corresponds to the following figure 10.3:

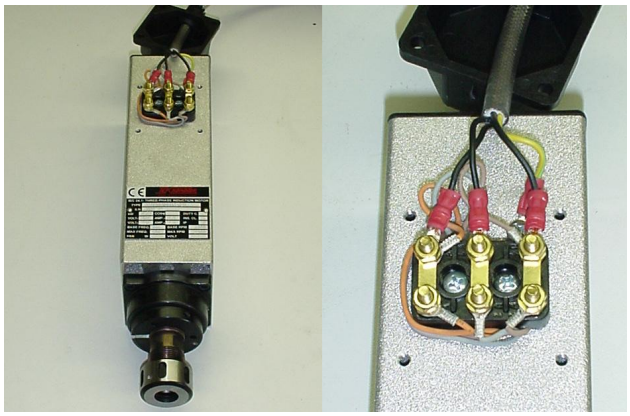


Figure 10.3: If you use a monophase 220V Delta VFD, mount the small gold metal connection plates as shown in order to configure the Teknomotor in **triangle** configuration

If you have a 380V 3 phase connection, leave the Teknomotor in the factory star setting.

Now that the configuration of the Teknomotor is set, we proceed to the wiring loom.

Wiring loom

Prepare your 4-pole cable by cutting the (preferably shielded **4x 0.75mm**^{2}) cable to the desired length, and trim away the sleeve of the cable on **5cm** of both ends such that the inner cables are visible.
Guide the cable through a cable gland with the corresponding black cover box.

On one end apply fork terminals to the four inner cables, to the other end apply cable ends. Now mount the cables as shown in table 10.2 and figure 10.4:

Wire number	Motor connection number
1	U_1
2	V_1
3	W_1
G	Ground (casing of the Teknomotor)

Table 10.2: Wiring diagram of the Teknomotor

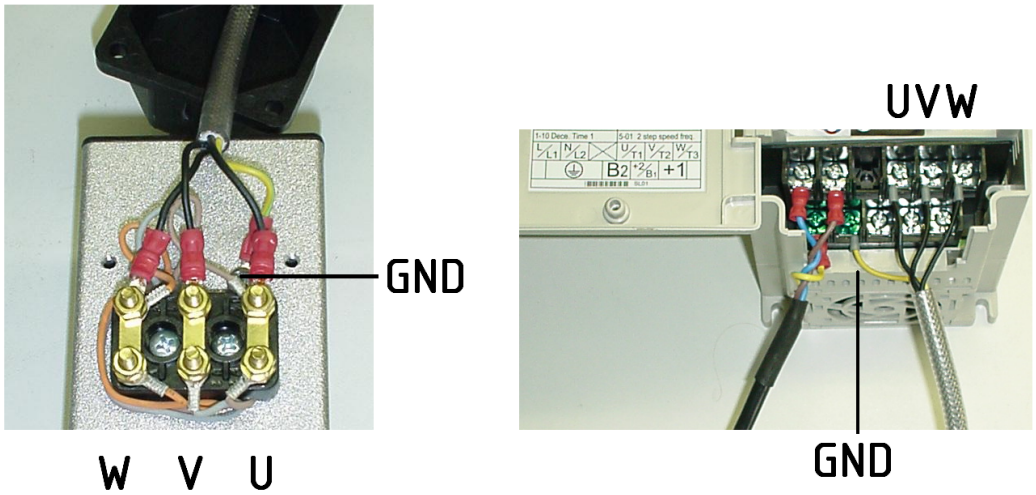


Figure 10.4: Wiring diagram on the Teknomotor (left) and Delta VFD (right)

When you have completed the wiring loom assembly, screw the Delta VFD plastic protection plate back in to its original position, such that the wire connections are protected.

6. Testing procedure

Now the assembly is ready for testing, according to the following testing procedure:

1. Remove the clamping nut from the Teknomotor
2. Rotate the potentiometer counterclockwise to a minimum value
3. Apply power to the Delta VFD
4. Press **RUN** on the Delta VFD
5. You should now be able to control the RPM of the Teknomotor with the potentiometer

Always try to accelerate the spindle by quickly rotating the potentiometer to full speed, then deccelerate the spindle by rotating the potentiometer to zero speed. If the Delta VFD reports an over-voltage error message while doing so, you need a Brake resistor.

Details on how to install a brake resistor can be found in step 2.

Always be sure to verify if the Teknomotor rotates in the right direction. The right direction is indicated in figure 10.5:



Figure 10.5: The correct rotation for any Teknomotor

In the case that the electrospindle rotates the wrong direction, unscrew the plastic Delta VFD protection plate and switch the motor cable **U** with the motor cable **W**. Be sure to leave terminal **V** intact.
This operation will reverse the rotation.

7. FAQ

Frequently Asked Questions are dealt with in this section.

Q 1: Why does my motor runs erratic at low RPM values?

A: Most probably you are using a different motor than a Teknomotor. In order to install your motor to the Delta VFD, try changing the parameters listed in table 10.3:

Parameter	Factory	DamenCNC setting	Function
6-00	D1	D0	Over voltage protection
7-02	D1	D0	Torque compensation

Table 10.3: Parameters which might solve an erratic operation at low RPM of your motor

Be sure to test one of these remedies at a time, as one of these settings might prove to be a solution.

Q 2: The Delta VFD returns an over voltage error during operation. How can I fix this?

A: This error can be solved by installing a brake resistor.
Please read Appendix 10.2.2 "Connect a brake resistor" on how to do this.

Q 3: The Delta VFD does not respond to the keypad. How can I fix this?

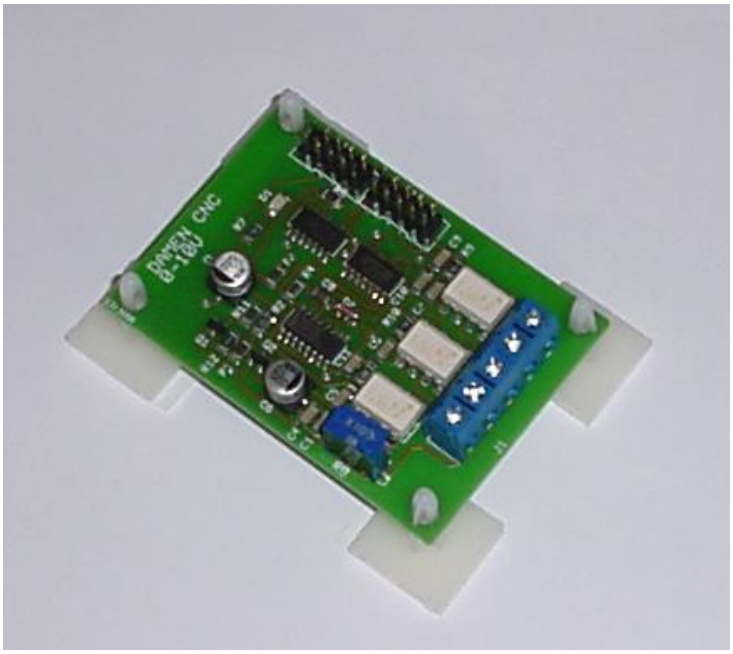
A: Parameter 2-01 is incorrectly set.
Try changing this parameter to d0 (factory setting) and try again.

Q 4: The Delta VFD does not respond to the potentiometer. How can I fix this?

A: Parameter 2-00 is incorrectly set.
Try changing this parameter to d3 and try again.

Chapter 11

Appendix F: Connecting a 0-10V Spindle Speed Control



11.1 Introduction

In this chapter we explain how you can connect a 0-10V Spindle Speed Control. A 0-10V Spindle speed control is a small PCB which is able to interface with USBCNC V3, V4 and V5B; allowing USBCNC software to control a (Delta) Variable Frequency Drive (VFD) which in its turn can control a (Tekno)motor HF - or electrospindle.

In order to install the 0-10V Spindle Speed Control PCB it is highly recommended to first read and perform Appendix E: Setup of a Delta VFD and a Teknomotor.

By doing so the user can control the (Tekno)motor without USBCNC, which might already exclude any errors.

After the user has performed the entire Appendix E procedure, the system can be prepared in the following sequence:

First the **hardware connection** procedure will be discussed:

1. **I have a DamenCNC RTR Performance (standard with 0-10V Spindle Speed Control) set and a Delta VFD with a cable assembly**

2. **I have a DamenCNC RTR without a 0-10V Spindle Speed Control Delta VFD**

3. **I have a USBCNC CPU V4.0 and a Delta VFD**

Afterwards the **software setup** is explained. The software setup consists of the setup of the DELTA VFD, and the setup of USBCNC V5.

11.2 Hardware Connection procedure

1. I have a DamenCNC RTR Performance (standard with 0-10V Spindle Speed Control) set and a Delta VFD with a cable assembly

Observe the RTR set: The connection for the 0-10V Spindle Speed Control is located as follows:

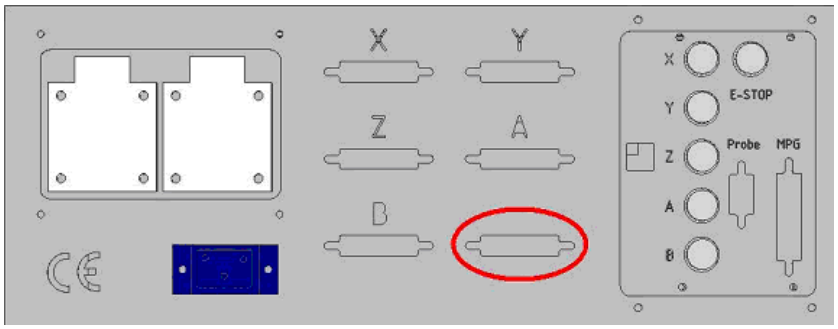
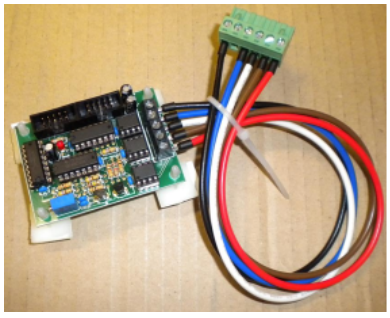
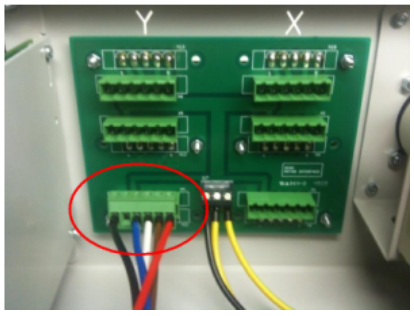


Figure 11.1: Connection for the 0-10V Spindle Speed Control on the RTR set

Visually verify that the connections are as follows:





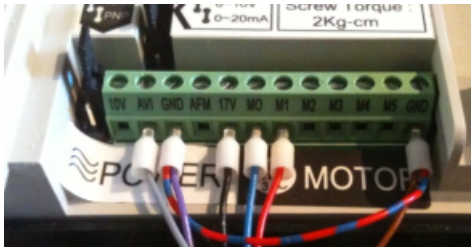
Connect the 0-10V Spindle Speed Control to the DELTA VFD by plugging the 5-pole connector which is assembled to the DELTA VFD, into the 5-pole bus outside the RTR system as shown in the figure 11.1.

