AEQ NETBOX RTC

REAL-TIME CONTROL SOFTWARE FOR
AEQ NETBOX 32 AD MX / 8 AD MX / DSP
AUDIO OVER IP MULTICHANNEL INTERFACES AND ROUTERS

USER'S MANUAL
ED. 01/24

V. 1.0 - 03/01/2024
Software version 2.0.0.20, December 2023

CONTENTS

1. INTRODUCTION. 4

2. SOFTWARE PACKAGES INSTALLATION. 4

2.1. "NetBox Tool". 4
2.2. Database engine. 5
2.3. "NetboxRTC". 6
2.4. "Netbox Server". 10
2.5. Creating a list of NETBOX devices. 13

3. APPLICATION START-UP SCREEN. 15

4. DROP-DOWN MENU: File. 16

4.1. New Logical XY View. 16
4.1.1. Creating customized views. Customize. 17
4.1.1.1. Selecting lines. 19
4.1.1.1.1. Test Tone. 20
4.1.1.2. New XY View. 20
4.1.2. Rename XY View. 23
4.1.3. Save XY View. 23
4.1.4. Save Status XY. 24
4.1.5. View Salvos. 24
4.1.6. View Macros. 25
4.1.7. AGC. 27
4.1.8. Processes. 27
4.2. New Vumeters View. 28
4.3. Add Talkback. 31
4.4. Status XY. 35
4.5. All Connections View. 36
4.6. Salvo to View. 36
4.7. Close and Close All. 37
4.8. Workspace. 37
4.8.1. Open Workspace. 37
4.8.2. Edit Workspace. 37
4.8.3. Save Workspace As. 38
4.9. Log - Macro & Salvo. 39
4.10. Backup Database. 39
4.11. Restore Database. 39
4.12. Backup Matrix. 39
4.13. Exit. 41

5. WORKING WITH CROSSPOINTS. 42

5.1. Operating the XY views in the workspace. 42
5.2. Actions on crosspoints. 42
5.3. Actions on lines or crosspoints. Context menus. 44
5.3.1. Precision vumeters. 46
5.3.2. Gain control. 47
5.3.3. Processes. 48

6. DROP-DOWN MENU: Config. 50

6.1. Netbox List. 51
6.2. Users. 53
6.3. Salvos. 55
6.4. Macros. 58
6.4.1. Programming a crosspoint in a macro. 60
6.4.2. Programming a salvo in a macro. 60
6.4.3. Programming a GPO in a macro. 60
6.5. Alarms. 61
6.6. Processes. 64

6.7. Talkback. 70
6.8. Logical Lines. 74
6.9. Virtual GPO. 80
6.10. Virtual GPI. 81
6.11. GPI. 84
6.12. Clear, Save & Reset. 86
6.13. Dante. 87
6.14. HW Switch. 91
6.15. Remote IP. 95
6.16. Vumeters. 97
6.17. Netbox - IP Server RTC. 97

7. DROP-DOWN MENU: Schedule 98

8. DROP-DOWN MENU: Window 104

1. INTRODUCTION.

The "**NetboxRTC**" is an application that allows to control the routing and mixing of inputs to outputs of the system on NETBOX 8 AD MX Audio Router 16x16, NETBOX 32 AD MX Audio Router 64x64 and NETBOX DSP and the processing of inputs and outputs on NETBOX DSP. As any mission-critical equipment this application has been designed to safeguard operation against user errors and equipment failures.

NOTE: A standard NETBOX 32/8 AD can be converted into a high-performance summing and distributing matrix for Dante™/AES67 AoIP, digital and analog audio switching by mixing specific firmware versions and a user license (for more information, please consult the user's manual of the unit).

The "NetboxRTC" application allows setup, modifying and controlling in real time the audio matrix. Any event or action such as salvos, macros, individual crosspoints, physical and virtual GPOs can be triggered as follows:

- Manually.
- Programmed to be executed automatically in a schedule.
- Manually executing a schedule.
- Through physical GPIs.
- Through virtual GPIs.

More effective learning is achieved through training given by specialized technicians with broad knowledge of the system. This document is for reference purposes, and it's organised according to the drop-down menus on the menus bar.

2. SOFTWARE PACKAGES INSTALLATION.

The following steps needs to be observed when installing the "NetboxRTC" Client and Server applications. Any installation of these software packages on a computer should be accomplished using Administrator privileges for Windows.

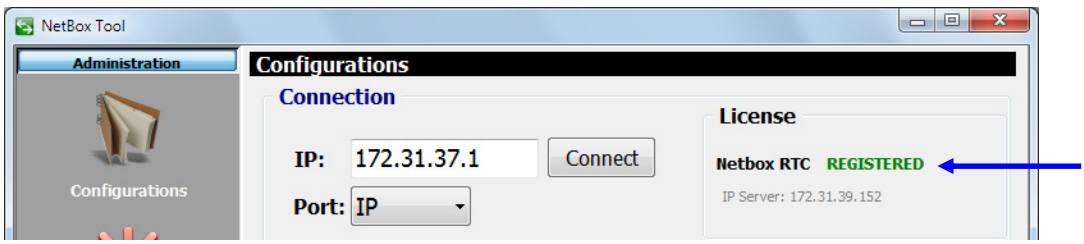
2.1. "NetBox Tool".

1. Install the "NetBox Tool" application (version 2.0.0.20 or higher). The auto run USB key furnished with the unit contains the executable file that installs this application (see section 3.2.1 of the user's manual of the unit).
2. In the case of the NETBOX 32/8 AD MX Audio Interfaces, these should have the 2.xx or newer firmware version for CPU and FPGA.

   If you need to upgrade the unit from a version 1.xx (older than the 2.xx) or setting up a new NETBOX 32/8 AD MX, it's necessary to execute a restart to the initial parameters by means of the "**Factory**" button (see section 3.2.2.1 of the user's manual of the unit).
3. In order to be able to use the "NetboxRTC" Client and Server application, it's necessary to obtain a "NetboxRTC" **license** for the unit.

   You can check if the unit has a license ("REGISTERED") or not ("LOCKED") in the "Configurations" submenu. If the unit is "LOCKED" you can activate the license in this submenu: see **section 3.2.2.1** of **user's manual** of the unit.

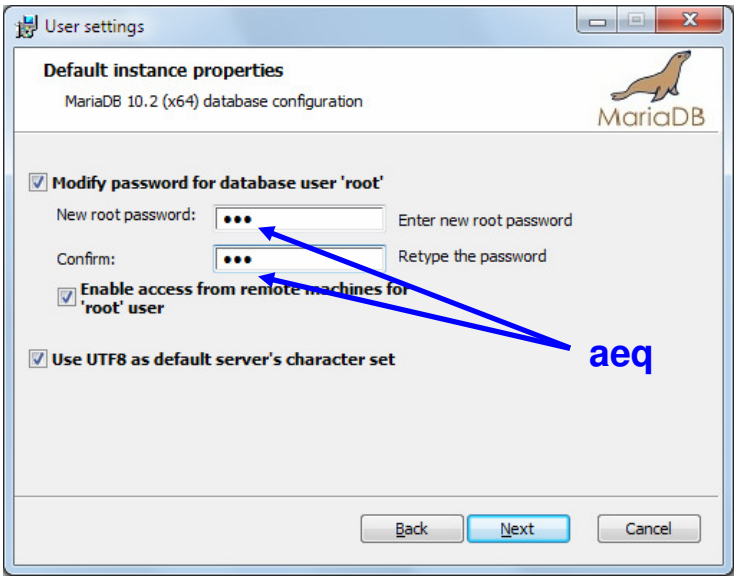
   It's recommended to turn off and on the unit after activating the license.



2.2. Database engine.

If there is no MySQL or MariaDB on the PC where the database is to be installed, install the corresponding version (32 or 64 bits) that is located in the auto run USB key furnished with the unit. You can find it going through the following screens:

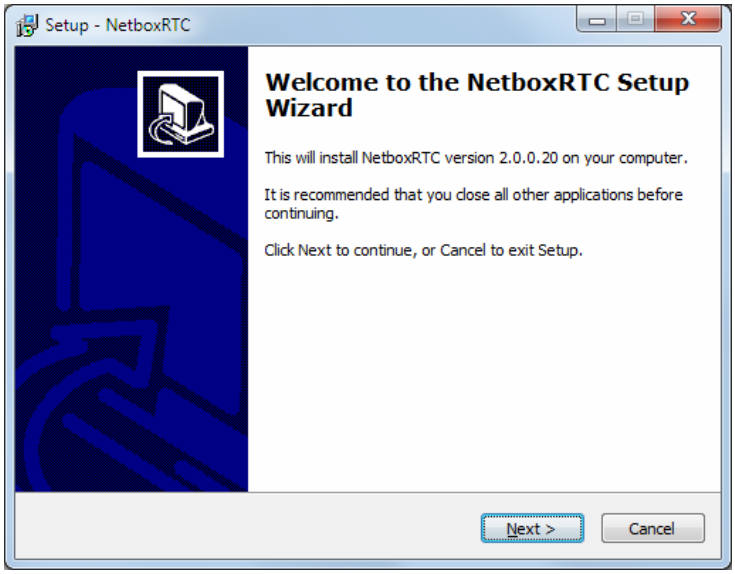
If you install **MariaDB**, when you reach the step "User settings" mark all the options and set as the password **aeq.** Continue by clicking "Next" for the remainder of dialogues popping up.



2.3. "NetboxRTC".

1. Install the "NetboxRTC" application (version 2.0.0.20 or higher). The auto run USB key furnished with the unit contains the executable file that installs this application. You can find it going through the following screens:

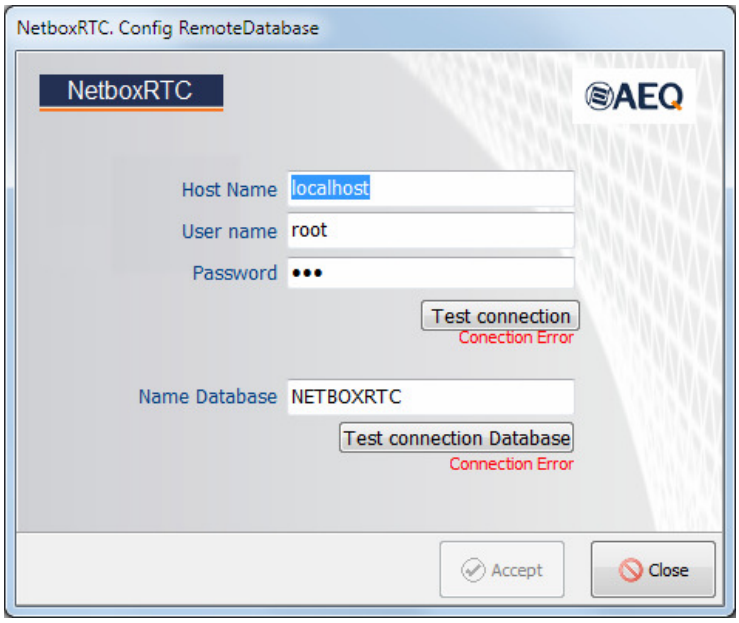
Installing the application is simply a matter of following the on-screen instructions as they come up:



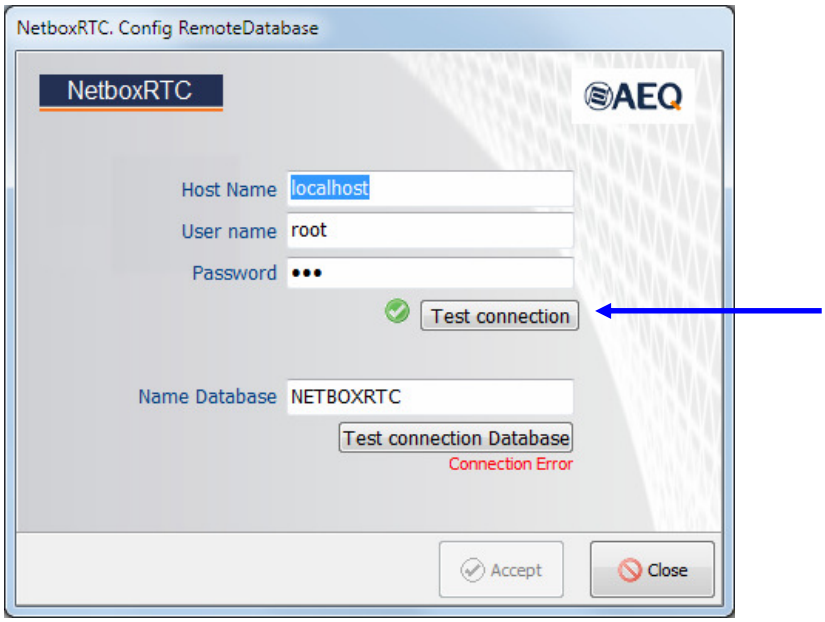
2. Once the "**NetboxRTC**" application is installed (by default, in C:\AEQ\NetboxRTC), start it up by double-clicking the icon displayed on the desktop:

3. The first step will be to create and connect to the new **NETBOXRTC** database.

4. When starting the application for the first time, the following screen will appear and in which you can check and configure the connection with the database that the "NetboxRTC" manages.



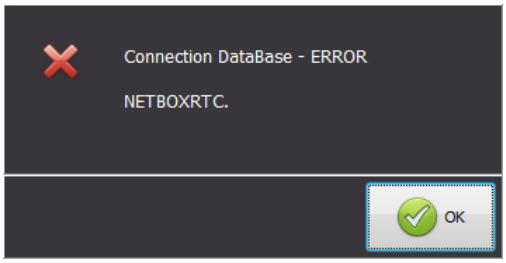
5. The "**Test connection**" button checks if there is a connection to the SQL server (by default, "Host Name" is **localhost**, "User name" is **root** and "Password" is **aeq**).



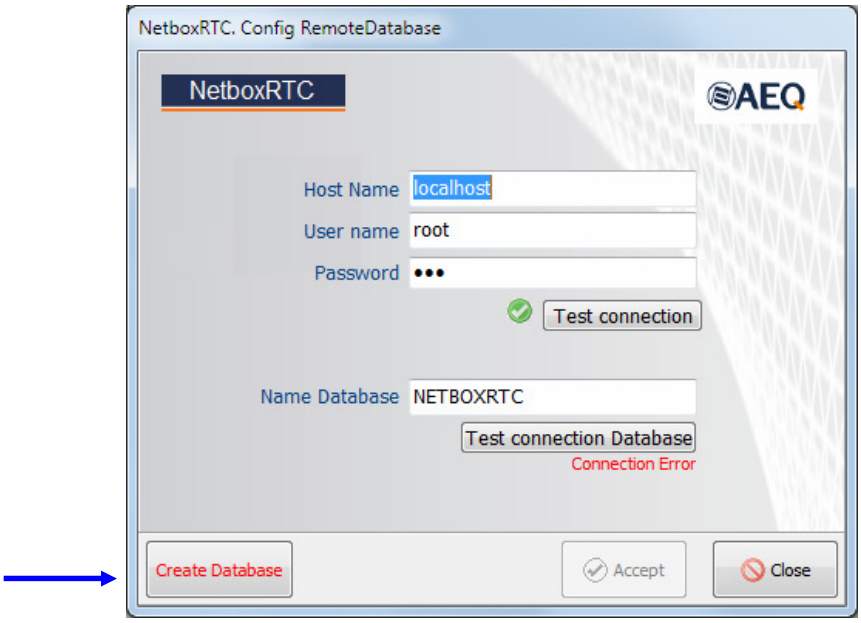
In case that you are installing "NetboxRTC" also in a different PC that the one where database is installed, the **netboxrtcuser** user with **aeq** password must be used and "Host Name" will be the IP address where the database is stored.

6. The "**Test connection Database**" button checks the connection to the database.

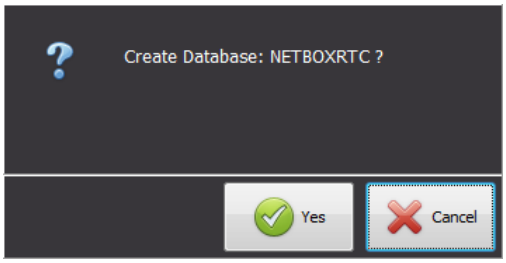
   If it's not possible to connect to the database but with the server, an error message will appear:

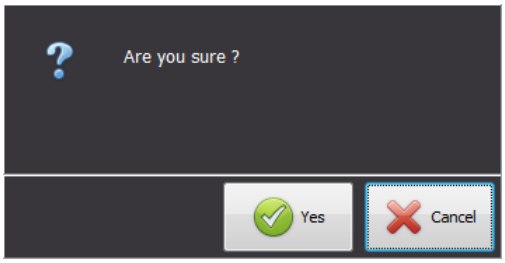


   This can be considered as normal as since it may be that the database has not been created yet. In this case a new button will appear at the lower left, "**Create Database**", which will allow you to create the database at that time.

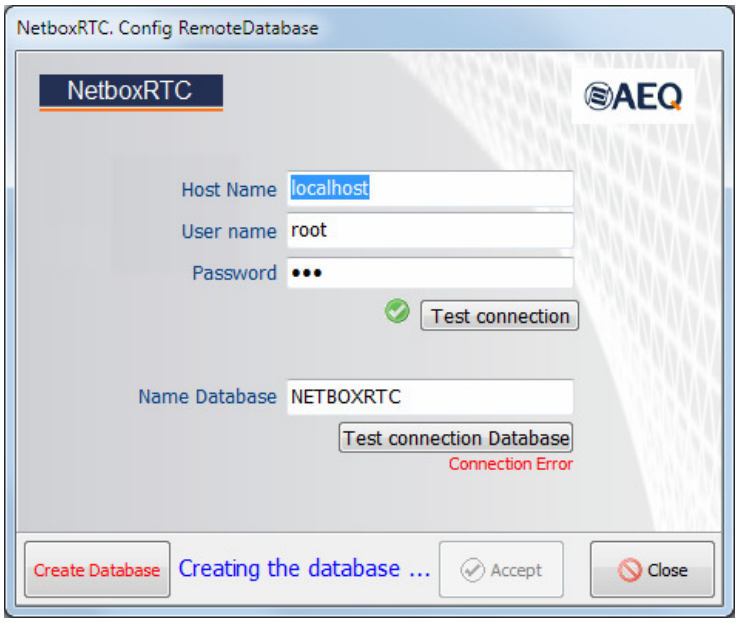


   Then two confirmation windows will appear.





   After confirmation, the database will be created.



NOTE: In the case of an already existing MySQL or MariaDB database engine, "UserName" and "Password" fields may be different from the ones that you have indicated above.

If you don't remember or are unaware of the "User name" and "Password", other possible combinations are root and aeqaeq, or root and aeq, or netboxrtcuser and aeqaeq, or systeluser and aeqaeq.

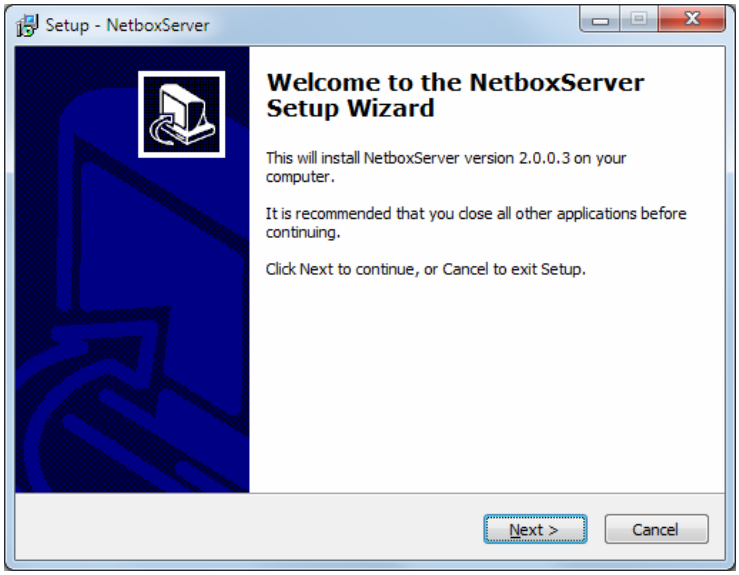
7. Once the database has been created, the validation window will appear for the application (this window will appear any time you open "NetboxRTC"). This is where our user and password must be validate: the default user is "ADMIN" and the password is "AEQ" (in this case there is no difference between capital and lower case letters).

If there are connection problems with the NETBOX, it may be necessary to assign administrator permissions to the "NetboxRTC.exe" executable.

2.4. "Netbox Server".

1. Install the "Netbox Server" application (version 2.0.0.3 or higher). The auto run USB key furnished with the unit contains the executable file that installs this application. You can find it going through the following screens:

Installing the application is simply a matter of following the on-screen instructions as they come up:



2. Once the application is installed (by default, in C:\AEQ\NetboxServer), start it up by double-clicking the icon displayed on the desktop:

Then the initial screen will appear:

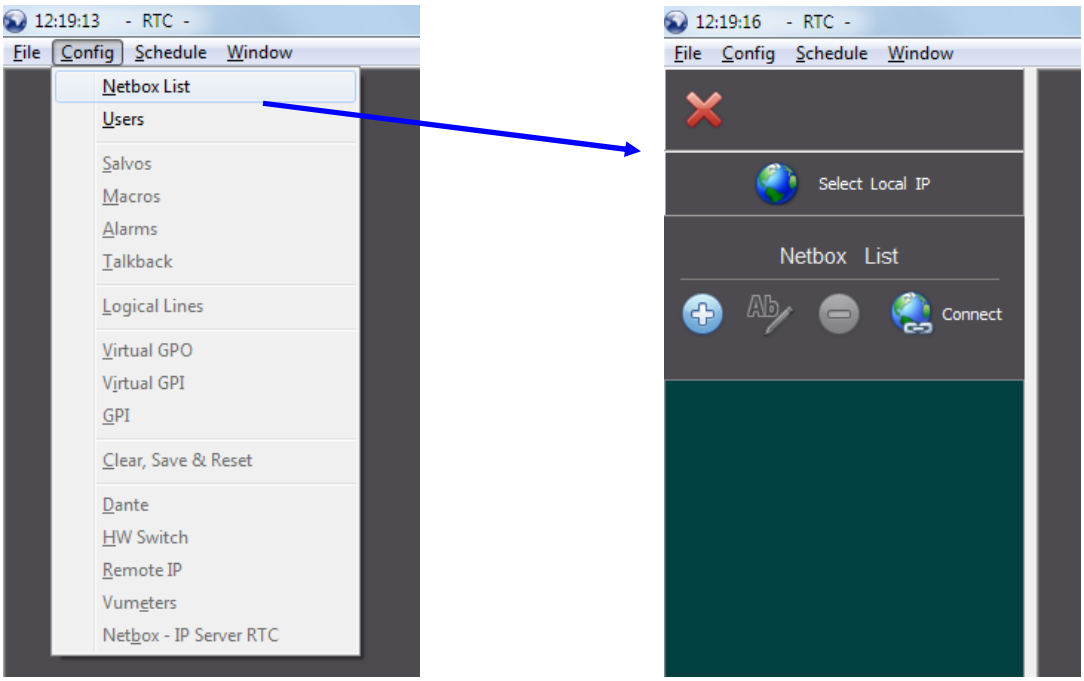
In the lower part of this window the IP address that will be used for the connection to NETBOX unit is shown (that IP address must be of the same range than the one for the unit). The date and time the application was started is also shown and, in the lower line, the database you're connected to and the application version (besides date and time of compilation) are shown.

If the Computer NIC has more than one IP address, it's necessary to select the one that should be used for the connection to NETBOX unit (of the same range that the one for the unit). In order to do that, you must press the "**Select Local IP**" button at the lower left corner (in that case it will appear available) and then select the IP address in the window that appears.

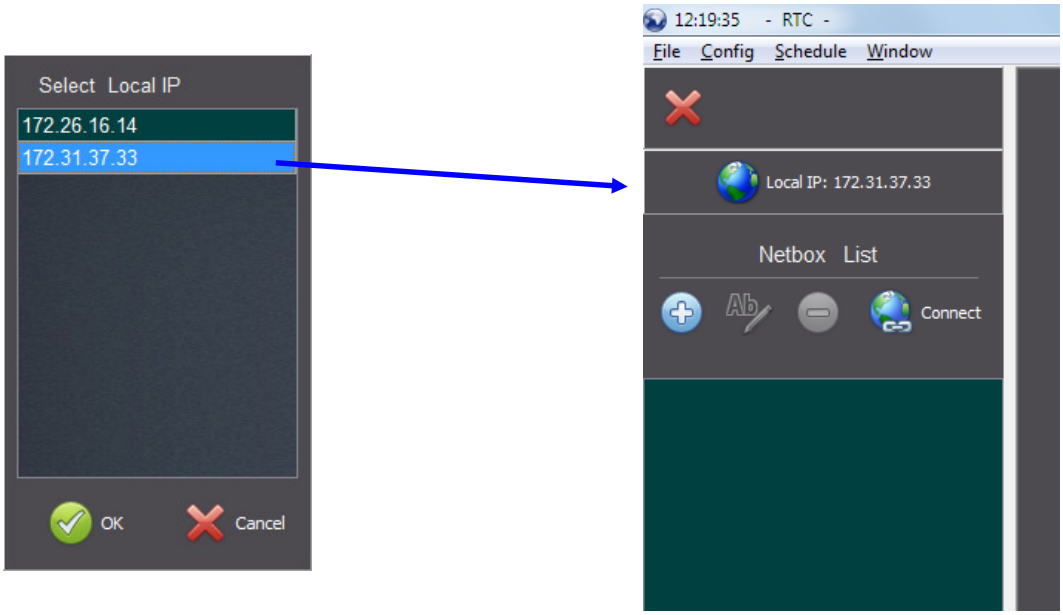
Observation: It's recommended that the database and "NetboxServer" are located on the same Computer. The .ini files of RTC and Server allows you to change some information as host, user, password or database.

2.5. Creating a list of NETBOX devices.

Once all the applications are installed, first thing to do is open "NetboxRTC" application in order to connect to the NETBOX unit you want to control and operate. In order to do that, you must go to the "Netbox List" option on the "Config" menu. Then the following tab will appear:



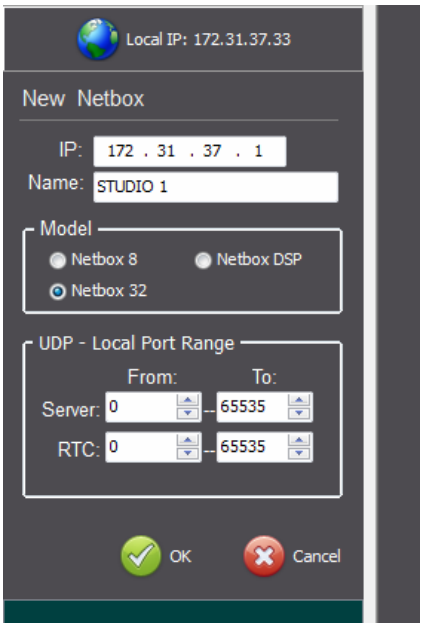
Firstly you must press the button with the Earth icon ("**Select Local IP**") in order to select the IP address corresponding to the PC where the application is installed. When you press this button a window will open that reflects several IP addresses, if you have configured your Network Interface or NIC with several IP addresses in different ranges. When you select the IP address that would be appropriate and in the same range as the device that you want to control with the application and confirm the selection by clicking on the "OK" button, that IP address will be shown in the previous button.



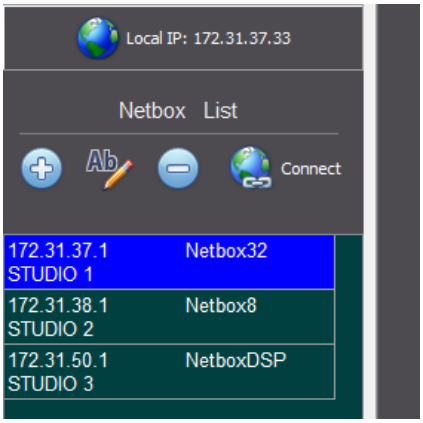
Below the "Local IP" button you will find the title "Netbox List" and immediately below this four buttons.

Allows to add a new NETBOX device to the "Netbox LIst". You must select the model (NETBOX 8, NETBOX 32 or NETBOX DSP), type in the IP Address of the device and give a name to it.

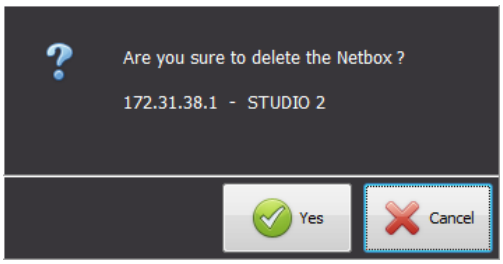
You can also select the UDP local ports range that will be used by the "Netbox Server" and "NetboxRTC" applications in order to communicate with that device; in case you configure or modify the range associated to "Netbox Server" it's necessary to restart this application/service. Its functionality is focused on network security, since it offers the option of being able to filter in the routers (that support it) the UDP ports that should be left open for communication between the NETBOX device and those applications. In case that this option is not configured, all UDP ports must be open since the port to be used by NETBOX applications is random.



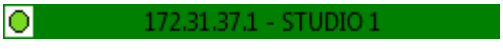
Allows to edit the selected device.

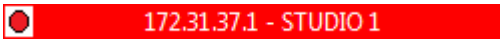


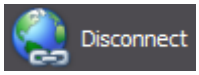
Allows to delete the selected device from the "Netbox List". The user will be prompted to confirm the deletion. All the device data will be lost (including salvos, macros, schedules...).



Allows to connect to the selected device. When the control computer NIC and the device are in the same range of IP addresses, the unit will connect to the control software, server application and database and signal to the software that the connection is established. At the footer of the "NetboxRTC" application the NETBOX device that is connected is indicated. When there is no connection to the selected device it appears in red.





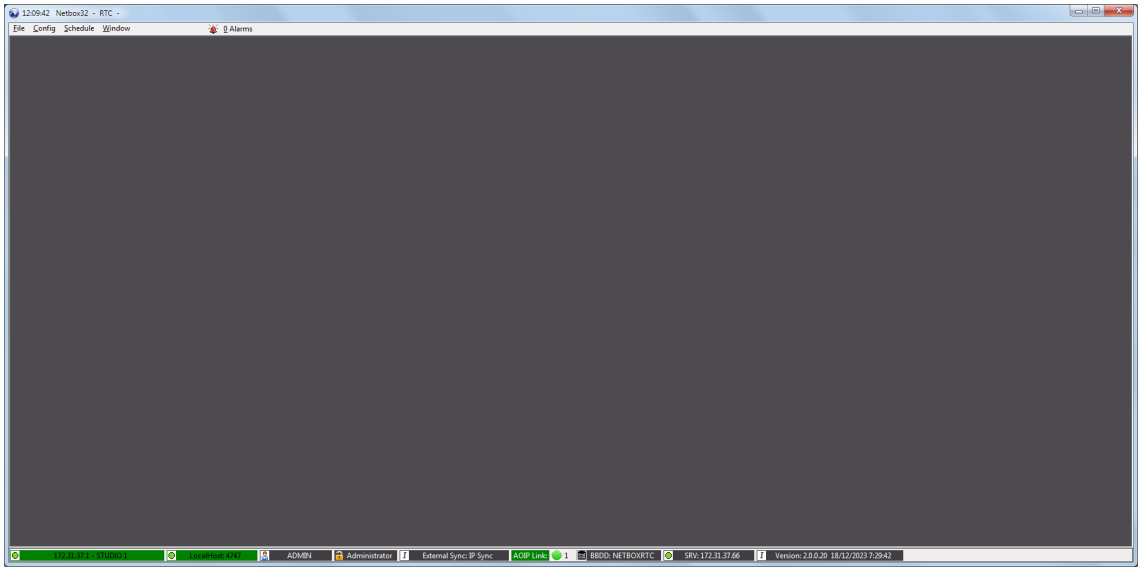


Once the selected device is connected, the previous button changes to "Disconnect" and allows to disconnect it in order to select another one and connect to it.

IMPORTANT NOTE: You must also configure a Multicast IP address for **vumeters** and the IP address where the **server** is hosted (see sections **6.15** and **6.16** of this manual).

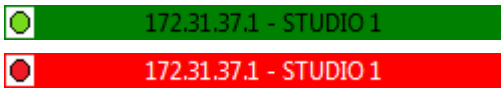
3. APPLICATION START-UP SCREEN.

When you access the "**NETBOX RTC**" application, the following window appears (if there is no default view selected). From this window you have access to all the drop-down menus that are explained in the subsequent sections of this document; if you connect to a device without the corresponding **license** activated (see section 2.1 of this manual), an informative screen will appear when you try to access to the options of those menus.

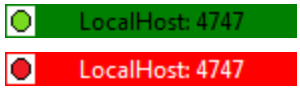


At the bottom of the window, the following information is displayed:

- **NETBOX IP Address and name:** shows the IP address and the name of the NETBOX that the application is pointing or trying to control. If in green this means that the connection is established and if in red this means that there is no connection.



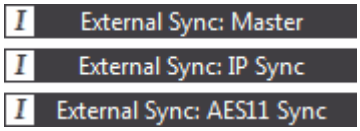
- **Local Host:** if in green indicates that there is connection to "NetboxServer" and if in red that there is no connection.



- **User:** when the active user in "NetboxRTC".

- **Group:** indicates the group that the current user belongs to.

- **External Sync:** indicates the configured synchronization mode of the unit. In the case of NETBOX DSP the third option is not available.

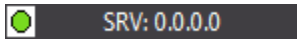


- **AOIP Link:** indicates the AoIP connection status of the unit. In the case of NETBOX 32/8 AD MX there is only 1 LED (it includes just one Dante module) and in NETBOX DSP there are 4 LEDs (1 per each possible Dante module: there is 1 Dante module in NETBOX DSP 64 model, 2 modules in NETBOX DSP 96, 3 in NETBOX DSP 128 and 4 in NETBOX DSP 160). The possible status are:

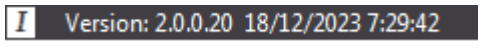
  - Grey: Dante module is not connected or not present.
  - Green: correct operation and synchronization of Dante module.
  - Red: Dante module is present but not synchronized.

- **BBDD:** the name of the system database that is in use to control the NETBOX devices.

- **SRV:** indicates the IP Address of Server RTC.

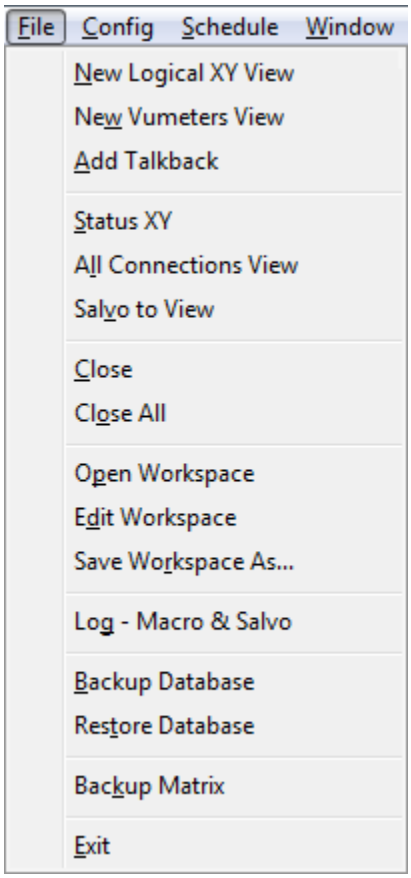


- **Application version in use, date and time of compilation.**



4. DROP-DOWN MENU: File.

The options that are available in the "File" drop-down menu are described in the following sections.



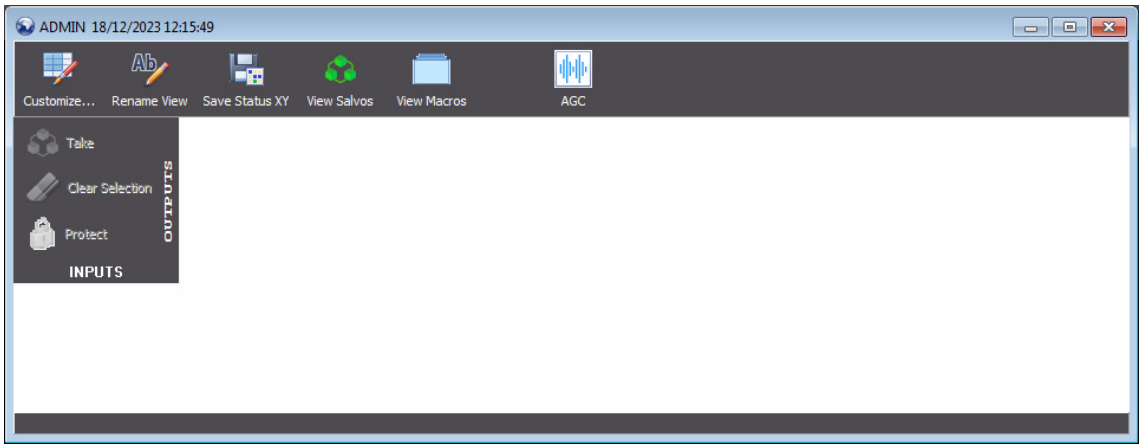
4.1. New Logical XY View.

By creating a logical XY View, enables you to control the connection between a selection of logical lines in XY format. Each connection of an input to an output included in the view is presented and made at the crosspoint between the row and the respective column.

The screens with real-time views can be configured according to the user's needs. Starting with the minimum possible size for each logical line, the space occupied by each line is readjusted to fit the size of the window, thus displaying all of the lines selected in that view. If there are more lines than can be shown in one window, scroll bars will be displayed.

The labels for each line can also be edited in real time, although when the session is closed, these changes will be lost, as they are momentary and depend on the use of the line at that particular moment.

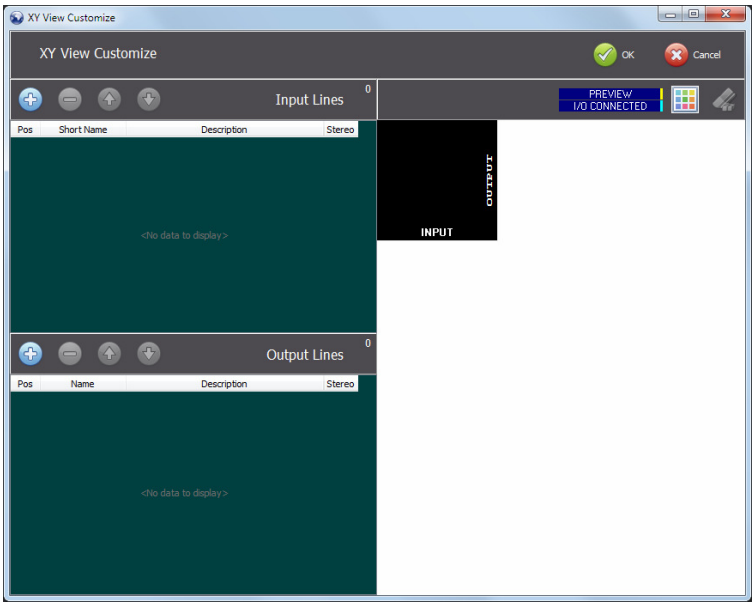
When you select "**New Logical XY View**", the following window appears:



4.1.1. Creating customized views. Customize.

When "**Customize…**" button is pressed, a window that allows you to select the input and output lines that should be part of the view appears. This window allows you to select from the available analog, digital and AoIP inputs and outputs. There is also a **test tone** of **1KHz** and **+4dBu** (-20dBFS) available.

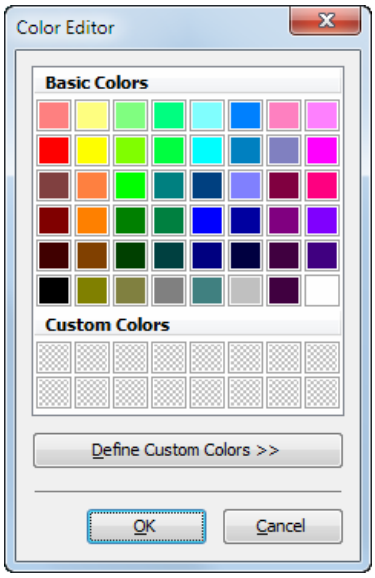
To access the list of available input or output lines, simply click the corresponding button. This will open the list of available lines to the right of this window.



