Table of contents

Introduction .................................................................................................................. 3
Installation ................................................................................................................... 4
EmpirBus studio ........................................................................................................ 5
  Overview .................................................................................................................. 5
  Features ................................................................................................................. 6
  Adding units .......................................................................................................... 6
  Adding components ............................................................................................... 6
  Adding input or output channels ......................................................................... 7
  Properties ............................................................................................................. 7
  Create connections between components ...................................................... 8
  Set connection line color ................................................................................. 9
  Rectangles ............................................................................................................ 10
  Textboxes ............................................................................................................. 10
  Tabs ....................................................................................................................... 11
  Portals .................................................................................................................. 12
  Snippets ................................................................................................................. 13
  Print ...................................................................................................................... 13
  Short commands ................................................................................................. 14
  WDU Finder ......................................................................................................... 14
  Log viewer ........................................................................................................... 14
Uploading process .................................................................................................... 15
  Connect to EmpirBus system ........................................................................... 16
  Set unit ID’s ........................................................................................................ 16
  Upgrade firmware ............................................................................................... 16
  Load configuration to EmpirBus system .......................................................... 17
Hardware ................................................................................................................ 18
  Terms of use and best practice ....................................................................... 18
Support .................................................................................................................. 18
Introduction

EmpirBus studio is a software developed to be used to build logical functions for EmpirBus systems. The logical functions will execute commands that control the EmpirBus modules input and output channels as well as integrating it to third party systems. All channels can be configurated in this software. It can be used to build advanced logical functions, set fuse size for outputs, build virtual buttons, convert and present analog sensor data, set alarm conditions and much more. This is a simple, yet powerful software where imagination and creativity probably will be the biggest limitation.
**Installation**

To Install the software, use the following link:


When the installation has been completed, download the latest firmware. The latest firmware can be found on the following webpage:

https://www.EmpirBus.com/#firmware

When the software has been installed and the firmware is downloaded. The next step will be to import the firmware.

1. Open EmpirBus studio.
2. In the EmpirBus studio, click on “File” in the top menu and select “Close Project”.
3. Click on “Settings” in the top menu and select “Import Firmware”.
5. Click import.
6. A message will inform that the firmware has been successfully imported.

**Note:** Whenever a new firmware is available, a popup will inform the user when EmpirBus studio is opened. There is no need to manually look for new updates.

The EmpirBus studio software is now ready for use.
EmpirBus studio

Overview

Starting from the left side there is the components menu. All the different components are listed in this menu and grouped together to make it easier to find each component. Components could be described as the building blocks that are used to build the logical functions.

Below the components, a window is displaying the available description for each component.

Note that there is also a statistics menu (below the component description) which can be activated by pressing “View” in the top menu and then select “Statistics”. The statistics window can be helpful when creating a large project where there is a need to keep track of how many components that have been used and how much memory there is left. Note also that several windows are available through the “View” menu in case any of the windows should close by mistake.

The logic schema can be found on the center of the screen. This is the area where the components and channels will be placed and linked together.

The log viewer is located below the logic schema. The log viewer is a tool that can be used to log channel statuses. See the log viewer section for more info.

On the top right side, the project explorer can be found. This is where the project units are added and listed (e.g. displays, DCM or connect 50 units, panels etc.). When expanding a unit, all channels related to that unit will be visible and can then be added to the logic schema and modified via the properties menu below the project explorer. The properties window is used to adjust the settings of a selected channel or component. Unnecessary
Features

Adding units
To add a unit (e.g. DCM), click the “Add unit” button in the project explorer. Select “DC I/O modules” > “DCM-CCM” and then select the model that match the physical unit. The model number can be used as reference. When the unit have been added, it can be renamed. This can also be done later by selecting the unit and change the name in the properties window.

The same procedure is used to add SP12 / SP8 panels, multifunctional displays (MFD), programming interface or MCU. In order to add channels that are used as buttons or indicators on the MFD, an MFD unit will need to be added.