The cable which is connected to the DELTA VFD has the following wiring scheme, when shown from above:



Figure 11.2: Wiring scheme from above: black, blue, purple, red, grey

On the other end of the cable, where the cable is introduced in the DELTA VFD, there are two different lay outs. For the VFD 015 and the VFD 022 the wiring scheme is as follows:



- AVI - Grey
- GND - Brown
- 17V - Black
- M0 - Blue
- M1 - Red
- GND (left) - Purple
- GND (right) - Brown (solder to cable shielding, connect with purple)

This concludes the hardware setup. Please proceed to the test section.

2. I have a DamenCNC RTR without a 0-10V Spindle Speed Control Delta VFD

When there is no 0-10V Spindle Control PCB already built in the RTR system, it has to be build in first.

Connect the 0-10V Spindle Control PCB cable to the lower left socket of the interface print (view from inside the RTR system) using the supplied cable, as can be seen in figure 11.3.

The 0-10V Spindle Control PCB itself should be fixed using sticky feet on the bacside of the USBCNC CPU holder (in figure 11.3, far left on the image on the right).

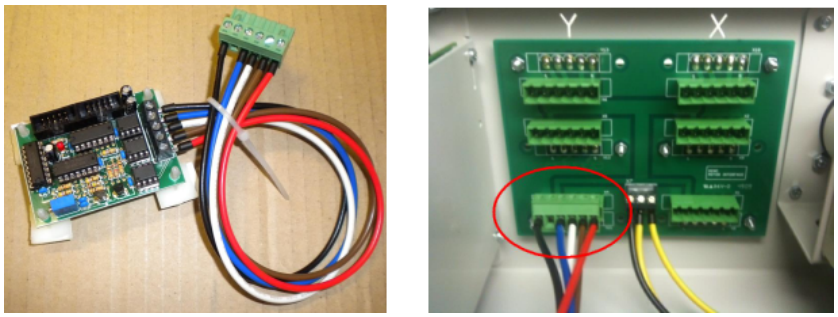


Figure 11.3: Plug the 0-10V Spindle Control PCB cable in the lower left cable slot

When you have an older version of the RTR, there might be not a interface print as shown in the photo above.

In this case one has to be make a custom connection.

This custom connection can be seen in the subsection which discusses

I have a USBCNC CPU V3.0 or V4.0 and a Delta VFD.

Connecting the 0-10V Spindle speed control PCB to a USBCNC CPU

The OUT signal from the USBCNC CPUV4.0 has to be connected to USBCNC CPU V4.0 socket from the 0-10V Spindle Control as shown in figure 11.4:

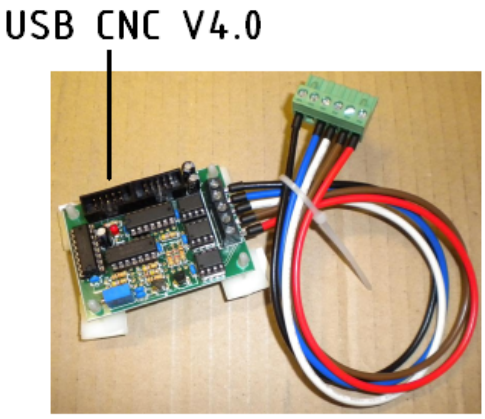


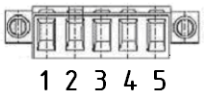
Figure 11.4: Terminal for USBCNC CPU V4.0

The OUT signal from the USBCNC CPU4.0 is connected to the same 10P ribbon cable which controls the relais PCB which controls Tool/Flood/Aux/Mist settings.

When this component is built in correctly, and you have a Delta VFD cable assembly, follow the steps of
I have a DamenCNC RTR Performance (standard with 0-10V Spindle Speed Control) set and a Delta VFD with a cable assembly.

If you do not have a Delta VFD cable assembly, you can create one using preferably a **5 x 0.25mm**^{2} shielded cable.

This cable should have the wiring scheme presented in figure 11.2:



RTR Connector	Delta VFD
1	17V
2	M0
3	GND
4	M1
5	AVI

Figure 11.5: Wiring scheme for connecting the 0-10V Spindle speed control PCB to the Delta VFD

Be sure to connect RTR connector 3 to both Ground (GND) terminals on the Delta VFD.

Now you can proceed to the software setup.

3. I have a USBCNC CPU V3.0, V4.0 or V5.0 and a Delta VFD

Connection to USBCNC CPU

Connect the supplied 10 pole connector to the SV6 port of USBCNC CPU V3 PCB or for USBCNC CPU V4 users on the OUT slot of CPUV4.

Connect the other end to the 0-10V output card on the respective slot for your USBCNC CPU; see figure 11.6 for your correct terminal.

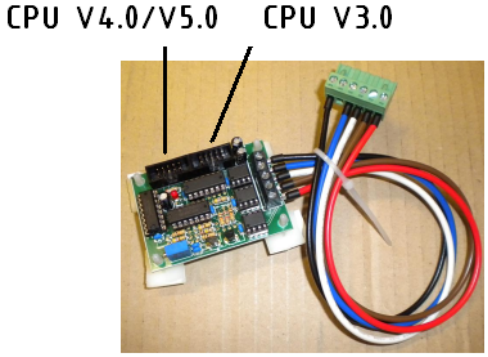


Figure 11.6: Terminals for USBCNC CPU V3.0 and USBCNC CPU V4.0

Connecting a 0-10V Spindle Speed Control PCB to a Delta VFD

Make sure the Delta VFD and the RTR set are both turned off while making these connections. The wiring scheme can be seen in figure 11.7 and figure 11.8:

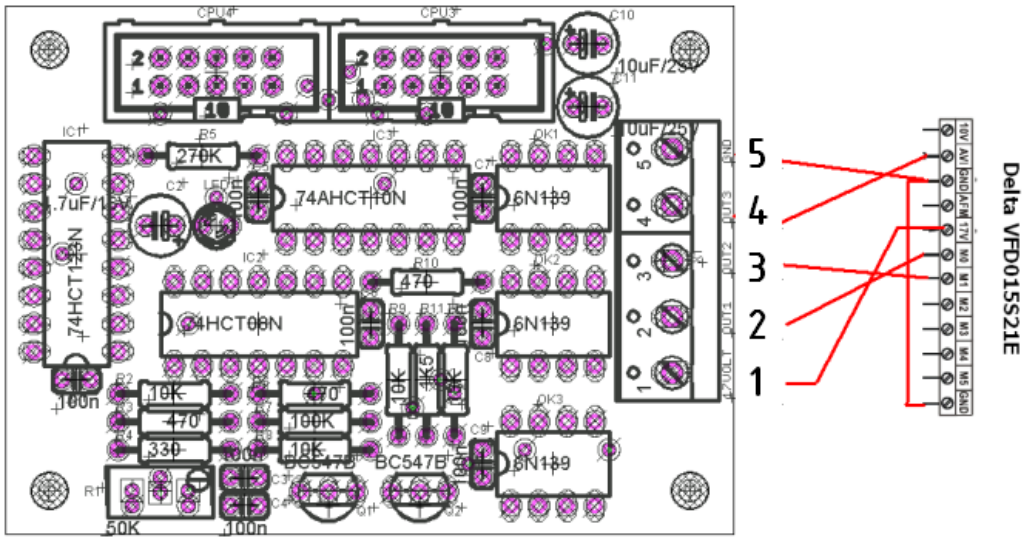


Figure 11.7: Wiring scheme for connecting a 0-10V Spindle speed control PCB to a Delta VFD

0-10V PCB		Delta VFD
1	17 V	17V
2	OUT 1	M0
3	OUT 2	M1
4	OUT 3	AVI
5	GND	GND

Figure 11.8: Wiring scheme for connecting a 0-10V Spindle speed control PCB to a Delta VFD

Be sure to attach 0-10V PCB pin 5 (GND) to both ground terminals on the Delta VFD connector.

Note:
Use a (twisted) shielded cable for the above mentioned connections.

The shielding of the used cable should only be connected at the connector at the side of the drive (or the RTR side of the cable).

Always connect the shielding of the used cable to the **Earth terminal** on the DamenCNC motor interface card.

Now you can proceed to the software setup.

11.3 Software setup

The software setup consists of two parts.

1. Setup of the Delta VFD
2. Setup of USBCNC Software

Software setup of the Delta VFD

The Delta VFD should be set up such that there is a communication between the USBCNC Software and the Delta VFD. This communication can be established by performing the following steps:

Unplug the connection between the Delta VFD and the 0-10V Spindle speed control PCB.

Apply power to the VFD; please bear in mind that you should always use the Delta VFD in an environment where your 220V socket is **NOT** attached to a group which supplies power to any important electronical (data) devices. This is because the Delta VFD might trigger your RCD (Dutch: aardlekschakelaar)!

In addition, **NEVER** use the 220V socket from your RTR set; instead use a seperate wall socket!