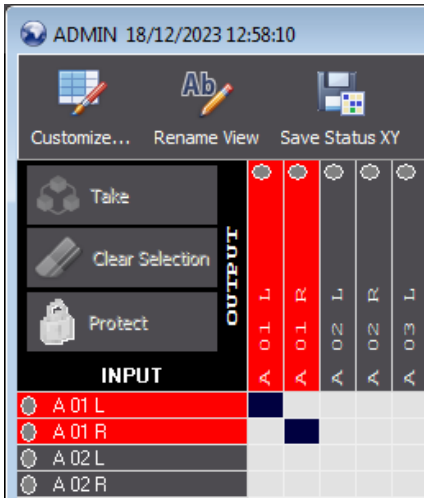
In the upper right part of the window there is a section that allows you to define the color that will appear in the XY view for the background of the labels of connected inputs and outputs:

A sample of the currently selected color appears on the left.

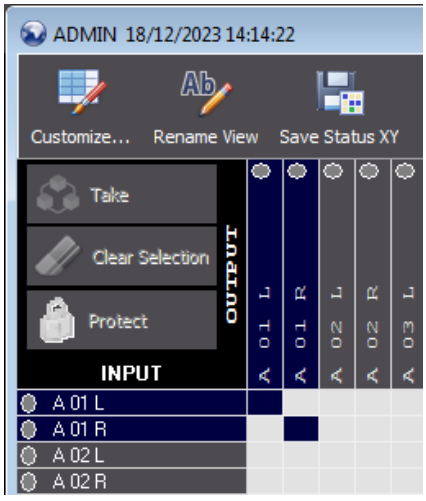
When the button is pressed the window that allows to select the desired color (from a series of basic colors or by defining custom colors) will appear:



Once selected, the new color will appear in the section and will be shown in the next XY views that are created (not in the current XY view):

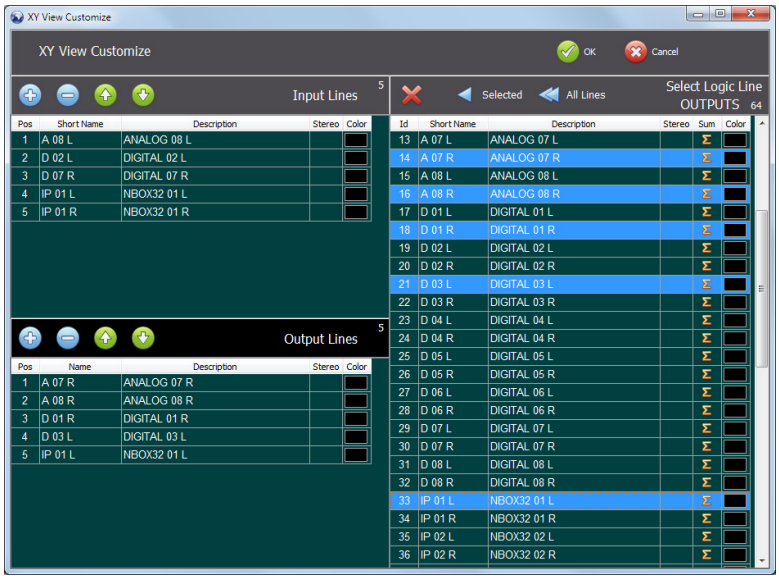


The button allows to return to default color. It will also be shown in the next XY views that are created (not in the current XY view):



4.1.1.1. Selecting lines.

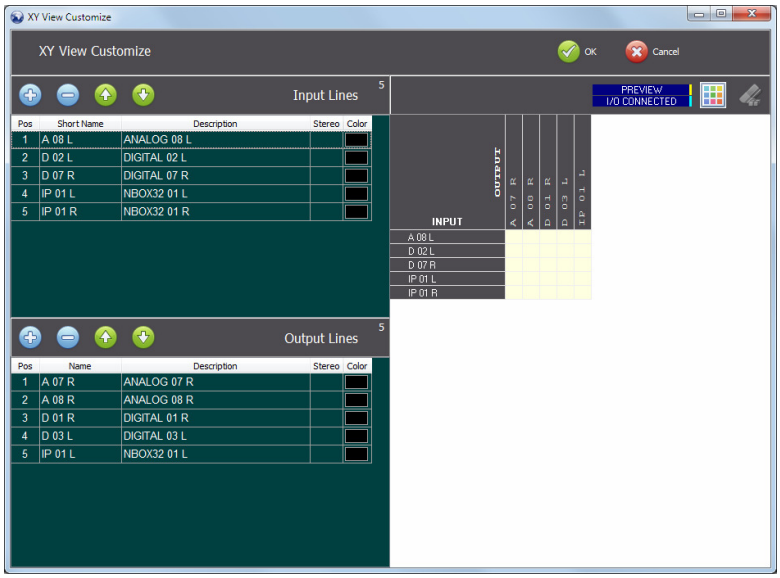
You can select the lines in any combination - one by one or making a multiple selection clicking the lines to the right and holding the Ctrl key on your keyboard down and when finished clicking on the "Selected" single arrow - or select all the lines by clicking on the "All Lines" double arrow. These actions will move the selection to the left side and into the inputs and outputs sections.



Any selection that is moved to the left will be part of the view and will also show at the last position of the list counting from above. If you need to move any of the inputs or output lines in the list, simply select the line and move it in the list with the green Up and Down arrows. If you need to remove any of the input or output lines, simply select the line in the corresponding inputs or outputs section and then click the button.

If you have made a multiple selection of either the input or output lines, this selection will remain on the list (inputs or outputs) to the right until you make a single selection on this list. This allows you to change the list view from inputs to outputs by alternatively clicking on the button for inputs and outputs. A second click on the button, e.g. if clicking on the button for the inputs when the list for inputs is active to the right, then you are toggling between the list and a preview of the XY view or panel that you are currently editing.

As you will be able to see, the list of selected inputs and outputs to the left in this window has a counter that indicates the current number of input and output lines that the XY view will have if you save the view. The counter is located in the upper right corner, slightly above the title bar "Input Lines" or "Output Lines". Once finished with the selection of input and output lines, you can either decide to confirm the view by clicking on the OK button or to discard it by clicking on the Cancel button.



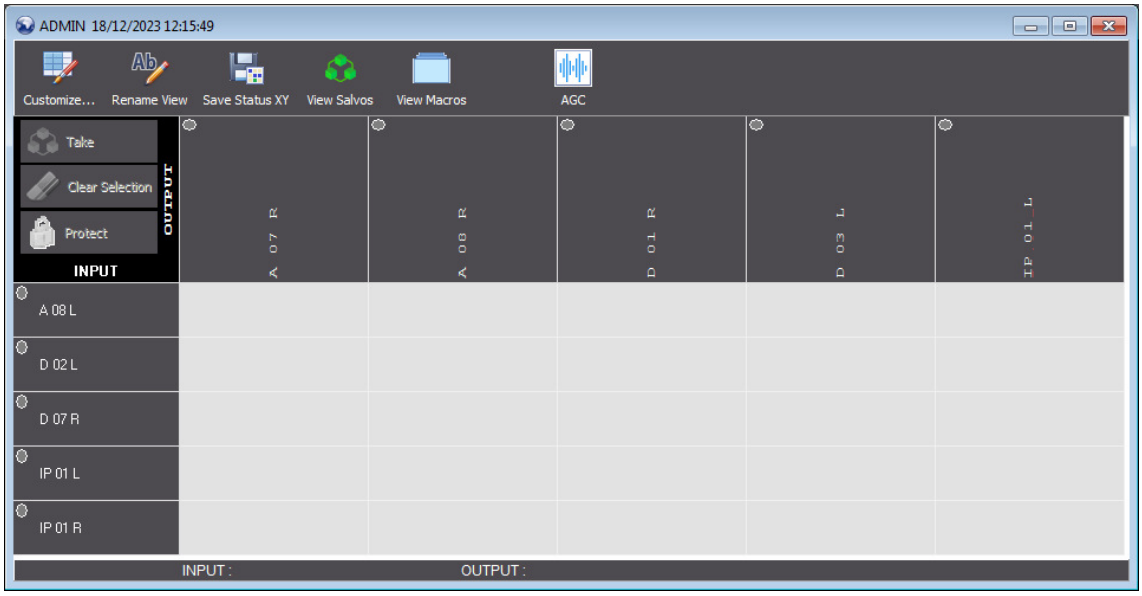
4.1.1.1.1. Test Tone.

As indicated in section 4.1.1, among the available input lines there is also a **test tone** of **1KHz** and **+4dBu** (-20dBFS), that appears at the end of the inputs list indicated in the previous section. That tone can be routed to any output from XY view by activating the corresponding crosspoint (see section 5.2) or from the context menu associated to that output line (see section 5.3).

4.1.1.2. New XY View.

If you confirm with the

button, the new XY view will appear as an operational window:



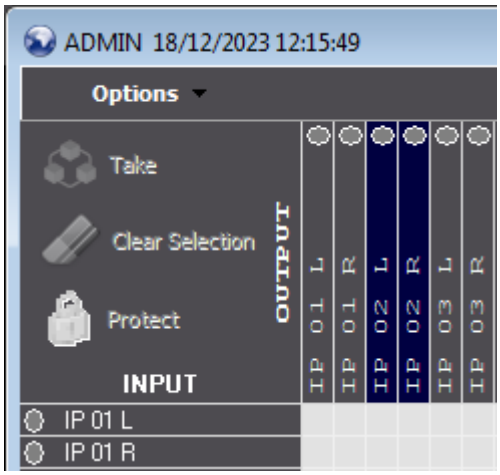
From left to right in the upper part of this window you will find the following buttons:

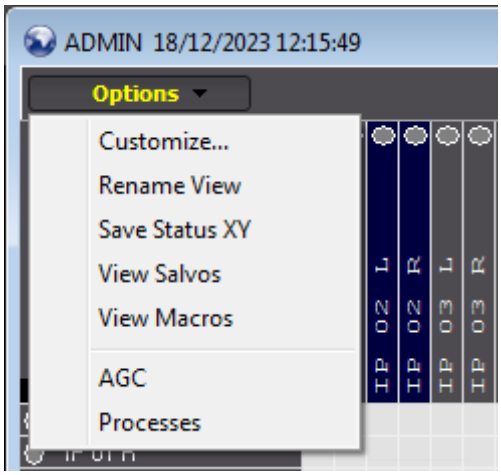
The "**Customize…**" button opens the window to select input and output lines as explained above, but with the inputs and outputs for this particular XY view in the sections to the left.

The remainders of the buttons are explained in the following sections.

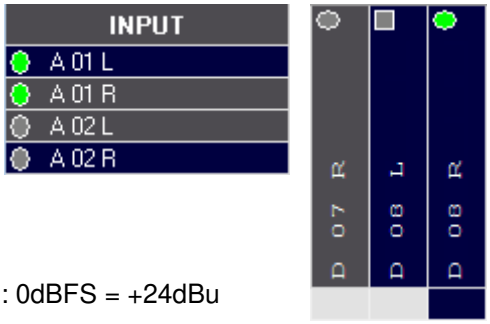
In the case of **NETBOX DSP** there is an additional button at the right of the previous ones (see section 4.1.8):

When you reduce the size of this window, at a certain moment those buttons are replaced by the "**Options**" drop-down menu with the same functionality:





The label of each one of the input and output lines that form the XY view shows an audio level indicator. In the case of the output lines the indicator will be square when the line is non summable. The levels and associated color are:

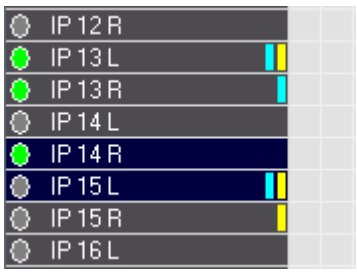


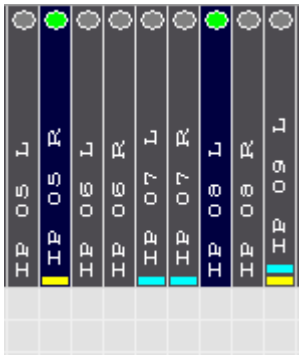
- Grey: less than -60dBFS
- Green: between -60 and -20dBFS
- Yellow: between -20 and -6dBFS
- Red: more than -6dBFS

NOTE: 0dBFS = +24dBu

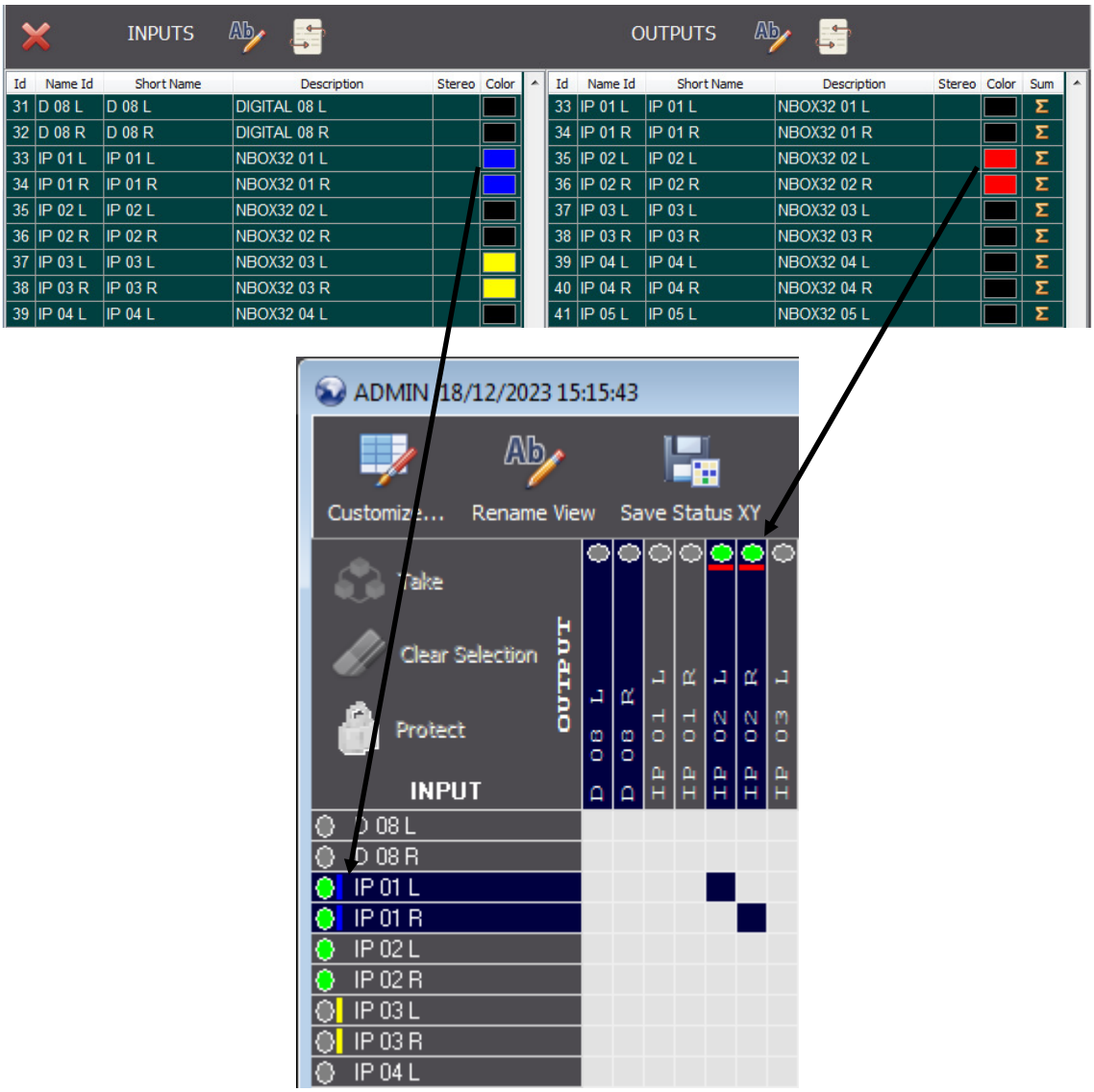
In the case of **NETBOX DSP**, at the right or under the label of the input or output lines where a **process** has been activated a yellow indicator of active process is shown.

In the case of **NETBOX DSP** and **NETBOX 32/8 AD MX**, at the right or under the label of the input or output lines where an **AGC** (Automatic Gain Control) has been activated a light blue indicator of active AGC is shown.

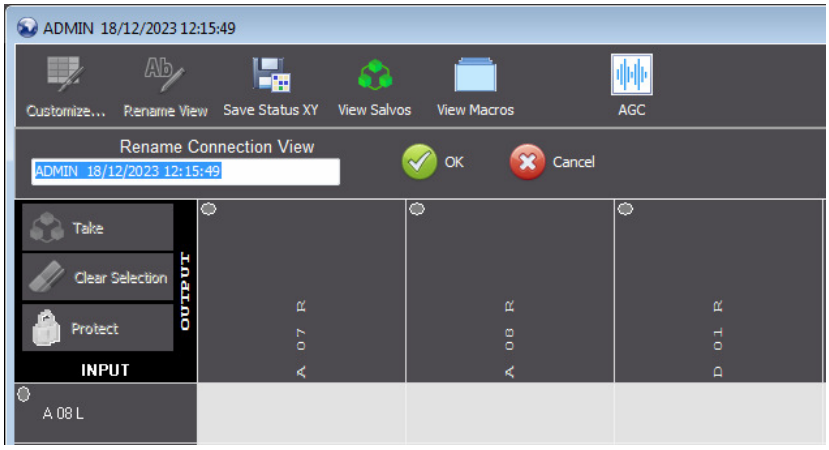




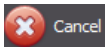
Finally, the label of the Dante input or output lines, if a color has been defined for the vumeters frames (see section 6.8), next to the audio level indicator an additional indicator appears with the color selected for those frames:


INPUTS
OUTPUTS
Id Name Id Short Name Description Stereo Color
31 D 08 L D 08 L DIGITAL 08 L
32 D 08 R D 08 R DIGITAL 08 R
33 IP 01 L IP 01 L NBOX32 01 L
34 IP 01 R IP 01 R NBOX32 01 R
35 IP 02 L IP 02 L NBOX32 02 L
36 IP 02 R IP 02 R NBOX32 02 R
37 IP 03 L IP 03 L NBOX32 03 L
38 IP 03 R IP 03 R NBOX32 03 R
39 IP 04 L IP 04 L NBOX32 04 L
Id Name Id Short Name Description Stereo Color Sum
33 IP 01 L IP 01 L NBOX32 01 L
34 IP 01 R IP 01 R NBOX32 01 R
35 IP 02 L IP 02 L NBOX32 02 L
36 IP 02 R IP 02 R NBOX32 02 R
37 IP 03 L IP 03 L NBOX32 03 L
38 IP 03 R IP 03 R NBOX32 03 R
39 IP 04 L IP 04 L NBOX32 04 L
40 IP 04 R IP 04 R NBOX32 04 R
41 IP 05 L IP 05 L NBOX32 05 L
ADMIN 18/12/2023 15:15:43
Customize...
Rename View
Save Status XY
Take
Clear Selection
Protect
INPUT
OUTPUT
D 08 L
D 08 R
IP 01 L
IP 01 R
IP 02 L
IP 02 R
IP 03 L
08 L
08 R
IP 01 L
IP 01 R
IP 02 L
IP 02 R
IP 03 L
IP 03 R
IP 04 L

4.1.2. Rename XY View.

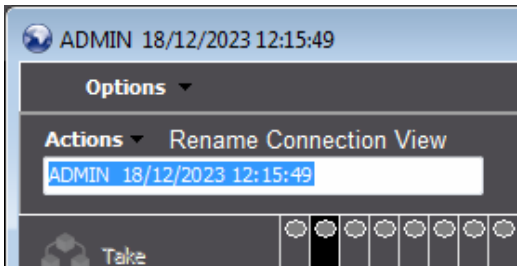


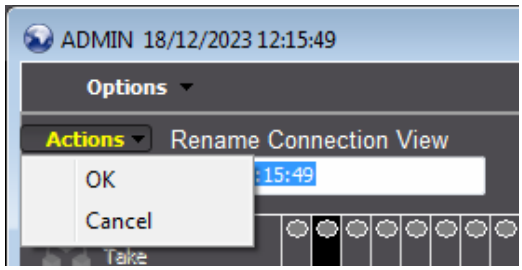
The "**Rename View**" button allows you to change the name of the view. By default, the application provides a name that consists in the name of the user that has created the view plus the date and time of the generation of the view. Just overwrite the default name with the preferred name in the "Rename Connection View" box, for example a name that correctly describes the XY view. When modified, either click the

button to accept the name change or the 

to discard the change.

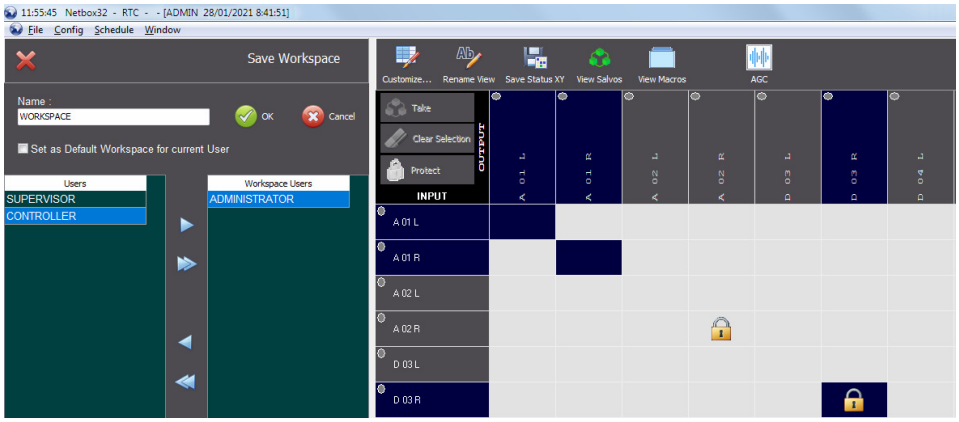
When you reduce the size of the window, at a certain moment those buttons are replaced by the "**Actions**" drop-down menu with the same functionality:





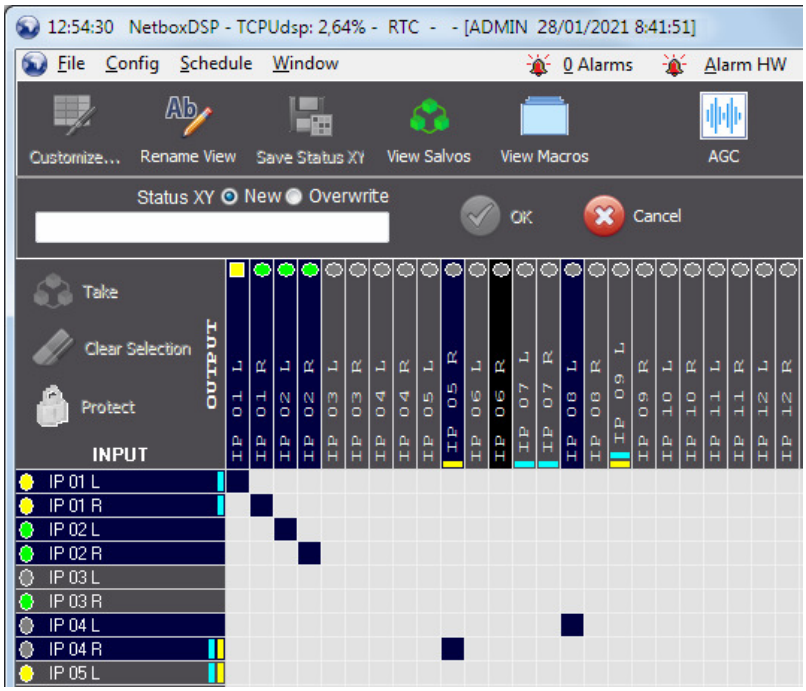
4.1.3. Save XY View.

When you want to **save** the XY view to the database you need to click on the "File" drop-down menu and then select "Save Workspace As…". This opens the "**Save Workspace**" window: here you can name the workspace, set the workspace as default when opening the application ("**Set as Default Workspace for current User**") and define what users should have access to the workspace in the application.

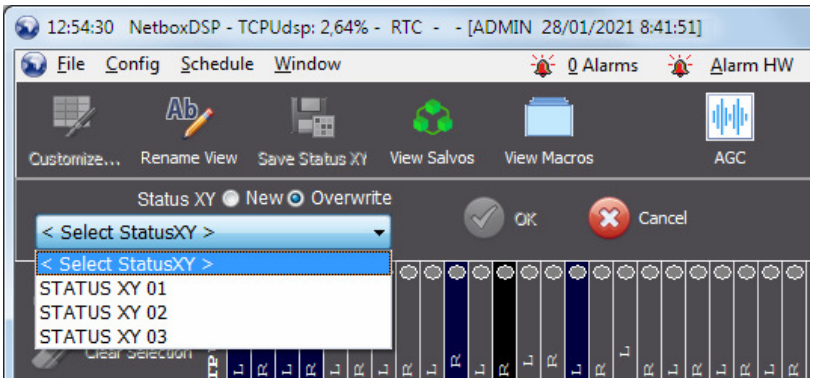


4.1.4. Save Status XY.

When you click on the "**Save Status XY**" button, this will open a line in the window that allows you to save the XY view as well as the connection/disconnection status of the crosspoints. See section 4.4 of this manual.

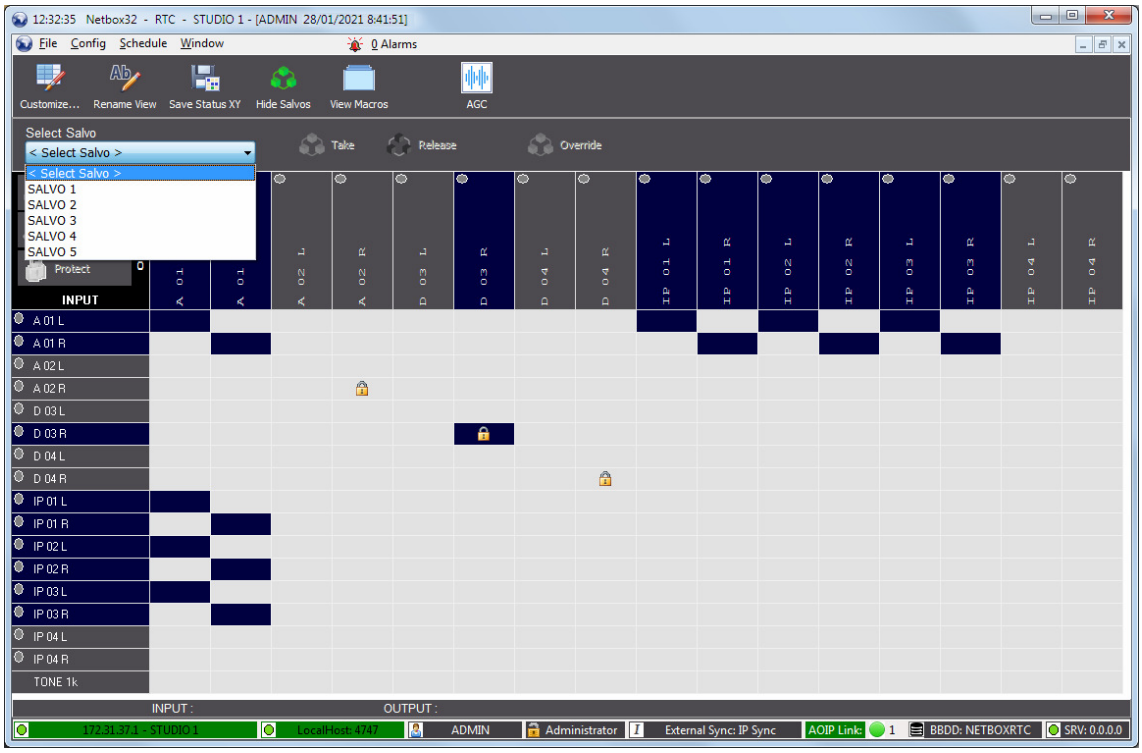


You can assign a new name to this "Status XY" or even overwrite an existing one by selecting the "Overwrite" option and then selecting one of the "Status XY" previously saved.



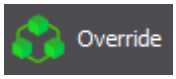
4.1.5. View Salvos.

When you click on the "**View Salvos**" button, this will open a line in the window that allows you to select a salvo from the "**Select Salvo**" drop-down list. If no salvos have been recorded, the list will for obvious reasons be empty. Also, when this line in the window is opened the "View Salvos" button will change legend and read "**Hide Salvos**" (when you click on it that new line is hidden). If you select a salvo from the drop-down list, the "Take", "Release" and "Override" buttons to the right of the list will become active.



The "**Take**" button will activate the salvo and execute the defined actions: connect/disconnect and/or protect/unprotect crosspoints. In case any of those crosspoints is protected yet, no action will be executed on it when activating the salvo.

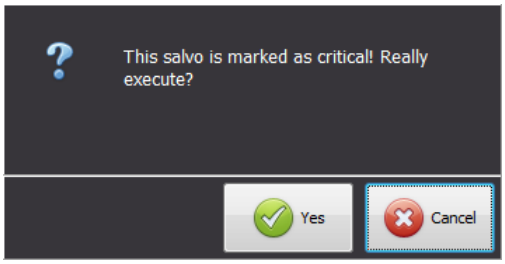
The clicking on the "**Release**" button will deactivate the salvo and execute the opposite actions on crosspoints. In case any of them is protected yet, no action will be executed on it when deactivating the salvo.



If the taking of a salvo implies such protected crosspoints, it's possible to execute the salvo completely, breaking those protections, by clicking on the "**Override**" button.

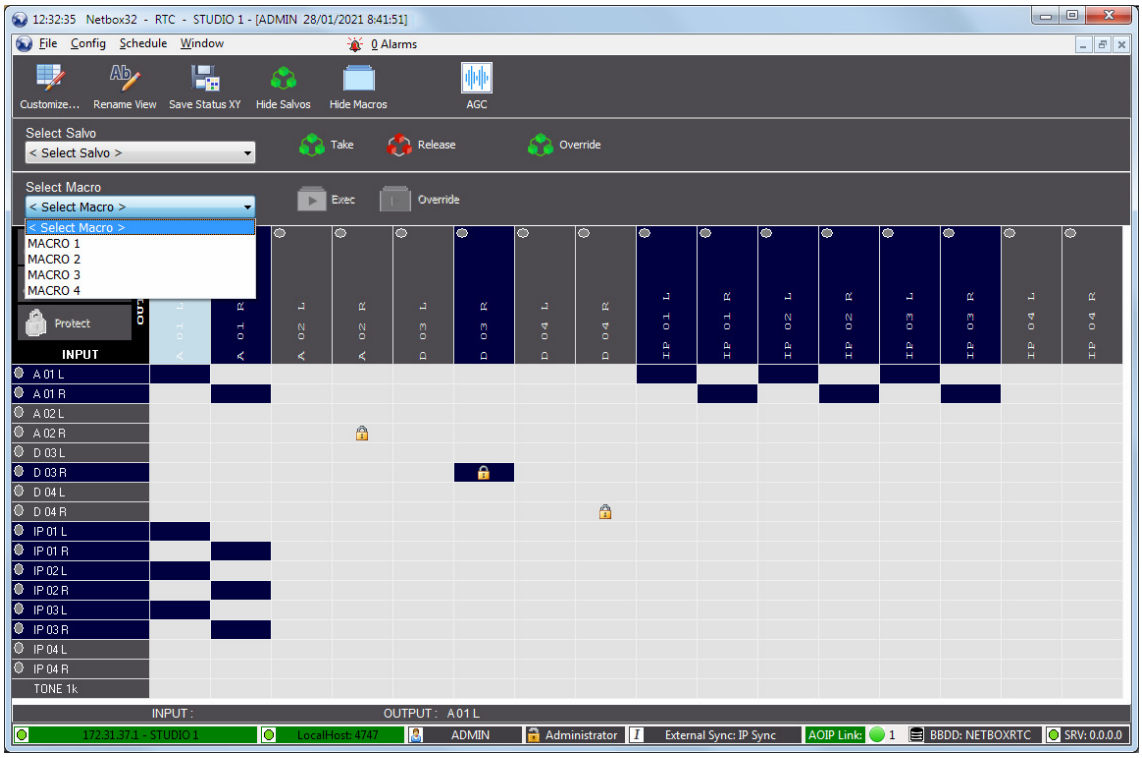
When you reduce the size of the window, at a certain moment those buttons are replaced by the "**Actions**" drop-down menu with the same functionality.

If a salvo is marked as **critical** ("Request confirmation" option has been selected when creating the salvo, as described later on in this manual) the user will be prompted to confirm the action with a warning message. If the salvo is not marked as critical, it will be executed immediately if the user has the rights or privileges for the selected salvo.

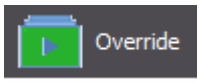


4.1.6. View Macros.

When you click on the "**View Macros**" button, this will open a line in the window that allows you to select a macro from the "**Select Macro**" drop-down list. If no macros have been recorded, the list will for obvious reasons be empty. Also, when this line in the window is opened the "View Macros" button will change legend and read "**Hide Macros**" (when you click on it that new line is hidden). If you select a macro from the drop-down list, the "Exec" button to the right of the list will become active.



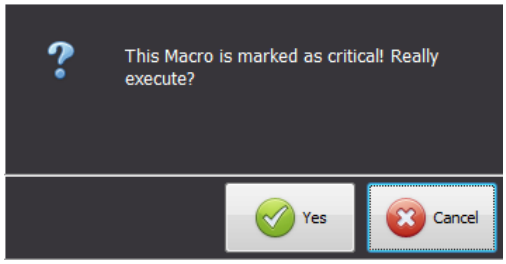
The "**Exec**" button will run all the actions that have been defined in the macro including connection/disconnection and/or protection/unprotection of crosspoints and activation/deactivation of salvos or GPOs. In case any of the those crosspoints is protected yet, no action will be executed on it when activating the macro.



If the taking of a macro implies such protected crosspoints, it's possible to execute the macro completely, breaking those protections, by clicking on the "**Override**" button.

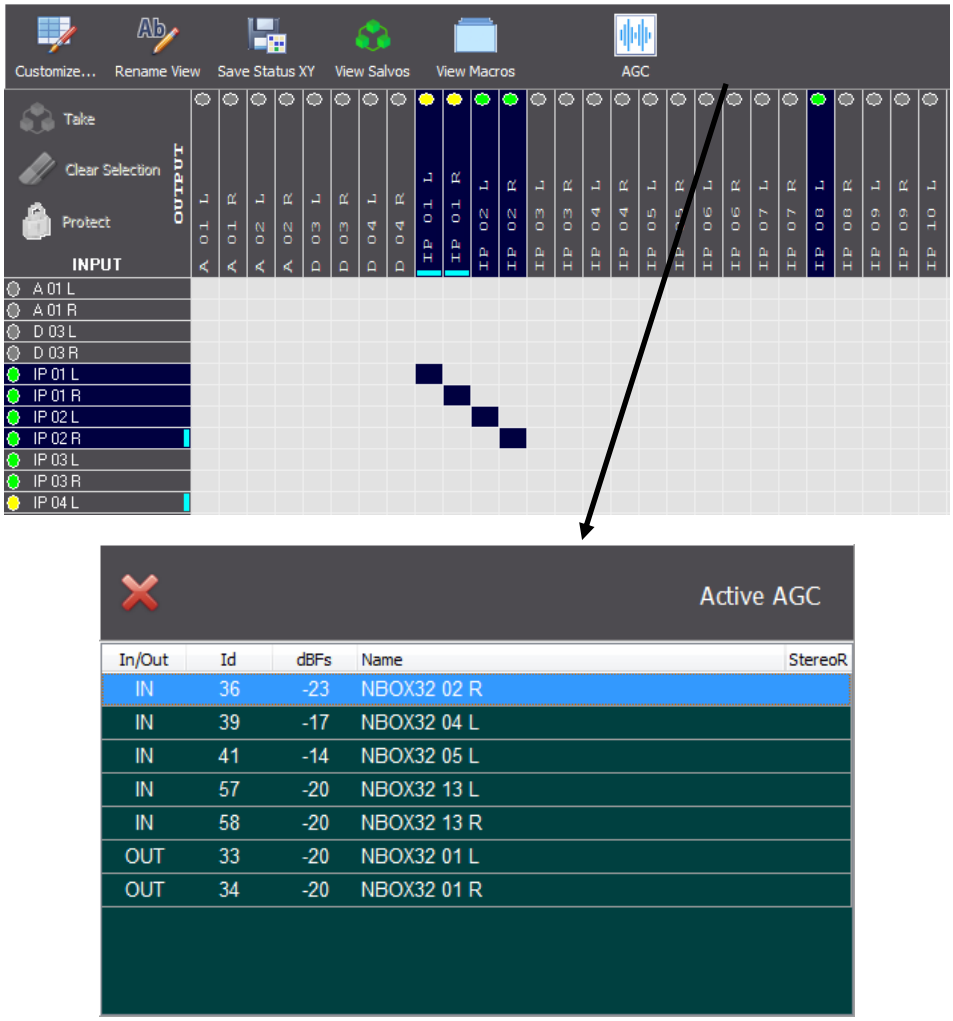
When you reduce the size of the window, at a certain moment those buttons are replaced by the "**Actions**" drop-down menu with the same functionality.

If a macro is marked as **critical** ("Request confirmation" option has been selected when creating the macro, as described later on in this manual) the user will be prompted to confirm the action with a warning message. If the macro is not marked as critical, it will be executed immediately if the user has the rights or privileges for the selected macro.



4.1.7. AGC.

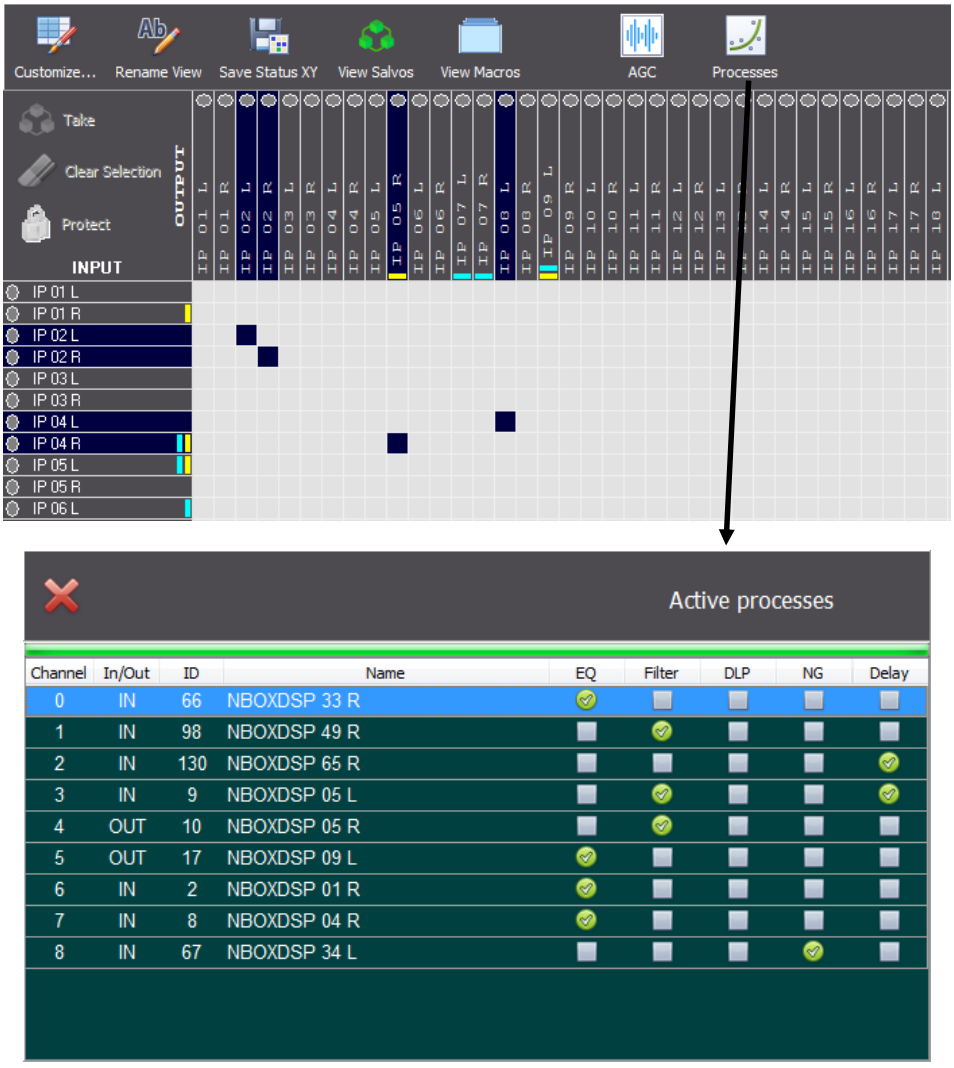
When you click on the "**AGC**" button, this will open the "Active AGC" informative window that shows a list of the input and output lines where an AGC (Automatic Control Gain) has been activated and also the configuration of those controls in dBFs. See section 5.3.2 of this manual.