In order to load the configuration, the system will need to have a master unit. An MCU, DCM or a Connect 50 can be used. In the project explorer, select the unit that should be used as a master unit. Drag and drop the whole unit out into the logic schema. Select the unit in the logic schema and look at the properties window. It should be set as master and not slave. When the unit is set as master, it will say “(Master)” right after the unit name in the project explorer. Keep in mind that system limitations changes depending on which unit that has been selected to be master.

Adding components
The components are located on the left-hand side. They are all grouped together in different categories to make it easier to find them. For new users the most common components used to build simple logic can be found under “Control Modules”, “General Components” and “Logic Gates”. Drag and drop the component into the logic schema. The properties window will display the available settings for each component whenever the component is selected. The pin(s) on the left side of the component are the signal(s) that goes into the component. The pin(s) on the right side of the component is the signal that have been processed by the component.
Adding input or output channels

Input and output channels are referred to as “I/O”. These are the actual channels that are physically available on the unit. E.g. if the project includes a DCM, it would have 16 channels total. Some of these channels can be used only as input or output while some can be used as both depending on how they are configurated in the properties window and which unit type that are used. In order to add a channel, a unit must first be added to the project.

Go to the chapter “Adding units” and add a unit before proceeding to add channels.

Once a unit have been added, select the unit in the project explorer and expand it. Also expand the “I/O Channels”. The green and blue arrows indicate if they are inputs or outputs where green are inputs and blue are outputs. To add an output (for example to drive a light), simply drag and drop it into the logic schema. As soon as the channel is released into the logic schema, a menu will appear where different channel settings can be set. By selecting “Plus (+)” the channel will be programmed to be an output that will drive the external load (e.g. lights).

To learn more about different channel types and settings, watch this video.

The above procedure is used for all units, even the SP8/SP12 panels or multifunctional display (MFD).

Properties

The properties window is used to set the properties for each component or channel. Different settings will be available depending on the currently selected object. The properties window is frequently used while programming. It is a good idea to keep an eye on this window whenever a new component or channel is added to the logic schema.
Create connections between components

**Figure 1: Circuit with 3 buttons controlling one output (Light).**

In the figure 1 above, three different buttons are used (an MFD unit is needed to access channels used for virtual switches) to control one output which in this example are connected to a light. The PBC in the middle are used to keep the signal fixed at ON or OFF as the buttons only gives a pulse for each press. Note that the PBC in the example above have been expanded through the properties window with a total of 3 pulse buttons. To connect the button to the PBC, simply press on the right-side pin of the button called “Button pressed”. A connection line between the pin and the cursor will appear. The connection line can now be either drawn straight to the left top pin of the PBC named “Pulse button 1” or through a specific path by clicking on several different points in the logic schema until finally connecting it to the PBC (“Pulse button 1”). Drawing a specific path is just to visually make it easier to follow the connection line. If a connection line needs to cross another (already existing) connecting line, then there is no limitation to do so. The connection lines won’t connect to each other but will only connect between the two points (pins) that have been selected as start and stop point.

**To delete a line (while drawing), press ESC.**

**To delete a line (that is already drawn), select it and press DEL.**
Set connection line color

It can be helpful to label connections or grouping them together. Sometimes, it can be easier to follow each connection if they have specific colors. For example, red color can be used for OFF signals and green color for ON signals. Line colors are only used to give a better overview visually and does not affect the logic.

Figure 2: Using colors on connections.

To set color, select a connection line and right click on it. Select “Set color”. Choose a color and press OK.

If the connection line needs to be changed back to its original color, then choose the black color.

It is also possible to set a different color than black as default. Select a color and press “Set as default”. Every new connection will be drawn with the selected color.
Rectangles
Rectangles can be used to highlight a specific area or function. It can be a helpful tool in a larger configuration when there are a lot of things to keep track of.

Figure 3: Right click and select create rectangles. Left click and drag the corners to resize the rectangle.