Usually a VFD obtains its inputs from the potentiometer and the interface. Since we want to control the speeds via a 0-10V input we need to change the parameters of the drives such that it takes the speed input at the 0-10V input. In table 11.1 one can find the parameters which need to be adjusted. After table 11.1 we explain how you can adjust each parameter.

Parameter	Factory	18.000 RPM	24.000 RPM	Function
2-00	D0	D1	D1	0-10V speed control
2-01	D0	D1	D1	External start / stop
1-00	D60	D300	D400	Maximum operating frequency
1-01	D60	D300	D400	Maximum frequency
1-09	D10	D3	D3	Acceleration time in seconds
1-10	D10	D3	D3	Deceleration time in seconds

Table 11.1: Parameters which need to be changed for the Delta VFD

Setting a parameter

In this example it is explained how you can change the first parameter 2-00; analogous to all other parameters.

1. Apply power by inserting the 220V plug in a socket which is **NOT** attached to a group which supplies power to any important electronical (data) devices or your RTR set; the Delta VFD might trigger your RCD (Dutch: aardlekschakelaar)!
2. Press **Prog data** button
3. Using the **Up** and **Down** buttons browse and select **2**
4. Press **Prog data** button
5. The display should now read 2-00
6. For the first parameter you do not need to change the values after the **2-**, press **Prog data** button again
7. The factory setting for the speed control is **D0**.
   Using the **Up** and **Down** buttons browse and select **D3**
8. Press **Prog data** button
9. The display should now display **END**, which indicates that this parameter is programmed.

All other respective parameters can be set by repeating step 2-9.

There are two jumpers pre installed on a Delta VFD. These jumpers must be set such that 0-10V - and NPN are connected, as can be seen in figure 11.9:

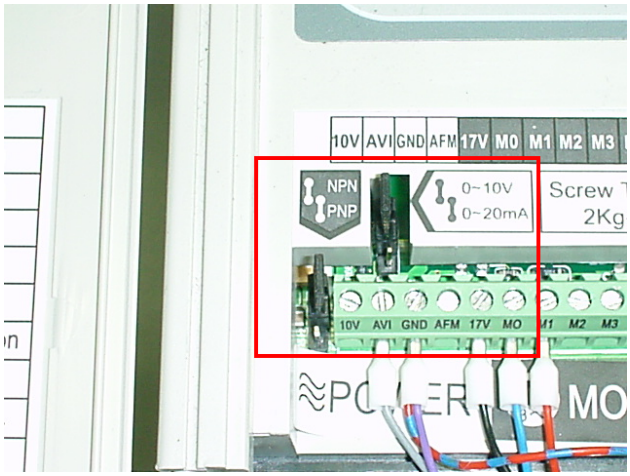


Figure 11.9: Jumper settings on 0-10V and NPN

Please proceed to the software setup of USBCNC Software.

Software setup of USBCNC Software

The Software setup of USBCNC consists of three steps.

1. Set the correct outputs
2. Testing without a Teknomotor
3. Testing with a Teknomotor

1. Set the correct outputs

Navigate in the USBCNC environment to the Tab Setup.

Make sure that under the Spindle box the parameters PWM is checked.

The term Ramp up time indicates the time the spindle needs to start.
The Ramp up time varies per spindle, but usually 0.5 to 2 seconds is sufficient. Furthermore Max S is the maximum allowable RPM of the spindle. In the case of a Teknomotor this is either 18000RPM or 24000RPM (this depends on the type of Teknmotor used).
It is also possible to indicate a minimum RPM value, as can be seen in figure 11.10.

As the name implicates, the option proportional ramp up time allows the user to poroportionally vary the acceleration time as a function of the required RPM value. So if you set the start up time to 2 seconds, it will require 2 seconds to reach 24000RPM; and 1 second to reach 12000RPM.

It might be required to adjust the USBCNC Software **Invert IO** functions. Ensure that the Tool Dir option is checked, and PWM is unchecked, as can be seen in figure 11.10.

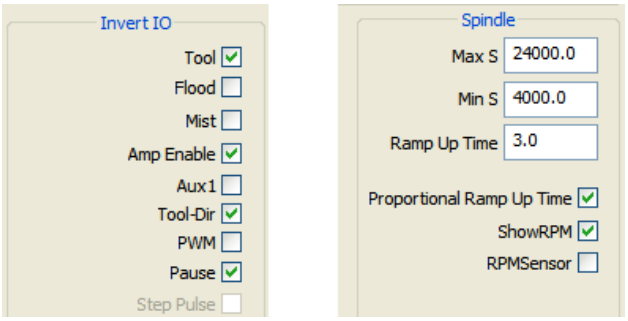


Figure 11.10: USBCNC V5 Setup tab: it is important to check Tool Dir, and uncheck PWM. Indicate the correct Teknomotor limits in the Spindle menu

2. Testing without a Teknomotor

In order to test the communication between USBCNC Software and your Delta VFD, follow this procedure:

1. Disconnect your Teknomotor from the Delta VFD
2. Connect your USBCNC CPU to your PC and connect the 0-10V Spindle speed control PCB to the Delta VFD
3. Apply power by inserting the 220V / 380V plug in a socket which is **NOT** attached to a group which supplies power to any important electronical (data) devices or your RTR set; the Delta VFD might trigger your RCD (Dutch: aardlekschakelaar)!
4. Open the MDI (Manual Data Input; F12; F6 in the Operate Tab) function. If you use a 18000RPM Teknomotor, input "M3 S18000" + Enter; as can be seen in figure 4

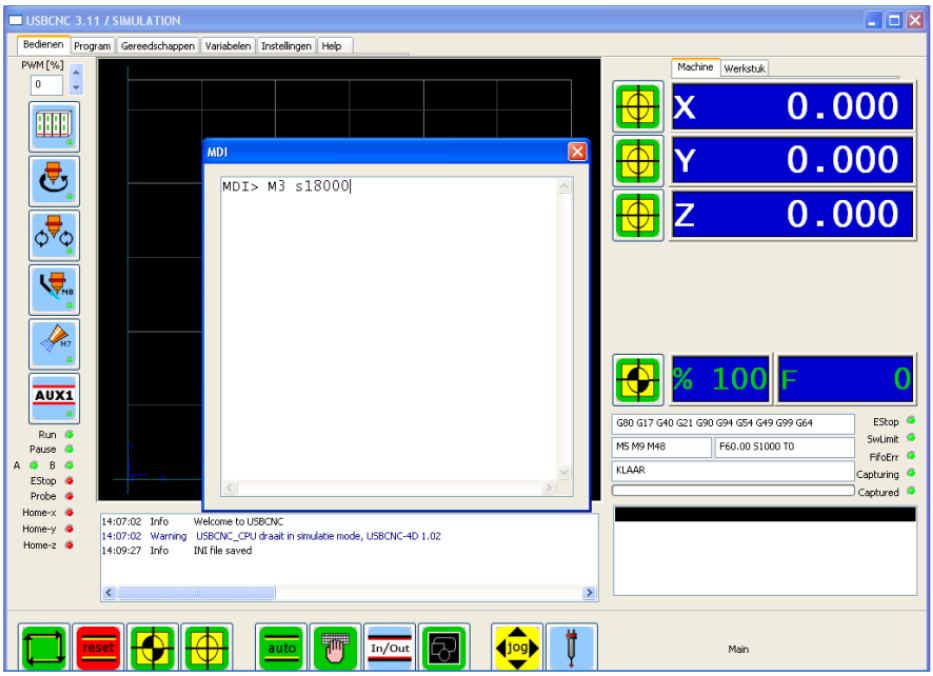
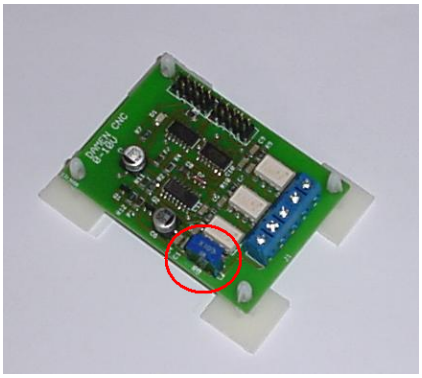


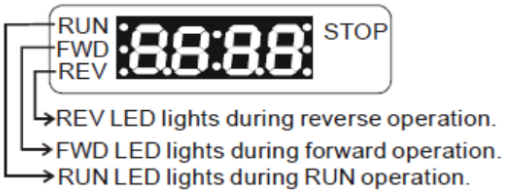
Figure 11.11: Open your MDI and type: "M3 S18000" + Enter

5. When the software is set to M3 S18000, the display on the VFD should read 300. If this is not the case please adjust the trim-potentio meter(the blue component with the bronze screw on your 0-10V Spindle speed control PCB) such that the value 300 is acquired.



6. On the VFD-Display the RUN LED should be lit, and also the FWD LED

7. If you use a 24000RPM Teknomotor, the display of the Delta VFD should read 300 when you input "M3 S24000" + Enter in the MDI.



8. Now input M4 S18000 in the MDI (F12; F6)

9. On the VFD-Display the RUN LED should be lit, and also the REV LED

10. With the M5 command the spindle is turned off

If all of the above checks are correct you can continue to the next step: Testing with a TeknoMotor.

3. Testing with a Teknomotor

First remove the clamping nut from the Teknomotor spindle.

Afterwards open the MDI (F12; F6), and prompt the following sentence: M3 S4000.

Verify that the spindle rotates in the anti clockwise direction (if seen from the front of the spindle) as can be seen in figure 11.3:



Figure 11.12: The arrow indicates the correct forward rotating direction

In the case that the Teknomotor rotates the reverse direction, unplug the 220V / 380V plug.