In the case of **NETBOX 32/8 AD MX** you can activate **AGCs only** in the input and output **Dante lines**, not in the analog or digital lines.

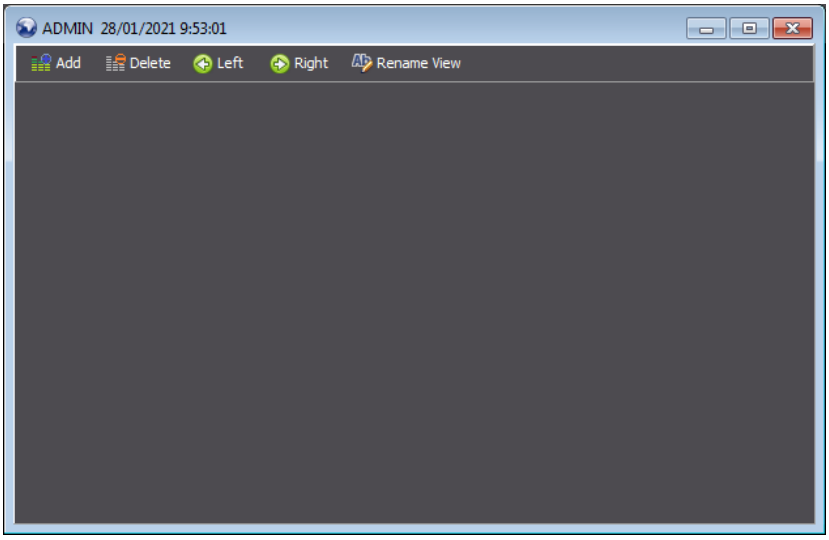
4.1.8. Processes.

This option only appears for **NETBOX DSP.** When you click on the "**Processes**" button, this will open the "Active Processes" informative window that shows a list of the input or output lines where a process has been activated and also the type of active process:



4.2. New Vumeters View.

The "**New Vumeters View**" option of the "File" menu allows you to create a vumeters view to visualize the levels of the selected Dante input and output lines in order to control them. When you select this option the following window will appear:

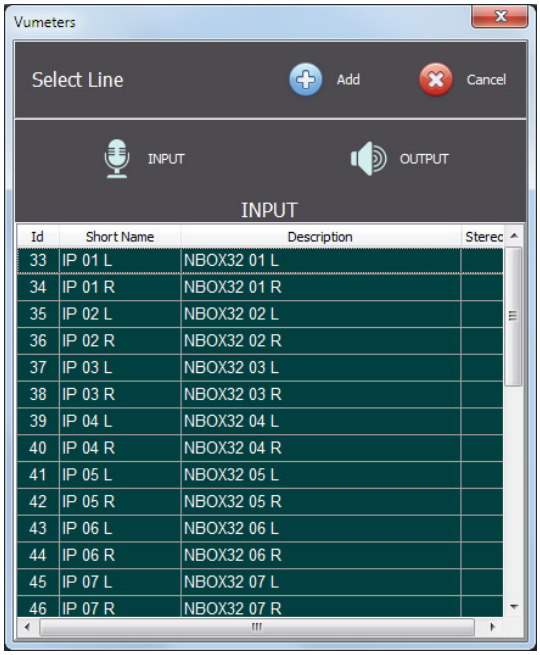


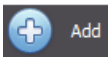
From left to right in the upper part of this window you will find the following buttons:

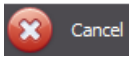
The "**Add**" button allows you to add new input and/or output vumeters and the "**Delete**" button allows you the eliminate the selected vumeters. The "**Left**" and "**Right**" buttons allow you to move the selected vumeter. The "**Rename View**" button allows you to rename the vumeters view.

The available vumeters will depend on the number of available input and output Dante lines. In order to be able to visualize the vumeters it's necessary to configure an associated Multicast IP address (see section 6.15 of this manual).

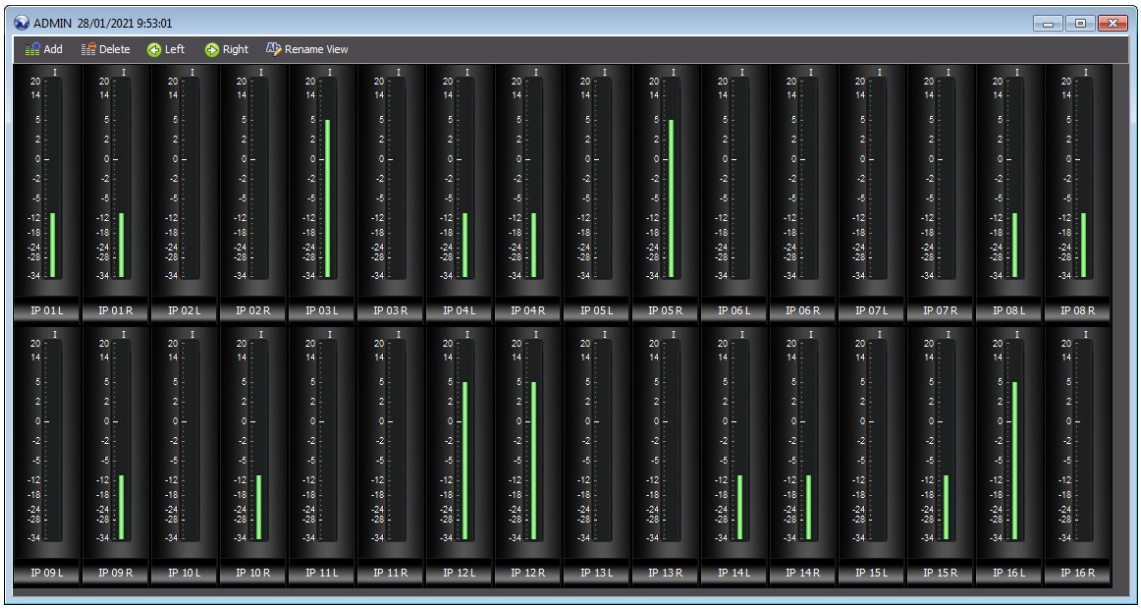
When you press the "Add" button the following window will appear:



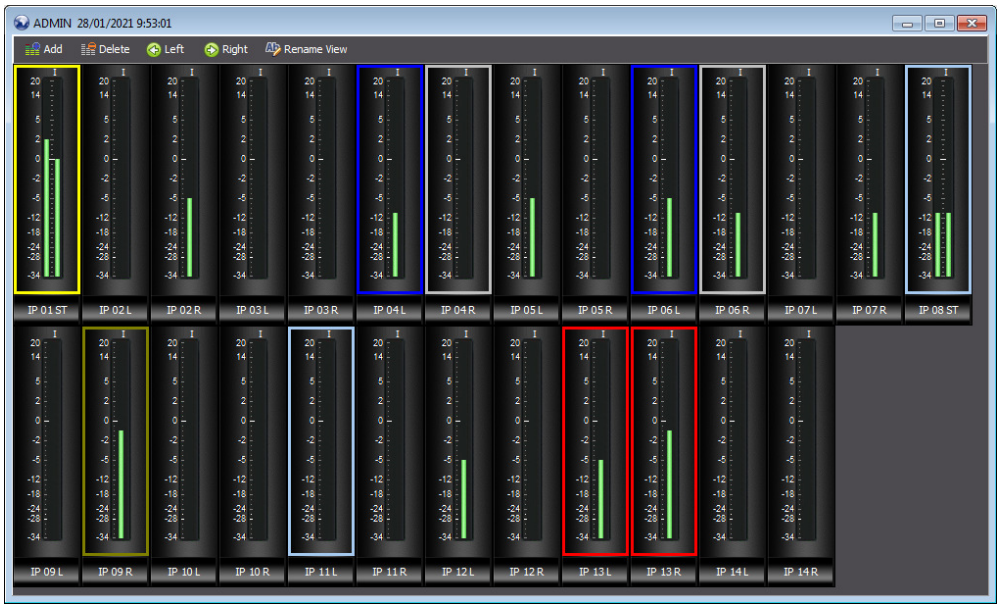
You can select Dante input or output lines, by pressing the corresponding button, one by one or making a multiple selection (clicking on one line and holding the Ctrl key in order to select more lines). Once the selection is finished, either click the 

button to add more lines or the 

button to discard the selection.

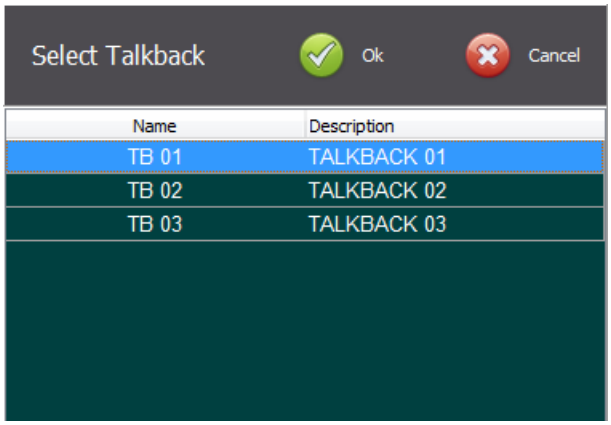


The "**Logical Lines**" option of the "Config" menu allows you to choose the **color** you want to visualize the frames of the vumeters (see section 6.8 of this manual).

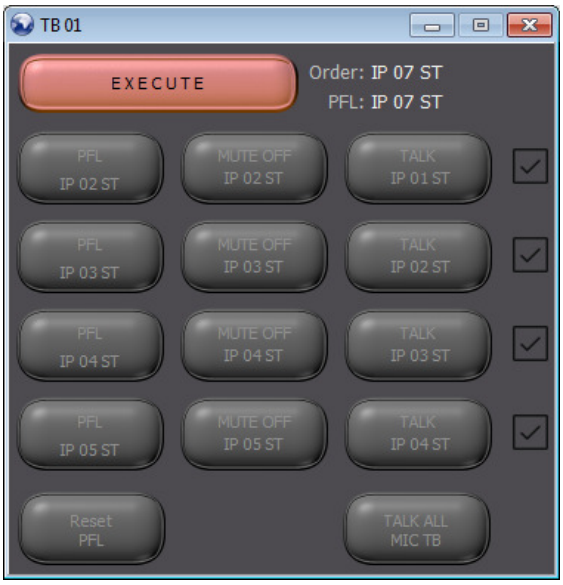


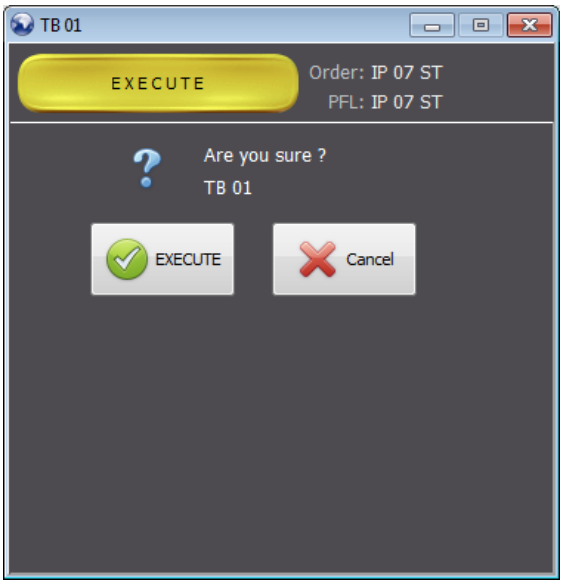
4.3. Add Talkback.

This option of the "File" menu will open the dialogue window that contains the list of "Talkback" groups previously created:



This window allows you to open the selected "Talkback" group by pressing the "**OK**" button. It will open the window that allows you to execute the selected "Talkback" by pressing the "**Execute**" button (confirmation is required).



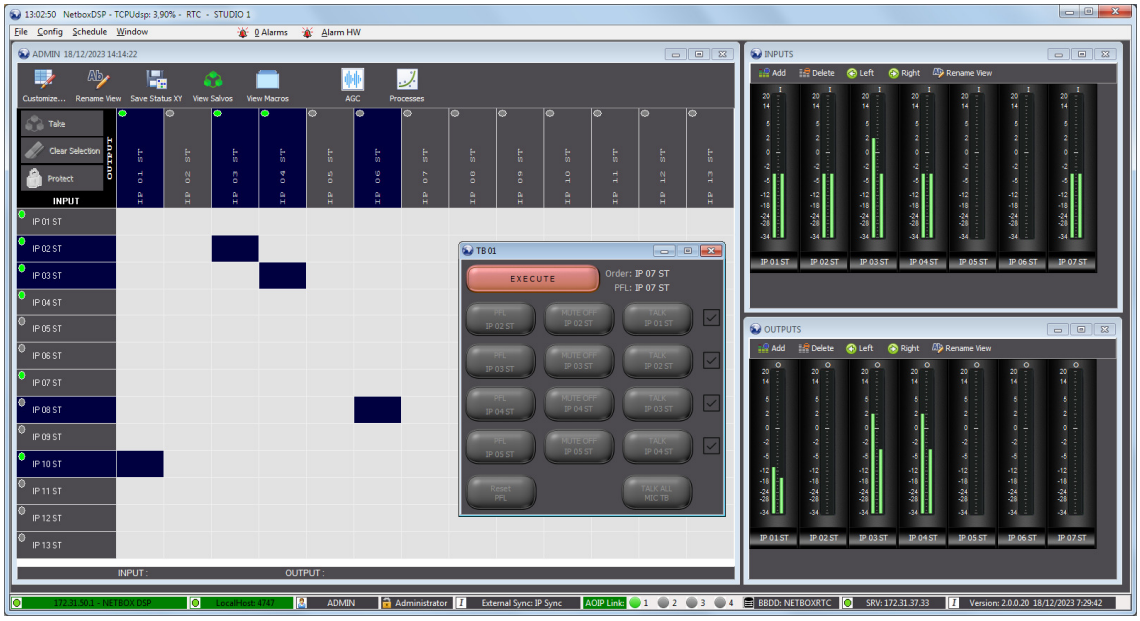


When the "Talkback" is executed all the existing crosspoints towards the output lines that are part of it will be deleted (towards the line defined as "PFL" as well as towards the ones defined as "Talk") and, only in the case of a NETBOX DSP, the gains of all the crosspoints (activated or not) towards those output lines will be restored to 0.

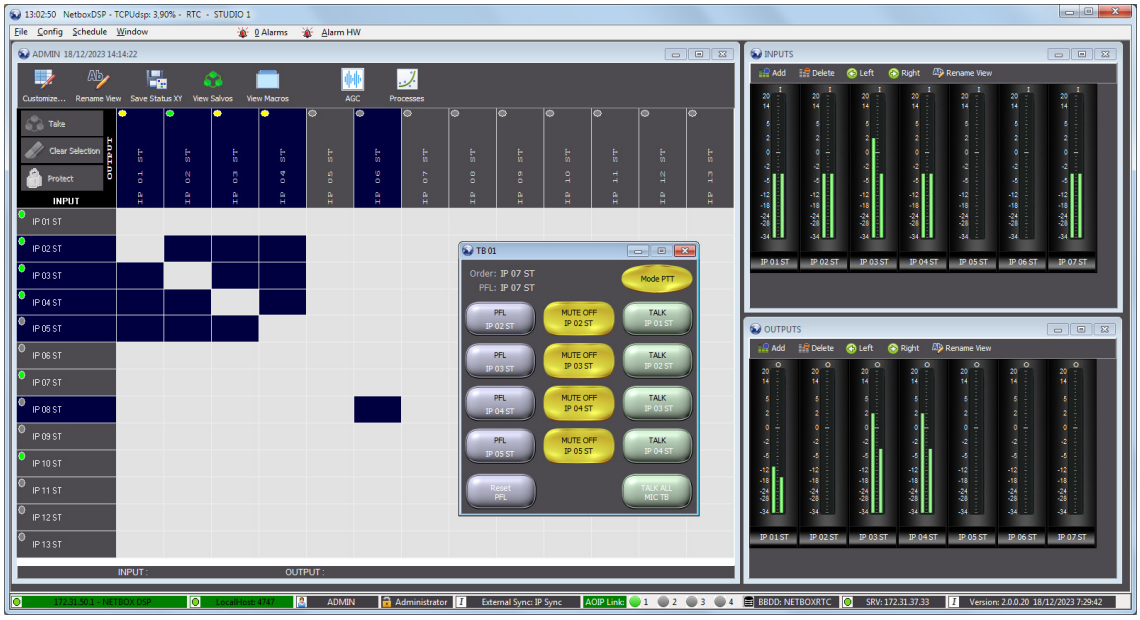
Additionally, the n-1 connections between the input lines of the "Talkback" defined as "Mute" and the output lines defined as "Talk" will be automatically created (each "Mute" input will be sent to all the "Talk" outputs except the one associated to that input when the "Talkback" was defined: see section 6.7 of this manual).

You can execute a previously defined "Talkback" without using some of the associated lines. In order to do that, the button of the lines that will not be used should be pressed (this button changes to in order to indicate that that line is out of the group; it can be added again by pressing the button again).

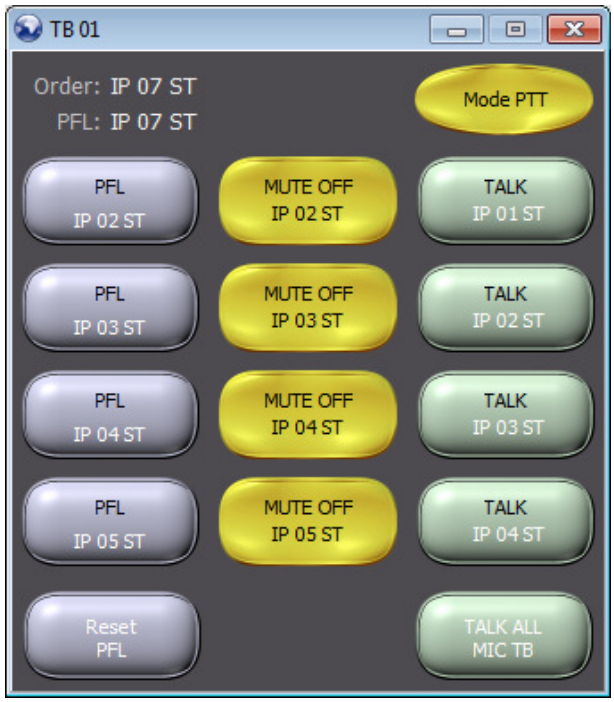
The following example shows a series of configured crosspoints:



When the **TB 01** "Talkback" is executed the 3 crosspoints on the left are deleted because those output lines are part of the selected "Talkback". Then the corresponding n-1 connections are created:



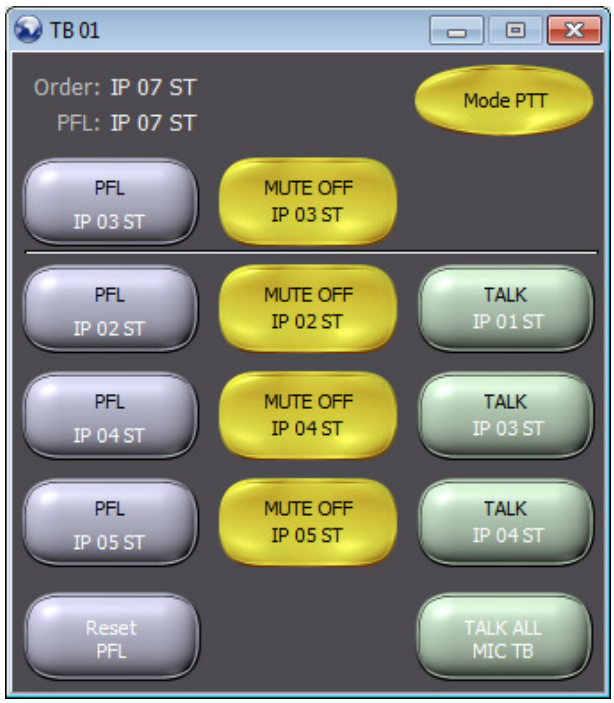
When the "Talkback" window gets active looks like this.



In the upper part of the window the input line defined as "Order" input and the output line defined as "PFL" output are shown. The following buttons are also shown:

- "**PFL":** When you activate it the input line displayed on the button is sent towards the "PFL" output.
- "**Reset PFL":** When you activate it the previously activated crosspoints to "PFL" are deactivated.
- "**MUTE OFF":** When you activate it (it changes to MUTE) the input line displayed on the button is muted by deleting the crosspoints that send that line towards the defined "Talk" outputs.
- "**Mode PTT":** Allows you to activate or not the PTT working mode (Push To Talk) of the TALK buttons placed below. By default it's active.
- "**TALK":** When you activate it all the crosspoints connected to the output line displayed on the button are deactivated and the "Order" input is sent to it. In case that the "Talkback Mode" option was configured as "Attenuation 20 dB" when the "Talkback" was created, those crosspoints will remain but with an attenuation of 20 dB (this option is available only for NETBOX DSP; see section 6.7 of this manual).
- "**TALK ALL MIC TB":** When you activate it, all the crosspoints connected to all the "Talk" outputs are deactivated (or attenuated) and the "Order" input is sent to all of them. This button will not appear in case that the "Talk All" option was configured as "Disabled" when the "Talkback" was created (see section 6.7 of this manual).

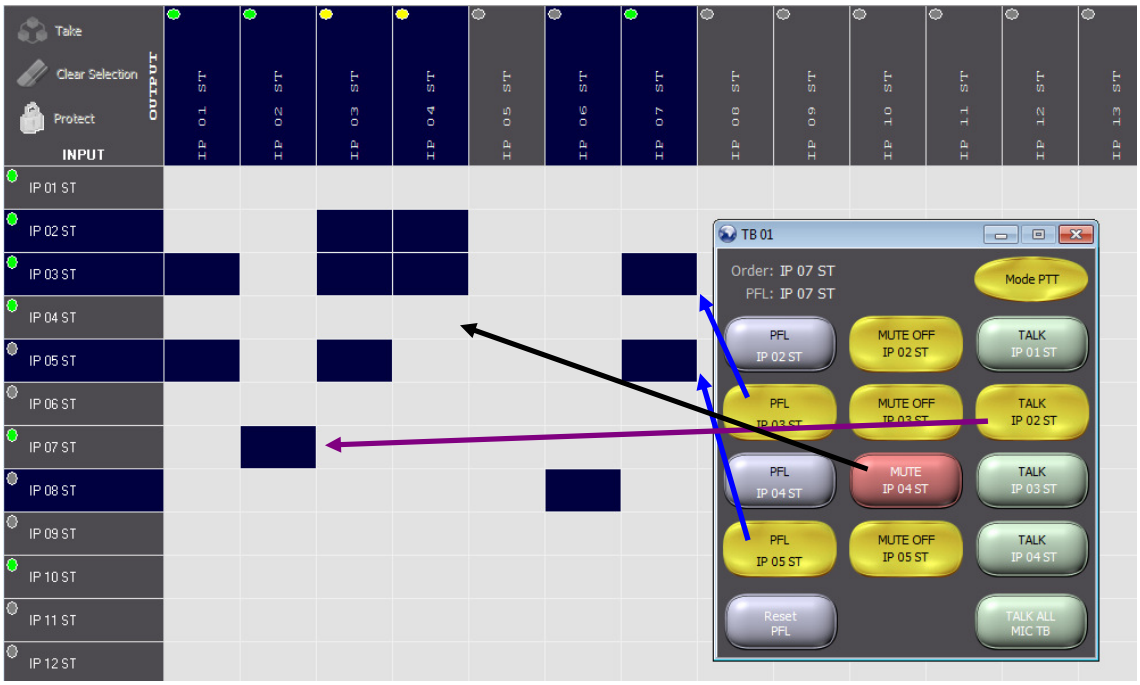
In case that one of the lines selection of the "Talkback" is configured as "**Program**", the previous window is slightly different (the "Program" lines selection appears in first place and separated by a line). In this case, the associated TALK button will not appear and the MUTE OFF button will appear just in case that the "Mute in Program" option is configured as "Enabled" (see section 6.7 of this manual).



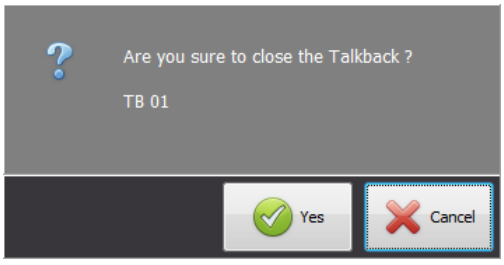
In the following example, the "PFL IP 03 ST" and "PFL 05 ST" buttons have been pressed to send the "IP 03 ST" and "IP 05 ST" inputs to the output line defined as "PFL" ("IP 07 ST").

Also the "MUTE OFF IP 04 ST" button has been pressed to deactivate the crosspoints that send the "IP 04 ST" input towards all the "Talk" outputs.

Finally the "TALK IP 02 ST" button has been pressed to deactivate all the crosspoints to the "IP 02 ST" output and send the "Order" input line ("IP 07 ST") towards that output.



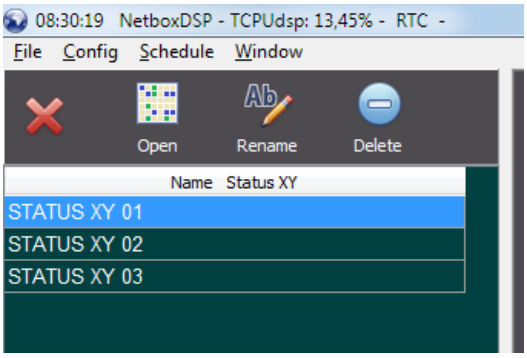
When the "Talkback" window is closed (by clicking on the right top cross) all the crosspoints created towards the output lines contained in the "Talkback" are deactivated. Confirmation is required:



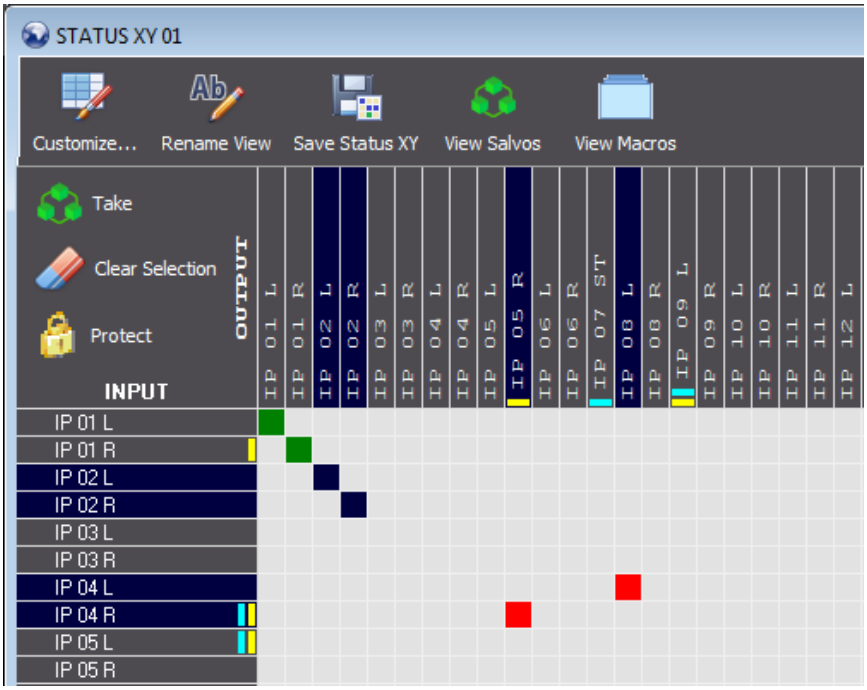
IMPORTANT NOTE: If any of the output lines included in a "Talkback" is defined as NON summable, when the "Talkback" is executed the line starts to act as a summable line (although on the XY view the audio level indicator will keep square). When the "Talkback" is closed the line will return to act as a NON summable line.

4.4. Status XY.

This option of the "File" menu will open the dialogue window that contains the list of "Status XY" previously saved:



This window allows you to open the selected "Status XY" (by means of the "**Open**" button). By doing it, the associated XY view is opened (with all the input and output lines selected for that view) and all the crosspoints that should be connected or disconnected in order to leave the system in the same status as it was when the "Status XY" was saved will be marked:

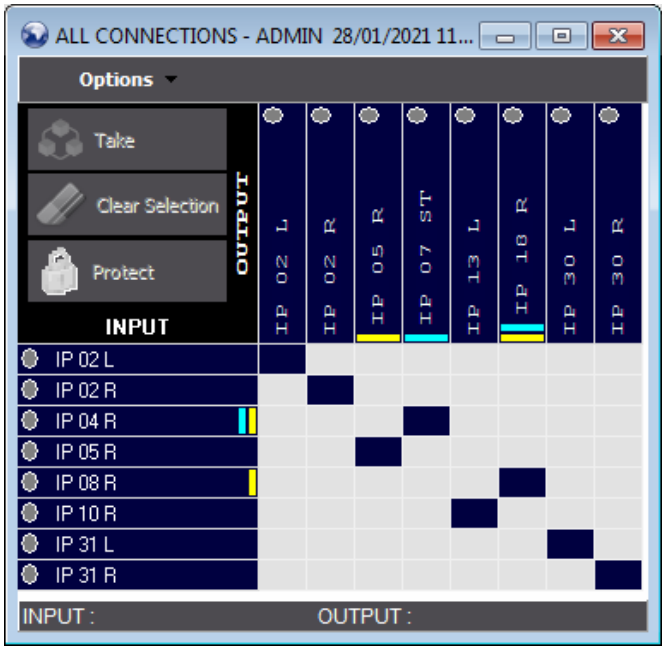


The crosspoints that will be connected/disconnected when the "**Take**" button is pressed are marked in green/red. You can also discard those connections/disconnections by pressing the "Clear Selection" button (or discard them and in addition protect those points by pressing the "Protect" button).

In the example, when you press "Take" 2 crosspoints will be connected and other 2 will be disconnected. The other 2 crosspoints that are shown were yet connected when that "Status XY" was save and their status will not change.

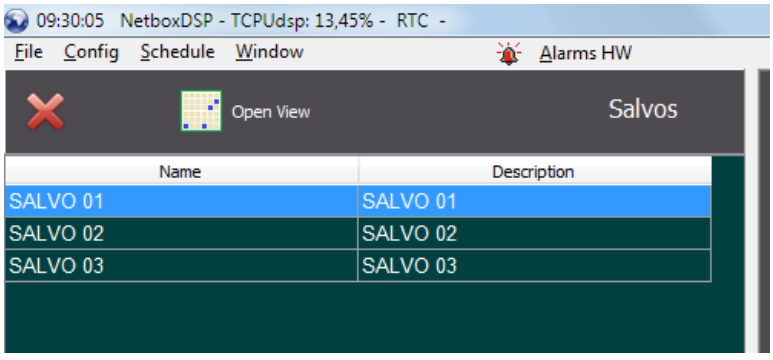
4.5. All Connections View.

This option of the "File" menu allows you to create an XY view including only all the input and output lines that are part of a crosspoint:



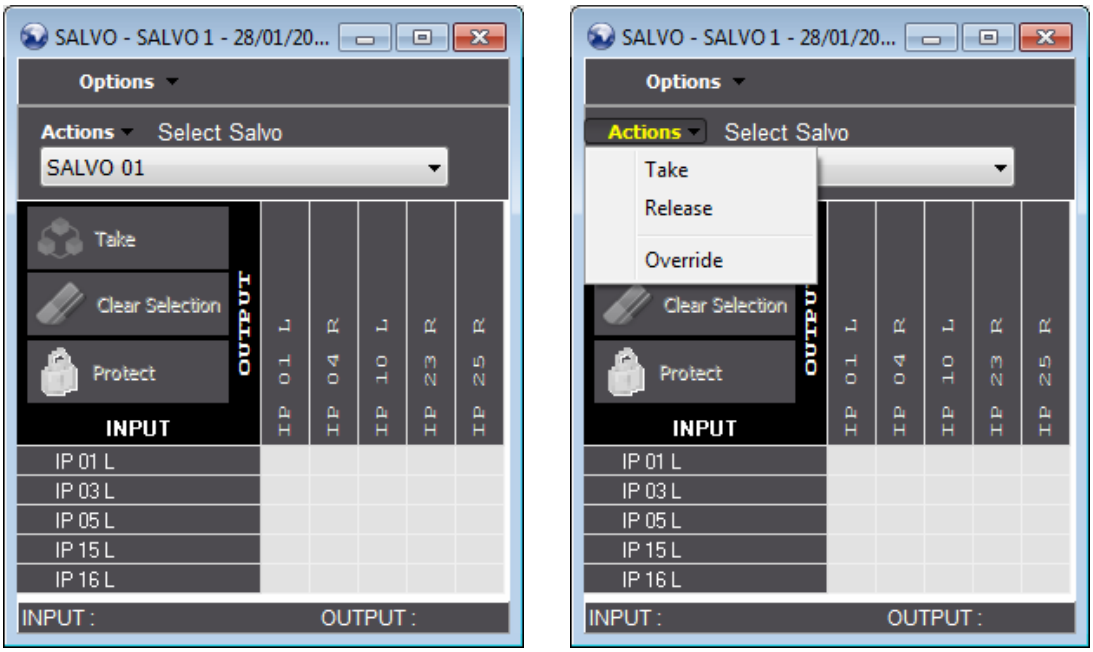
4.6. Salvo to View.

This option of the "File" menu will open the dialogue window that contains the list of "Salvos" previously created:



When you select one of the salvos and press the "**Open View**" button, it will open an XY view that includes only the input and output lines that are part of that salvo.

From that view you can also execute the desired action ("Take", "Release" or "Override") from the "Actions" drop-down menu or from the associated buttons (depending on the size of the salvo and therefore on the size of the associated XY view):



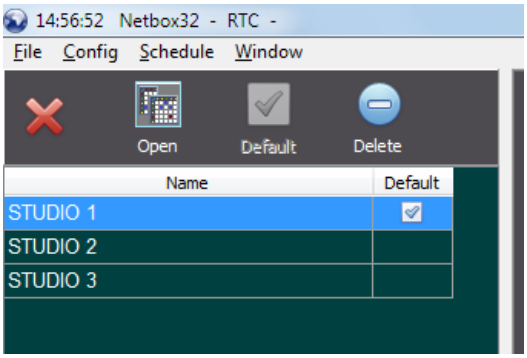
4.7. Close and Close All.

The "**Close**" option of the "File" menu allows you to close the active XY or vumeters view and the "**Close All**" option allows you to close all the views currently open.

4.8. Workspace.

4.8.1. Open Workspace.

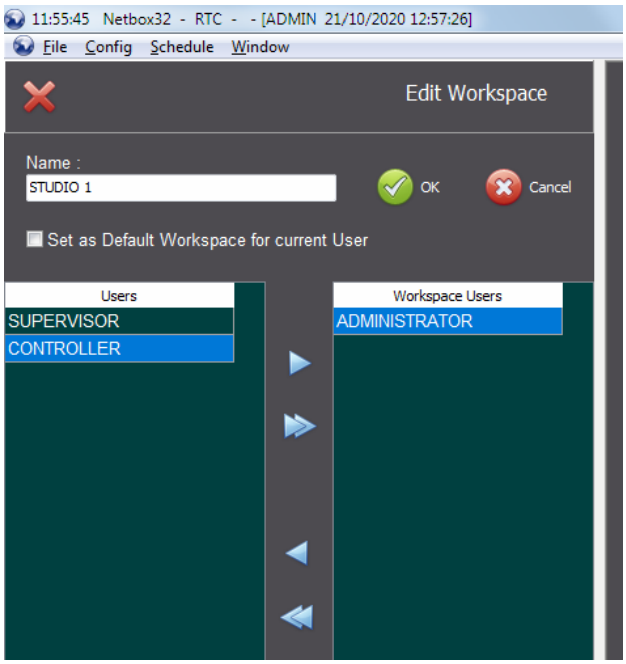
This option of the "File" menu will open the dialogue window that contains the list of workspaces that are created, visible and available to the user currently operating the "NetboxRTC" application.



This window allows you to open the selected workspace (by means of the "**Open**" button), make the selected workspace the default one for the user that is operating the "NetboxRTC" application (by means of the "**Default**" button) and also eliminate the workspace if the user has sufficient application privileges (by means of the "**Delete**" button).

4.8.2. Edit Workspace.

This option of the "File" menu allows you to edit the properties of the active workspace, i.e. modify what was defined when you saved the original workspace and as described above: here you can re-name the workspace, set the workspace as default when opening the application and define what users should have access to this particular workspace in the application.

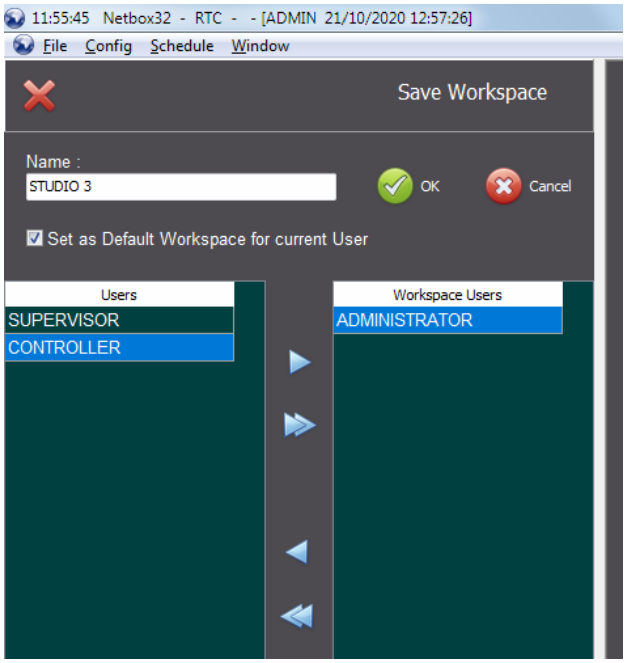


4.8.3. Save Workspace As.

This option of the "File" menu allows you to save the visible workspace (including XY and vumeters views) and assign a name that allows to recognize it from "Open Workspace" window.

For each workspace you also must select the users that will have access to it and if that will be the default workspace for the current user ("**Set as Default Workspace**").

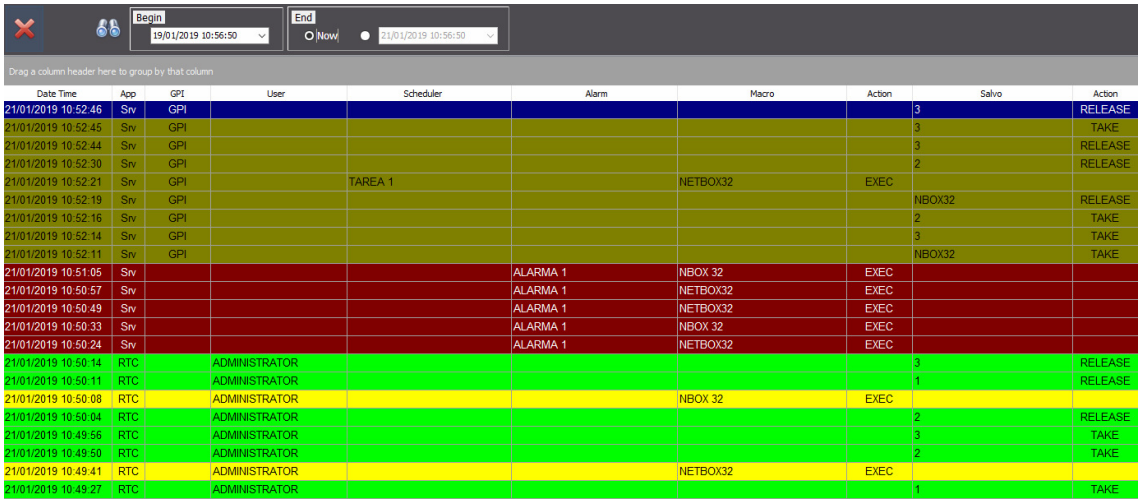
When you save a workspace previously saved, you will basically duplicate the workspace with a different name.