Textboxes
Textboxes can be very helpful when overviewing the whole project. Textboxes can be used to mark specific functions or even specific connections.

Figure 4: Creating a textbox
Right click on an empty area on the logic scheme and select “Create textbox”. A new window will open where the text can be edited. Enter the text and press OK. The textbox will appear in the logic schema. Textboxes can be moved around and placed wherever in the logic schema.

Tabs

Tabs can be added to split up the logic schema. Right click on it and choose properties. It is now possible to change name and order in the properties window. To create a new tab, click on “Add”. The benefit with several tabs is that different functions can be separated. Note that no matter how many tabs there is in the project, they will still be within the same config file and project. In other words, the project can be drawn on several different tabs, or only on one tab. It will function the same. The tabs are there only to make it easier for the programmer.
Portals
Portals are used to make a connection between two points if the components are far away from each other or if the components are drawn on different tabs. Rather than drawing a very long line a portal can be used instead. In the example below, a button with ON/OFF pulse has been added to a portal. As soon as a portal and signal (connected to the specific portal) have been created, an instance of a portal can be created to output the signal. An instance of a portal can be created unlimited times and be used on several different locations. Even on other tabs.

Figure 5: Right click on an empty area and select “Create new portal”. Give the portal a name and press OK.

Figure 6: Draw a new connection, in the above case from the button ON/OFF pulse pin and connect it to the portal. Give the signal a name within the portal. Right click on an empty area in the logic schema and select “Create instance of portal”. Select the portal that was created. The portal will now appear in the logic schema.

Figure 7: Click on the right-side pin of the portal. A list of all the available signals will open. Select signal and click OK. A connection line will appear and can now be connected to a component pin. In this case to the PBC.
If a connection is drawn between two pins or if it is drawn between two pins with the help of a portal will result in the exact same function. The benefit with the portal is that it reduces the lengths of connection lines and makes the logic easier to follow. It can be wise to use the portal to collect several different signals if they all should be outputted at the same place. Useful to collect alarm signals from multiple alarm components which might be outputted to one single output (e.g. a buzzer).

It is recommended to name the signals with a word that describe the type of signal e.g. fixed or pulse. This will help a lot when using the signals in the other end.

The use of portals can sometimes lead to confusion if there are many portals and signals to keep track of. Sometimes it can be necessary to verify which component a specific signal origin from. To get this kind of information there is an option called “Find origin”. To find the origin of a signal, select and right click on the signal. Select “Find origin”. This will mark the component of which the signal origins from. If all references are needed, right click on a specific signal and select “Find all references”. This will open a window which will show every place in the config where the specific signal is being used.

Snippets
A snippet is used to export a selection of components. It can be a complete function or a part of a function. To create a snippet, select the components and I/O channels that should be included in the snippet (also works for text and rectangles). Hold down CTRL while selecting components to select them one by one. When the components have been selected, click “Edit” > “Save snippet”. It is also possible to right click on one of the selected components to access the “Save snippet” option. Give the snippet a name and click on save. Whenever this specific set of components needs to be accessed, click on “Insert snippet” which can be found either under the “Edit” menu or by right clicking on an empty area in the logic schema and select “Insert snippet”. Logic blocks can be saved for later use. Snippets can also be downloaded from a snippet library which can be helpful. Download the library here.

Print
It is possible to print a part of the Logic schema or the channel name together with the channel settings. The size of the logic schema is unlimited, meaning that logical blocks can be built vertically or horizontally at an unlimited area. To avoid that a logic schema that contains too many components are printed, there is a limit of how large area that can be printed. Activate the printing area by clicking on “View” and then on “Print area”. A rectangle will appear in the top left corner of the logic schema. The rectangle symbolizes the paper. Fit the components inside the rectangle and click on “File” > “Print” in order to print. Note that the print size can be changed to A3 if needed.
### Short commands