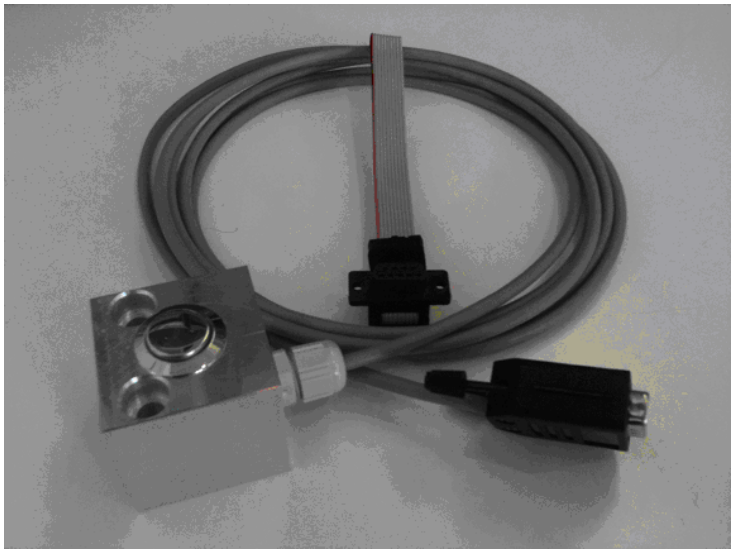
After a few minutes you can unscrew the plastic Delta VFD protection plate and switch the motor cable **U** with the motor cable **W**. Be sure to leave terminal **V** intact.
This operation will reverse the Teknomotor's direction of rotation. Mount the plastic Delta VFD protection plate again and plug in your 220V / 380V plug, and try again. The system should now rotate in the correct direction.

Note:
Always make sure the Emergency Stop of the machine is working properly and run the machine with limit and home switches active.

Even a 1,6 kW motor can cause serious hazard to operators and spectators.

Chapter 12

Appendix G: Connecting a Toolsetter



12.1 Introduction

Thank you for purchasing the DamenCNC toolsetter.
This device is capable of measuring Z-axis coordinates, which enables the operator to perform 2 actions:

1. **Zero-ing the Z-axis**
   The operator can zero the Z-axis by placing the toolsetter on top of the to-be-machined material. The USBCNC V3 software can detect the toolsetter, and therefore the thickness of the material becomes evident.

2. **Perform tool length measurements**
   The operator can also let different tools touch the toolsetter. The different tool lengths are stored in the tooltable of USBCNC. Also the tool diameter can be entered here manually.

In this Appendix we explain how to connect and use the toolsetter in the following order:

- Hardware Connection procedure
- Testing procedure
- Using the toolsetter to zero the Z-axis
- Using the toolsetter to measure tool length
- Detailed info on hardware connection

Scope of delivery

- The toolsetter itself with 3 meter cable and connector
- A conversion cable
- A Sub-D Mount Screw Set
- This user manual



Figure 12.1: Scope of delivery

12.2 Hardware Connection procedure

In order to use and connect your toolsetter, please select your system.

1. You have a DamenCNC RTR Classic or Performance, with USBCNC CPU V4 or V5
2. You have an USBCNC CPU V3, V4 or V5 (No RTR set)

First the DamenCNC RTR set with USBCNC CPU V4 or V5 is explained. After this section we discuss the installation of USBCNC CPU's without a RTR set.

1. You have a DamenCNC RTR Classic or Performance, with USBCNC CPU V4 or V5

If you have a DamenCNC RTR Classic or Performance, select if you purchased this set before January 2010, or after January 2010:

You purchased your RTR system after January 2010

In this configuration, you do not need the delivered 10 pole cable. This is because this component is already integrated in the RTR housing. Simply plug the toolsetter's SUB-D connector into the designated slot (with the notification "PROBE") at the back of your RTR housing, as shown below.

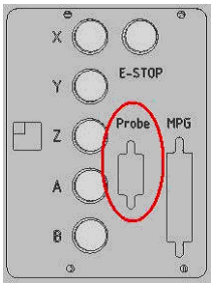


Figure 12.2: Plug the SUB-D connector into the Probe slot

You purchased your RTR system before January 2010
In this configuration, you do need the 10 pole conversion cable, since the required component is not yet integrated in the RTR housing.

- Open the RTR set by unscrewing the 8 screws, and lift the top cover.
- Remove one of the white cover plates at the backside of the RTR set as shown in the figure below.
- Mount the conversion cable's SUB-D connector in one of the free slots, with the cable on the inside of the RTR set.
- plug the 10-pole connector inside the RTR set into your CPUV4, in the terminal labeled "PHW".
- plug the toolsetter's SUB-D connector into the SUB-D connector you just connected to the USBCNC CPU V4 PCB.

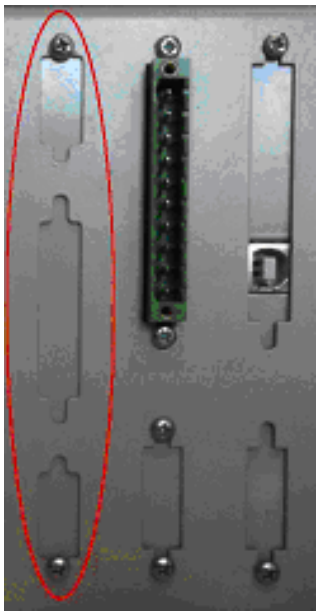


Figure 12.3: Plug the SUB-D connector into the connector you implemented

2. You have a USBCNC CPU V3, V4 or V5 without a DamenCNC RTR set

The installation of your toolsetter depends on the version of your USBCNC CPU. First we will discuss USBCNC CPU V3, then V4, and finally V5.

- **USBCNC CPU V3**
  In the unmodified configuration of the connector cable, the toolsetter will not work. When you press the button on the toolsetter, connector pin 1 and 6 of the SUB D connector will be connected. In the unmodified case, pin 1 and pin 6 of the SUB D connector match with pin 1 and 2 of the 10P connector. Follow the following steps in order to prepare your toolsetter with USBCNC CPU V3.

  - Cut the connector cable (the 200mm cable with a SUB D Connector and a 10P connector attached) in two equal pieces, orthogonal to the wire direction
  - From the SUB D connector piece, strip the cable which is attached to pin 1 and 2 of the ribbon cable (this is the RED cable and the cable adjacent to the RED cable; see Figure 11.4)
    Warning: Ribbon wire 1 and 2 of the ribbon cable are attached to pin 1 and 6 of the SUB-D connector. Ensure that you install the wires to ribbon wire 1 and 2 of the ribbon cable; **don't** refer to the pin numbering of the SUB-D connector!
  - Install heat shrinks
  - Solder this cable to the other halve (with the 10P connector) by connecting the stripped ribbon you made in the previous step to the other halve; connect them to the ribbon wires attached to pin 4 and 10. Bear in mind that the RED cable is numbered one. Apply heat to the heat shrinks.

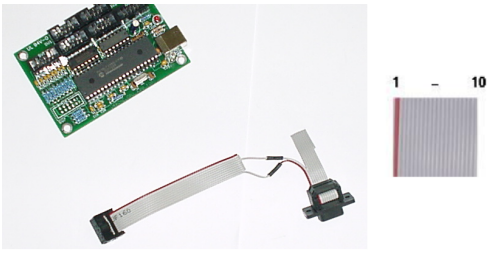


Figure 12.4: Attach ribbon wire 4 and 10 (ribbon connector side, left on this figure) to ribbon wire 1 and 2 (SUB-D side, right on this figure)

With your custom connector cable now ready, mount the 10-pole connector in the CPU3 slot which is labeled as "SV-5".

Now proceed with the Testing procedure.

- **USBCNC CPU V4**
  Connect the SUB-D connector from the 10 pole cable with the SUB-D connector from the toolsetter.
  10-pole connector from the 10-pole cable should be connected with the CPU4 slot labeled "PHW".

  Now proceed with the Testing procedure.

- **USBCNC CPU V5**
  In the unmodified configuration of the connector cable, the toolsetter will not work. When you press the button on the toolsetter, connector pin 1 and 6 of the SUB D connector will be connected. In the unmodified case, pin 1 and pin 6 of the SUB D connector match with pin 1 and 2 of the 10P connector. Follow the following steps in order to prepare your toolsetter with USBCNC CPU V5.

  - Cut the connector cable (the 200mm cable with a SUB D Connector and a 10P connector attached) in two equal pieces, orthogonal to the wire direction
  - From the SUB D connector piece, strip the cable which is attached to pin 1 and 2 of the ribbon cable (this is the RED cable and the cable adjacent to the RED cable; see Figure 11.5)
    Warning: Ribbon wire 1 and 2 of the ribbon cable are attached to pin 1 and 6 of the SUB-D connector. Ensure that you install the cables to ribbon wire 1 and 2 of the ribbon cable; **don't** refer to the pin numbering of the SUB-D connector!
  - Install heat shrinks
  - Solder this cable to the other halve (with the 10P connector) by connecting the stripped cables you made in the previous step to the other halve; connect them to the ribbon wires attached to pin 1 and 10. Bear in mind that the RED cable is numbered one. Apply heat to the heat shrinks.

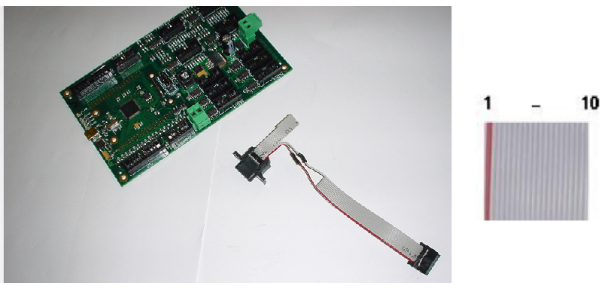


Figure 12.5: Attach ribbon wire 1 and 2 (SUB-D side, left on this figure) to ribbon wire 1 and 10 (ribbon connector side, right on this figure)

With your custom connector cable now ready, mount the 10-pole connector in the CPU V5 slot which is labeled as "IN-1".

Now proceed with the Testing procedure.

12.3 Testing procedure

To test your toolsetter with any RTR System, or USBCNC CPU PCB, follow the next steps.

- Connect your USBCNC CPU to your PC and make sure the toolsetter is connected.
- Start the USBCNC V3 Software
- Ensure that the USBCNC V3 Software is not in simulation mode!
- Press the button of the toolsetter, and verify that at the lower-left corner of the main menu, the red light next to "Probe" changes color to green when you press the button.



* As an alternative, the status of the probe input can also be monitored in the IO tab. (CPU / PROBE IN)

12.4 Using the toolsetter to zero the Z-axis

The toolsetter can be used in 2 different ways; Zero-ing the Z-coordinate of the raw material, and measuring the tool length. First we will explain how the operator can Zero the Z-coordinate of the raw material.