4.9. Log - Macro & Salvo.

The "**Log - Macro & Salvo**" option of the "File" menu allows you to have a log of the salvos and macros executed by means of alarms, schedule, GPIs and Virtual GPIs. Each action has a different color and application associated:

- **Server:**
  - Alarm (red).
  - Schedule (green).
  - GPI (olive).
  - Virtual GPI (blue).
- **RTC:**
  - Salvo (lime).
  - Macro (yellow).



Date Time	App	GPI	User	Scheduler	Alarm	Macro	Action	Salvo	Action
21/01/2019 10:52:46	Srv	GPI						3	RELEASE
21/01/2019 10:52:45	Srv	GPI						3	TAKE
21/01/2019 10:52:44	Srv	GPI						3	RELEASE
21/01/2019 10:52:30	Srv	GPI						2	RELEASE
21/01/2019 10:52:21	Srv	GPI		TAREA 1		NETBOX32	EXEC		
21/01/2019 10:52:19	Srv	GPI						NBOX32	RELEASE
21/01/2019 10:52:16	Srv	GPI						2	TAKE
21/01/2019 10:52:14	Srv	GPI						3	TAKE
21/01/2019 10:52:11	Srv	GPI						NBOX32	TAKE
21/01/2019 10:51:05	Srv				ALARMA 1	NBOX 32	EXEC		
21/01/2019 10:50:57	Srv				ALARMA 1	NETBOX32	EXEC		
21/01/2019 10:50:49	Srv				ALARMA 1	NETBOX32	EXEC		
21/01/2019 10:50:33	Srv				ALARMA 1	NBOX 32	EXEC		
21/01/2019 10:50:24	Srv				ALARMA 1	NETBOX32	EXEC		
21/01/2019 10:50:14	RTC		ADMINISTRATOR					3	RELEASE
21/01/2019 10:50:11	RTC		ADMINISTRATOR					1	RELEASE
21/01/2019 10:50:08	RTC		ADMINISTRATOR			NBOX 32	EXEC		
21/01/2019 10:50:04	RTC		ADMINISTRATOR					2	RELEASE
21/01/2019 10:49:56	RTC		ADMINISTRATOR					3	TAKE
21/01/2019 10:49:50	RTC		ADMINISTRATOR					2	TAKE
21/01/2019 10:49:41	RTC		ADMINISTRATOR			NETBOX32	EXEC		
21/01/2019 10:49:27	RTC		ADMINISTRATOR					1	TAKE

4.10. Backup Database.

This option of the "File" menu allows you to make a backup copy of the database and choose the directory to store it.

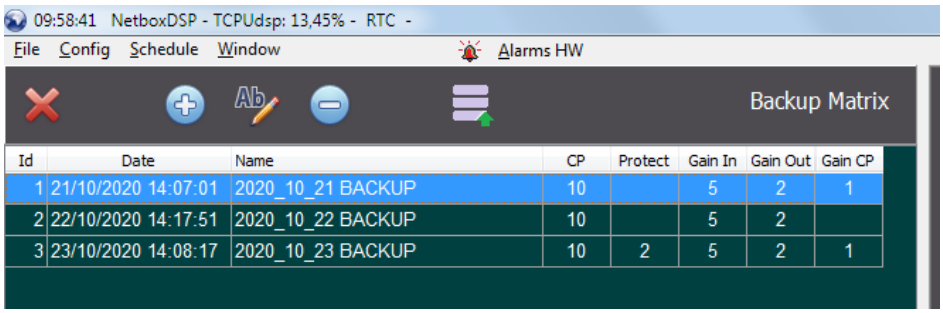
4.11. Restore Database.

This option of the "File" menu allows you to restore the backup copy of the database. In order to do that all the applications (Server and all the RTCs) should be closed and reopened then.

4.12. Backup Matrix.

This option of the "File" menu allows you to make a backup copy of the matrix configuration or restore a backup copy previously saved.

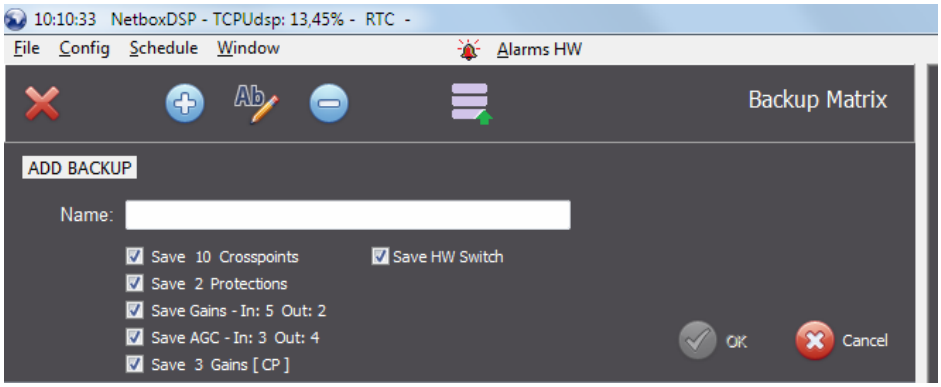
When this option is selected, it will open the dialogue window that contains the list of matrix backups previously created:



For each backup it indicates the creation date, the name and the number of crosspoints and inputs, outputs and crosspoints gains the backup contains.

The buttons available in the upper part of the window are the following:

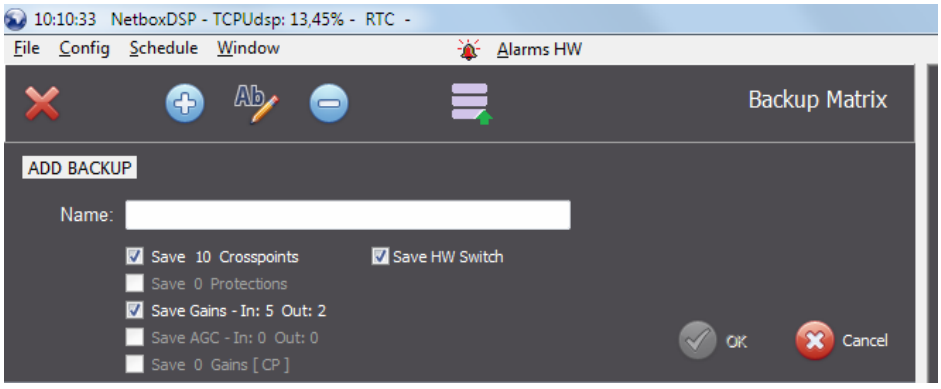
Allows you to create a new matrix backup. In order to do that you must assign a name and select what will be stored in that backup.



The tic-boxes under the name allows the user to choose whether the following parameters will be stored in the backup or not:

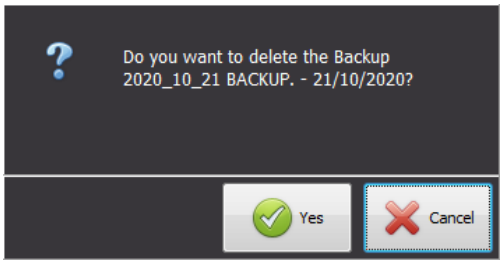
- crosspoints.
- protections.
- inputs and outputs manual gains.
- inputs and outputs AGCs (Automatic Gain Control).
- crosspoints gains (available only for NETBOX DSP).
- HW Switch configurations.

When any of the 5 first parameters is not configured in matrix, the associated tic-box is not selectable. In the following example there are no protections, AGCs and crosspoints gains configured in matrix and that's why those tic-boxes are not available:

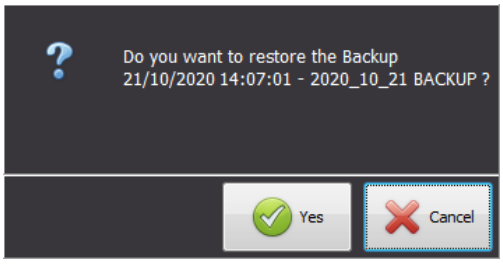


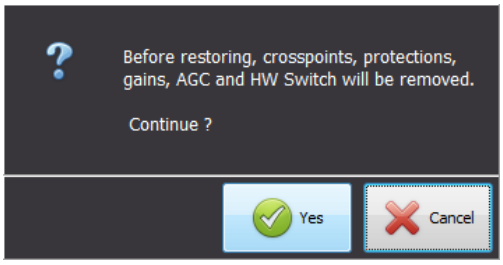
Allows you to edit the name of the selected backup.

Allows you to delete the selected backup from the list. Confirmation is required.

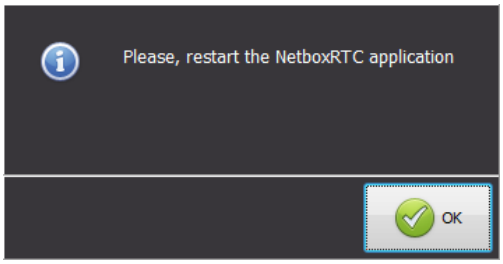


Allows you to restore the selected backup from the list. Double confirmation is required because before restoring that backup all crosspoints, protections, gains, AGCs and HW Switch configurations of matrix will be removed.



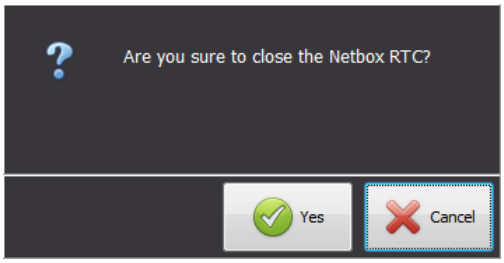


Once the selected backup is restored the "NetboxRTC" application must be restarted:



4.13. Exit.

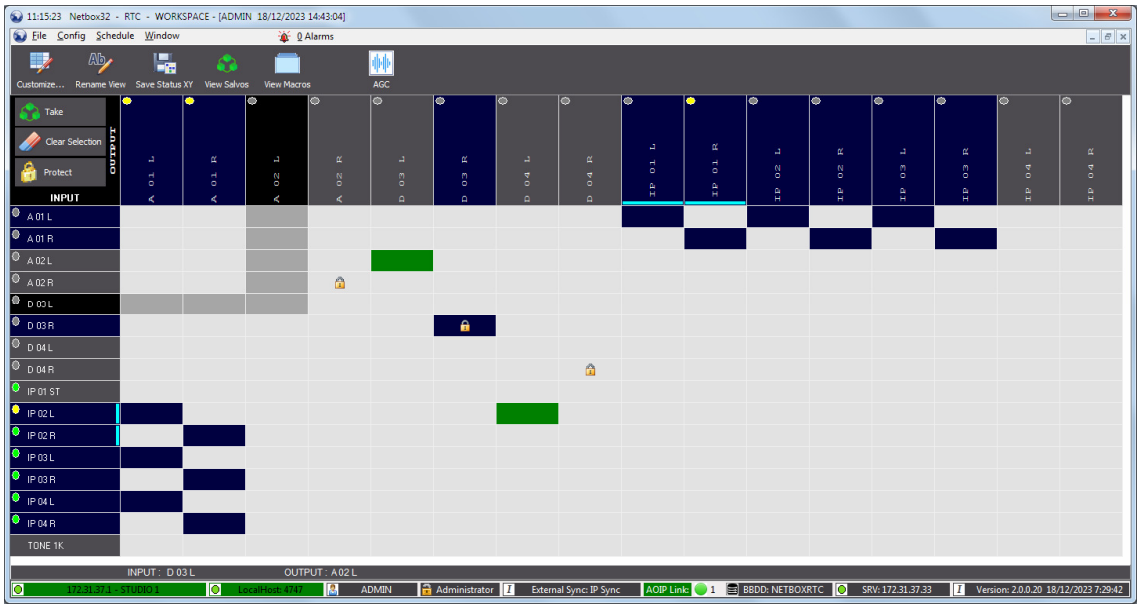
This option of the "File" menu allows the user to exit the "NetboxRTC" application. Confirmation is required.



5. WORKING WITH CROSSPOINTS.

5.1. Operating the XY views in the workspace.

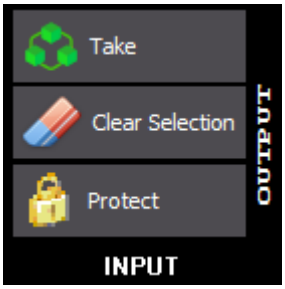
Once you have adjusted the XY view, the operational window will have this appearance:



Any modification made in any of the actions detailed in sections 5.2 or 5.3 (except the ones regarding the precision vumeters) will be stored in the non volatile memory of the NETBOX after at least one minute with no changes detected in these actions.

5.2. Actions on crosspoints.

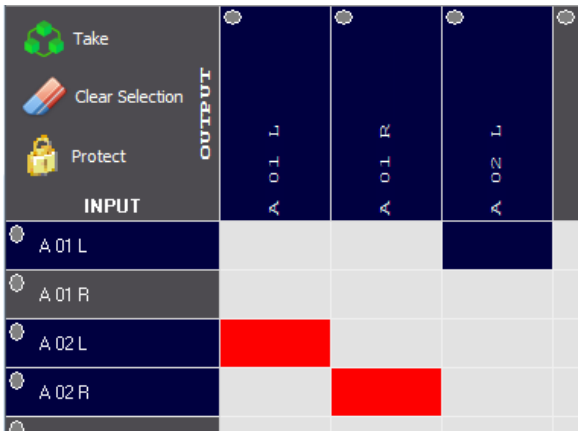
The following buttons, at the left corner of the XY view, allows the user to operate the matrix:



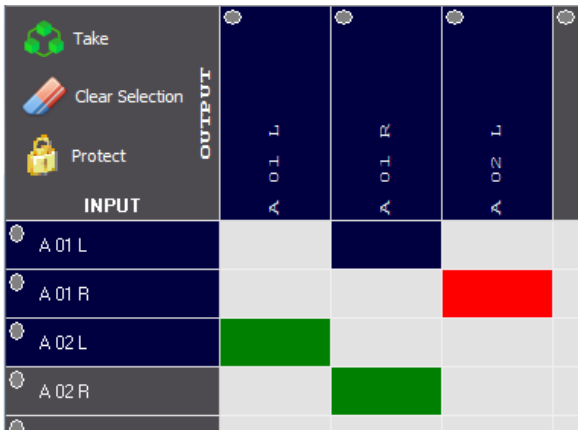
- "**Take":** Activates or deactivates the selected crosspoints to connect (green crosspoints) or disconnect (red crosspoints) the respective logical lines.
- "**Clear Selection":** Clears any selection of crosspoints on the XY view.
- "**Protect":** Sets the status of the crosspoint to protect against incorrect actions: once activated, the crosspoint will show a padlock.

When you place the cursor over the crosspoints you wish to establish and then select them, the crosspoints will be marked **green.** When you press the "**Take**" button, the selected crosspoints will be activated. An active crosspoint is always indicated as **blue.**

If you place the cursor over one or several active crosspoints (marked in blue) and then select them, they will be marked in **red.** Pressing the "**Take**" button deactivates the selected crosspoints, that will be cleared from any colour.

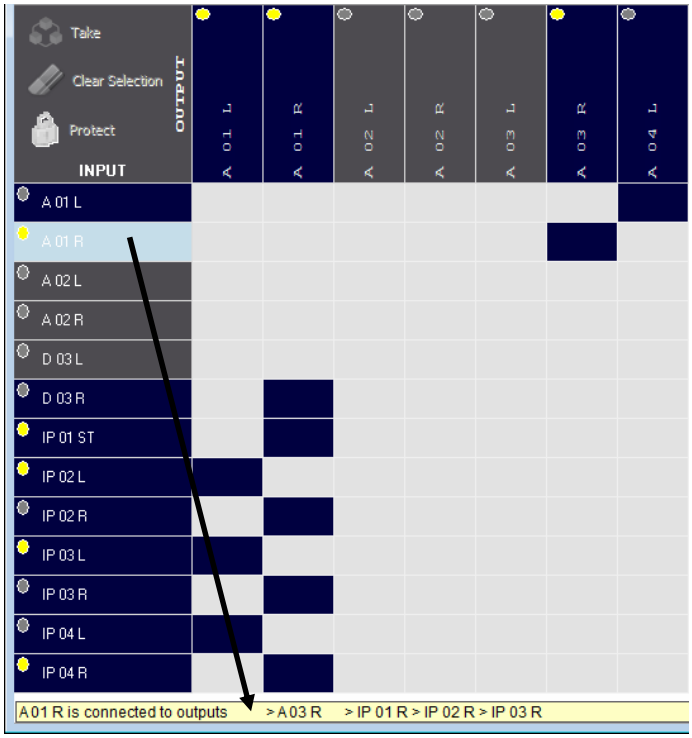


If a set of active and inactive crosspoints are selected (they will be marked in red and green respectively) and "Take" is executed, the "NetboxRTC" application will execute both the connections and disconnections, i.e. it's possible to both activate and deactivate multiple crosspoints with the same "Take" action.



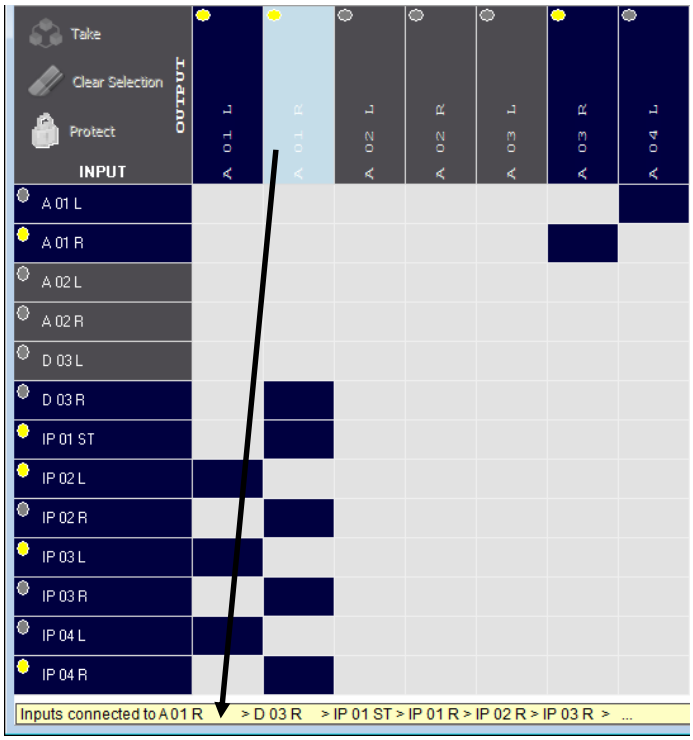
At the bottom of the window, the input and output associated with each crosspoint will be displayed when the mouse cursor is placed over that crosspoint.

If you click over an input of the XY view, the active connections of that input appear in the lower bar of the view, that is, the outputs that input is connected to.



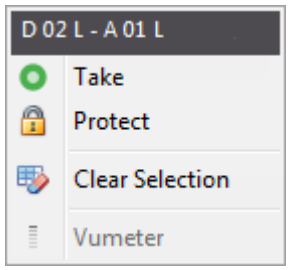
If you do the same over an output, its active connections will also appear, that is, the inputs connected to that output.

In both cases a maximum of 5 signals is shown: if there are more than 5 signals suspension points will be shown.



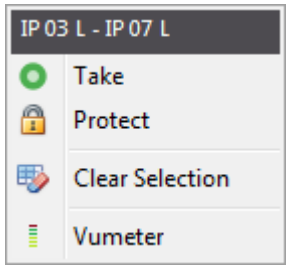
5.3. Actions on lines or crosspoints. Context menus.

In the XY view you can use the following context menu displayed by right-clicking the mouse over a **crosspoint:**

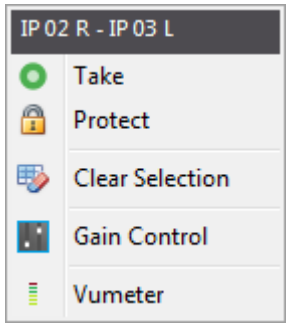


For each crosspoint you can "**Take**" (activate or deactivate) or "**Protect**" it and also see the physical connections (input connected to output). The "**Clear Selection**" option will clear any selection of crosspoints on the XY view.

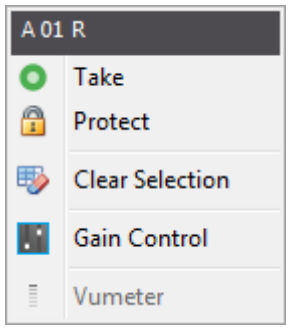
In the case of crosspoints **between Dante lines** the "**Vumeter**" option appears enabled so you can open the precision vumeters corresponding to the input and output associated to that crosspoint. This option also appears for crosspoints between a Dante line (input or output) and analog or digital line (output or input), but in this case only the precision vumeter of the Dante line will appear. See section 5.3.1 of this manual.



In the case of **NETBOX DSP** the "**Gain Control**" option appears so you can adjust the **crosspoint gain.** See section 5.3.2 of this manual.

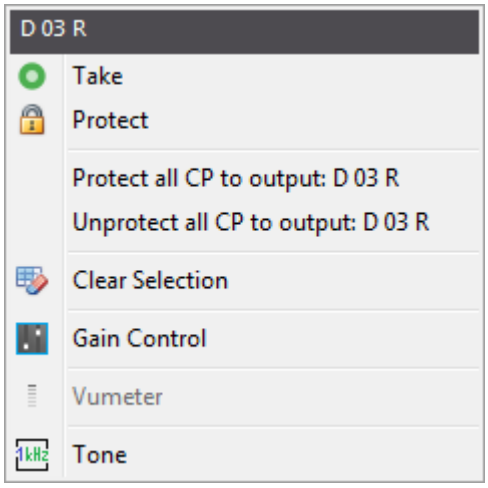


If you locate the mouse over the label of an **input line**, a right mouse click will open the following context menu:



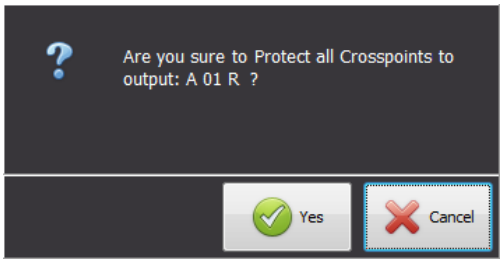
As you can see the "Gain Control" option has been added to the menu and has also changed the information concerning the physical connection - only indicating the name of the input/output line. The remainder actions are intact i.e., you can still execute the "Take" action or "Protect" the selected crosspoints or clear the selected crosspoints.

If you locate the mouse over the label of an **output line**, a right mouse click will open the following context menu:

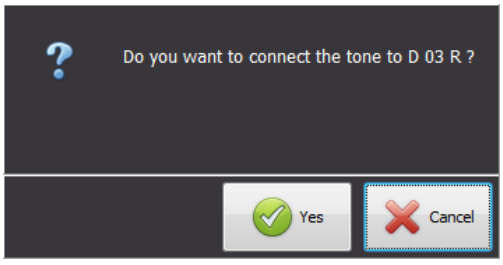


As you can see the same options are shown and 3 new actions appear. The first 2 ("**Protect all CP to output**" and "**Unprotect all CP to output**") allow to protect or unprotect all the established crosspoints towards that output.

For both actions, confirmation is required:

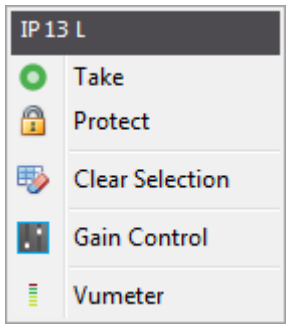


The "**Tone**" option allows to activate the routing of the **test tone** of **1KHz** and **+4dBu** (-20dBFS) towards the output line. Confirmation is required:

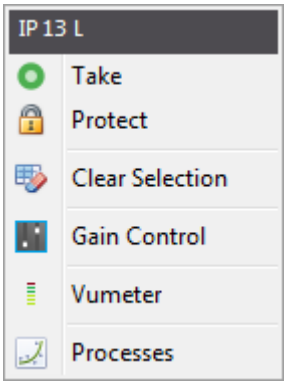


In case that this routing is already active, the option appears as "**Tone OFF**" and allows to deactivate it. Confirmation is also required.

In case that **input or output line** is a **Dante** line the "**Vumeter**" option also appears enabled so you can open the associated precision vumeter. See section 5.3.1 of this manual.



In the case of **NETBOX DSP**, the context menu associated to input or output lines includes also the "**Processes**" option. See section 5.3.3 of this manual.



5.3.1. Precision vumeters.

As indicated in the previous section, the "**Vumeter**" option allows you to open the precision vumeter corresponding to the selected Dante input or output line. This option also appears enabled in the case of crosspoints where at least one of the lines is a Dante line.

Those vumeters look like this:

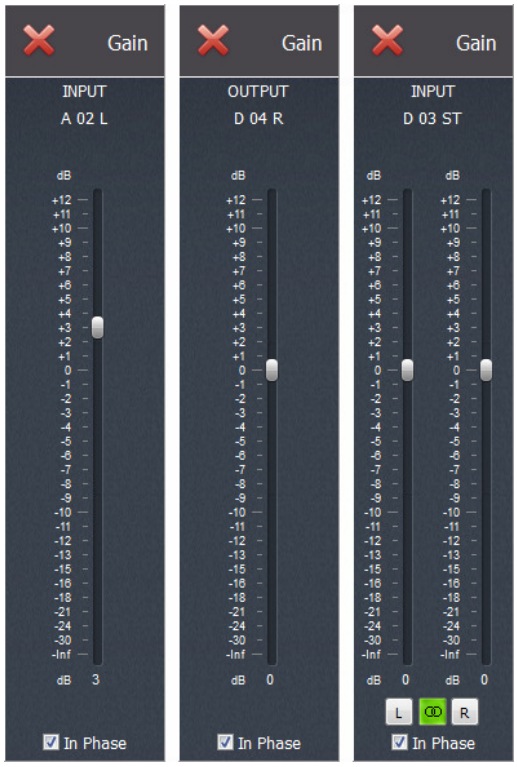
The upper part of the window indicates whether it's an input (I) or output line (O). Next, the line name is displayed. In the example, the first vumeter corresponds to the mono input line "IP 13 L" and the second one to the stereo output line "IP 02 ST".

5.3.2. Gain control.

As indicated in section 5.3, the "**Gain Control**" option allows you to set the individual gain for each of the NETBOX inputs, outputs and, only in the case of NEBOX DSP, crosspoints. The adjustments is accomplished by selecting the "Gain Control" in the contextual menu and then acting on the gain with the Fader tool that will pop-up. Digital gain adjustment is from -infinite to +12dB. It's also possible to inverse the **phase** of the signal if so required.

The gain of an input or output line can be adjusted even if these are not part of a crosspoint.

In the case of **analog and digital inputs and outputs** (NETBOX 32/8 AD MX) and **crosspoints** (NETBOX DSP), the adjustment looks like this:



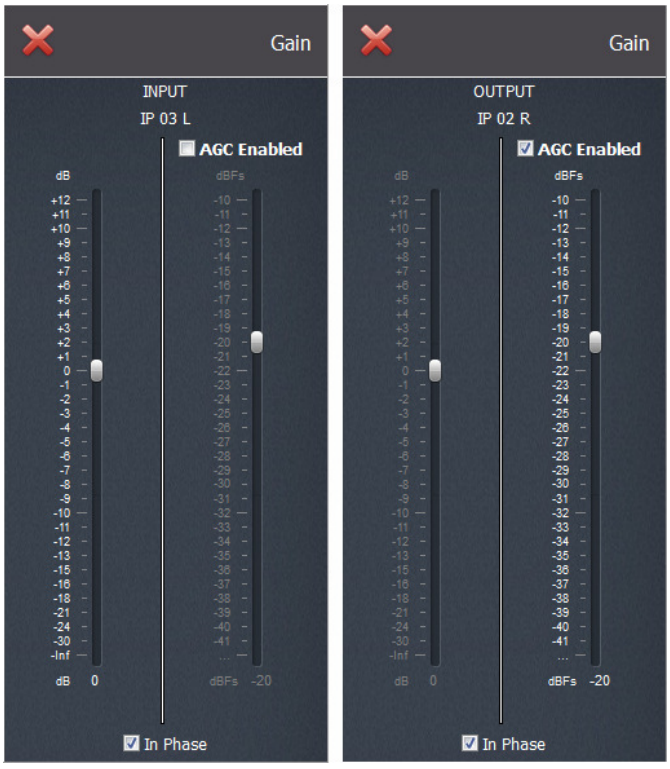
In the case of **stereo** lines the gain of each channel can be configured separately. In the lower part of this window 3 buttons are shown:

allows to align the R channel level with the L channel level.

allows to link (or not) both channels in order to have (or not) a synchronous movement of both faders.

allows to align the L channel level with the R channel level.

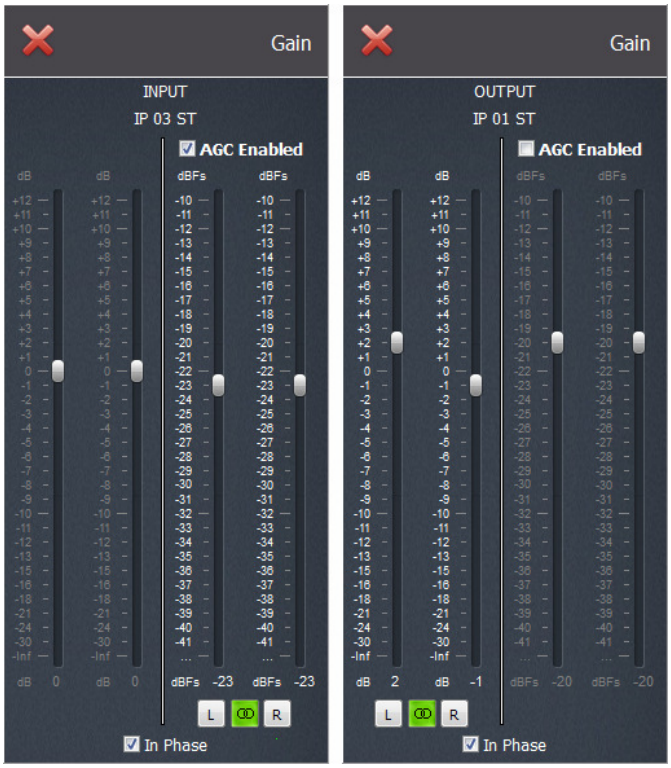
In the case of **AoIP inputs and outputs**, the adjustment looks like this:



In this case you can configure manually the gain (default option) or establish an **Automatic Gain Control** (by activating the "**AGC Enabled**" tic-box).

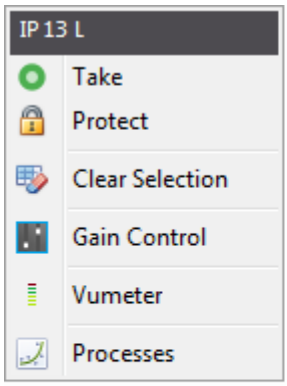
By default, AGC is configured as -20dBFs.

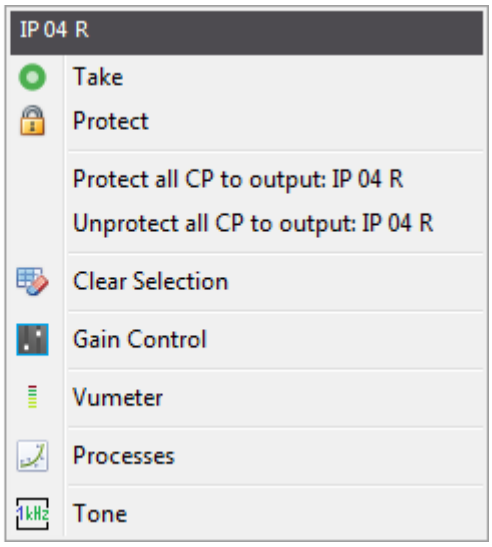
In the case of **stereo AoIP inputs and outputs**, the adjustment looks like this:



5.3.3. Processes.

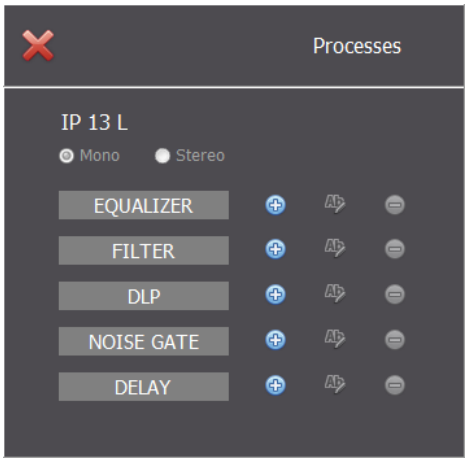
As indicated in section 5.3, only in the case of **NETBOX DSP**, the context menu displayed by right-clicking the mouse over an **input or output line** includes also the "**Processes**" option.



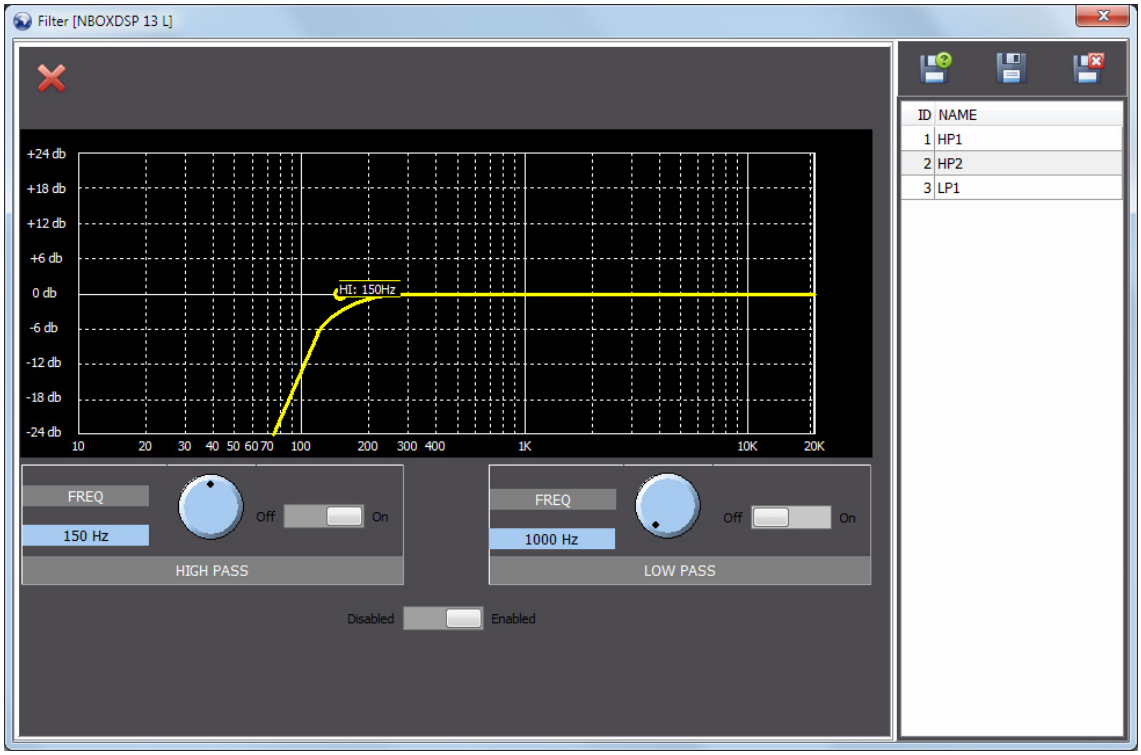


When you select this option the following window will appear.

In the upper part of the window the short name of the line is shown as well as it's mono or stereo.



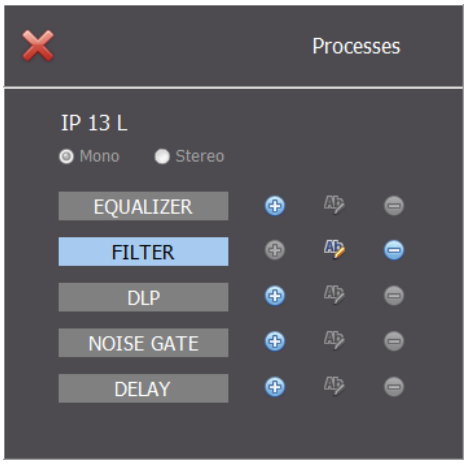
Next the available types of processes are shown (for more information, please consult section 6.6 of this manual). In order to activate a process just press the associated button and, in the window that appears, configure the parameters associated to the selected process (or select one of the **presets** previously configured by double-clicking one of them in the list on the right) and then select the "**Enabled**" option:



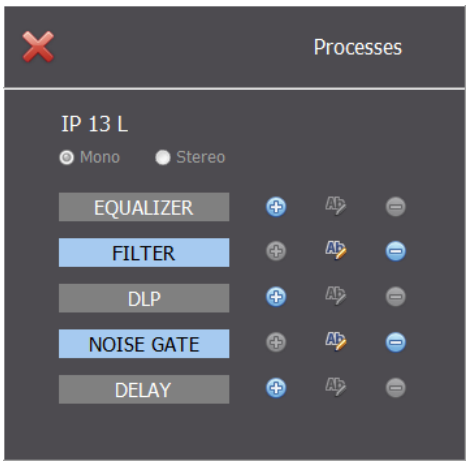
In the example, we press the button of "FILTER" option, then select preset 2 ("HP2") and activate it by means of the button

From this window, you can also create, delete and modify presets, by means of the buttons that appear up on the right. The first one allows to save configuration into a new preset, the second one allows to overwrite the selected preset (confirmation is required) and the third one allows to delete the selected preset (confirmation is also required).

When the window is closed by means of the button, the activation of that filter is shown in the "Processes" window:

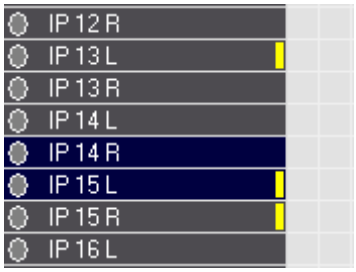


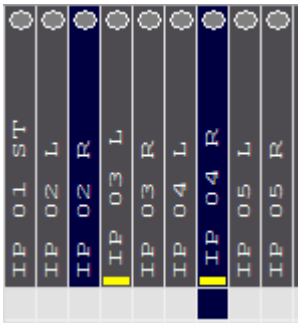
You can activate several processes in the same line. It will be reflected the same way in the "Processes" window.



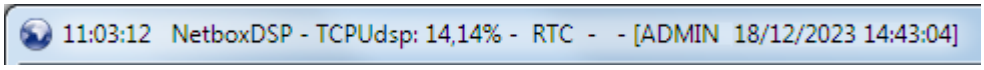
An active process can be edited by means of the button or deactivated by means of the button.

On the other hand, in the XY view the label of the input or output lines where a **process** has been activated shows a yellow indicator of active process:





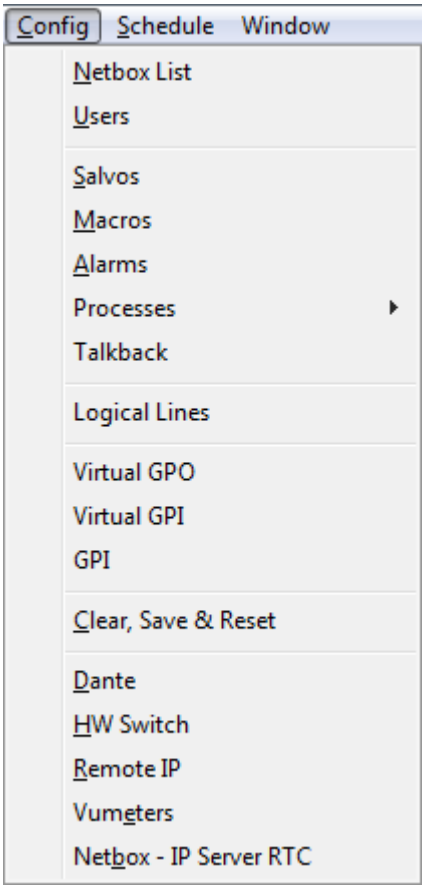
Besides, in the upper bar of the application, the DSP module percentage of use is shown.