<table>
<thead>
<tr>
<th>Command</th>
<th>Action</th>
</tr>
</thead>
<tbody>
<tr>
<td>Delete already drawn connection</td>
<td>DEL</td>
</tr>
<tr>
<td>Delete connection, not yet connected</td>
<td>ESC</td>
</tr>
<tr>
<td>Add anchor point on connection line</td>
<td>ALT+Left click (On connection line)</td>
</tr>
<tr>
<td>Move</td>
<td>Hold SPACE+Left click and drag</td>
</tr>
<tr>
<td>Zoom</td>
<td>Hold CTRL+Scroll</td>
</tr>
<tr>
<td>Zoom area</td>
<td>Hold CTRL+select area (hold left click and drag)</td>
</tr>
<tr>
<td>Copy</td>
<td>CTRL+C</td>
</tr>
<tr>
<td>Paste</td>
<td>CTRL+V</td>
</tr>
<tr>
<td>While drawing connection, hold to create 45-degree angles</td>
<td>SHIFT</td>
</tr>
<tr>
<td>Select multiple components</td>
<td>Hold CTRL+Left click</td>
</tr>
</tbody>
</table>

### WDU Finder

The WDU finder is a tool included in the EmpirBus studio which is located under “Tools” in the top menu bar. It is used to find the WDU’s IP address. Once the PC is connected to the WDU with the ethernet cable or wifi, it will be listed here for easy access.

### Log viewer

The logviewer can be found on the bottom of the EmpirBus studio software (see overview on page 6). The log viewer is used to see I/O status and sensor values in real time. Different massages can also be filtered.
Uploading process

In order to connect a PC to the NMEA2000 network, which contain all units, one of the following interfaces must be used:

- MCU
- Programming interface
- Kvaser Leaf Light v2
Connect to EmpirBus system
Once the interface is connected to the PC it will show up in the top toolbar. If not, check connection and make sure the drivers have been installed properly (Kvaser only).

Set unit ID’s
In the top menu, click “Communication” > “Set system unit IDs”. The software will start to scan for units. If two units have the same ID, there will be a conflict within the system. All units must have a unique ID. The easiest way to do this is to physically look on the units and write down the serial numbers and ID on a paper and then set ID’s here with the serial numbers as reference.

Upgrade firmware
In the top menu, click “Communication” > “Upgrade firmware”. The software will start to scan for units and tell which units that need to be updated. Update all units that need to be updated. Note that the master unit must be updated individually and it is recommended to start with the master unit. If the update fails, try to update one unit at a time. Note that this step will only be needed the first time when uploading a config to the system.
Load configuration to EmpirBus system

In the top menu, click “Communication” > “Send configuration”. The software will start to search for the available units and list the units on the left side labeled “System units”. On the right side, labeled “System units” there is a list of all the units that have been added to the project (the units added in the project explorer). Normally the software will match each unit automatically if the unit IDs and serial numbers match. If not, they need to be matched manually. To match the units manually, drag and drop the units from the left side list to the right-side list. If the match were successful, then the unit will appear in the middle as “Matched units”. This step will only have to be done the first time. Next time this will happen automatically.

Sometimes the programmer has a reason to upload the configuration to the system without one or two units while testing the system. That will result in a match warning that will be displayed on the left bottom side of the “Send configuration” window. To upload the configuration despite the match warning, check the “Ignore match warnings” checkbox which will allow the user to press “Send”.

When the matching has been done, click “Send”. Once the config has been successfully loaded, the master will reboot.
Hardware
To get more information about our system, please visit our website where datasheets, manuals and much more can be found.
https://www.EmpirBus.com/

Terms of use and best practice

Support
A lot of useful information can be found on our website. In case answers can’t be found there, feel free to contact us anytime by phone or email.

Website
https://www.EmpirBus.com/#faq
https://www.EmpirBus.com/#support

Email:
Support@EmpirBus.com

Phone:
+46 522 44 22 22
8 am to 5 pm (CET/CEST), Monday to Friday