This section covers:

- Calibrating the height of the toolsetter
- Using the toolsetter to zero the Z-axis

Warning:
DamenCNC assumes that you are familiar with your CNC machine and with USBCNC software.
If you have just purchased your CNC machine or system, it is not recommend to start using the toolsetter directly. It is better to first get familiar with your machine before using this feature.
If the toolsetter is not operated correctly, damage to your machine and tools will occur!
We assume you have USBCNC software V3.49 or higher, which has the user defined cycles.

Calibrating the height of the toolsetter
The first time the toolsetter is used the height needs to be calibrated.
The switching point of the toolsetter is about 43mm above the zero plane. This can differ slightly, so be sure to calibrate this for the first time you use the toolsetter.

During a standard installation of USBCNC V3 the required files you need to use for the calibration procedure should be stored in the following directory:

C:/Program Files/USBCNCV3

(it can differ, if you have installed USBCNC in a different location)

- In this folder there is a file called macro.cnc
- Open the macro.cnc file using Wordpad or any other basic text editing program (**do NOT use Microsoft Word**)
- The lines that need to be edited are stored in Sub user1; at the top of the macro.cnc file

Zero tip example
This is a macro.cnc textfile. In this file we point out which lines you must modify.

Sub user1
msg "user1, Zero Z (G92) using toolsetter"
f30 (Start probe move, slow)
g38.2 z-100
g0 z#5063 (Move back to touch point)
G92 z43.0 (Set position, the measuring device is 43mm in height, adapt for your measuring device)
G91 (incremental distance mode)
g0 z5.0 (move 5 mm above measuring device)
g90 (absolute distance mode)
m30
Endsub

- Adjust the macro.cnc file as indicated above (from the line Set position,... up until ...distance mode) , save the macro.cnc file and restart the USBCNC software.
- After the compensation has been set to 43mm, you need to verify if this is indeed the correct value, often a small change in the order of 0.1mm needs to be made
- The procedure will leave the tool at 5mm above the toolsetter
- Using the MPG handwheel or the normal JOG keyboard keys, JOG the machine down to the top plane of the material to be milled.
  If all is set correctly, the DRO in the upper right corner of USBCNC should read zero for the Z-axis (in the work CS!). If the Z-axis value is not equal to zero, repeat the previous steps.(re adjust your macro.cnc file)
- **Note:**
  All changes made in the macro.cnc file take affect only after Saving your changes, and restarting the USBCNC V3 Software!

Using the toolsetter to Zero the Z-axis in Work CS (Sub User 1)
Now the operator is ready to use the toolsetter for the Zero-ing of the raw material. In order to do this, follow this procedure:

- Clamp the raw material which you wish to machine on the table of your CNC machine
- Place the toolsetter on the top surface of the material to be machined
- Using the MPG handwheel or just the manual JOG keys, move the machine to the approximately 10mm above the center of the toolsetter
- Go to the user defined cycles menu and choose the first toolsetting option (In the 'Operate' tab: press F12, F11, F1)
- As a result, the machine will now move down until the switch is activated. As soon as the tool has touched the toolsetter, the tool will move to 5mm above the toolsetter.

Your raw material is now correctly Zero-ed.

12.5 Using the toolsetter to measure tool lengths

On more advanced CNC machines (i.e. with toolchanger) the toolsetter can be used in order to measure the length of tools (Sub User 2). It is also possible to automatically store the tool lengths in the tool table of USBCNC.

When you are using the sub user 2 cycle in order to automatically measure a tool's length, you really need to be familiar with your machine and USBCNC. This is because this is an advanced feature which needs to be calibrated correctly before it can be used.

It is very convenient to have your tools in designated tool-holders such that when they are placed back into the machine, the total tool length has not changed. Sometimes spacing rings are also used for this purpose.

If you don't have tool-holders or spacing rings the length of the tool protruding from the toolholder/collet wil always be different. Be aware that in this case you need to re-measure the toollength every time you use it after a changing a tool! Using Sub User 2 in the software this is an easy task.

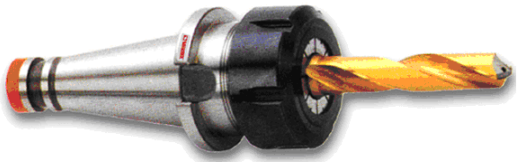


Figure 12.6: Tool-holders prevent changes in the height offset of your tool

Section outline

This section consists of the following 2 parts:

- Calibrating the XYZ position of the toolsetter
- Using the toolsetter to measure tool lengths

Calibrating the XYZ position of the toolsetter

In order to calibrate the toolsetter for tool measuring, you need to take the following steps.

- Determine a permanent place for the toolsetter within the working range of your machine; mark this location or mount the toolsetter in this position.
- Clamp a milling cutter in the milling motor
- Measure the distance between a reference point which is suited for your milling machine (e.g. the clamping nut) and the tip of the clamped tool; as can be seen in Figure 11.7.
  You are free to choose your own convenient reference point; as long as you are consequent in using the same convenient reference point after your choice!

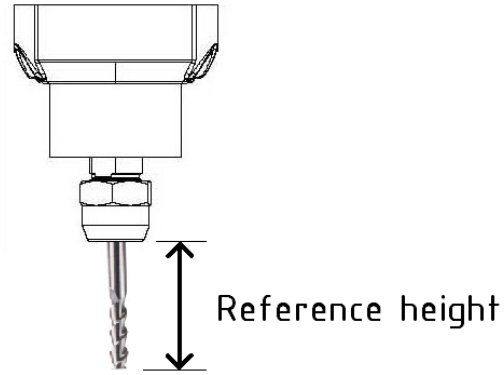


Figure 12.7: Measure the height of the tool with respect to a convenient reference point (e.g. the clamping nut)

- Go to USBCNC V3's Tool tab
- Enter the measured length in the tooltable (ZOffset of **Tool 16**) and click Save
- Open MDI (F12, F6)
- Type "gosub calibrate_tool_setter" and press Enter
- Close the MDI pop-up window
- Click Run (F1)
- In the Jog menu, Jog to the maximum (safe) Z-height.
  This is very important, as USBCNC V3 remembers this "Safe Z coordinate"!
  Be sure to choose this coordinate such that you can load all your tools from this position.
- Click Run (F1)
- Jog to just above the toolsetter. The tip of the mill has to be right above the **center** of the toolsetter (max 10mm higher).
- Click Run (F1) when done.

This concludes the calibration procedure.

If you wish to re-calibrate your system, you can always go to the MDI (F6) and type "gosub calibrate_tool_setter".

Note: Many clients have 2 toolsetters, one has a permanent position on the machine and is used for measuring tool lengths, the other can be moved freely and is used to Zero the Z axis. Using only one toolsetter does not have to be a problem. If for some reason you would forget to put the toolsetter in the correct position, the software stops the measurement procedure if it does not find a tool after 20mm of movement.

Using the toolsetter to measure tool lengths

The calibration procedure has been done. The USBCNC V3 software knows the position where the toolsetter is located, and will move there automaticly when a tool length measurement is done. So you are ready to start measuring tools. The procedure to measure a tool length:

- Go to USBCNC V3's Tool table in the USBCNC v3 software navigate to the user defined cycles menu and choose the second option (In 'Operate' tab: F12, F11, F2).
- A dialog will pop up, in this dialog enter the tool number of the tool to be measured
- Enter the approximate tool length, Simply measure the distance between the tip of your tool and your (unchanged) reference point. Your measurement does not need to be very precise; aim to measure your tool with an accuracy of around ± 2mm. The toolsetter will do the precise measurement for you.
- Enter the tool diameter (only used when G41/G42 is active)
- Press OK (Lower right corner of the Pop Up dialog)
- The machine will move to the XY position where the toolsetter is located and start its probing move
- The length of the tool is automatically stored in the tool table.

In order to use the length offset, it needs to be activated via the **g43** command. Deactivation happens through the **g49** command.

There are three configuration options, and they will be discussed below:

1. **Procedure with Toolchanger and Toolrack**
   If the operator has a machine with an automatic tool changer, it is required to initially measure the tool length of each tool. Each measurement will be stored in the tool table. The operator does not need to measure the tool lengths again, as long as the tools remain in their holders. If the operator installs a new tool in a holder, he needs to measure the new tool length.

2. **Procedure with manual tool changer**
   Many machines have manual toolchangers. These can be pneumatic or mechanical. Each millingtool has its own toolholder which can be numbered, that is the important issue.This means that each individual tool holder has its own height offset.
   It is important that the operator organizes the tools such that the tool numbering does not get mixed up. If this is the case the tool table can be used in order to save the tool lengths. Since when a tool holder is placed back into the machine, the height offset has not changed. All you need to do is set the correct toolheight offset corresponding the the tool number. For example using g43h1, height offset voor toolholder 1 set activated.

3. **Procedure for tool changing by hand**
   In this configuration you exchange tools without a tool holder. This is quite common in the hobby CNC world, using mainly KRESS motors, that allow the tool to be directly mounted in the collet. After each tool change, the operator must measure the toollength.

Note: When using height offsets (g43h command), the order in which offsets are set is quite important. There are many different procedures. But we would recommend always first measuring the tool lengths, activating the length via the g43h command. And only after the tool length is active zeroing the Z axis using the toolsetter. Using this procedure no errors can be made with incorrect height offsets.
Note: When using the g49 to cancel the height offset, be careful with G0 or G1 codes in the same line. Our practice is to cancel tool offset g49, and in the next line move in machine coordinates g53 g1 z0 In our machines the Z0 position is always the safe height, this can differ per machine.

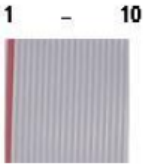
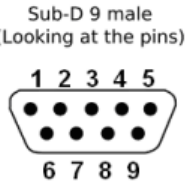
On the following page the operator can find an example G-code concerning the toolsetter.