6. DROP-DOWN MENU: Config.

The options that are available in the "Config" drop-down menu are described in the following sections.

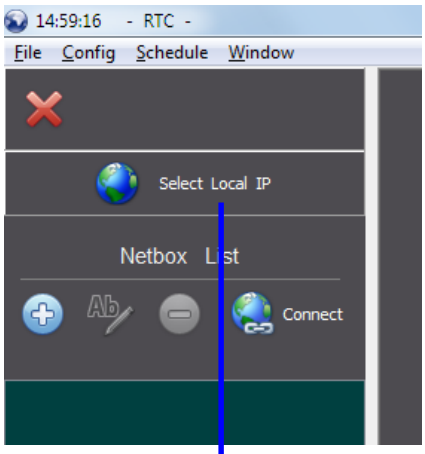
The "Processes" option is available only for NETBOX DSP.



6.1. Netbox List.

The first thing that needs to be accomplished once the "Netbox RTC" application has been installed, is to connect to the NETBOX that you want to control and operate. To do so, click on "Config" drop-down menu and select the "Netbox List" option.

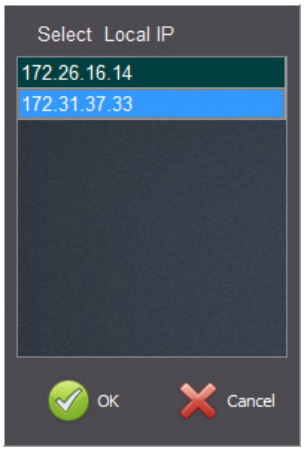
The following window will open to the left:

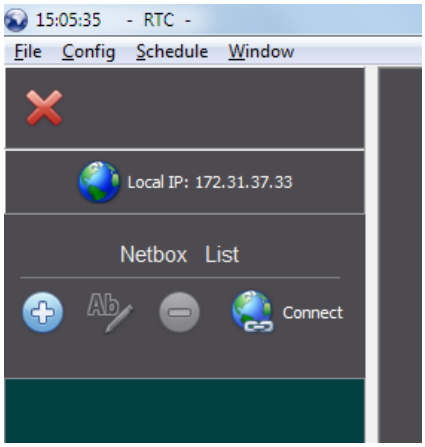


Firstly you must press the button with the Earth icon ("**Select Local IP**") in order to select the IP address corresponding to the PC where the application is installed.

When you press this button a window will open that reflects several IP addresses, if you have configured our Network Interface or NIC with several IP addresses in different ranges.

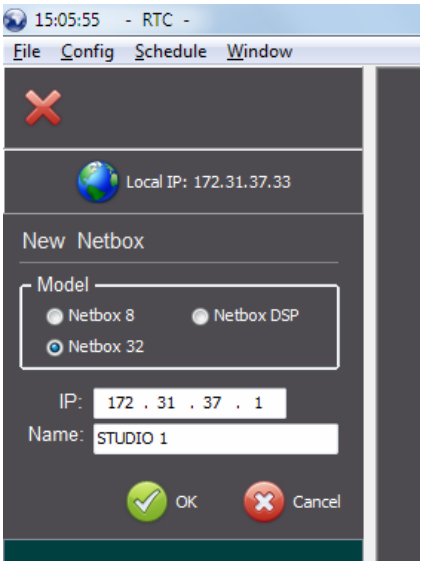
When you select the IP address that would be appropriate and in the same range as the device that you want to control with the application and confirm the selection by clicking on the "OK" button, that IP address will be shown in the previous button.



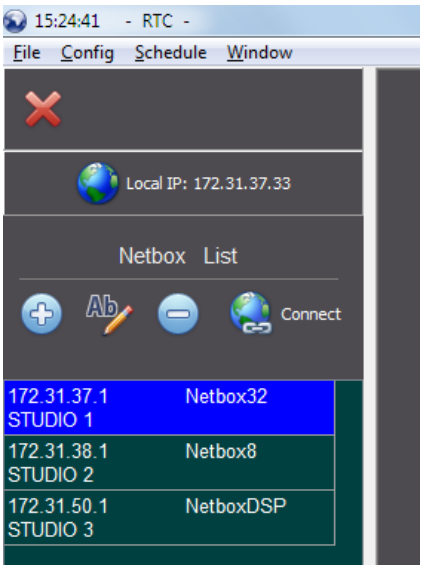


Below the "Local IP" button you will find the title "Netbox List" and immediately below this four buttons:

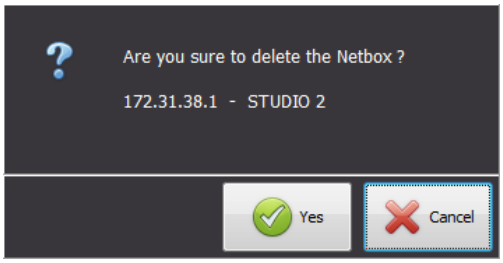
Allows to add a new NETBOX device to the "Netbox LIst". You must select the model (NETBOX 8, NETBOX 32 or NETBOX DSP), type in the IP Address of the device and give a name to it.



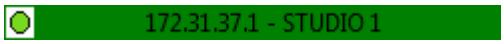
Allows to edit the selected device.



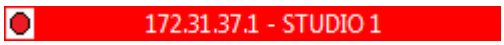
Allows to delete the selected device from the "Netbox List". The user will be prompted to confirm the deletion. All the device data will be lost (including salvos, macros, schedules...).

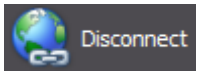


Allows to connect to the selected device. When the control computer NIC and the device are in the same range of IP addresses, the unit will connect to the control software, server application and database and signal to the software that the connection is established. At the footer of the "NetboxRTC" application the NETBOX device that is connected is indicated.



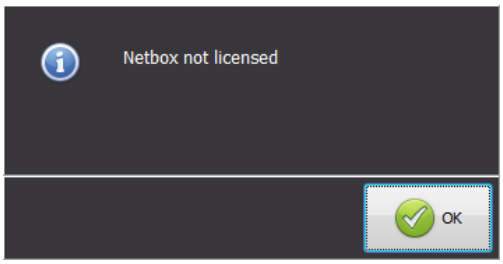
When there is no connection to the selected device it appears in red.





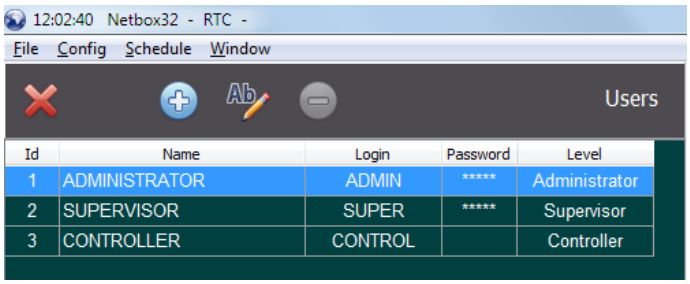
The previous button changes to "Disconnect" and allows to disconnect the selected NETBOX device in order to select another one and connect to it.

If you connect to a device without the corresponding **license** activated (see section 2.1 of this manual), an informative screen will appear when you try to access to any of the other options of the drop-down menus.



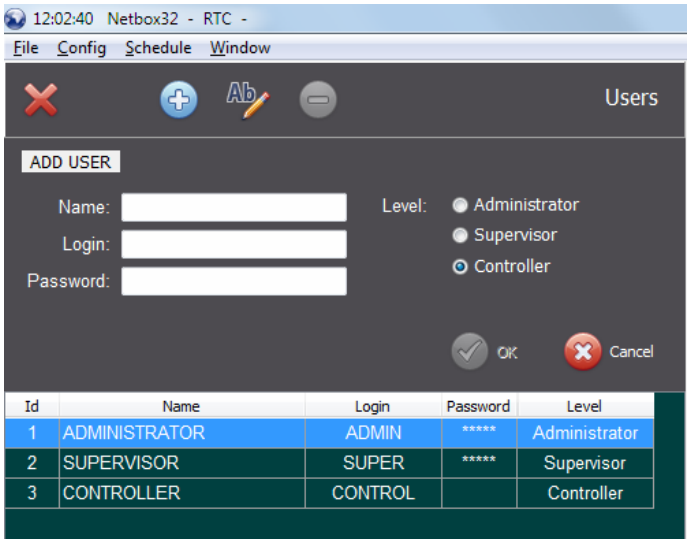
6.2. Users.

The "NetboxRTC" application provides different levels of users to be implemented to control the NETBOX matrix. There are three levels of operators or users: Administrator, Supervisor and Controller.



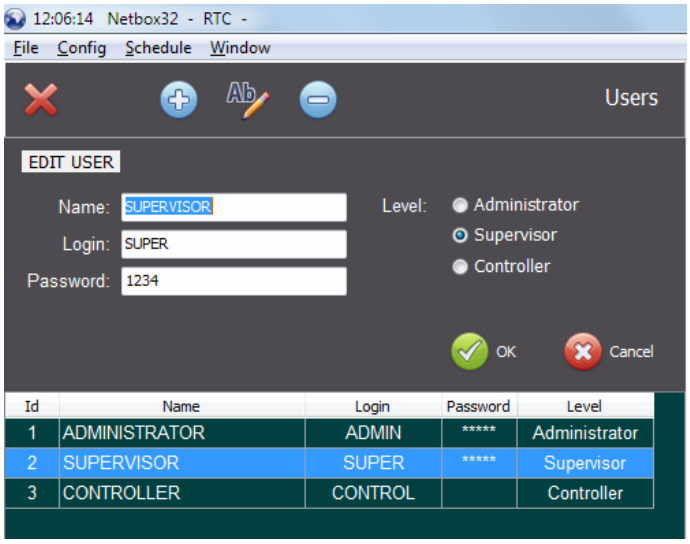
When clicking on the button you are opening the "ADD USER" dialogue/edit box:

You can now provide the "Name" of the user, the "Login" name and a "Password". Also, you can decide the level of the user to the right of these fields and as indicated above. When the creation of the new user has concluded you click on OK to confirm or Cancel to abort the adding of the new user.

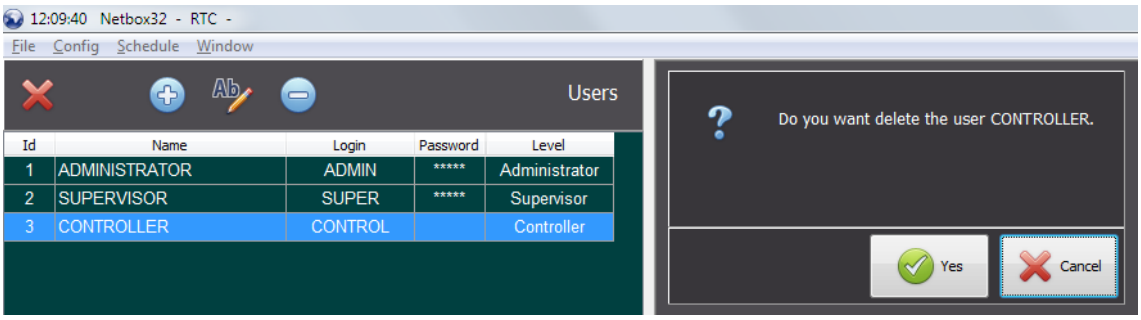


The button will open the "EDIT USER" dialog/edit box, identical to the previous one:

You can now edit the "Name" of the user, the "Login" name and modify the "Password". Also, you can change the level of the user to the right of these fields and as indicated above. When the editing has concluded you click on OK to confirm or Cancel to abort the editing the user.



If you select a user and click on the button you will be prompted to confirm that you want to delete the user selected:



Press "Yes" to proceed or "Cancel" to abort the deletion of the user.

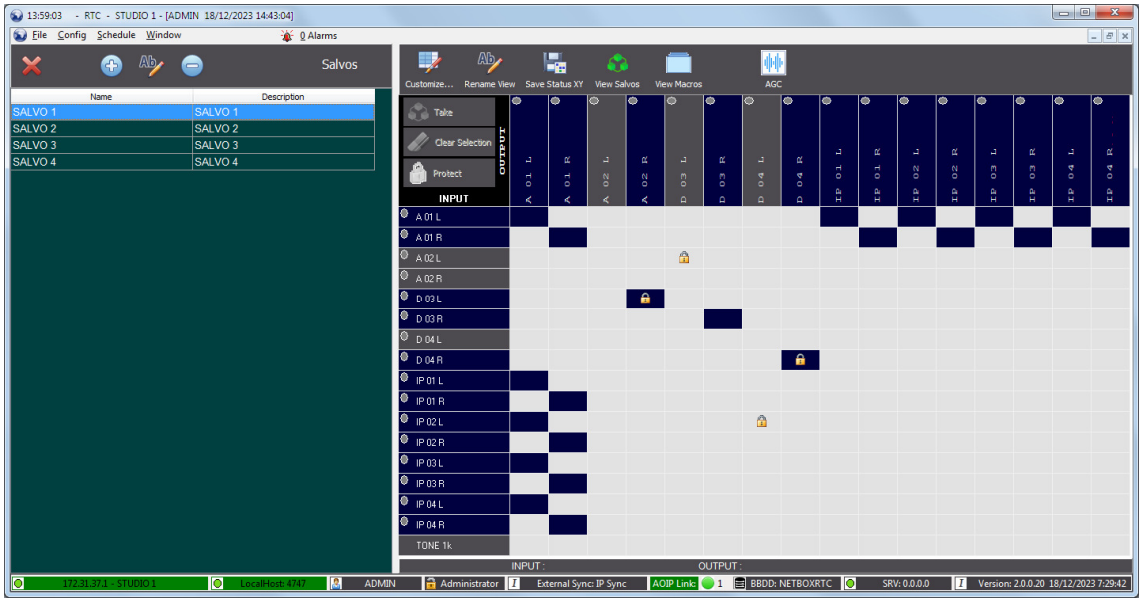
There are three levels of users: Administrator, Supervisor and Controller.

All actions are allowed for Administrators. Supervisors and Controllers have restricted access to certain menus or actions, that are shown in the following table:

Menu - Action	Supervisor	Controller
File		
New Logical XY View		✗
Customize		✗
Salvo Override		✗
Macro Override		✗
New Vumeters View		
Add Talkback		
Status XY		
All Connections View		✗
Salvo to View		
Close		
Close All		
Open Workspace		
Edit Workspace		
Save Workspace As...		✗
Log - Macro & Salvo		
Backup Database	✗	✗
Restore Database	✗	✗
Backup Matrix		✗
Exit		
Config		
Netbox List		
Add / Edit / Delete		✗
Local IP Configuration	✗	✗
Users		✗
Salvos		✗
Macros		✗
Alarms		✗
Processes		
Talkback		✗
Logical Lines		✗
Virtual GPO		✗
Virtual GPI		✗
GPI		✗
Clear, Save & Reset	✗	✗
HW Switch		✗
Remote IP	✗	✗
Vumeters	✗	✗
Netbox - IP Server RTC	✗	✗
Schedule		
Schedule		
Add / Edit / Run / Remove		✗
Holiday Calendar		✗

6.3. Salvos.

A salvo is a configuration of crosspoints or connections between inputs and outputs, pre-configured and that can be activated manually, through a schedule or through GPI's triggered by remote or local equipment or devices. When you click on "**Salvos**" in the "Config" drop-down menu, the list of salvos window will open:



The buttons available in the upper part of the list of salvos are the following:

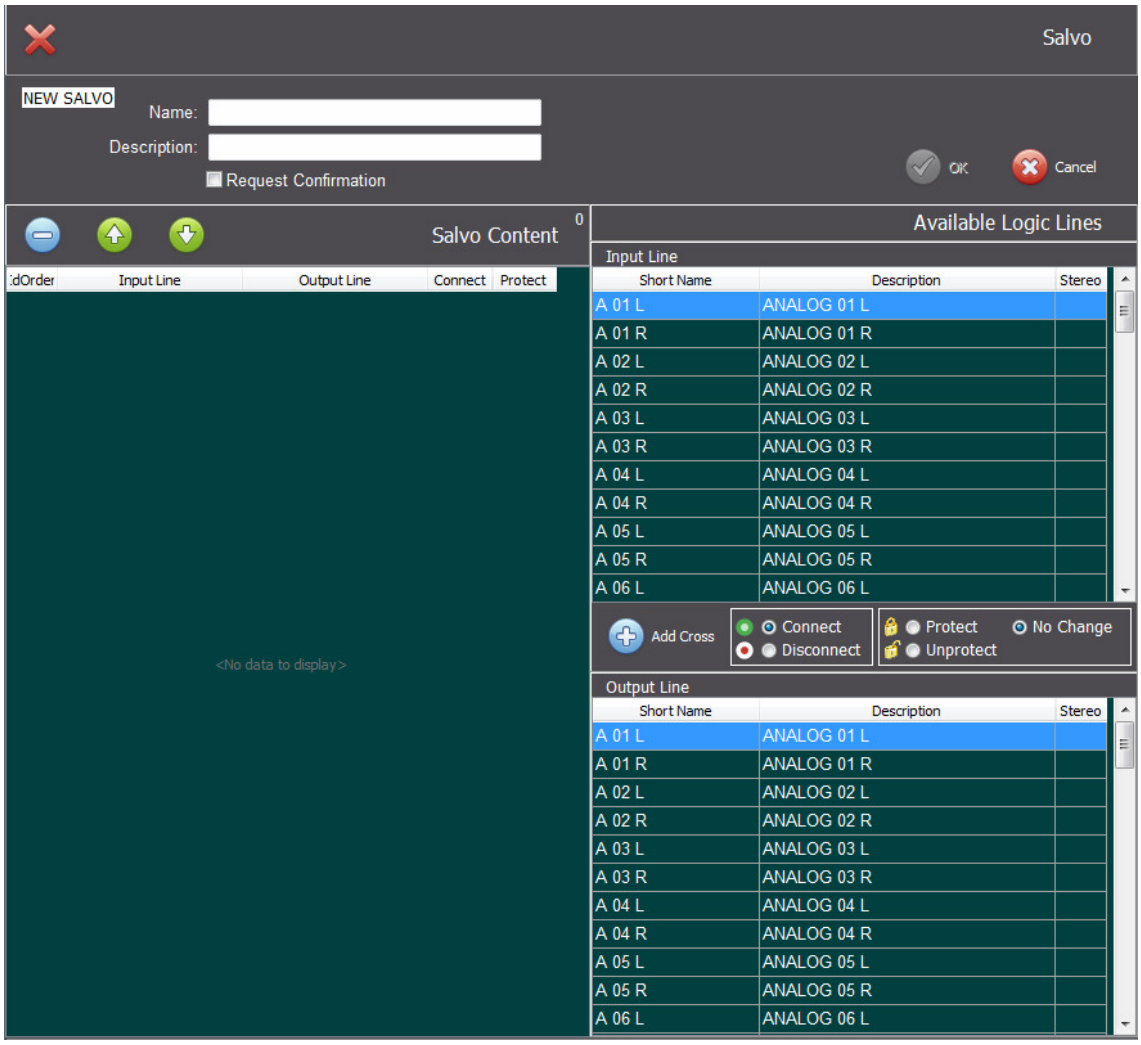
allows you to close the window.

allows you to create a new salvo.

allows you to edit an existing salvo.

allows you to delete the selected salvo from the list. Confirmation is required.

If no salvo has been created yet, the list will be empty. Click on the button and the window for creating salvos will appear (this is the same window that will allow you to edit an existing salvo). From the upper left part of the window from left to right and going downwards, the following fields and controls are available:



allows you to close the "NEW SALVO" / "EDIT SALVO" window.

Below this area, the salvo information area will appear:

- "**NEW SALVO**" or "**EDIT SALVO**" field indicates if you are creating a new salvo or editing an existing one.
- "**Name":** field used to name the salvo alphanumerically.
- "**Description":** field allowing us to provide a brief description of the salvo.
- "**Request Confirmation":** this tic-box should be activated when you want any user to confirm the activation of a salvo. This will treat the salvo as **critical** and will prompt the user to confirm the activation or deactivation.

OK allows you to confirm the creation or edition of the salvo and save it to the database.

Cancel allows you to discard the creation or edition of the salvo to the database.

Following the salvo information the "**Salvo Content**" area will appear:

allows you to remove the crosspoint action from the salvo.

allows you to move the crosspoint action up in the sequence of the salvo.

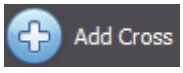
allows you to move the crosspoint action down in the sequence of the salvo.

Below these buttons the following columns will also appear:

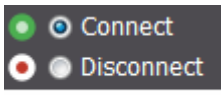
- "**IdOrder":** order of the action in the salvo.
- "**Input Line":** short name of the input line of the crosspoint.
- "**Output Line":** short name of the output line of the crosspoint.
- "**Connect":** type of connection: if green this means the action is to connect or generate the crosspoint; if red this means that the crosspoint will be deactivated or disconnected.
- "**Protect":** you can determine if a crosspoint action in the salvo should be protected from deactivation by locking or protecting this. This is shown with a closed padlock. If the salvo action includes the unprotect of this crosspoint, this is shown with an open padlock.

To the right of the "Salvo Content" area you will find the "Available Logic Lines" that you can use for matrix crosspoints. The upper part of this section contains the input lines and the lower part the output lines. The list of input and output lines is organised in three columns with the following headings: "Short Name" of the input/output line, "Description" and "Stereo" (indicates with a stereo symbol if the line has been configured as stereo).

Between these sections, the possible crosspoint actions are located:

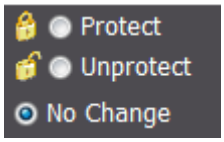


Allows you to add the salvo action for the crosspoint between the selected input line and output line. The different options for the action are:



Connects the input line to the output line selected.

Disconnects the input line from the output line selected.

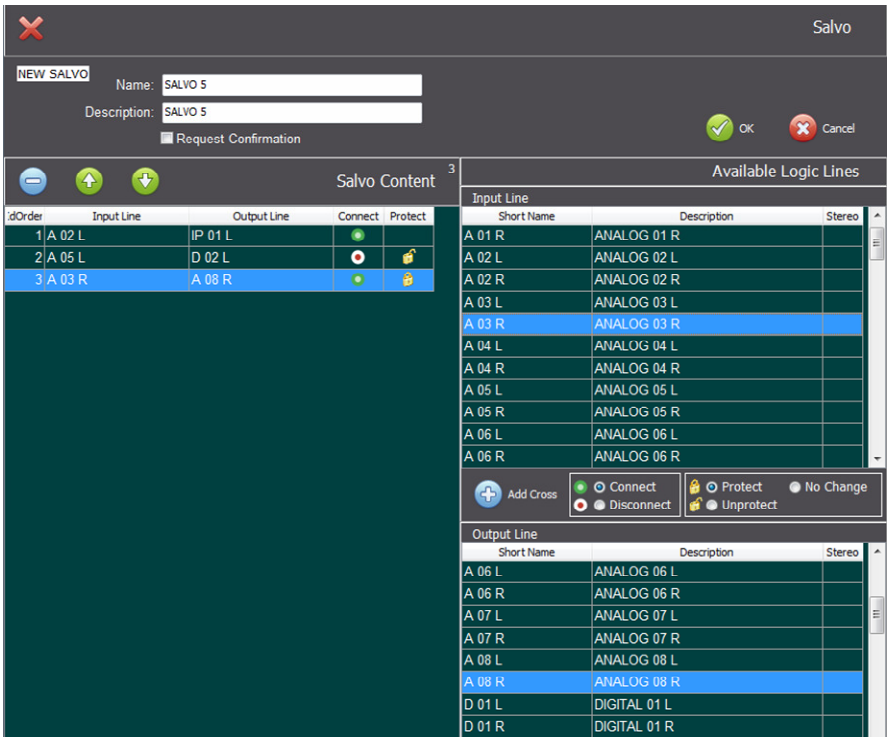


Marks the crosspoint as protected.

Un-marks the crosspoint as protected.

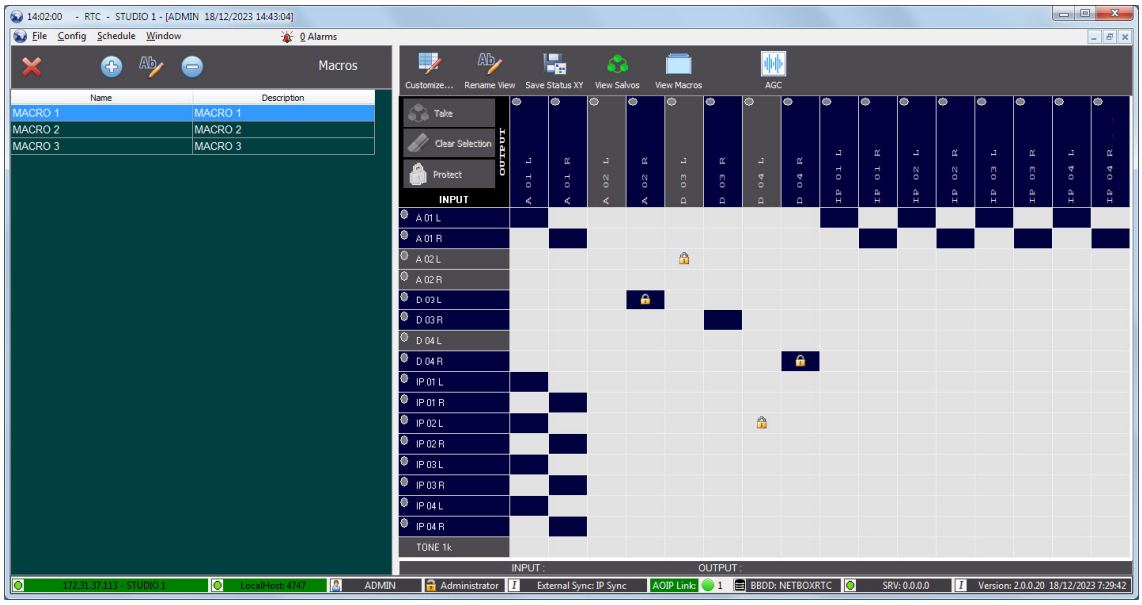
Will make no change in regards to the protection of the crosspoint.

In the example we have made a crosspoint (connection) between the "A 02 L" input line and the "IP 01 L" AoIP output line (no change in regards to the protection of this crosspoint will be taken when you execute the salvo), a disconnection and unprotect of the crosspoint between the "A 05 L" input line and the "D 02 L" output line and a connection and protect of the crosspoint between the "A 03 R" input line and the "A 08 R" output line.



6.4. Macros.

A macro is an action or a set of several actions that can include already generated salvos, activation or deactivation of independent crosspoints or the resulting combination of both, and the configuration of GPO's that can trigger specific actions related to external equipment or devices. A macro is a very useful tool when using the scheduler and when automating a matrix operation. When you click on "**Macros**" in the "Config" drop-down menu, the list of macros window will open:



The buttons available in the upper part of the list of macros are the following:

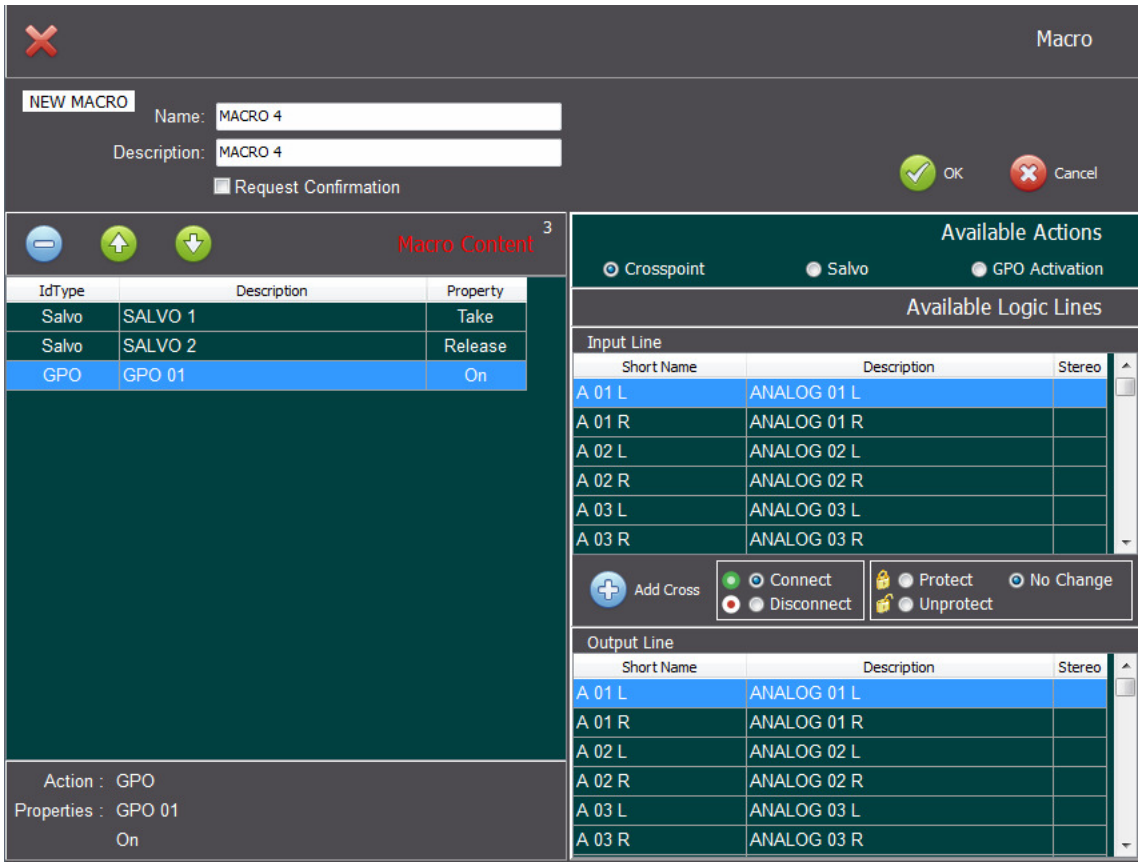
allows you to close the window.

allows you to create a new macro.

allows you to edit an existing macro.

allows you to delete the selected macro from the list. Confirmation is required.

If no salvo has been created yet, the list will be empty. Click on the button and the window for creating macros will appear (this is the same window that will allow you to edit an existing macro). From the upper left part of the window from left to right and going downwards, the following fields and controls are available:



allows you to close the "NEW MACRO" / "EDIT MACRO" window.

Below this area, the macro information area will appear:

- "**NEW MACRO**" or "**EDIT MACRO**" field indicates if you are creating a new macro or editing an existing one.
- "**Name":** field used to name the macro alphanumerically.
- "**Description":** field allowing us to provide a brief description of the macro.
- "**Request Confirmation":** this tic-box should be activated when you want any user to confirm the activation of a macro. This will treat the macro as **critical** and will prompt the user to confirm the activation or deactivation.

OK allows you to confirm the creation or edition of the macro and save it to the database.

Cancel allows you to discard the creation or edition of the macro to the database.

Following the macro information the "Macro Content" area will appear:

allows you to remove the action from the macro actions list.

allows you to move the action up in the sequence of the macro.

allows you to move the action down in the sequence of the macro.

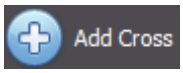
Below these buttons the following columns will also appear:

- "**IdType":** identifies what type of action the event in the macro is: crosspoint, salvo or GPO.
- "**Description":** indicates the name of the action, whether is a crosspoint, a salvo or a GPO.
- "**Property":** provides information regarding the action in the macro - for example the release of a salvo, the connection or disconnection of input and output lines to conform a crosspoint or the activation or deactivation of a GPO.

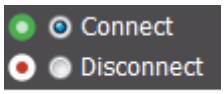
6.4.1. Programming a crosspoint in a macro.

To the right of the "Macro Content" area you will find the "Available Actions" field and three options: "Crosspoint", "Salvo" and "GPO Activation". If you select "**Crosspoint**", immediately below this field the "Available Logic Lines" that you can use for matrix crosspoints will appear. The upper part of this section contains the input lines and the lower part the output lines. The list of input and output lines is organised in three columns with the following headings: "Short Name" of the input/output line, "Description" and "Stereo" (indicates with a stereo symbol if the line has been configured as stereo).

Between these sections, the possible crosspoint actions are located:

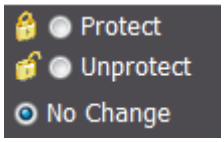


Allows you to add the macro action for the crosspoint between the selected input line and output line. The different options for the action are:



Connects the input line to the output line selected.

Disconnects the input line from the output line selected.



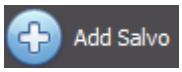
Marks the crosspoint as protected.

Un-marks the crosspoint as protected.

Will make no change in regards to the protection of the crosspoint.

6.4.2. Programming a salvo in a macro.

If you select "**Salvo**" in the "Available Actions" field, the "Select Salvo" dialogue box will appear:

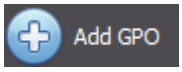
The 

button allows you to add the selected salvo from the "Select Salvo" drop-down list.

"Select Salvo" drop-down list will show the existing, already created salvos in a list that the user can select from. The "State" field has two options: "Take" or "Release",. i.e. to take the complete salvo in the list of actions defined in the macro or release the selected salvo in the list of actions defined in the macro.

6.4.3. Programming a GPO in a macro.

If you select "**GPO Activation**" in the "Available Actions" field, the "Select GPO" dialogue box will appear:

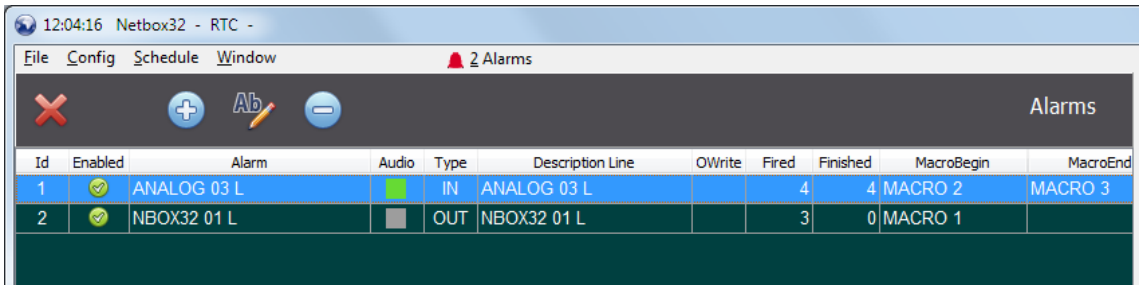
The 

button allows you to add the selected GPO from the "Select GPO" drop-down list.

"Select GPO" drop-down list will show the existing GPO's in a list that the user can select from. The "State" field has two options: "On" or "Off", i.e. the GPO will generate an On for the GPO contact or address in the list of actions defined in the macro or to turn the GPO Off in the list of actions defined in the macro.

6.5. Alarms.

The "**Alarms**" option of the "Config" drop-down menu allows to configure system alarms that will be activated when there is no signal in a line or when that signal returns. When you select this option the following window will appear:



The buttons available in the upper part of the window are the following:

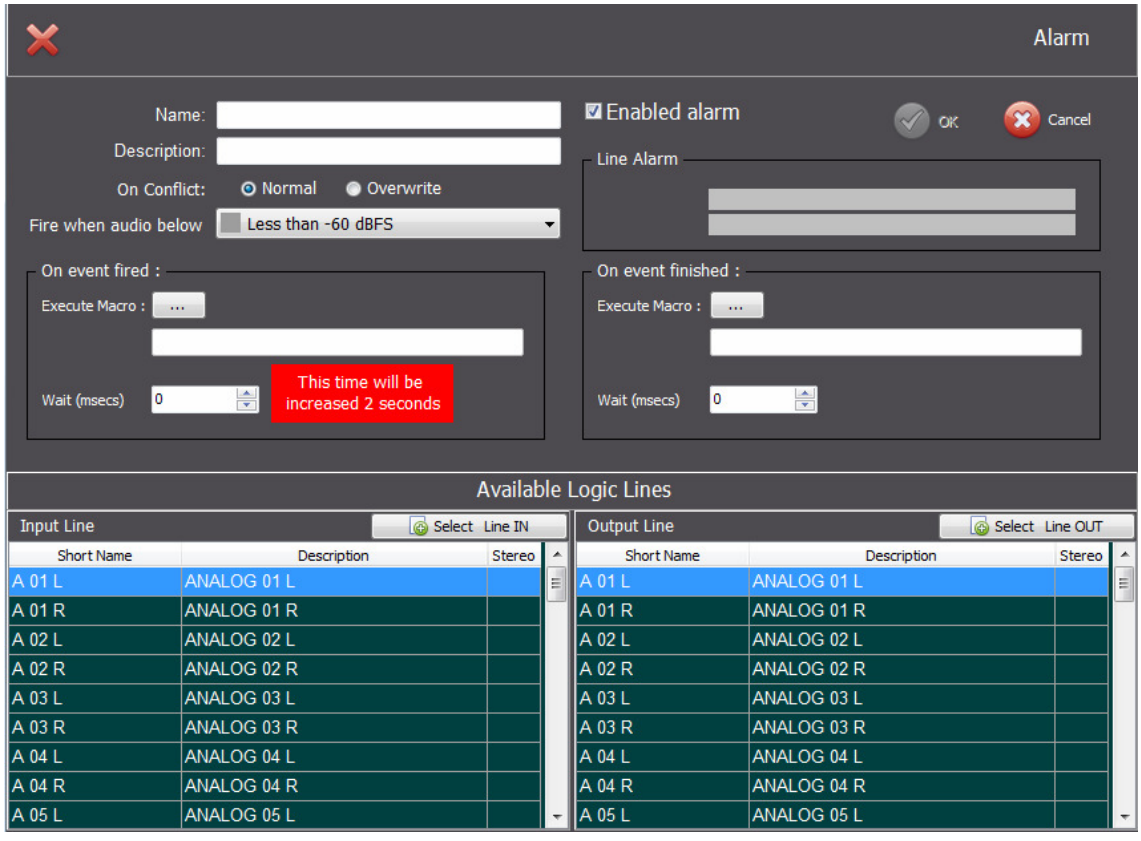
allows you to close the window.

allows you to create a new alarm.

allows you to edit an existing alarm.

allows you to delete the selected alarm from the list. Confirmation is required.

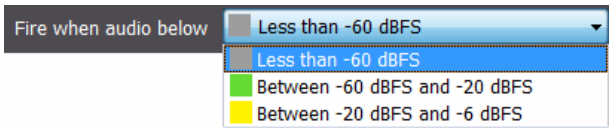
If no alarm has been created yet, the list will be empty. Click on the button and the window for creating alarms will appear (this is the same window that will allow you to edit an existing alarm).



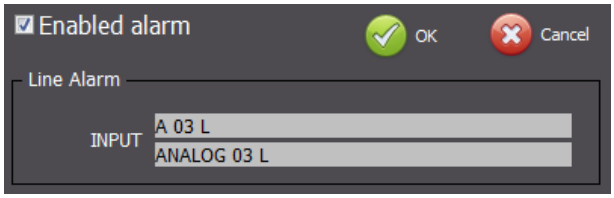
The available fields are the following:

- "**Name":** field used to name the alarm alphanumerically.
- "**Description":** field allowing us to provide a brief description of the alarm.
- "**On Conflict":** allows you to choose what to do in case of conflict (when there are protected crosspoint). You can select a "Normal" execution or an "Overwrite" (make an execution opening the pad-locks).
- "**Fire when audio below":** drop-down field that allows you to choose when the alarm will be fired depending on the level in dBFS of the selected line audio. There are 3 options with 3 associated colors:
  - Grey: less than -60dBFS
  - Green: between -60 and -20dBFS
  - Yellow: between -20 and -6dBFS

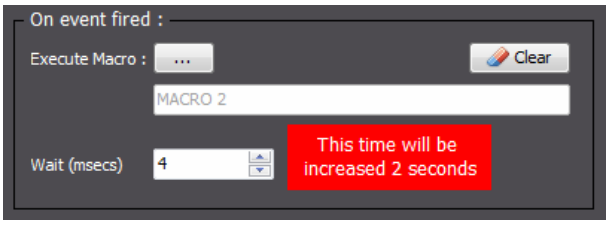
  NOTE: 0dBFS = +24dBu



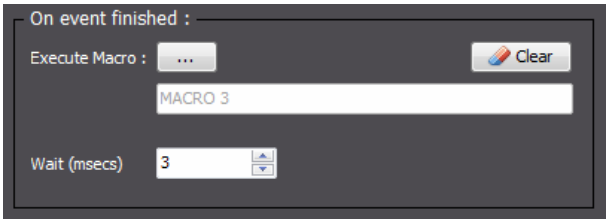
- "**Enabled alarm":** this tic-box allows the user to choose whether the alarm is initially enabled or not.
- "**Line Alarm":** this filed shows the line the alarm is applied to. It indicates if it's an input or output line, as well as the line short name and description.