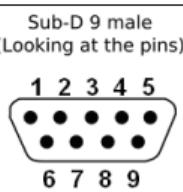
Example G-code Program

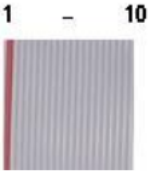
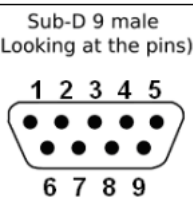
```
m6 t1 (change tool to Tool 1)
g43 h1 (activate height offset Tool number 1)
g0 z50
g0 x0 y0
g0 z5
g1 z-2
g1 x100 y100
g0 z50
g49 (deactivate height offset)
g53 z0 f1000 (move to safe z height)
```

12.6 Detailed information of the toolsetter's wiring scheme

For most clients this information will not be relevant, however the technical data is still attached and can be seen in the figure below:

CPU V3.0 (SV-5 input)		
Function in USBCNC	Pin nr 10 pole flatcable	Pin nr SUB D
		
EOS-X	1 Red	**1**
E-stop	2	**6**
EOS-Y	3	**2**
Probe	**4**	**7**
EOS-Z	5	**3**
Pause	6	8
EOS-A	7	**4**
Run	8	9
+5 USB	9	**5**
GND	10	Not Connected

CPU V4.0 (PHW input)		
Function in USBCNC	Pin nr 10 pole flatcable	Pin nr SUB D
		
Probe Input	**1 Red**	**1**
GND	**2**	**6**
Handwheel A	3	2
GND	4	7
Handwheel B	5	3
GND	6	8
+V ext	7	4
GND	8	9
+5 USB	9	5
GND	10	Not Connected

CPU V5.0 (**IN-1** input)		
Function in USBCNC	Pin nr 10 pole flatcable	Pin nr SUB D
		
Probe	1 Red	1
Spindle Pulse	2	6
E-Stop	3	2
Extern Error	4	7
AUX-IN-6	5	3
AUX-IN-5	6	8
+5V	7	4
GND	8	9
+5V	9	5
GND	10	Not Connected

Chapter 13

Appendix H: Connecting a Probe

13.1 Introduction

Thank you for purchasing the DamenCNC EMP-03-TouchProbe. In this Appendix we explain how to connect and use the EMP-03-TouchProbe in the following order:

- Hardware Connection procedure
- Testing the Toolsetter
- Using the EMP-03-TouchProbe
- Detailed info on hardware connection

As the name of the Probe already implies, this device is used to use your CNC machine as a measuring device. The 3D probe allows to probe in the X, Y and Z direction. It can also be used in combination with a 4th Axis (A,B or C axis)

Scope of delivery

- The EMP-03-TouchProbe with 3 meter cable and connector
- A 10-pole ribbon cable with two connectors
- This user manual

13.2 Hardware Connection procedure

In order to connect and use your EMP-03-TouchProbe, there are two options for your system.

1. You have a DamenCNC RTR Classic or Performance, with USBCNC CPU V4 or V5
2. You have an USBCNC CPU V3, V4 or V5 (No RTR set)

First the DamenCNC RTR set with USBCNC CPU V4 or V5 is explained. After this section we discuss the installation of USBCNC CPU's without a RTR set.

1. You have a DamenCNC RTR Classic or Performance, with USBCNC CPU V4 or V5

If you have a DamenCNC RTR Classic or Performance, select if you purchased this set before January 2010, or after January 2010:

You purchased your RTR system after January 2010

In this configuration, you do not need the delivered 10 pole cable. This is because this component is already integrated in the RTR housing. Simply plug the toolsetter's SUB-D connector into the designated slot (with the notification "PROBE") at the back of your RTR housing, as shown below.

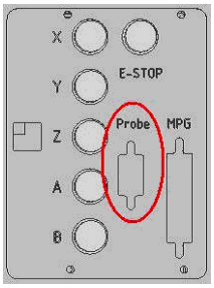


Figure 13.1: Plug the SUB-D connector into the Probe slot

You purchased your RTR system before January 2010
In this configuration, you do need the 10 pole conversion cable, since the required component is not yet integrated in the RTR housing.

- Open the RTR set by unscrewing the 8 screws, and lift the top cover.
- Remove one of the white cover plates at the backside of the RTR set as shown in the figure below.
- Mount the conversion cable's SUB-D connector in one of the free slots, with the cable on the inside of the RTR set.
- plug the 10-pole connector inside the RTR set into your CPUV4, in the terminal labeled "PHW".
- plug the toolsetter's SUB-D connector into the SUB-D connector you just connected to the USBCNC CPU V4 PCB.

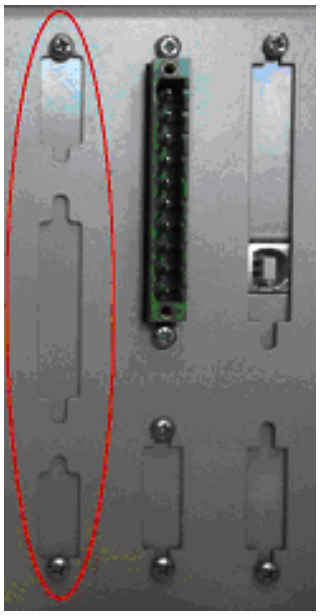


Figure 13.2: Plug the SUB-D connector into the connector you implemented

2. You have an USBCNC CPU V3, V4 or V5 without a DamenCNC RTR set

The installation of your toolsetter depends on the version of your USBCNC CPU. First we will discuss USBCNC CPU V3, then V4, and finally V5.

- **USBCNC CPU V3**
  In the unmodified configuration of the connector cable, the toolsetter will not work. When you press the button on the toolsetter, connector pin 1 and 6 of the SUB D connector will be connected. In the unmodified case, pin 1 and pin 6 of the SUB D connector match with pin 1 and 2 of the 10P connector. Follow the following steps in order to prepare your toolsetter with USBCNC CPU V3.

  - Cut the connector cable (the 200mm cable with a SUB D Connector and a 10P connector attached) in two equal pieces, orthogonal to the wire direction
  - From the SUB D connector piece, strip the cable which is attached to pin 1 and 2 of the ribbon cable (this is the RED cable and the cable adjacent to the RED cable; see Figure 12.3)
    Warning: Ribbon wire 1 and 2 of the ribbon cable are attached to pin 1 and 6 of the SUB-D connector. Ensure that you install the wires to ribbon wire 1 and 2 of the ribbon cable; **don't** refer to the pin numbering of the SUB-D connector!
  - Install heat shrinks
  - Solder this cable to the other halve (with the 10P connector) by connecting the stripped ribbon you made in the previous step to the other halve; connect them to the ribbon wires attached to pin 4 and 10. Bear in mind that the RED cable is numbered one. Apply heat to the heat shrinks.

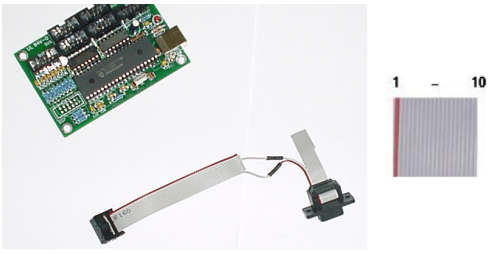


Figure 13.3: Attach ribbon wire 4 and 10 (ribbon connector side, left on this figure) to ribbon wire 1 and 2 (SUB-D side, right on this figure)

With your custom connector cable now ready, mount the 10-pole connector in the CPU3 slot which is labeled as "SV-5".

Now proceed with the Testing procedure.

- **USBCNC CPU V4**
  Connect the SUB-D connector from the 10 pole cable with the SUB-D connector from the toolsetter.
  10-pole connector from the 10-pole cable should be connected with the CPU4 slot labeled "PHW".

  Now proceed with the Testing procedure.

- **USBCNC CPU V5**
  In the unmodified configuration of the connector cable, the toolsetter will not work. When you press the button on the toolsetter, connector pin 1 and 6 of the SUB D connector will be connected. In the unmodified case, pin 1 and pin 6 of the SUB D connector match with pin 1 and 2 of the 10P connector. Follow the following steps in order to prepare your toolsetter with USBCNC CPU V5.

  - Cut the connector cable (the 200mm cable with a SUB D Connector and a 10P connector attached) in two equal pieces, orthogonal to the wire direction
  - From the SUB D connector piece, strip the cable which is attached to pin 1 and 2 of the ribbon cable (this is the RED cable and the cable adjacent to the RED cable; see Figure 12.4)
    Warning: Ribbon wire 1 and 2 of the ribbon cable are attached to pin 1 and 6 of the SUB-D connector. Ensure that you install the cables to ribbon wire 1 and 2 of the ribbon cable; **don't** refer to the pin numbering of the SUB-D connector!
  - Install heat shrinks
  - Solder this cable to the other halve (with the 10P connector) by connecting the stripped cables you made in the previous step to the other halve; connect them to the ribbon wires attached to pin 1 and 10. Bear in mind that the RED cable is numbered one. Apply heat to the heat shrinks.

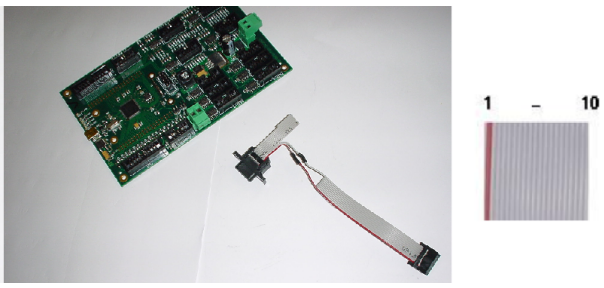


Figure 13.4: Attach ribbon wire 1 and 2 (SUB-D side, left on this figure) to ribbon wire 1 and 10 (ribbon connector side, right on this figure)

With your custom connector cable now ready, mount the 10-pole connector in the CPU V5 slot which is labeled as "IN-1".

Now proceed with the Testing procedure.

13.3 Testing procedure

To test your toolsetter with any RTR System, or USBCNC CPU PCB, follow the next steps.