- "**On event fired":** allows you to select the macro that will be executed on alarm fired and the wait time after the detection of the signal absence (the defined time will be increased by 2 seconds, as the absence of a signal on a line for less than 2 seconds should not be a reason to generate an alarm).



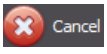
- "**On event finished":** allows you to select the macro that will be executed when alarm finishes and the wait time after the detection of the signal return.



In the lower part of this window you will find the list of input and output logical lines that are available in order to select the line the alarm will be associate to, by means of the "Select Line IN" or "Select Line OUT" button.

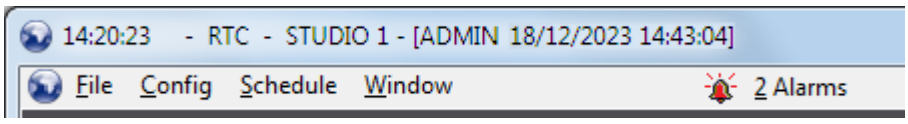
The

button, located in the upper right corner, allows you to confirm the creation or edition of the alarm and save it to the database.

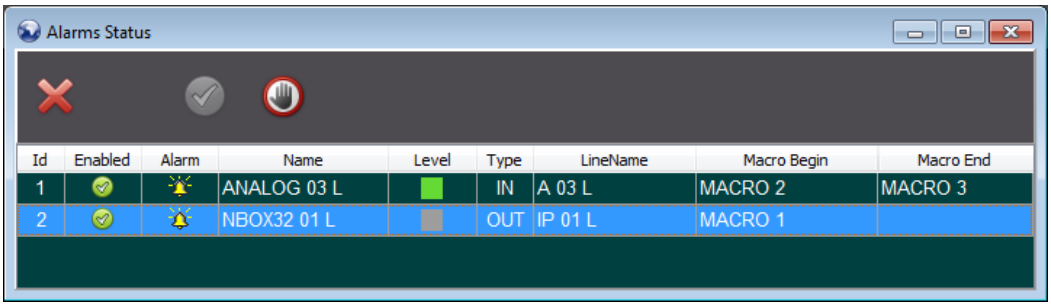
The 

button allows you to discard the creation or edition of the alarm.

In the upper bar of the application, beside a red bell icon, the number of active alarms is shown (this indication will appear only when the "Netbox Server" application is running).



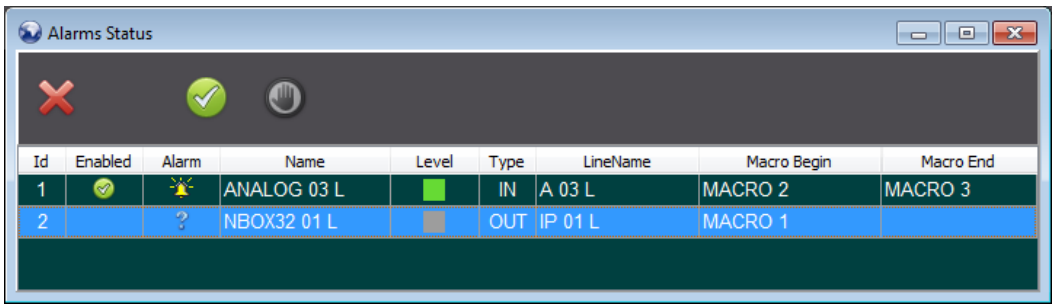
When you press this indication a window will open that shows the list of defined alarms as well as the status and information about them. Here you can activate or deactivate any alarm at any time.



In this window the following columns appear:

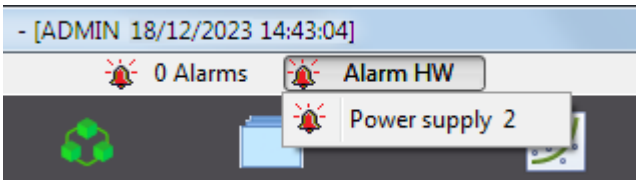
- "**Id**": alarm numerical identifier.
- "**Enabled**": indicates whether the alarm is enabled or not.
- "**Alarm**": indicate whether the alarm is fired or not.
- "**Name**": alarm name.
- "**Level**": indicates the configured level for the alarm to fire.
- "**Type**": indicates if the associated line is an input or an output.
- "**Line Name**": indicates the short name of the associated line.
- "**Macro Begin** ": indicates the macro that will be executed when the alarm is fired.
- "**Macro End**": indicates the macro that will be executed when the alarm is finished.

The button allows you to deactivate the selected alarm that changes to:



The button allows you to reactivate the alarm.

On the other hand, in the case of **NETBOX DSP** a second indicator of hardware alarms appears (this is independent of the "Netbox Server" application). When you press this indication a list will be unfolded that shows if there is any active hardware alarm in the unit:



The available alarms are:

- "**Power supply 1**": indicates that the main power supply is not present or fed or that it's damaged.
- "**Power supply 2**": indicates that the redundant power supply is not present or fed or that it's damaged.
- "**Link Lan Eth AoIP 1**": indicates that there is no connection in Ethernet LAN 1 port.
- "**Link Lan Eth AoIP 2**": indicates that there is no connection in Ethernet LAN 2 port.
- "**Temperature**": indicates that the unit is overheated.
- "**Fan**": indicates that the unit has reached the operating temperature that should starts the cooling fan but it didn't start because of being damaged or blocked.
- "**HW not configured**": indicate that the FPGA is not configured or it's resetting.

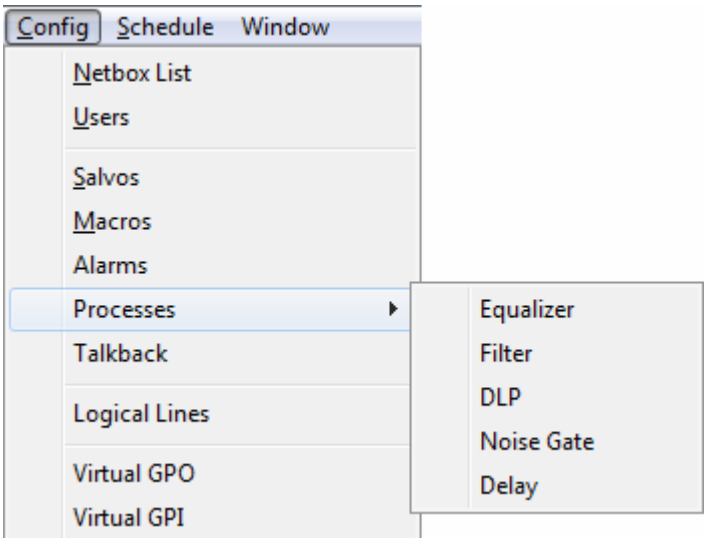
6.6. Processes.

The "**Processes**" option of the "Config" drop-down menu allows to create presets of the different processes types available in the system. This option is available only for **NETBOX DSP.**

Processes can be applied to a maximum of 64 selectable lines. These are the available processes:

- "**Equalizer":** Four-band parametric equalizer.
- "**Filter":** High pass, low pass and band pass filter.
- "**DLP" (Dynamic Level Processor):** 5-points compressor-limiter-expander.
- "**Noise Gate".**
- "**Delay".**

You can access to processes configuration through the drop-down menu that appears when the "Processes" option of the "Config" menu is selected:



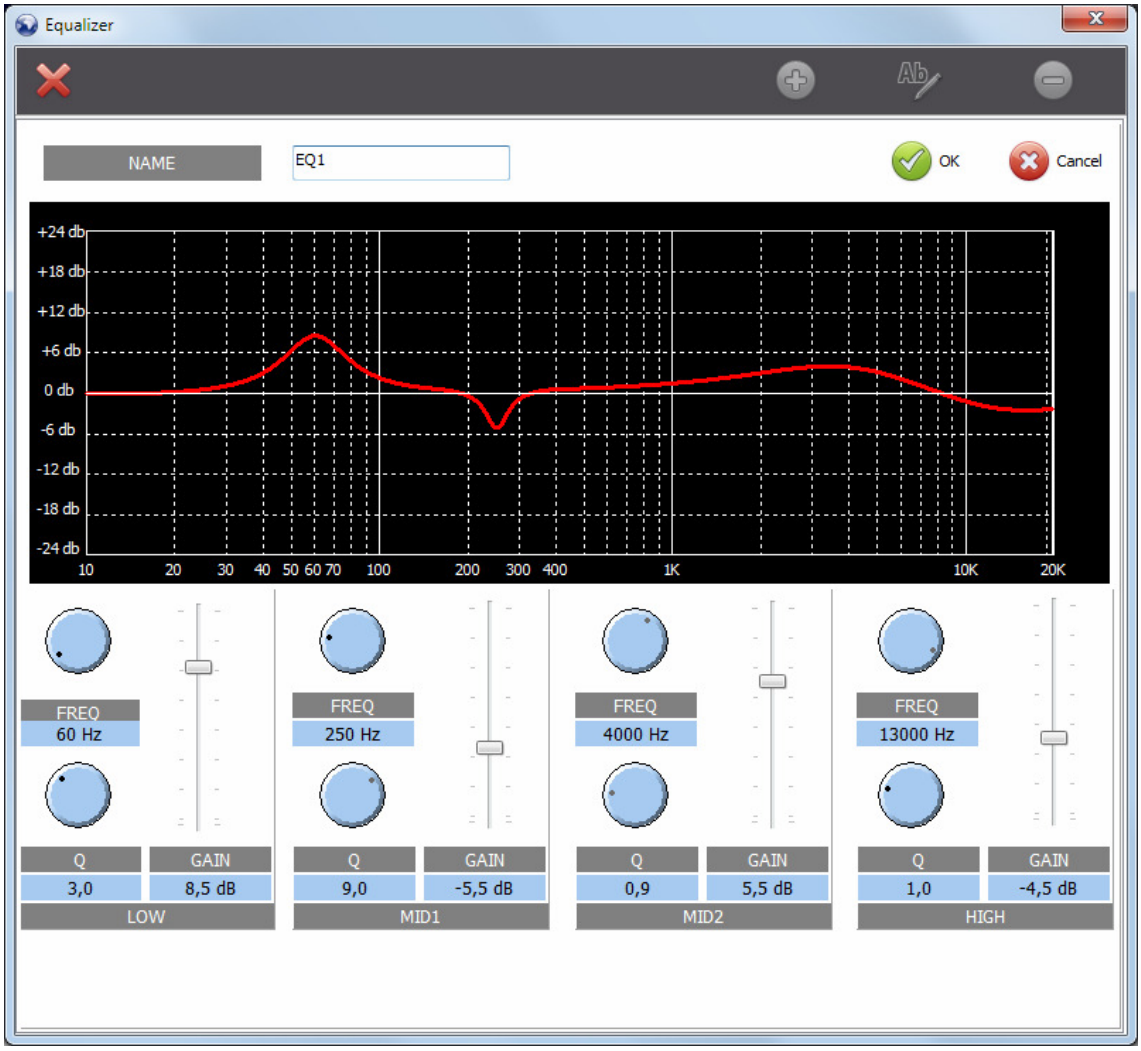
When clicking on one of the options, its presets editing window is presented, where these presets can be created, deleted or modified. If no preset has been created yet, the list will be empty.

As an example, here you can see the equalizer selection window, where new presets can be created by means of the button, deleted by means of the button or edited by means of the button.

Next, the adjustment windows for each of the different processes are described.

Equalizer.

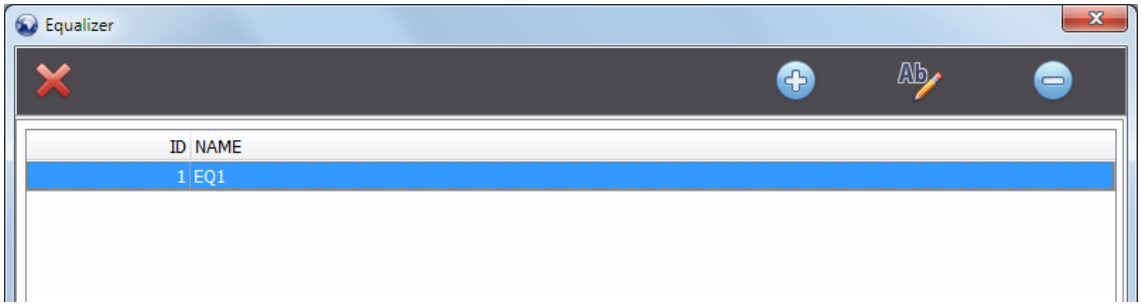
A four-band parametric equalizer is available.



An identifier name is typed in the "NAME" field. Afterwards, the adjustment of parameters for each of the four bands is performed:

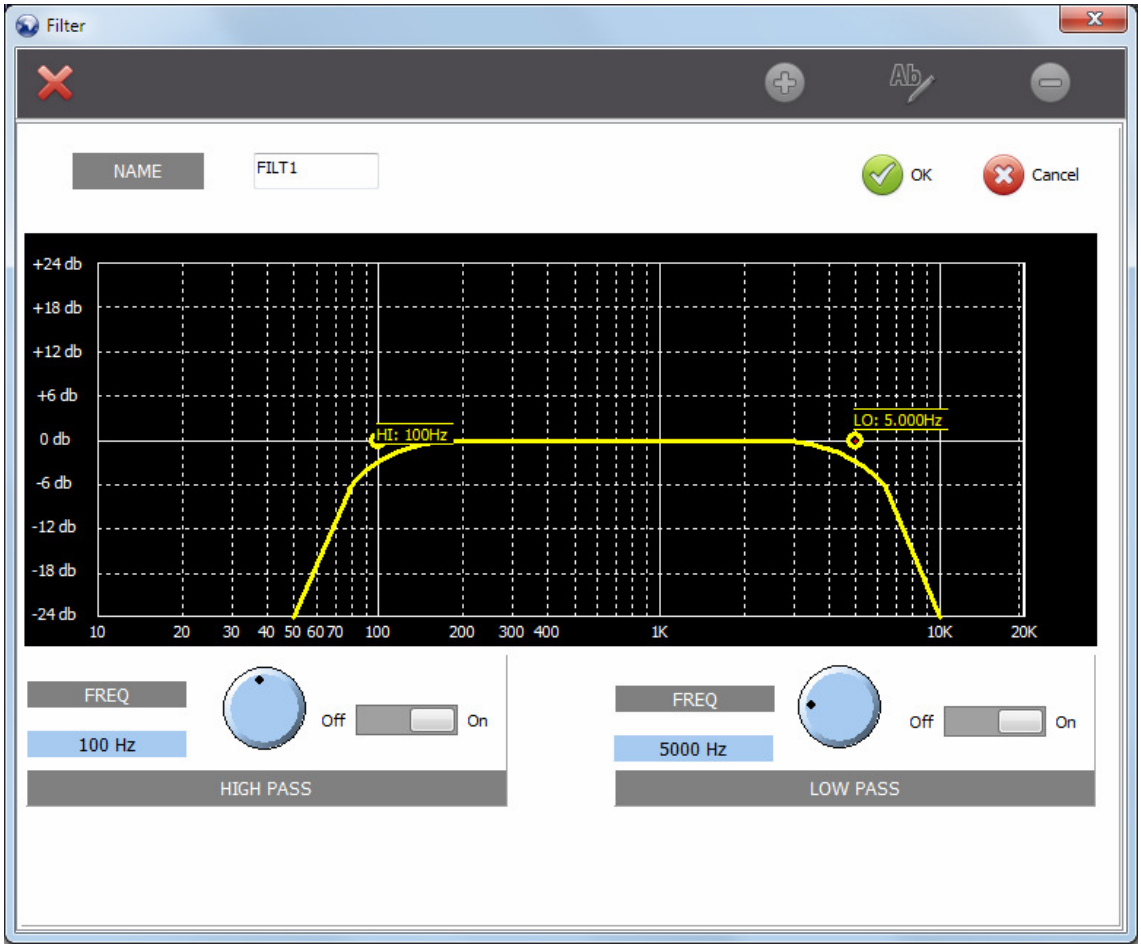
- "**FREQ**": Band's center frequency in a range from 10Hz to 20KHz.
- "**Q**": Quality factor of the band, adjustable from 0.1 to 20.
- "**GAIN**": Band's gain, in a range from -18dB to +18dB.

Once edition is finished, click on the OK button to validate the changes or click on the Cancel button to discard the changes. After confirmation, the edition window will close and the new preset will appear in the list.



Filter.

Low-pass and/or high-pass filter can be activated for the input, so only the desired range of frequencies passes to the system.



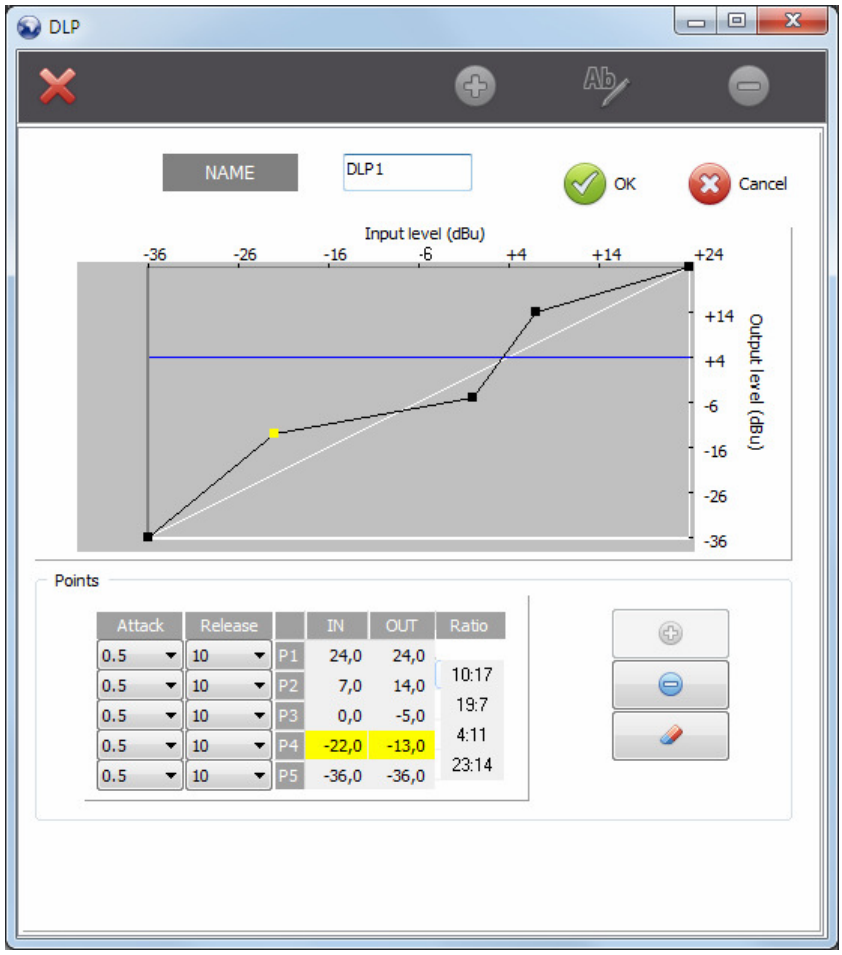
As for the rest of processes, an identifier name is given to the preset, and then its parameters are set. These are the parameters applicable for filters:

- "**HIGH PASS":** High-pass filter adjustments.
  - "**FREQ":** High-pass filter cutoff frequency, from 20Hz to 1KHz.
  - "**On/Off":** Activate or deactivate high-pass filter.
- "**LOW PASS":** Low-pass filter adjustments.
  - "**FREQ":** Low-pass filter cutoff frequency, from 1KHz to 20KHz.
  - "**On/Off":** Activate or deactivate low-pass filter.

Once edition is finished, click on the OK button to validate the changes or click on the Cancel button to discard the changes. After confirmation, the edition window will close and the new preset will appear in the list.

DLP (Dynamic Level Processor).

This window allows for the creation of a DLP (Dynamic Level Processor) adjustment preset, basing on compression and expansion, creating up to five points as required to control the voice dynamics.



As for the rest of processes, an identifier name is given to the preset, and then its parameters are set.

This option allows you to define up to 4 configurable sections for OUT/IN transfer function, each one of them working as compressor/expander or limiter with different ratios.

The graph shows the variation produced at the different levels of the signal. The input levels are plotted along the horizontal axis, while the output levels obtained are plotted along the vertical axis.

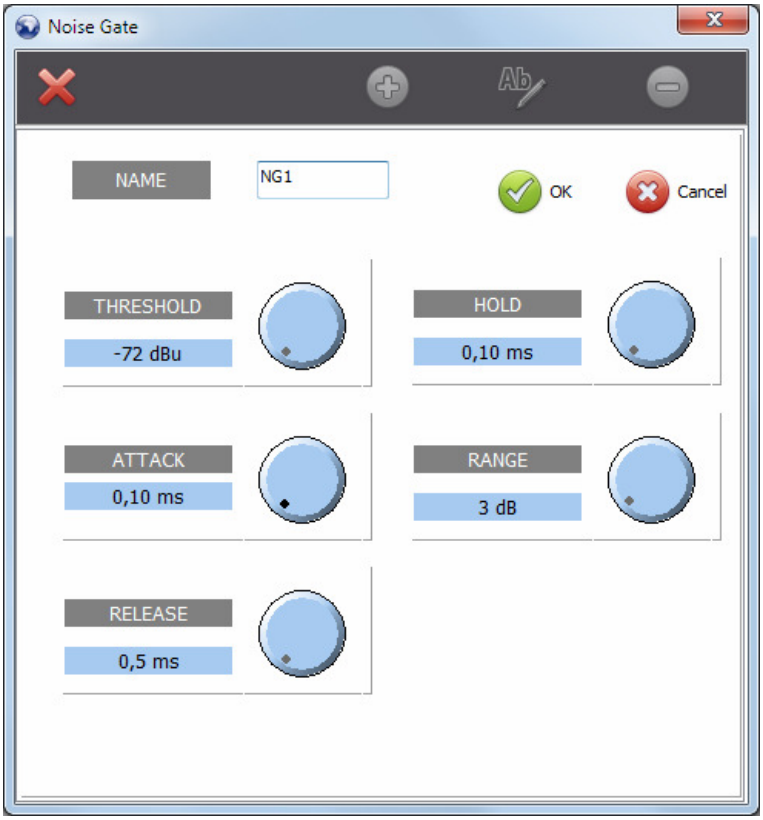
You can add up to 5 points to that graph (from “Points” box) in order to define up to 4 processing sections. By selecting and moving each one of those points you can modify the different compression ratios. You can also configure attack and release times.

Proper DLP configuration demands training and experience, and is beyond the scope of this manual.

Once edition is finished, click on the OK button to validate the changes or click on the Cancel button to discard the changes. After confirmation, the edition window will close and the new preset will appear in the list.

Noise Gate.

Noise gate is a special dynamics processor that attenuates or completely mutes audio below a certain level.



A name has to be assigned to each preset, which can be configured afterwards.

The parameters to adjust for a noise gate are as follows:

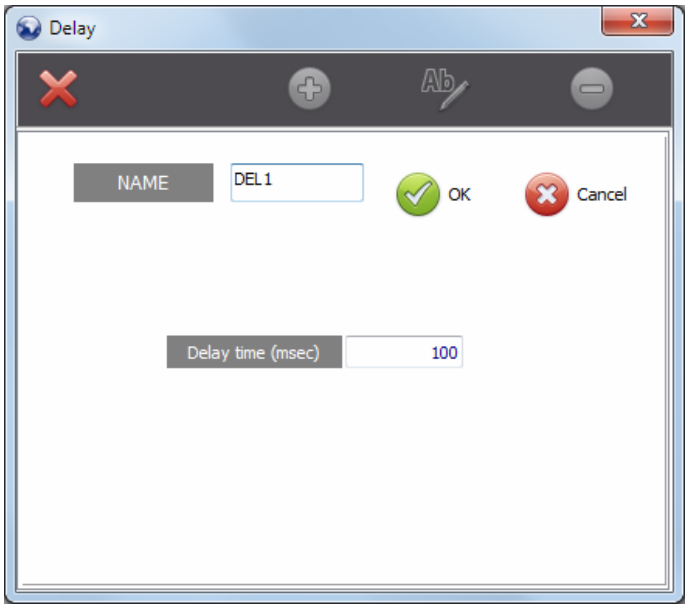
- "**THRESHOLD**": Nominal level below which the noise gate starts operating.
- "**ATTACK**": Attack time. The time it takes to start operating.
- "**RELEASE**": Release time, this is the time that the adjustment still operates for after receiving a sound impulse.
- "**HOLD**": Time that the effects still doesn't operate for, even in presence of a silence.
- "**RANGE**": Number of dB that the sound is attenuated when the processor is operating.

Once edition is finished, click on the OK button to validate the changes or click on the Cancel button to discard the changes. After confirmation, the edition window will close and the new preset will appear in the list.

Delay.

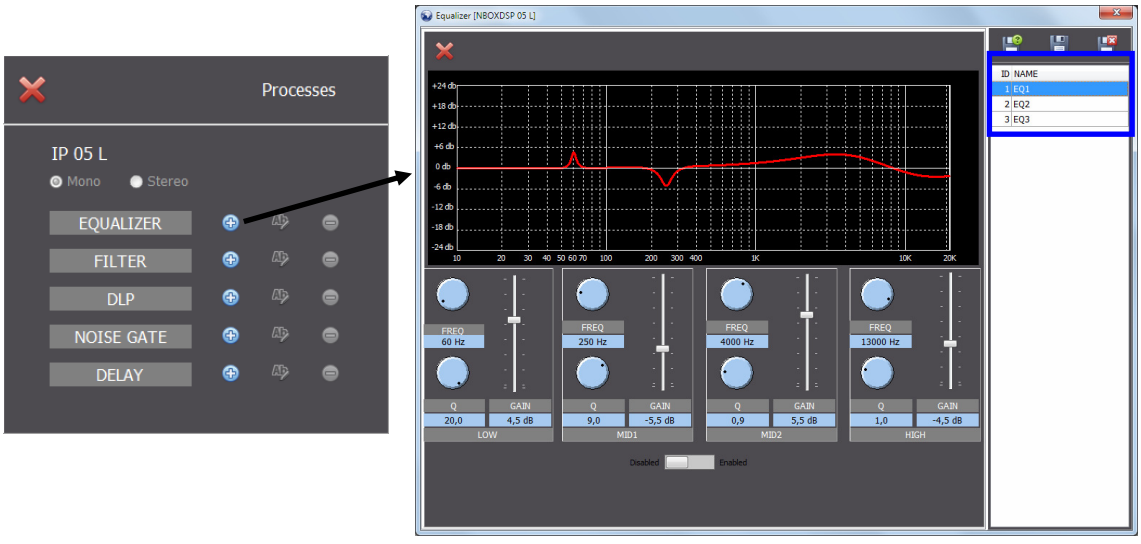
This process is a time delay of a signal.

In this case, an identifier name is also given to the preset, and then configure its only parameter ("**Delay time**") that allows to adjust the delay time in milliseconds.



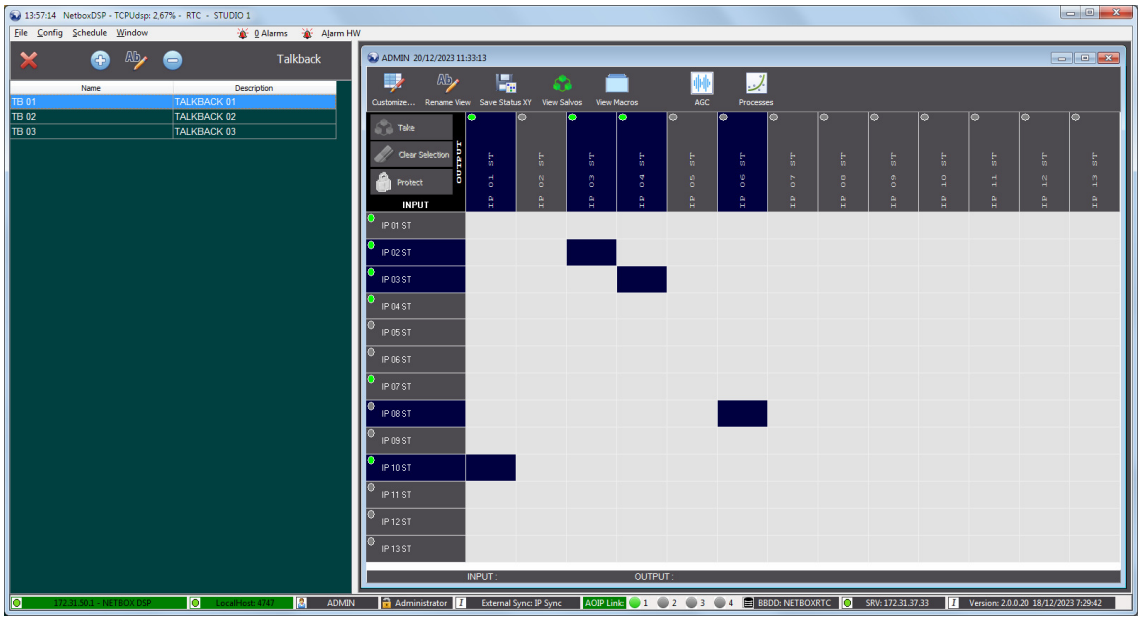
Once edition is finished, click on the OK button to validate the changes or click on the Cancel button to discard the changes. After confirmation, the edition window will close and the new preset will appear in the list.

Once the required presets are created, they will be available in the system and will appear when you try to activate a process in any of the input lines. From that window you can also create, delete and modify presets. See section 5.3.3 of this manual.



6.7. Talkback.

A "Talkback" group is a combination of input and output lines that allows to automatically execute a n-1 routing between those lines (additionally, the existing crosspoints towards the outputs lines are deleted and, in the case of a NETBOX DSP, the gains of all the crosspoints, activated or not, towards those inputs are restored to 0). When you click on "**Talkback**" option in the "Config" drop-down menu, the list of "Talkback" groups window will open, where they can be created, edited or deleted:



The buttons available in the upper part of the list of "Talkback" groups are the following:

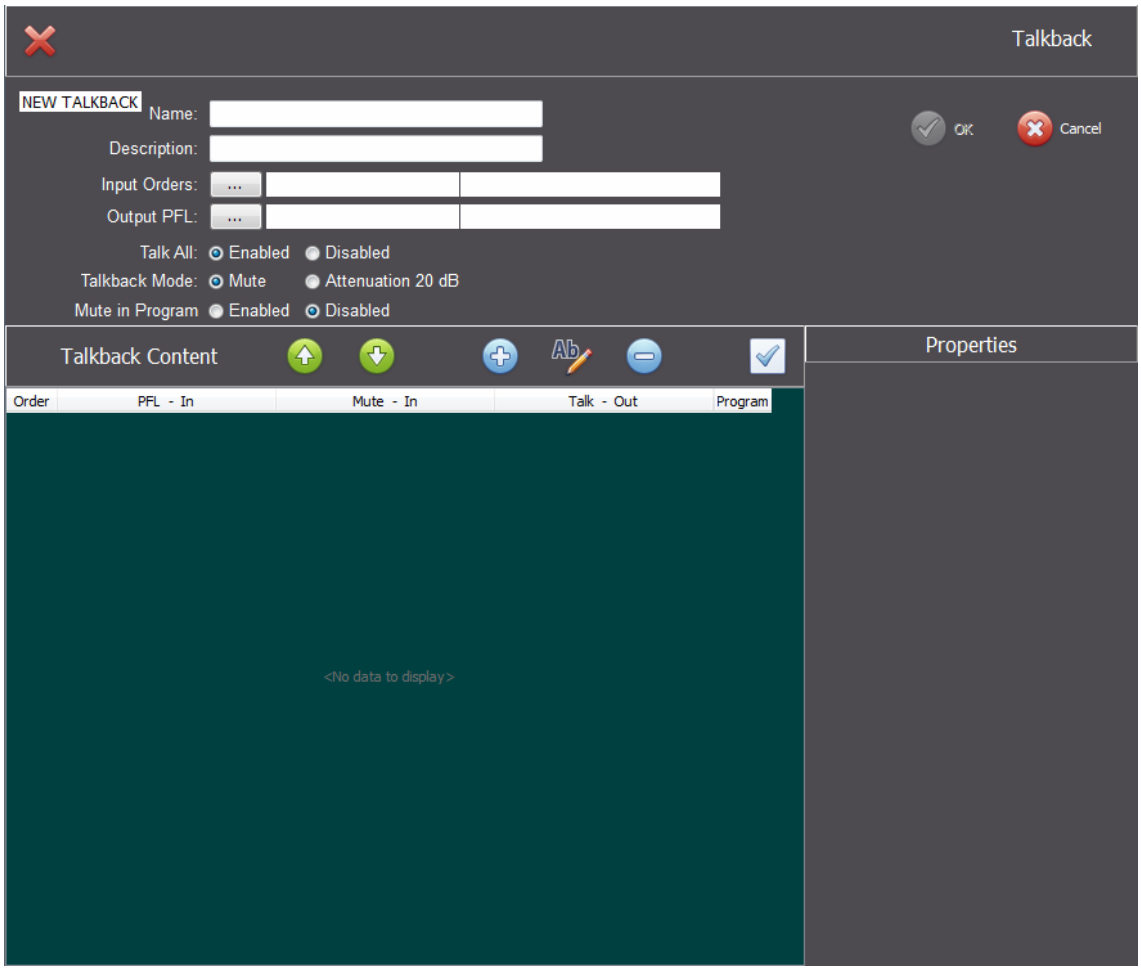
allows you to close the window.

allows you to create a new "Talkback".

allows you to edit an existing "Talkback".

allows you to delete the selected "Talkback" from the list. Confirmation is required.

If no "Talkback" has been created yet, the list will be empty. Click on the button and the window for creating "Talkback" will appear (this is the same window that will allow you to edit an existing "Talkback"). From the upper left part of the window from left to right and going downwards, the following fields and controls are available:

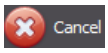


allows you to close the "NEW TALKBACK" / "EDIT TALKBACK" window.

Below this area, the "Talkback" information area will appear:

- "**NEW TALKBACK**" or "**EDIT TALKBACK**" field indicates if you are creating a new "Talkback" or editing an existing one.
- "**Name":** field used to name the "Talkback" alphanumerically.
- "**Description":** field allowing us to provide a brief description of the "Talkback".
- "**Input Orders":** field allowing us to select the input line that will work as "Order" input.
- "**Output PFL":** field allowing us to select the output line that will work as "PFL" output.
- "**Talk All":** when the "Enabled" option is selected, the "TALK ALL MIC TB" button will be shown in the "Talkback" window. When the "Disabled" option is selected, that button will not appear. See section 4.3 of this manual.
- "**Talkback Mode":** when the "Mute" option is selected and a "TALK" button of the "Talkback" window is pressed, all the crosspoints connected to the associated output line are deactivated and the "Order" input is sent to it. When the "Attenuation 20 dB" is selected, those crosspoints will remain but with an attenuation of 20 dB. This option is available only for NETBOX DSP. See section 4.3 of this manual.
- "**Mute in Program":** when the "Enabled" option is selected, the "MUTE OFF" button will be shown in the "Talkback" window. When the "Disabled" option is selected, that button will not appear. This option take effect when one of the lines selections of the "Talkback" is configured as "Program". See section 4.3 of this manual.

allows you to confirm the creation or edition of the "Talkback" and save it to the database.

 allows you to discard the creation or edition of the "Talkback" to the database.

Following the "Talkback" information the "**Talkback Content**" area will appear:

allows you to move the selected line up in the sequence.

allows you to move the selected line down in the sequence.

allows you to open the selection of the lines that will be included in the "Talkback" (in the "Properties" area).

allows you to edit the selected line.

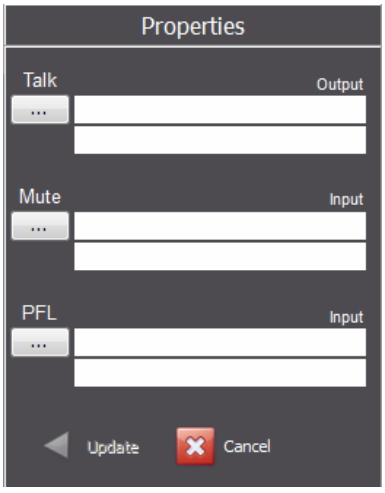
allows you to remove the selected line.

allows you to configure the selected line as "Program".

Below these buttons the following columns will also appear:

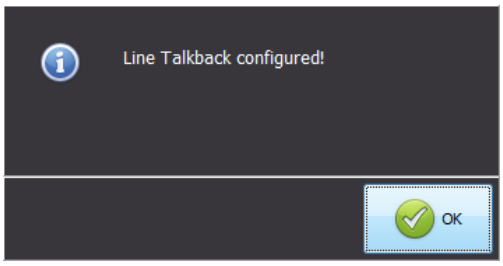
- "**Order**": order of the line in the "Talkback".
- "**PFL - In**": input line that will be sent to the "PFL" output when the associated "PFL" button of the "Talkback" window is pressed.
- "**Mute - In**": input line that will be muted when the associated "MUTE OFF" button of the "Talkback" window is pressed (all the crosspoints that send that line towards the defined "Talk" outputs are deleted).
- "**Talk - Out**": "Talk" output line associated to the previous input lines.
- "**Program**": indicates if the lines selection is configured as "Program", in which case, it will be shown in a different way in the "Talkback" window (see section 4.3 of this manual).

When the button of the "Talkback Content" area is pressed, the following buttons will appear in the "Properties" area allowing us to select the lines that will be included in the "Talkback":

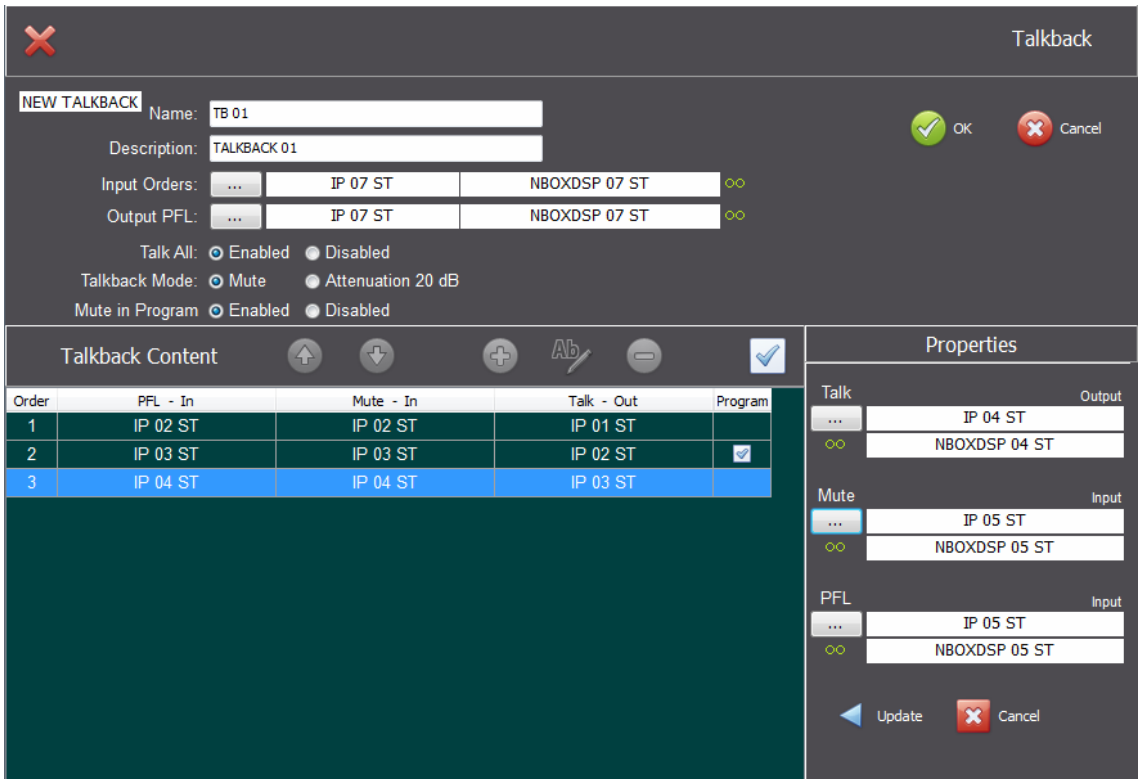


When you press the button of each one of the three sections a window will open that allows us to select the output line that will appear in the "Talk - Out" column, the input line that will appear in the "Mute - In" column and the input line that will appear in the "PFL - In" column. By default, when the "Mute" input is selected the "PFL" input is automatically configured with the same input line, although you can change this selection by pressing the associated button. In order to add the lines selection to the "Talkback" just press the "Update" button.

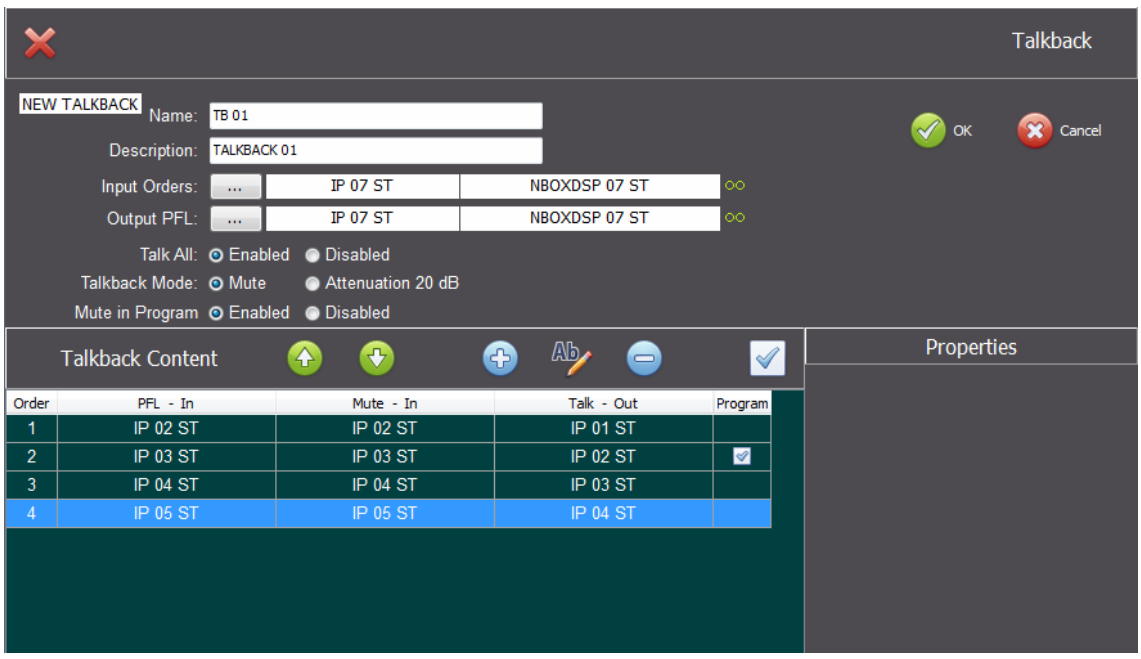
NOTE: An output line selected as "Talk" can be repeated in a "Talkback" group. n that case an informative window will appear:



In the example, a "Talkback" group (TB 01) is being created where the "IP 07 ST" input line is defined as "Order" input and the "IP 07 ST" output is defines as "PFL" output. Besides, three combinations of input and output lines are defined (the second one is configured as "Program") and a fourth one is being added where the "IP 04 ST" output line is selected as "Talk" output and the "IP 05 ST" input line is selected as "Mute" and "PFL" inputs.



When you press the "Update" button the new lines combination will appear in the "Talkback Content" area:

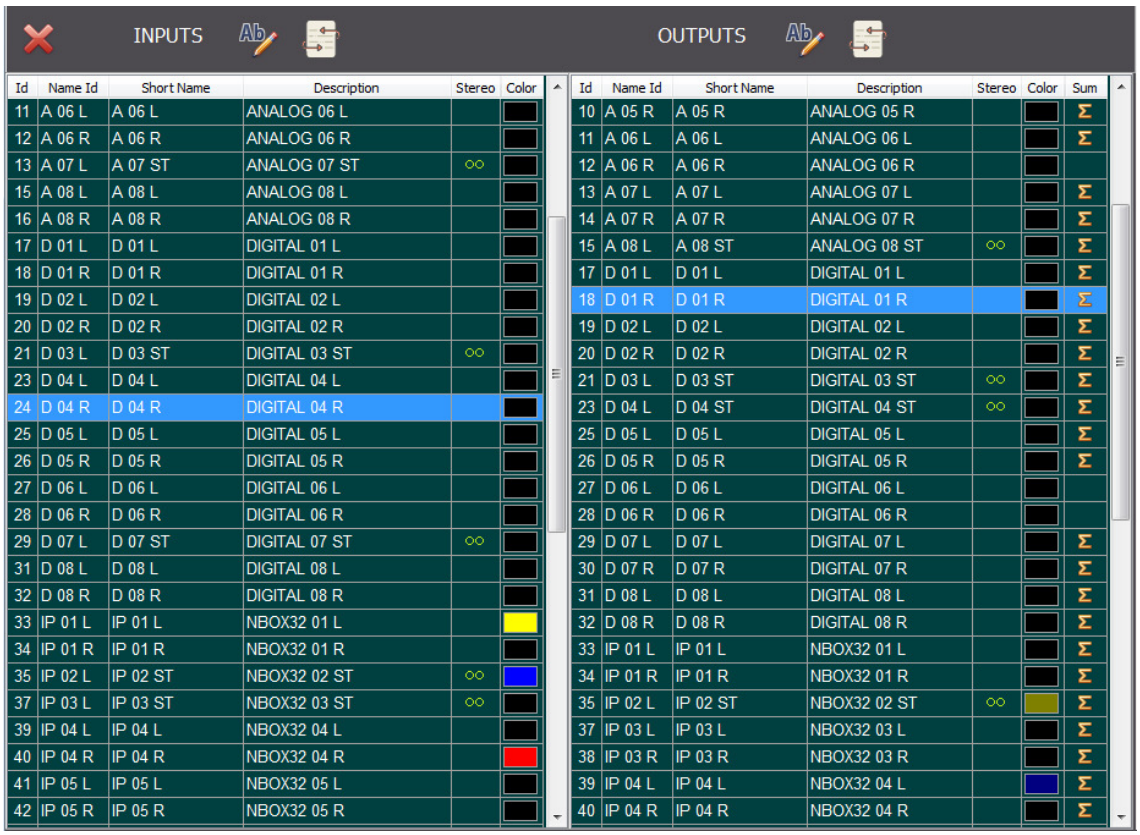


Once you have finished the "OK" button should be pressed to save the "Talkback" to the database.

6.8. Logical Lines.

The "**Logical Lines**" option of the "Config" menu gives access to the area of the application where you can edit the physical or logical input and output lines of the NETBOX in regards to their "Short Name" and "Description". Here you can also make or break "Stereo" pairs, choose the color you want to visualize the vumeters frames (only for Dante lines; it's also reflected in the XY view) and define if the output line will be summable or not.

NOTE: The "Dante" option of the "Config" menu allows you to rename the transmission and reception channels of the unit's Dante module with the names of the associated logical lines defined here (see section 6.13 of this manual).



Id	Name Id	Short Name	Description	Stereo	Color
11	A 06 L	A 06 L	ANALOG 06 L		
12	A 06 R	A 06 R	ANALOG 06 R		
13	A 07 L	A 07 ST	ANALOG 07 ST	∞	
15	A 08 L	A 08 L	ANALOG 08 L		
16	A 08 R	A 08 R	ANALOG 08 R		
17	D 01 L	D 01 L	DIGITAL 01 L		
18	D 01 R	D 01 R	DIGITAL 01 R		
19	D 02 L	D 02 L	DIGITAL 02 L		
20	D 02 R	D 02 R	DIGITAL 02 R		
21	D 03 L	D 03 ST	DIGITAL 03 ST	∞	
23	D 04 L	D 04 L	DIGITAL 04 L		
24	D 04 R	D 04 R	DIGITAL 04 R		
25	D 05 L	D 05 L	DIGITAL 05 L		
26	D 05 R	D 05 R	DIGITAL 05 R		
27	D 06 L	D 06 L	DIGITAL 06 L		
28	D 06 R	D 06 R	DIGITAL 06 R		
29	D 07 L	D 07 ST	DIGITAL 07 ST	∞	
31	D 08 L	D 08 L	DIGITAL 08 L		
32	D 08 R	D 08 R	DIGITAL 08 R		
33	IP 01 L	IP 01 L	NBOX32 01 L		
34	IP 01 R	IP 01 R	NBOX32 01 R		
35	IP 02 L	IP 02 ST	NBOX32 02 ST	∞	
37	IP 03 L	IP 03 ST	NBOX32 03 ST	∞	
39	IP 04 L	IP 04 L	NBOX32 04 L		
40	IP 04 R	IP 04 R	NBOX32 04 R		
41	IP 05 L	IP 05 L	NBOX32 05 L		
42	IP 05 R	IP 05 R	NBOX32 05 R		

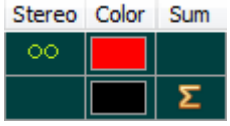
Id	Name Id	Short Name	Description	Stereo	Color	Sum
10	A 05 R	A 05 R	ANALOG 05 R			Σ
11	A 06 L	A 06 L	ANALOG 06 L			Σ
12	A 06 R	A 06 R	ANALOG 06 R			
13	A 07 L	A 07 L	ANALOG 07 L			Σ
14	A 07 R	A 07 R	ANALOG 07 R			Σ
15	A 08 L	A 08 ST	ANALOG 08 ST	∞		Σ
17	D 01 L	D 01 L	DIGITAL 01 L			Σ
18	D 01 R	D 01 R	DIGITAL 01 R			Σ
19	D 02 L	D 02 L	DIGITAL 02 L			Σ
20	D 02 R	D 02 R	DIGITAL 02 R			Σ
21	D 03 L	D 03 ST	DIGITAL 03 ST	∞		Σ
23	D 04 L	D 04 ST	DIGITAL 04 ST	∞		Σ
25	D 05 L	D 05 L	DIGITAL 05 L			Σ
26	D 05 R	D 05 R	DIGITAL 05 R			Σ
27	D 06 L	D 06 L	DIGITAL 06 L			
28	D 06 R	D 06 R	DIGITAL 06 R			
29	D 07 L	D 07 L	DIGITAL 07 L			Σ
30	D 07 R	D 07 R	DIGITAL 07 R			Σ
31	D 08 L	D 08 L	DIGITAL 08 L			Σ
32	D 08 R	D 08 R	DIGITAL 08 R			Σ
33	IP 01 L	IP 01 L	NBOX32 01 L			Σ
34	IP 01 R	IP 01 R	NBOX32 01 R			Σ
35	IP 02 L	IP 02 ST	NBOX32 02 ST	∞		Σ
37	IP 03 L	IP 03 L	NBOX32 03 L			Σ
38	IP 03 R	IP 03 R	NBOX32 03 R			Σ
39	IP 04 L	IP 04 L	NBOX32 04 L			Σ
40	IP 04 R	IP 04 R	NBOX32 04 R			Σ

Input lines are listed to the left side and the output lines to the right.

Apart from the button to cancel or exit, there is an edit button located in the header of the lists - the number of inputs and their generic names are imported by the application automatically when a connection to the NETBOX is established. There is also a button that allows you to generate/select an import file in order to modify the labels associated to each line ("Short Name" and "Description") from a text file.

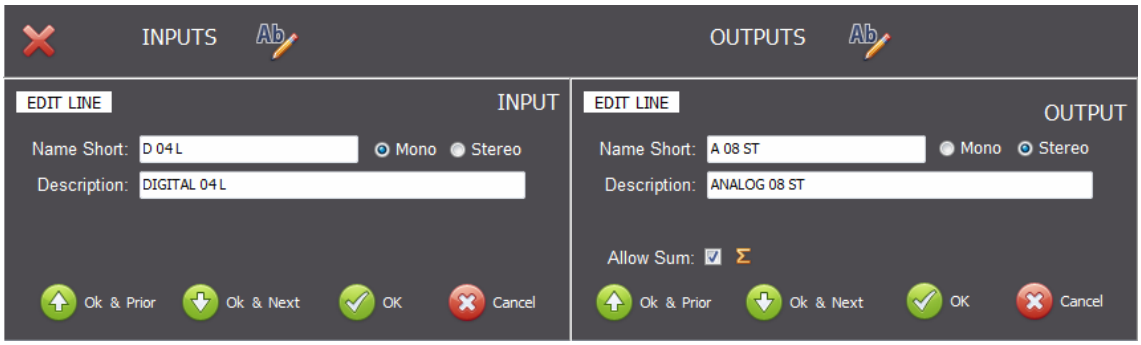
The "Id" and "Name Id" columns are 2 informative fields, that can't be modified and show the line identifier number and the line internal name.

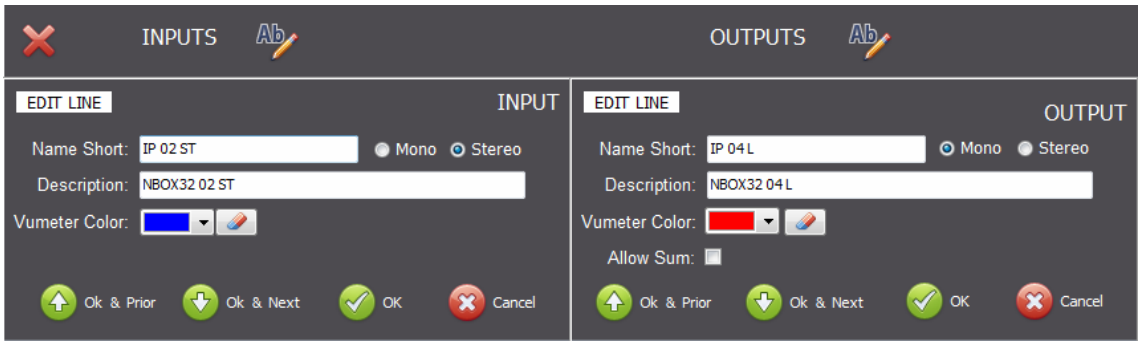
If you click on the button on either the input or the output side of the lines, an edit dialogue box will open above the listed lines and that allows us to change the "Short Name", "Description" and grouping and ungrouping the lines in Stereo pairs ("Mono" or "Stereo"). In the case of the Dante lines you can also choose the color of the vumeters frames (it's also reflected in the label of the line in the XY view) and in the case of the output lines if the line will be summable or not.



It also allows the user to confirm the edit and move to the prior or next line in the list without leaving this dialogue box which becomes very handy when setting up the system for the first time.

The following examples show a digital input line configured as mono, an analog output line configured as stereo and summable, a Dante input line configured as stereo and with blue vumeters frames and a Dante output line configured as mono, non summable and with red vumeters frames.



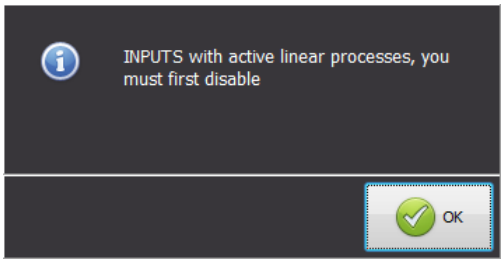


Once you have finished the editing you confirm the changes with the OK button or not with the Cancel button, both of which will close the "EDIT LINE" dialogue box. The changes will be automatically updated to the list.

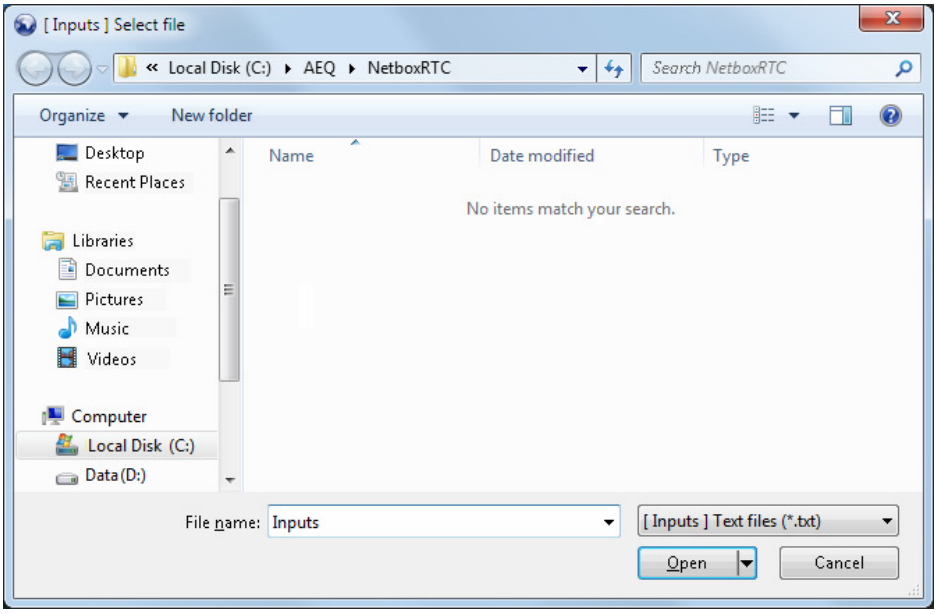
If you use the button Ok & Next or Ok & Prior instead of the OK button, you will also confirm the changes and go to edit the next or previous line respectively.

When done editing, you exit the logical lines edit by clicking on the button in the upper left corner.

In the case of **NETBOX DSP**, when you try to change from mono to stereo (or vice versa) an input or output line where one or several **processes** have been activated, you must first disable these processes in order to be able to make that change. In that case an informative window will appear:

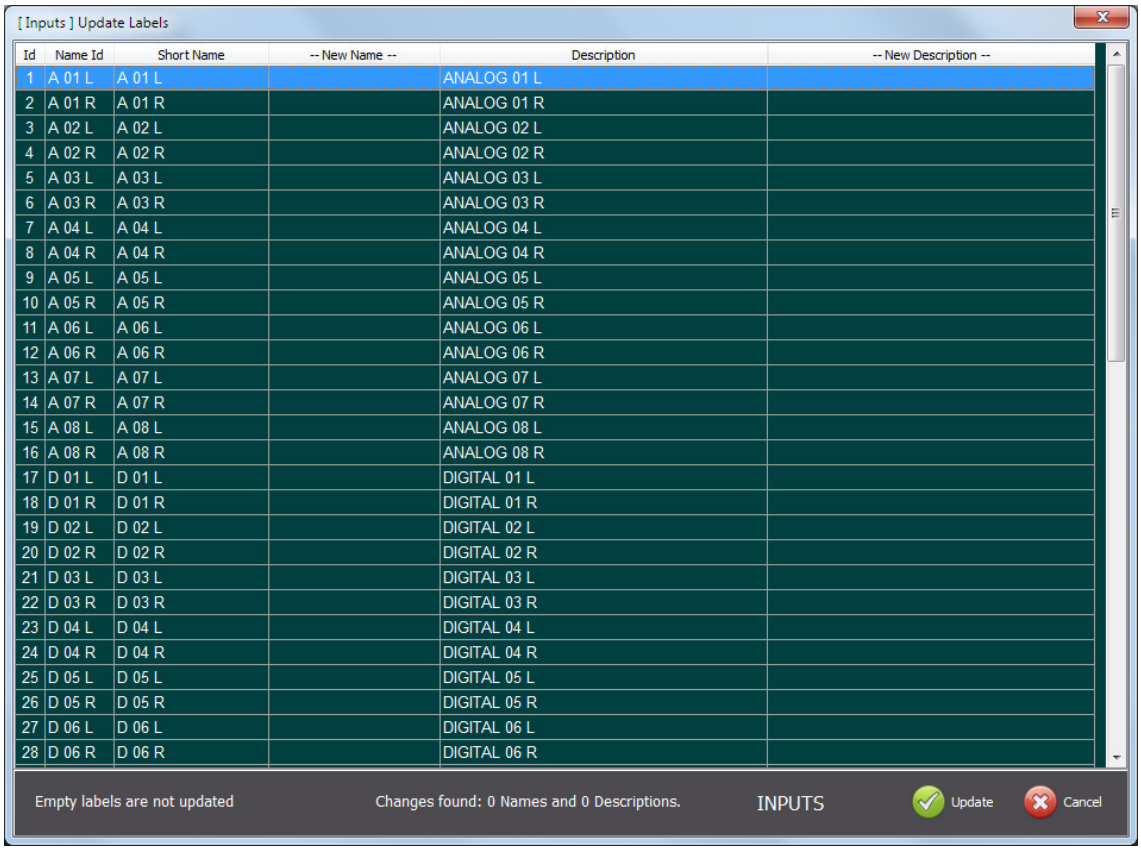


If you click on the button on either the input or the output side of the lines, a window will open that allows us to generate or select an import file in order to modify the labels associated to each line from a text file:

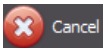


When there is no import file previously created, you must select the file location (by default C:\AEQ\NetboxRTC) and give it a name ("Inputs" in the example).

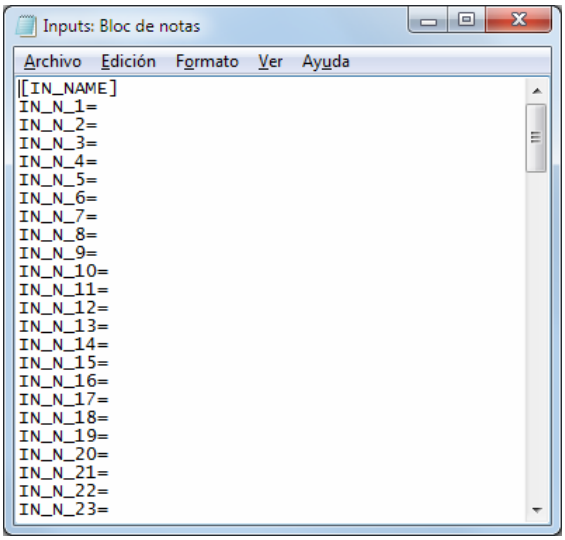
When you press the "Open" button the file is created in the selected location and the following window will appear:



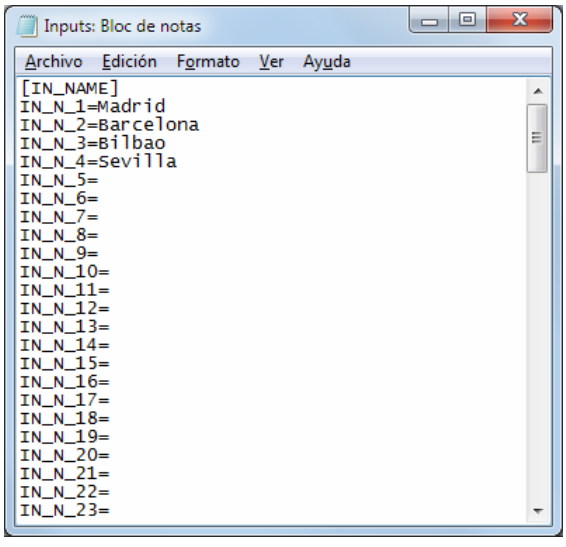
where the lines (input lines in the example) and the new labels are shown. In this case, given that we have just created the file, there is no content in the "-- New Name --" and "-- New Description --" columns and the lower part of the window indicates that no changes are detected in those labels:

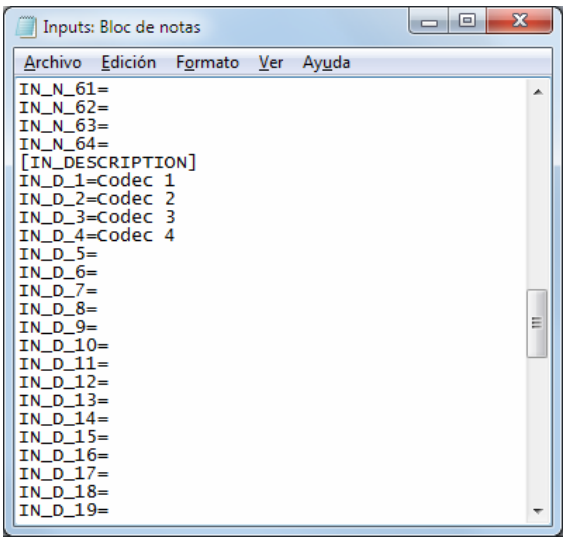
Since there are no changes, just press the  button to close that window.

When you open the created "Inputs" file (using "Notepad" or "WordPad" for instance), it will look like this:



There you can configure the desired "Short Name" or "Description" in the list that appears:

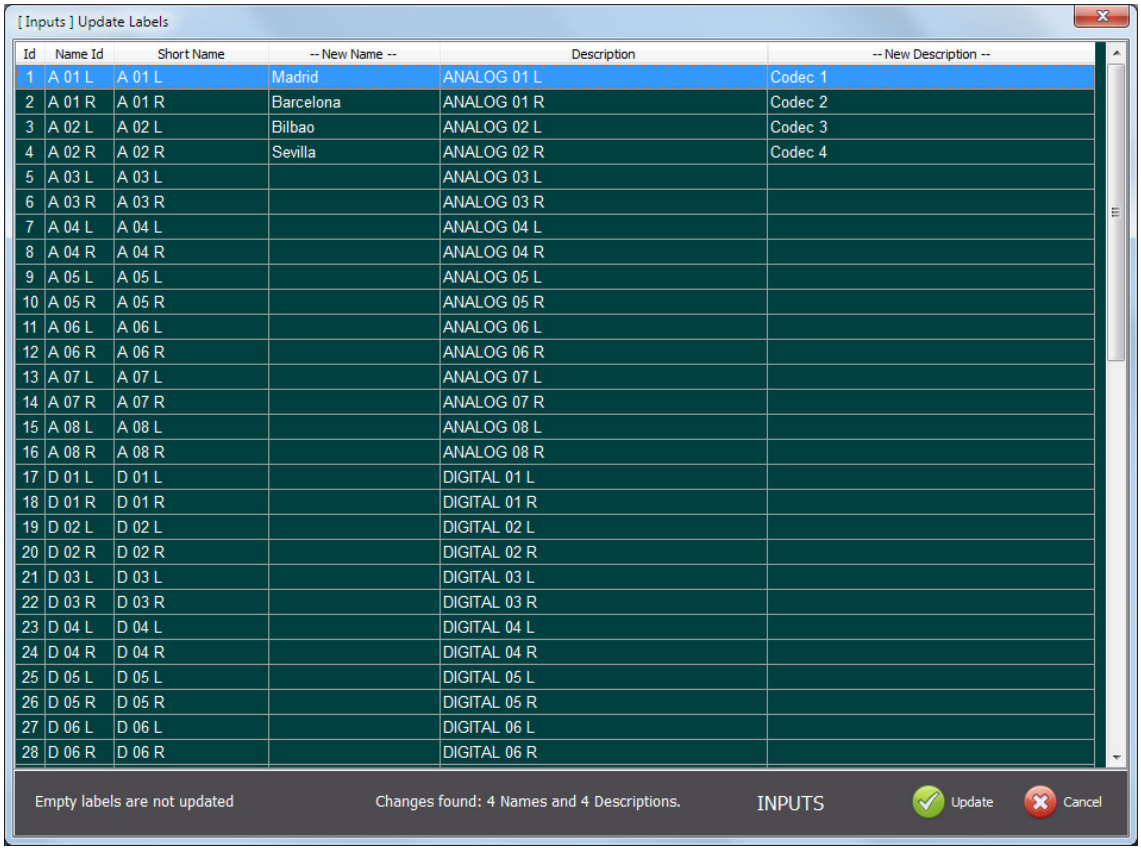




In the example, we have modified the name and description of the first 4 input lines.

The labels that are left blank will be ignored during the import process.

If you click again on the button of input lines and you select the modified "Inputs" file in the window that appears, the following window will appear again when you press the "Open" button:

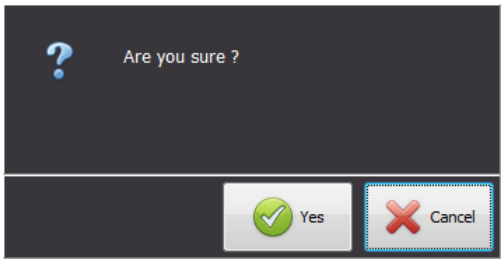


Id	Name Id	Short Name	-- New Name --	Description	-- New Description --
1	A 01 L	A 01 L	Madrid	ANALOG 01 L	Codec 1
2	A 01 R	A 01 R	Barcelona	ANALOG 01 R	Codec 2
3	A 02 L	A 02 L	Bilbao	ANALOG 02 L	Codec 3
4	A 02 R	A 02 R	Sevilla	ANALOG 02 R	Codec 4
5	A 03 L	A 03 L		ANALOG 03 L	
6	A 03 R	A 03 R		ANALOG 03 R	
7	A 04 L	A 04 L		ANALOG 04 L	
8	A 04 R	A 04 R		ANALOG 04 R	
9	A 05 L	A 05 L		ANALOG 05 L	
10	A 05 R	A 05 R		ANALOG 05 R	
11	A 06 L	A 06 L		ANALOG 06 L	
12	A 06 R	A 06 R		ANALOG 06 R	
13	A 07 L	A 07 L		ANALOG 07 L	
14	A 07 R	A 07 R		ANALOG 07 R	
15	A 08 L	A 08 L		ANALOG 08 L	
16	A 08 R	A 08 R		ANALOG 08 R	
17	D 01 L	D 01 L		DIGITAL 01 L	
18	D 01 R	D 01 R		DIGITAL 01 R	
19	D 02 L	D 02 L		DIGITAL 02 L	
20	D 02 R	D 02 R		DIGITAL 02 R	
21	D 03 L	D 03 L		DIGITAL 03 L	
22	D 03 R	D 03 R		DIGITAL 03 R	
23	D 04 L	D 04 L		DIGITAL 04 L	
24	D 04 R	D 04 R		DIGITAL 04 R	
25	D 05 L	D 05 L		DIGITAL 05 L	
26	D 05 R	D 05 R		DIGITAL 05 R	
27	D 06 L	D 06 L		DIGITAL 06 L	
28	D 06 R	D 06 R		DIGITAL 06 R	

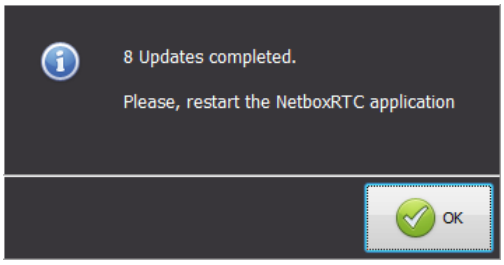
where the 4 new names and descriptions are shown in the "-- New Name --" and "-- New Description --" columns and the lower part of the window indicates the detected changes:

If you press now the

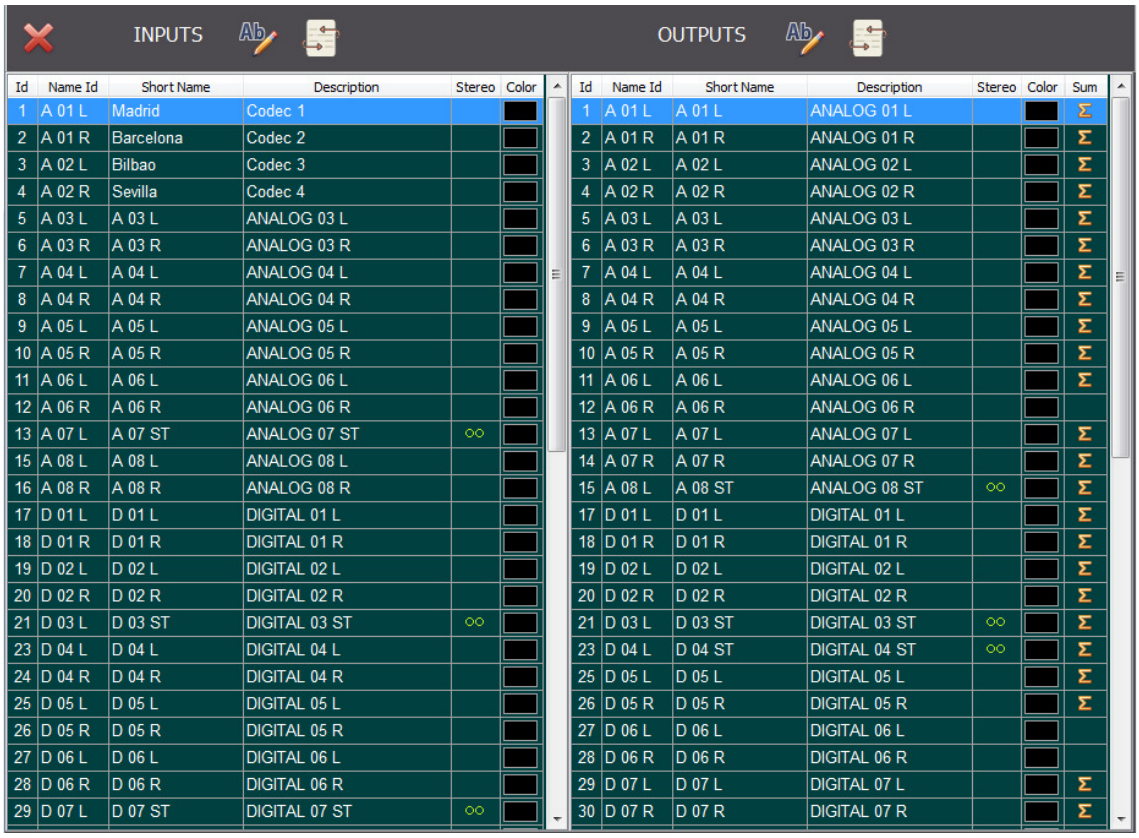
button, a confirmation window will appear:



After confirmation, an informative window indicates that the "NetboxRTC" application must be restarted:



Once restarted, the updated list of input lines will appear:

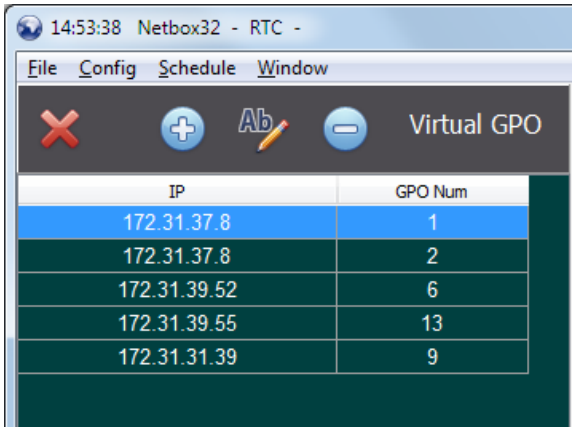


Id	Name Id	Short Name	Description	Stereo	Color
1	A 01 L	Madrid	Codec 1		
2	A 01 R	Barcelona	Codec 2		
3	A 02 L	Bilbao	Codec 3		
4	A 02 R	Sevilla	Codec 4		
5	A 03 L	A 03 L	ANALOG 03 L		
6	A 03 R	A 03 R	ANALOG 03 R		
7	A 04 L	A 04 L	ANALOG 04 L		
8	A 04 R	A 04 R	ANALOG 04 R		
9	A 05 L	A 05 L	ANALOG 05 L		
10	A 05 R	A 05 R	ANALOG 05 R		
11	A 06 L	A 06 L	ANALOG 06 L		
12	A 06 R	A 06 R	ANALOG 06 R		
13	A 07 L	A 07 ST	ANALOG 07 ST	oo	
15	A 08 L	A 08 L	ANALOG 08 L		
16	A 08 R	A 08 R	ANALOG 08 R		
17	D 01 L	D 01 L	DIGITAL 01 L		
18	D 01 R	D 01 R	DIGITAL 01 R		
19	D 02 L	D 02 L	DIGITAL 02 L		
20	D 02 R	D 02 R	DIGITAL 02 R		
21	D 03 L	D 03 ST	DIGITAL 03 ST	oo	
23	D 04 L	D 04 L	DIGITAL 04 L		
24	D 04 R	D 04 R	DIGITAL 04 R		
25	D 05 L	D 05 L	DIGITAL 05 L		
26	D 05 R	D 05 R	DIGITAL 05 R		
27	D 06 L	D 06 L	DIGITAL 06 L		
28	D 06 R	D 06 R	DIGITAL 06 R		
29	D 07 L	D 07 ST	DIGITAL 07 ST	oo	

Id	Name Id	Short Name	Description	Stereo	Color	Sum
1	A 01 L	A 01 L	ANALOG 01 L			Σ
2	A 01 R	A 01 R	ANALOG 01 R			Σ
3	A 02 L	A 02 L	ANALOG 02 L			Σ
4	A 02 R	A 02 R	ANALOG 02 R			Σ
5	A 03 L	A 03 L	ANALOG 03 L			Σ
6	A 03 R	A 03 R	ANALOG 03 R			Σ
7	A 04 L	A 04 L	ANALOG 04 L			Σ
8	A 04 R	A 04 R	ANALOG 04 R			Σ
9	A 05 L	A 05 L	ANALOG 05 L			Σ
10	A 05 R	A 05 R	ANALOG 05 R			Σ
11	A 06 L	A 06 L	ANALOG 06 L			Σ
12	A 06 R	A 06 R	ANALOG 06 R			
13	A 07 L	A 07 L	ANALOG 07 L			Σ
14	A 07 R	A 07 R	ANALOG 07 R			Σ
15	A 08 L	A 08 ST	ANALOG 08 ST	oo		Σ
17	D 01 L	D 01 L	DIGITAL 01 L			Σ
18	D 01 R	D 01 R	DIGITAL 01 R			Σ
19	D 02 L	D 02 L	DIGITAL 02 L			Σ
20	D 02 R	D 02 R	DIGITAL 02 R			Σ
21	D 03 L	D 03 ST	DIGITAL 03 ST	oo		Σ
23	D 04 L	D 04 ST	DIGITAL 04 ST	oo		Σ
25	D 05 L	D 05 L	DIGITAL 05 L			Σ
26	D 05 R	D 05 R	DIGITAL 05 R			Σ
27	D 06 L	D 06 L	DIGITAL 06 L			
28	D 06 R	D 06 R	DIGITAL 06 R			
29	D 07 L	D 07 L	DIGITAL 07 L			Σ
30	D 07 R	D 07 R	DIGITAL 07 R			Σ

6.9. Virtual GPO.

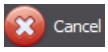
The NETBOX devices are capable of generating a Virtual GPO command that is sent over the audio and control network. This command can be interpreted by any AEQ standard AoIP equipment and many third-party SW applications. This section allows us to prepare these GPO commands so that these can be used for, for example, macros and scheduled tasks when you need to command or automate the control of other equipment from the "NetboxRTC" application. What needs to be accomplished in this section is to set up these commands that are stored in the database of "NetboxRTC" and will be triggered by the NETBOX whenever so requested.

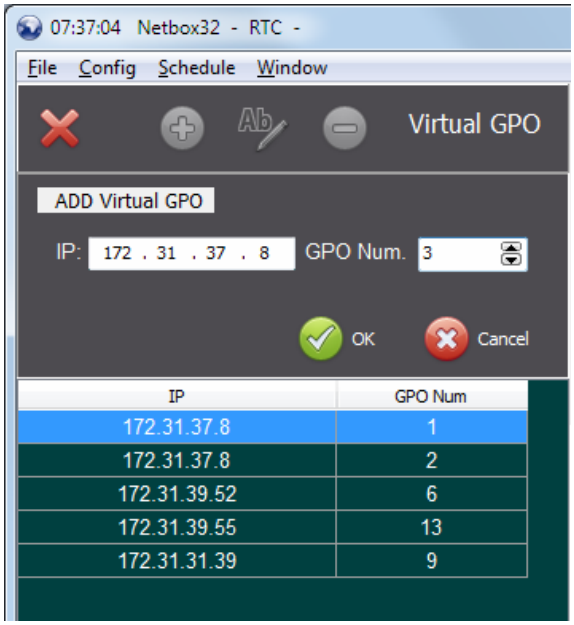


For this you need to indicate the IP address that the GPO command is intended - addressed to, and the Id number of the GPO. Theoretically you could generate up to 256 GPO commands or Id Numbers for each and every IP address that you are able to communicate with.