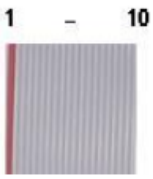
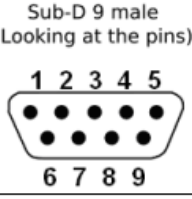
- Connect your USBCNC CPU PCB to your PC and make sure the toolsetter is connected.
- Start the USBCNC V3 Software
- Ensure that the USBCNC V3 Software is not in simulation mode!
- Press the button of the toolsetter, and verify that at the lower-left corner of the main menu, the red light next to "Probe" changes color to green when you press the button.

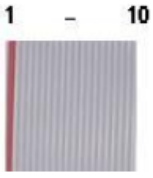
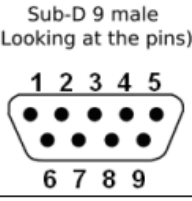


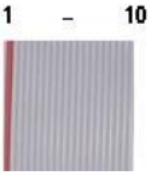
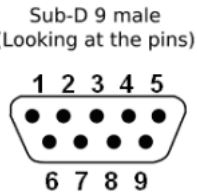
13.4 Using the Probe

This section still needs to be made.

Detailed information or the wiring scheme of the Tool setter
For most clients this information will not be relevant, however the technical data is still attached and can be seen in the figure below:

CPU V3.0 (SV-5 input)		
Function in USBCNC	Pin nr 10 pole flatcable	Pin nr SUB D
		
EOS-X	1 Red	**1**
E-stop	2	**6**
EOS-Y	3	**2**
Probe	**4**	**7**
EOS-Z	5	**3**
Pause	6	8
EOS-A	7	**4**
Run	8	9
+5 USB	9	**5**
GND	10	Not Connected

CPU V4.0 (PHW input)		
Function in USBCNC	Pin nr 10 pole flatcable	Pin nr SUB D
		
Probe Input	**1 Red**	**1**
GND	**2**	**6**
Handwheel A	3	2
GND	4	7
Handwheel B	5	3
GND	6	8
+V ext	7	4
GND	8	9
+5 USB	9	5
GND	10	Not Connected

CPU V5.0 (**IN-1** input)		
Function in USBCNC	Pin nr 10 pole flatcable	Pin nr SUB D
		
Probe	1 Red	1
Spindle Pulse	2	6
E-Stop	3	2
Extern Error	4	7
AUX-IN-6	5	3
AUX-IN-5	6	8
+5V	7	4
GND	8	9
+5V	9	5
GND	10	Not Connected

Chapter 14

Appendix I: Connecting a Brake (Relay) module

14.1 Introduction

In this Appendix we explain how you can connect a Brake (Relay) module. The Brake module is used to prevent (heavy) Z-axis components from falling downward after e.g. a power failure. Next to braking, the relay module can be used for a variety of purposes.

14.2 Hardware Connection procedure

If you are using this module for a Brake, please continue reading. When you intend to use the module for an other purpose than for braking, please continue reading in the Relay section.

This Brake section is set up as follows:

Brake

- Hardware Setup

    1. You have a DamenCNC RTR Performance (standard with a brake(relay) module) set and you would like to connect a brake
    2. You have a DamenCNC RTR set without a brake(relay) module inside
    3. You have an external USBCNC CPU V4.0 and a brake(relay) module

- Software Setup

Relay

- Hardware Setup
- Software Setup

Connecting procedure

1. You have a DamenCNC RTR Performance (standard with a brake module) set and you would like to connect a brake

If you have a DamenCNC RTR Performance and you wish to connect your brake, verify that you have the Brake component which is shown on the left and ensure that it is connected as can be seen on the right. The other end of the cables should be attached to the **NO** terminal from the brake PCB.

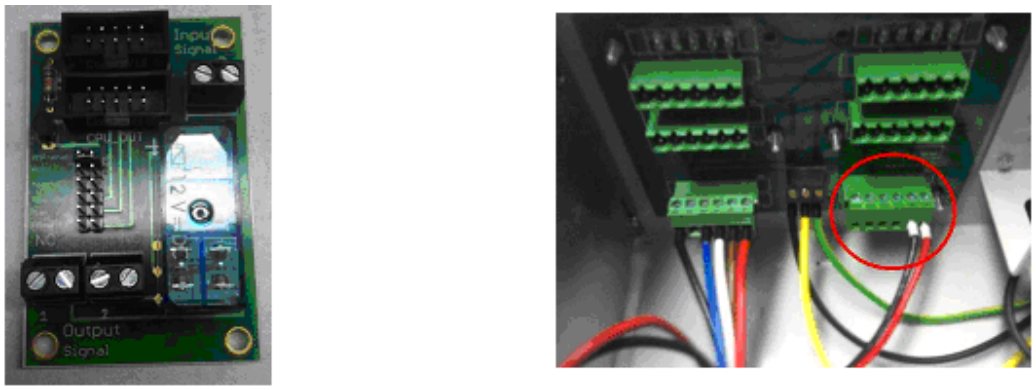


Figure 14.1: The components and wiring for the brake module. The brake PCB may look slightly different. Verify that on the right figure, +Vcc is on the right of the connector (as seen from the figure's perspective), and GND is left.

The Brake (relay) module is engaged by the AMP-enable signal from USBCNC. This standard setting can be altered by switching the jumper at the Brake (relay) Module. For normal brake operations, ensure that it is set to the AMP-enable setting.

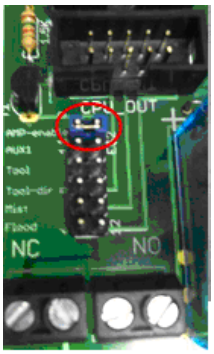
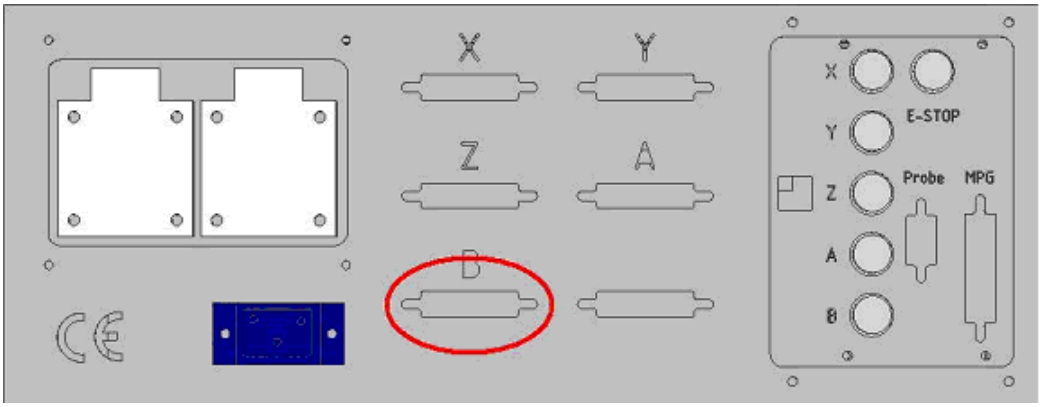


Figure 14.2: Verify that the jumper is set to AMP-enable

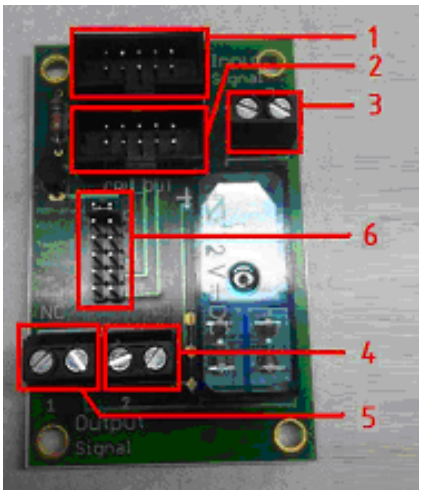
When all the wiring is set up correctly you can insert the connector of the brake into the hub on your RTR set:



Continue with the Software Setup Relay to test the brake.

2. You have a DamenCNC RTR set without a brake (relay) module inside

In this section the installation of the Brake inside your RTR set is discussed. In order to do so, the following information about the Brake module is essential:

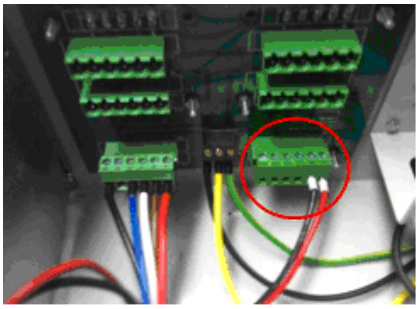


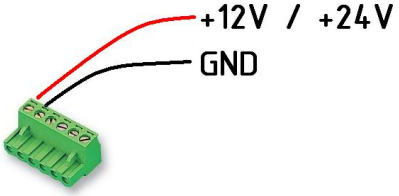
The wiring scheme is as follows:

- 1. USBCNCAXIS
- 2. USBCNCOUT
- 3. Signal Input Terminal
- 4. Signal Output Terminal NO
- 5. Signal Output Terminal NC
- 6. Jumpers

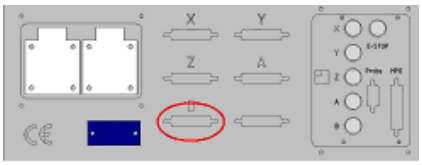
The Brake can be installed by performing the following procedure:

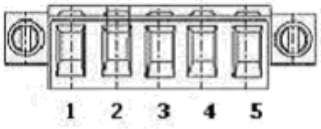
- 1. Make sure the jumper is correctly set to the AMP enable position
- 2. Make a connection between the Signal Input from the Brake (relay) Module and the 12V power supply.
  Signal 1 should be connected with +12V and Signal 2 with the GND.
  The 12V power can be recognized because it also supplies power to the cooling fan and USBCNC CPU.
  In case of doubt, use a Multimeter to verify, if you are indeed connecting the wiring to a 12V powersource.
- 3. Establish a connection between USBCNC and Brake (relay) Module.
  The Brake (relay) Module has 2 terminals for ribbon cable connectors. One is labeled CPUOUT and the other is labeled CPUAXIS.
  CPUOUT should be connected with the OUT terminal from USBCNC CPU.
  CPUAXIS should be connected with any axis terminal from the USBCNC CPU. Most common is to use axis 5 or 6 since most customers have a 3 or 4 axis system, which leaves axis 5 and 6 open for use.
- 4. Establish a connection between the Brake (relay) Module and the interface card inside the RTR that provides the output.
  Connect the output terminals NO to the interface card as shown in the following figure:





The connection on the outside of the RTR controller should be as follows:





The wiring scheme is as follows:

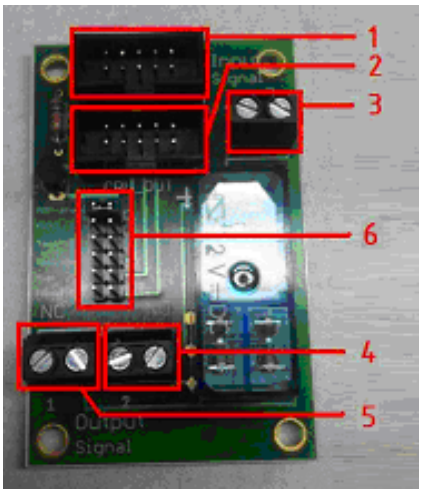
- 1. +12V (+24V)
- 2. GND
- 3. NC
- 4. NC
- 5. NC

The brake (relay) module is now ready to be attached nside the RTR. Sticky feet prove to be a good solution to fix the brake PCB inside a RTR system. Please bear in mind that the brake (relay) module is sensitive for electronic noise originating from the USBCNC CPU. For this reason it must be placed as far as possible from the USBCNC CPU.
DamenCNC RTR systems are factory assembled with the Brake PCB mounted on the backside of the CPU holder, which proves to be a good solution.

When you have performed all these steps successfully, you can proceed to test your Brake module in the Test section.

3. You have an external USBCNC CPU V4.0 and a Brake (relay) module

For this operation it is necessary to have the following information about the Brake PCB



The wiring scheme is as follows:

1. USBCNCAXIS
2. USBCNCOUT
3. Signal Input Terminal
4. Signal Output Terminal NO
5. Signal Output Terminal NC
6. Jumpers

- **1. Input signals**

  The Brake (relay) Module contains a relay that connects two inputs signals (signal 1 and signal 2) with four output signals (Signal 1 NC, Signal 1 NO, Signal 2 NC and Signal 2 NO).
  In order to use a brake you need to connect 12V or 24V as input signal. The brake itself should be connected with signal 1 and 2 from the **NO** output.

- **2. Connection from Brake (relay) Module and USBCNC CPU**
  In order to control the relay from USBCNC the following connections have to be made:
  The Brake (relay) Module has 2 terminals for ribbon cable connectors. One is labelled CPUOUT and the other is labelled CPUAXIS.
  CPUOUT should be connected with the OUT terminal from USBCNC CPU.
  CPUAXIS should be connected with any axis terminal from the USBCNC CPU.

Warning:
The relay uses the external power input of the USBCNC CPU that has to be 12V.
Without this input or an input with different voltage the Brake (relay) Module will not function and will be damaged.

Testing the Brake and Software setup
Now the brake is connected, we can test the brake functionality.
We assume that the system is correctly configured and that the machine is working properly.
This implicates that the only change performed to your system is adding this brake module. If you don't have the XYZ(AB) axis of your machine working properly, first address this issue. This is because the signal that enables the brake is the same signal that enables the stepper drives. In the case that you invert this signal, both the brake module and the stepper drives will get affected.
For that reason it is important the steppermotors are correctly functioning before attaching the brake module.

1. Start your PC and start the USBCNC Software

2. Ensure that USBCNC is not running in Simulation mode!

3. When you are in the USBCNC operating environment, look at the top left where the buttons can be found as shown in the figure below:



   Click on the drivers such that the LED turns red and the drivers are thus engaged. With the drives engaged, the brake should be disabled. If the drivers are disabled, the brake should be enabled.

Hardware Setup

The Brake relay module can also be used to control other functionalities using USBCNC Software. If you want more information about how to connect your Brake relay module and which signals can be used, please continue reading.

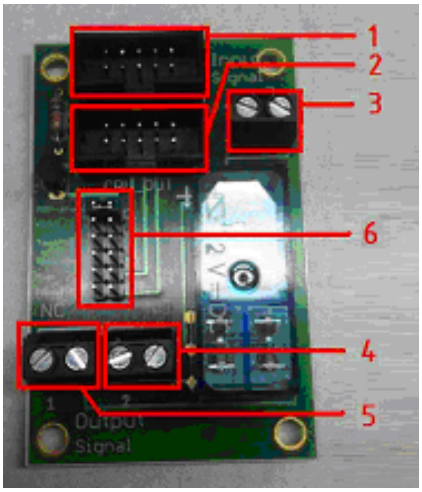


Figure 14.3: Your Brake module may look slightly different

On the following page you can find the wiring scheme corresponding with Figure 13.3

The wiring scheme is as follows:

- 1. USBCNCAXIS
  This connector is for the AXIS signal of your USBCNC CPU. Connect this slot with an AXIS signal from the USBCNC CPU if you wish to trigger the relay using the AMP-Enable signal. (**Note:** the OUT signal also needs to be connected in order to use the AMP-Enable signal)

- 2. USBCNCOUT
  This connector needs to be connected with the OUT signal of your USBCNC CPU. Without this connection the brake relay module does not function.

- 3. Signal Input Terminal
  There are 2 Signal inputs. The input signal should be within the range of (AC/DC) 50V and approximately 500mA. These values are not equal to the specifications from the relay, but these are limited because of the PCB specifications.

- 4 and 5. Signal Output Terminal NC
  There are two different output terminals; **NO** (4) (Normally open) and **NC** (5) (Normally Closed). The output signals are the same as the used input signals.

- 6. Jumpers
  In this section you can chose which signal must control the relay. You can choose from the following options:

  1. AMP-Enable
  2. AUX 1
  3. Tool
  4. Tool - dir
  5. Mist
  6. Flood

  Please bear in mind that it is not possible to have more than one signal controlling the relay.

Warning:

- When you are using one of the signals (except signal 1), make sure there are no other devices such as relays connected with the same signal you want to use.
  For example; when the OUT from the CPU is also connected with one of our Tool/Flood modules (and the jumpers are set on Tool and Flood of the PCB from the Tool/Flood module), you can't use signal 3 and 6.

- When you also use a **0-10V Spindle Speed Control**, you can **NOT** use signal 3 and 4.

Software Setup Relay
The Relay can be controlled from USBCNC with the following M-codes.
The Relay will only switch when the jumper is attached to the signal with the corresponding M-codes.

Nr.	**Signal**	**M-Code**	
		ON	OFF
1	AMP-enable	M80	M81
2	AUX1	M82	M83
3	Tool	M3/M4	M5
4	Tool-dir	M4	M5
5	Mist	M7	M9
6	Flood	M8	M9

14.3 Brake Module FAQ

- **I have connected the brake as specified but it doesn't seem to work properly.**

  Answer: Did you remove the springs from the brake?

- **I have connected the brake as specified but it works exactly the wrong way around. So it is already disabled when the drives are still disabled too.**

  Answer: On the The Brake (relay) Module there are two output terminals: **NO** and **NC**. Our standard connection is made with **NO** so when the brake works inverted at your machine the connection should be changed to **NC**.

  Note: Do **NOT** invert the AMP enable input in the software.
  The steppermotors use the AMP enable, so when the signal is inverted the steppermotors not to work anymore with your current machine configuration.

Chapter 15

Appendix J: Using a Relay module

15.1 Introduction

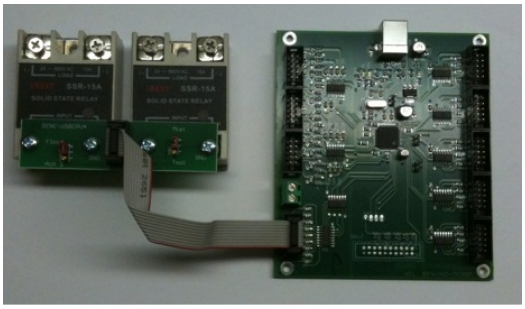
Thank you for purchasing the DamenCNC RTR set or relay module. This chapter explains how to connect and use external tools in combination with your DamenCNC RTR set or relay module. Topics are covered in the following order:

- Hardware setup
- Software setup
- Using external tools

15.2 Hardware setup

There are two power sockets positioned at the back of the module. These can be used to power external tools. The power sockets are switched on and off using solid state relays. These solid state relays are positioned behind the power sockets. To prepare your RTR set/relay module, follow the next steps:

1. Shut off powersupply
2. If necessary, open the RTR set by unscrewing the 8 screws, and lift the top cover.
3. Make sure the 10 pole connector on the relay module is connected to the "OUT" port of CPU 4 or CPU 5. (see image below)
4. Position the jumpers on the relay PCB correctly.



2 x SolidState relais connector +relais

Unit is plug&play with USBCNC CPU v4.0. Jumpers allow Flood/Aux1 Mist/Tool to be selected as the source, its possible to couple two units to a single CPU V4.0 allowing control of four 220V devices.

How to position the jumpers:
The default settings of the relay module are "flood" and "tool". This means that the "flood" command in USBCNC would enable the left solid state relay. If you want the solid state relays to react to other commands, place the jumpers accordingly. For example: if the jumper would be placed on the bottom two pins of the left relay, the relay would be enabled by the "AUX1" signal.

15.3 Software setup

To enable your tools automatically during production, it is necessary to implement the right M-codes. USBCNCs Help menu shows a list of G- and M-codes. Those that apply to this are shown in the table below.

Signal	M-Code	
	ON	OFF
Tool	M3/M4	M5
Mist	M7	M9
Flood	M8	M9
AUX1	M82	M83

This is an example of a CNC program:

---

m3 (turn on the tool)
g0 x0 y0
g1 z-5 f400
g1 z100 f200
g0 z5
m5 (turn off the tool)

How to test the module:
Before you begin production with your CNC program, first test if the software is configured correctly. Do this by connecting a device (for example a lamp) to the power socket. Then activate the Tool/Mist/Flood/AUX1 in the "Operate" menu. If the relay module works inverted, change the settings in USBCNC (Settings - Invert IO).



15.4 Using external tools

This section still needs to be made.