When clicking on the or button the "ADD Virtual GPO" or "EDIT Virtual GPO" dialogue box is opened: here you can indicate the "IP" address that the command is addressed for and the "Id Number" of the GPO command.

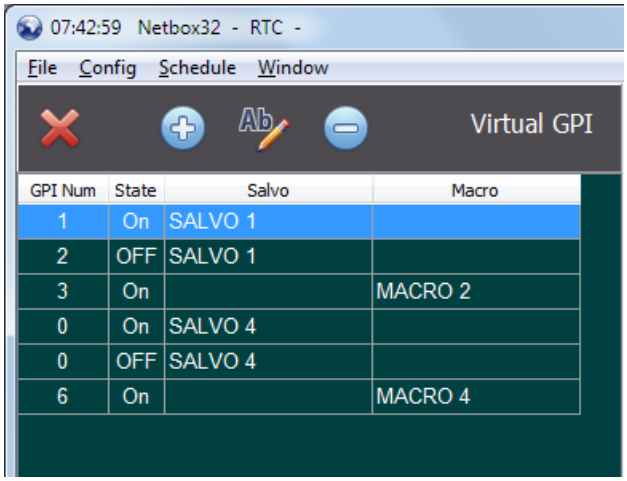
will confirm the creation or edition of the Virtual GPO and save this to the database.

 will discard the creation or edition of the Virtual GPO.



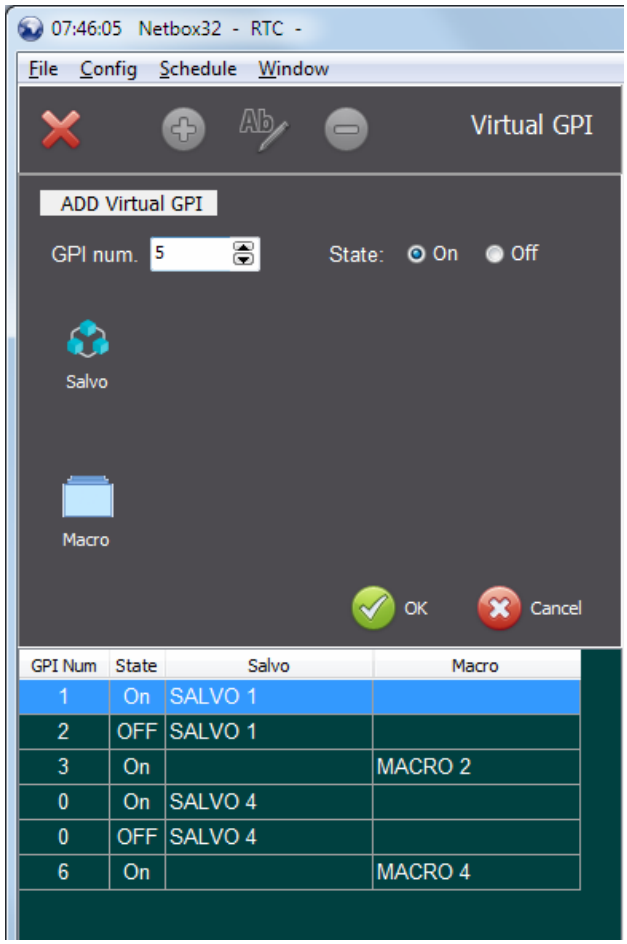
6.10. Virtual GPI.

The NETBOX devices are capable of receiving a Virtual GPI command that is sent over the audio and control network. This command can be generated by any AEQ standard AoIP equipment and many third-party SW applications. This section allows us to prepare these GPI commands so that these can be used to, for example, trigger macros or salvos when you need to automate the operation of the "NetboxRTC" application from external equipment or SW applications. What needs to be accomplished in this section is to set up these commands that are stored in the database of "NetboxRTC" and define what such commands should trigger in terms of macros or salvos will be triggered by the NETBOX whenever so requested.

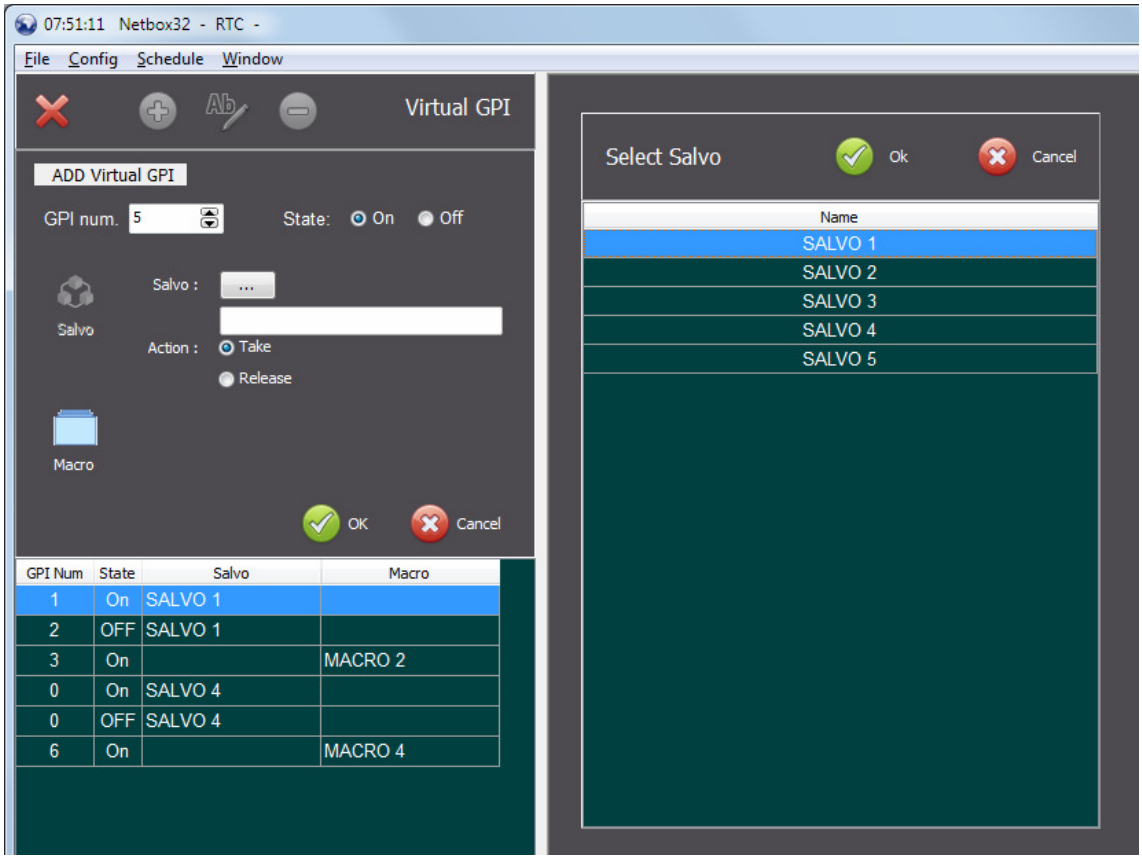


For this you need to indicate the Id number of the GPI. Theoretically you could receive up to 256 GPI commands or Id numbers from each and every IP address that are able to communicate with the NETBOX.

When clicking on the or button the "ADD Virtual GPI" or "EDIT Virtual GPI" dialogue box is opened: here you can indicate the "Id Number" of the expected GPI and if the "State" expected should be interpreting the GPI as "On" or "Off". Next, there are two buttons that defines whether the GPI is going to trigger a salvo or a macro.

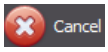


If you click on the "Salvo" button, the selection of salvo and the action definition ("Take" or "Release") appear. When you click on the salvo selection, the list of existing salvos will open to the right:



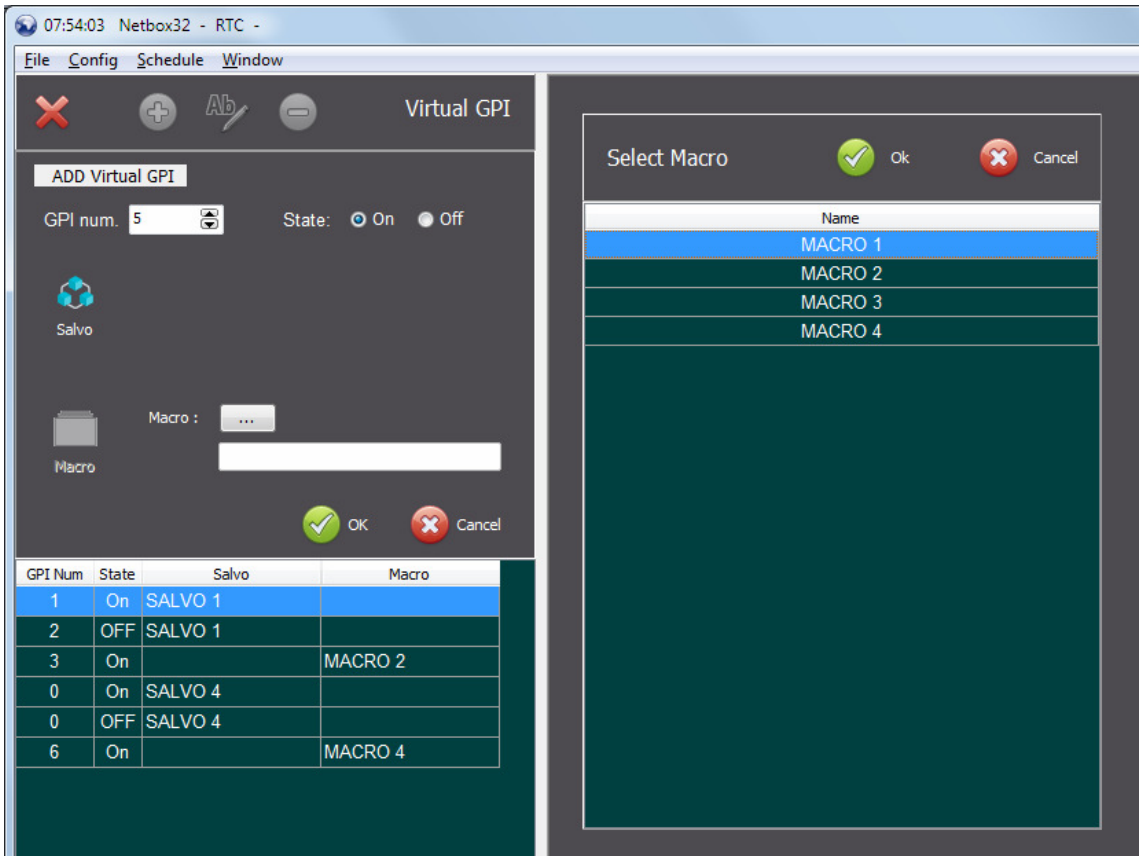
If you select a salvo on the list and double click this it will confirm the selection. The other way to confirm the selection is by clicking the

button. This will confirm the triggering of the selected salvo when receiving the Virtual GPI with the corresponding Id Number and save the information to the system database.

If you press the 

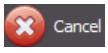
button, this will discard the selection of the salvo and the association to the Virtual GPI Id Number.

If you click on the "Macro" button, the selection of macro appears. When you click on the macro selection, the list of existing macros will open to the right:



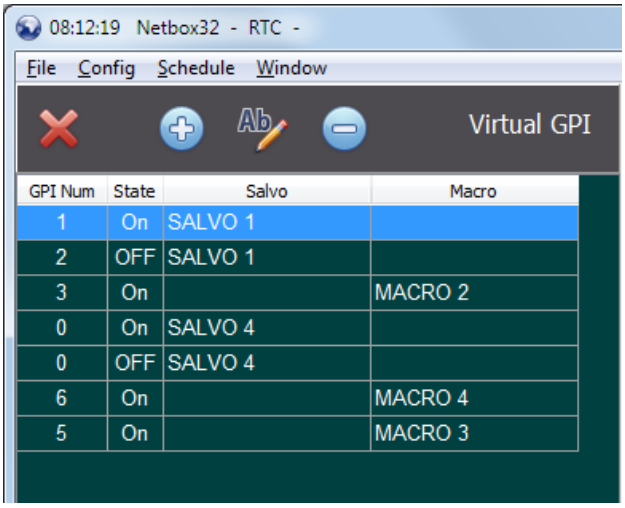
If you select a macro on the list and double click this it will confirm the selection. The other way to confirm the selection is by clicking the

button. This will confirm the triggering of the selected macro when receiving the Virtual GPI with the corresponding Id Number and save the information to the system database.

If you press the 

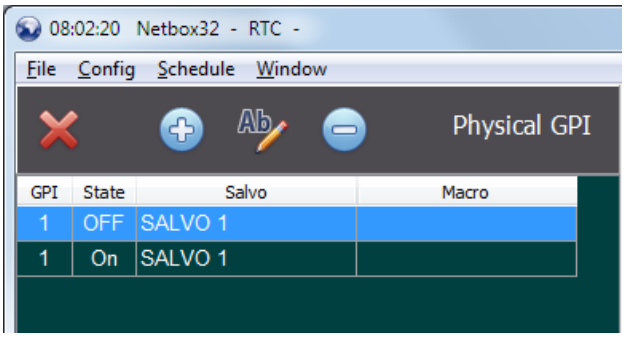
button, this will discard the selection of the macro and the association to the Virtual GPI Id Number.

Once you have finished editing the Virtual GPI Id Number and the associated action (macro or salvo) will be shown in the list of defined Virtual GPI's:

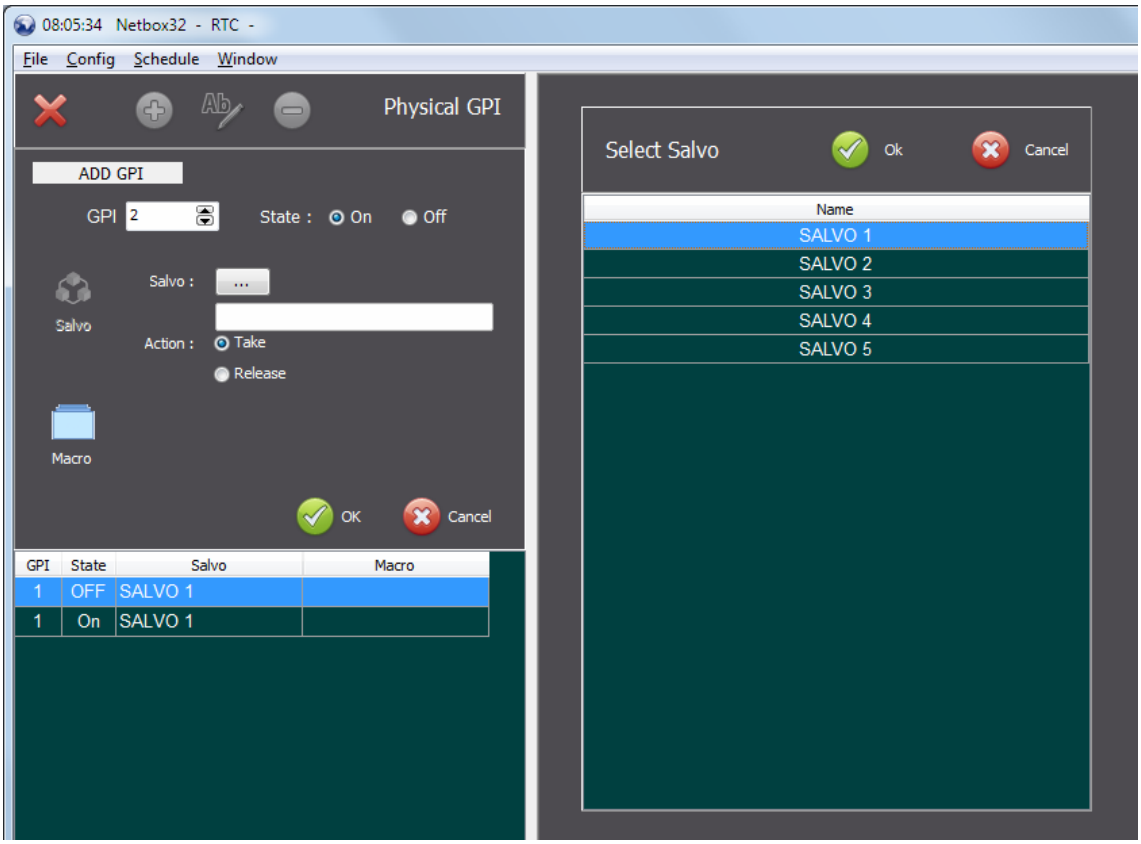


6.11. GPI.

The NETBOX 32 AD MX and NETBOX 8 AD MX have 16 and 4 physical opto-coupled GPI/O's respectively. This section allows the user of "NetboxRTC" to define what physical GPI corresponds to a specific salvo or macro, i.e. if an equipment or device that is physically connected to the GPI's of the NETBOX and this device sends a GPO signal with the Id of the GPI and the status "On" or "Off", the NETBOX will be able to interpret this and signal to the "NetboxRTC" application of what changes in the matrix needs to be accomplished.



When clicking on the or button the "ADD GPI" or "EDIT GPI" dialogue box is opened: here you can indicate the "Id Number" of the expected GPI and if the "State" expected should be interpreting the GPI as "On" or "Off". Next, there are two buttons that defines whether the GPI is going to trigger a salvo or a macro.

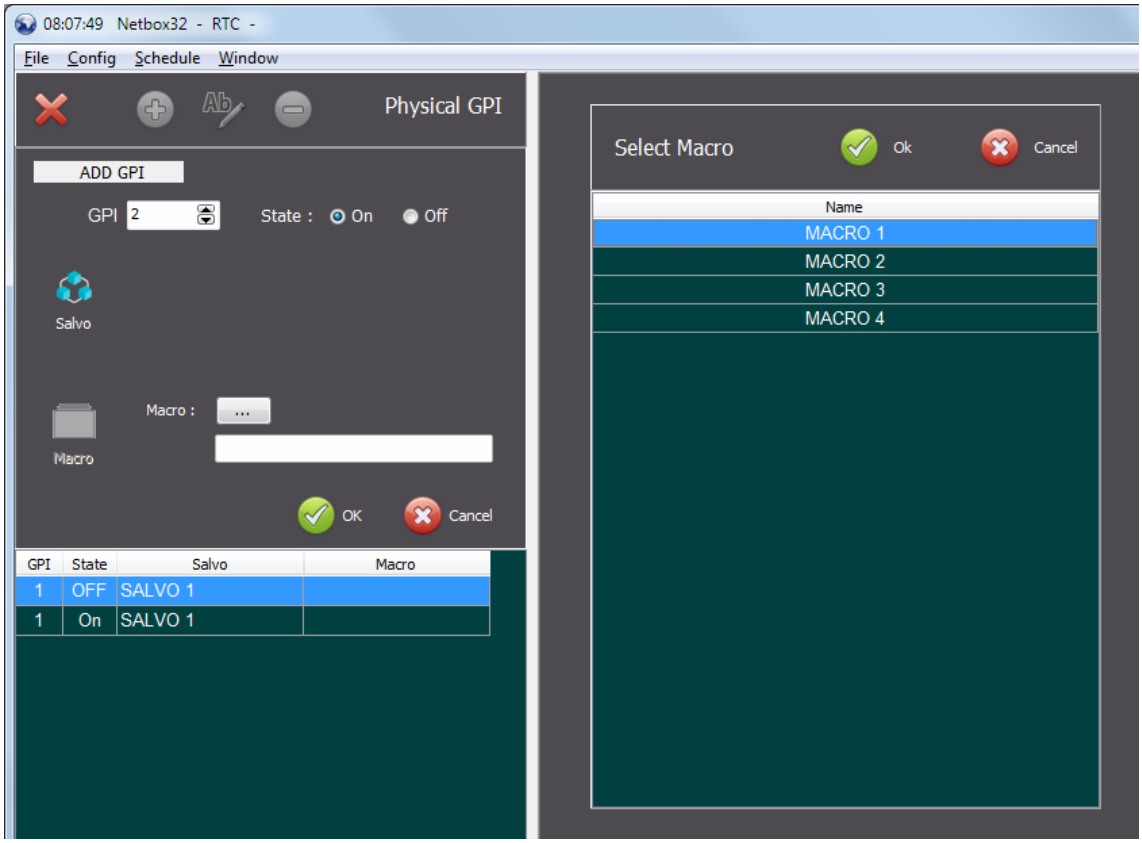


If you click on the "Salvo" button, the selection of salvo and the action definition ("Take" or "Release") appear. When you click on the salvo selection, the list of existing salvos will open to the right.

If you select a salvo on the list and double click this it will confirm the selection. The other way to confirm the selection is by clicking the button. This will confirm the triggering of the selected salvo when receiving the GPI with the corresponding Id Number and save the information to the system database.

If you press the Cancel button, this will discard the selection of the salvo and the association to the GPI Id Number.

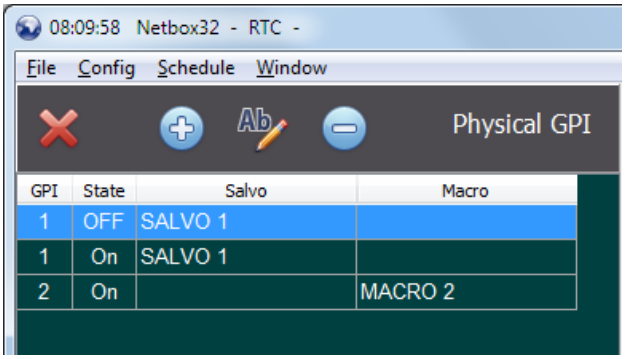
If you click on the "Macro" button, the selection of macro appears. When you click on the macro selection, the list of existing macros will open to the right:



If you select a macro on the list and double click this it will confirm the selection. The other way to confirm the selection is by clicking the OK button. This will confirm the triggering of the selected macro when receiving the GPI with the corresponding Id Number and save the information to the system database.

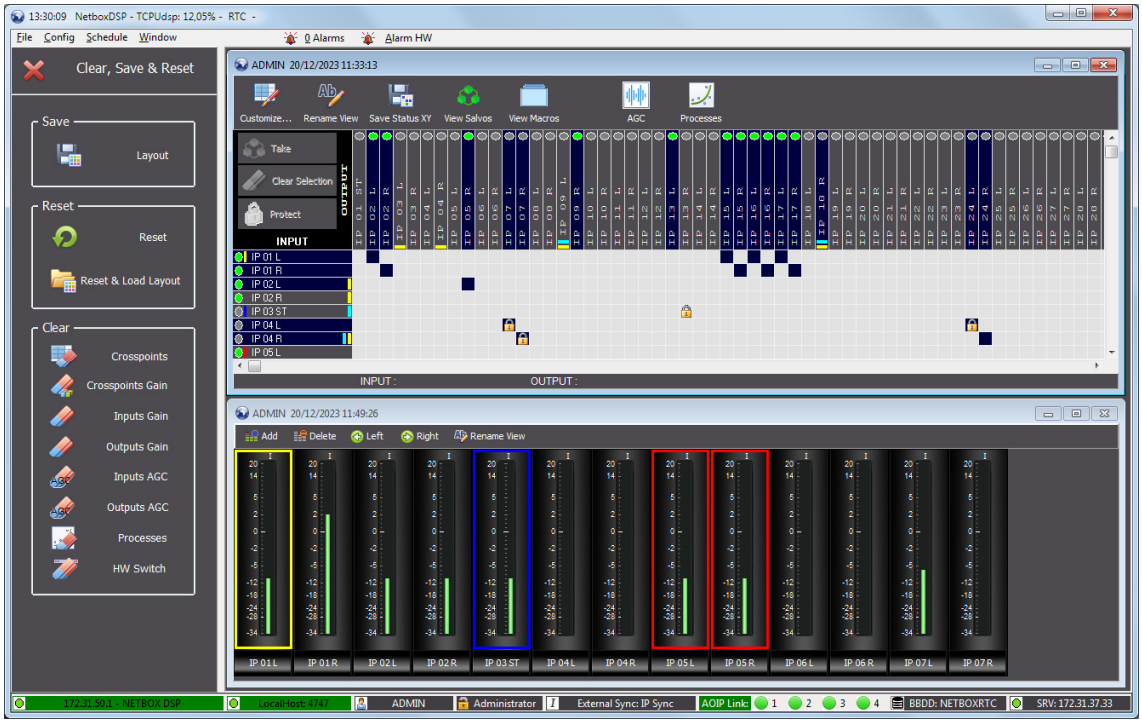
If you press the Cancel button, this will discard the selection of the macro and the association to the GPI Id Number.

Once you have finished editing the GPI Id Number and the associated action (macro or salvo) will be shown in the list of defined GPI's:

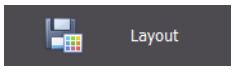


6.12. Clear, Save & Reset.

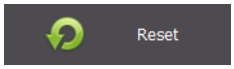
This option of the "Config" menu will open the following window to the left of the current XY view or layout:



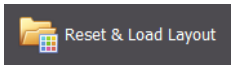
The buttons in each of the "Save", "Reset" and "Clear" sections are explained hereunder.



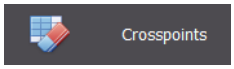
The NETBOX have a memory bank to store a "Master Layout" of the matrix that includes all data concerning crosspoints and gains. The "Layout" button of the "Save" section will save the current layout of the resulting matrix to this part of the non-volatile memory of the NETBOX.



The "Reset" button of the "Reset" section will reset the NETBOX. The unit will in this case startup with the last saved layout stored to its non-volatile memory. Take into account that this is not the "Master Layout" that is stored in a specific part of the NETBOX memory.

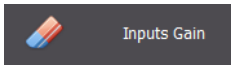


The "Reset & Load Layout" button of the "Reset" section will reset the NETBOX and load the last stored "Master Layout" from the non-volatile memory of the NETBOX.

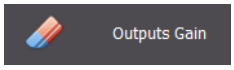


The "Crosspoints" button of the "Clear" section will clear all the existing crosspoints and protections in the NETBOX Matrix.

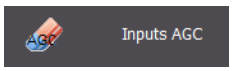
The "Crosspoints Gain" button of the "Clear" section will clear all the existing crosspoint gains in the NETBOX Matrix. This option is available only for NETBOX DSP.



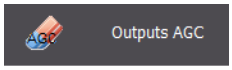
The "Inputs Gain" button of the "Clear" section will clear all the existing input lines gain that have been set in the NETBOX Matrix.



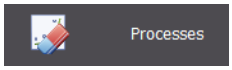
The "Outputs Gain" button of the "Clear" section will clear all the existing output lines gain that have been set in the NETBOX Matrix.



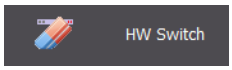
The "Inputs AGC" button of the "Clear" section will deactivate all the AGCs (Automatic Gain Control) of the input lines with that option activated. Additionally, AGCs are preconfigured as -20dBFs (default value).



The "Outputs AGC" button of the "Clear" section will deactivate all the AGCs (Automatic Gain Control) of the output lines with that option activated. Additionally, AGCs are preconfigured as -20dBFs (default value).



The "Processes" button of the "Clear" section will clear all the processes of the input and output lines with configured processes. This option is available only for NETBOX DSP.



The "HW Switch" button of the "Clear" section will clear all the HW Switch configurations that have been stored on the NETBOX.

For all these actions the user will be prompted to confirm the operation.

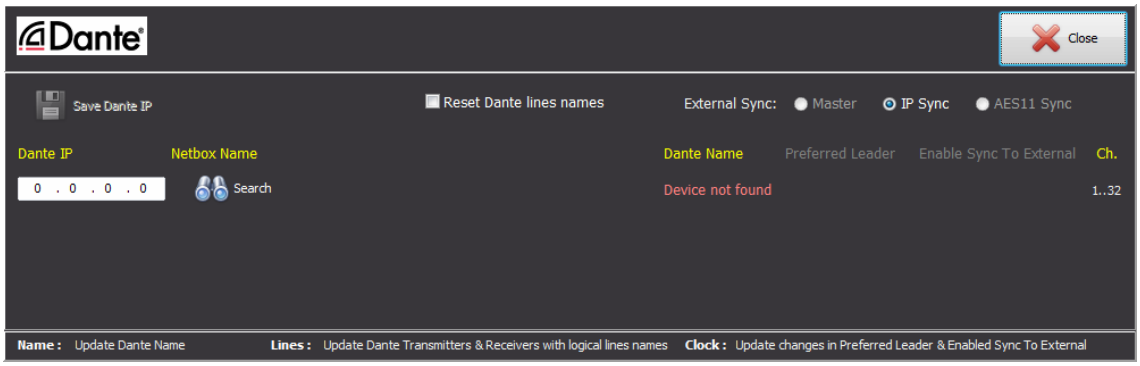
6.13. Dante.

This option of the "Config" menu allows you to configure the Dante module of the unit (1 in the case of NETBOX 32/8 AD MX and up to 4 in the case of NETBOX DSP) with the device name defined in the "NetboxRTC" application (see section 2.5 of this manual).

It also allows you to rename the transmission and reception channels of the Dante module with the names of the associated lines defined in the "Logical Lines" option of the "Config" menu (see section 6.8 of this manual). If any line is configured as stereo, at Dante level the 2 associated lines will be renamed with the L and R ending.

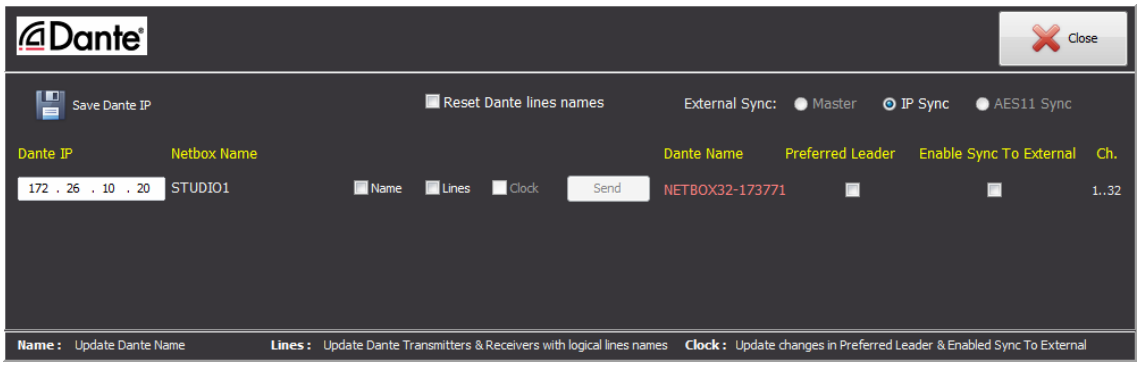
Finally, this option also allows you to modify the clock configuration at Dante level, taking into account the way that the device's synchronization is configured at local level.

When you select the "Dante" option, the following window appears:



The first thing to do is configure the IP address of the Dante module in the "Dante IP" field and press the associated "Search" button.

When connection to the Dante module is established, the previous window changes to:

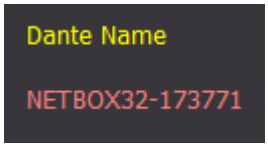


IMPORTANT NOTE: If changes are made to the Dante module from the "Dante Controller" application (for example, its name is changed), we must close and open again this window in order to refresh the information.

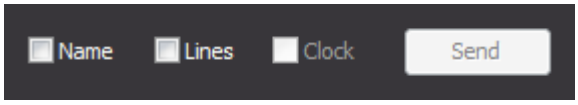
The button

allows to save in the database the IP address of the Dante module that has been configured so you don't need to write it every time you enter this window. If you close the window and open it again, the connection to the Dante module is established automatically.

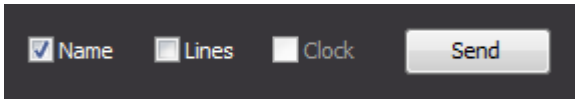
The "**Netbox Name**" field shows the device name defined in the "NetboxRTC" application and the "**Dante Name**" field shows the name of the Dante module (if they don't match, the second one appears in red):



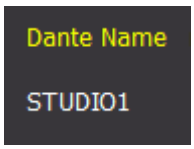
In the central part of the window, the "**Name**", "**Lines**" and "**Clock**" tic-boxes allow you to select which changes you want to make to the Dante module: its name, the name of the Dante channels and/or the clock configuration. Once one or more of these tic-boxes is selected, the "Send" button is enabled, allowing the configuration to be sent to the Dante module.



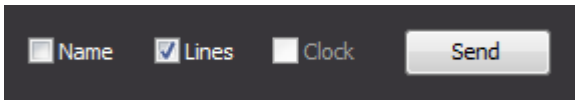
When you check the "**Name**" box and then press the "**Send**" button, the name of the Dante module (the one that appears in the "Dante Controller" application) is changed to the device name defined in "NetboxRTC" ("Netbox Name"):



Once the change is made, the name is refreshed and no longer appears in red in the "Dante Name" field:



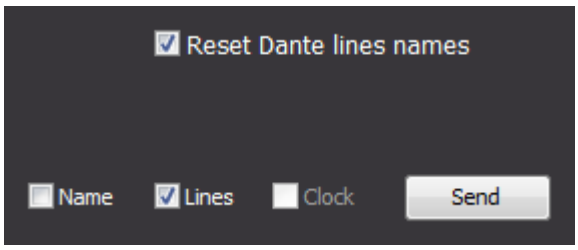
When you check the "**Lines**" box and then press the "**Send**" button, the names of the transmission and reception channels of the Dante module (the ones that appear in the "Dante Controller" application) are changed to the names of the associated lines defined in "NetboxRTC":



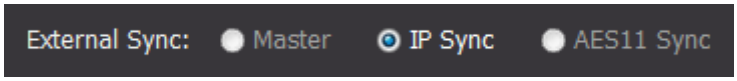
The "**Reset Dante lines names**" tic-box allows you to rename the transmission and reception channels of the Dante mode with the default names, as they leave the Factory, that is:

NBOX32_1L, NBOX32_1R, NBOX32_2L, etc.
NBOX8_1L, NBOX8_1R, NBOX8_2L, etc.
NBOXDSP_1L, NBOXDSP_1R, NBOXDSP_2L, etc.

In order to do that, just check that box (automatically, the "Lines" box is also checked) and press the "Send" button:

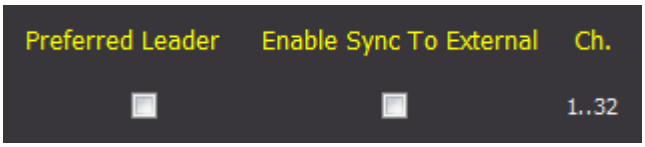


In the upper right part of the window, the "**External Sync**" informative section appears, which shows the way that the device's synchronization is configured at local level:

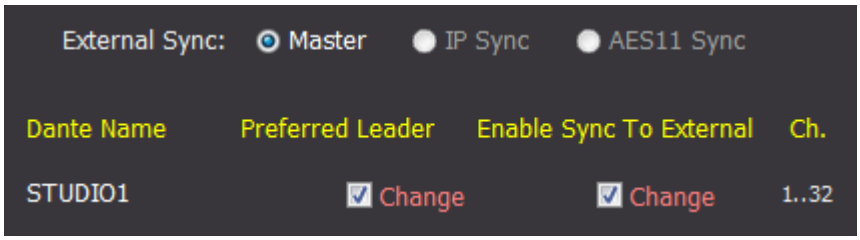


If you want to change that configuration, you must do it from the "AEQ NetBox Tool" application (see section 3.2.2.1 of the user's manual of the unit).

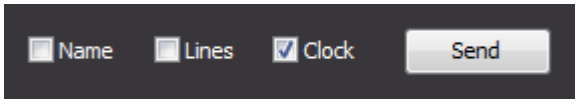
Below this section, the information about the way that clock is configured at Dante level appears (and, at the right, the available Dante channels of the module):



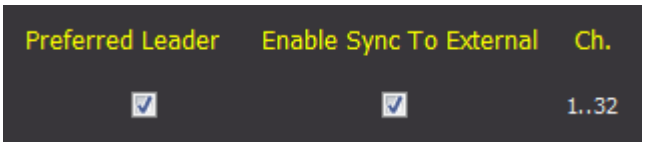
Depending the way the synchronization is configured at local level, the clock should be configured at Dante level, and vice versa (see section 4.7.4.3 of the AEQ AoIP user's manual). If configuration is correct, the information will appear as in the image above; if not, in the "**Preferred Leader**" and/or "**Enable Sync To External**" boxes the "Change" indication will appear in red and it will indicate the way that the associated box in "Dante Controller" should be (checked or not):



In this example, the unit is configured as "Master" at local level and therefore the "Preferred Leader" and "Enable Sync To External" boxes should be checked in "Dante Controller", but they are not (so the "Change" indication appears). In order to change it, you can do it from the "Dante Controller" application or, if you prefer, from this window, by checking the "Clock" box and then pressing the "Send" button:

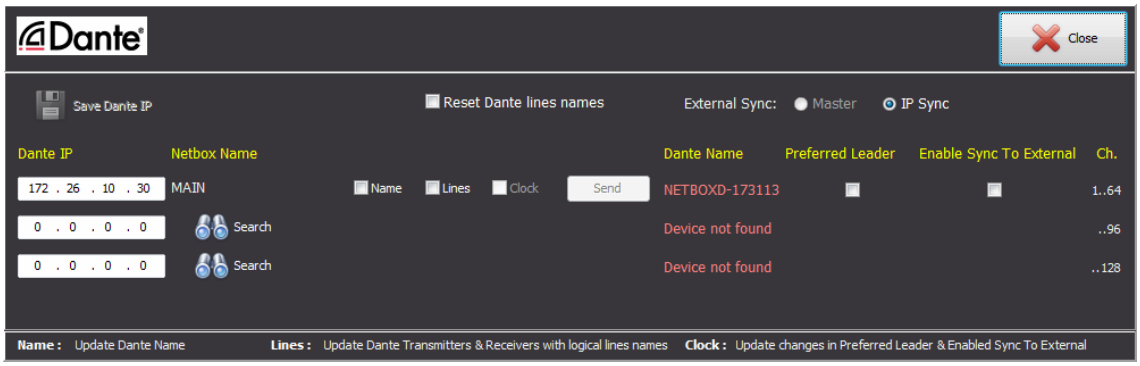


The "Change" indication then disappears to indicate that configuration is now correct:

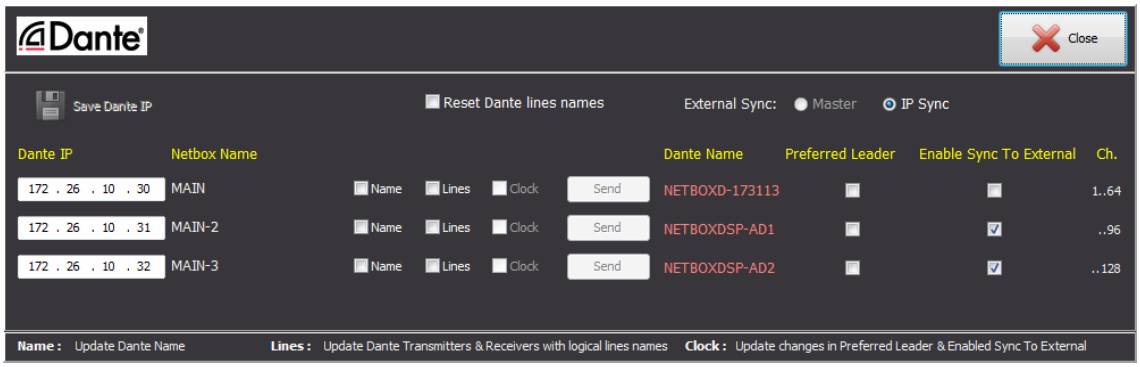


As previously indicated, in the case of **NETBOX DSP**, depending on the model, 1 Dante module (NETBOX DSP 64), 2 (NETBOX DSP 96), 3 (NETBOX DSP 128) or 4 (NETBOX DSP 160) will appear reflected.

For instance, in the case of NETBOX DSP 128, the window accessed by the "Dante" option looks like this (after configuring the "Dante IP" field with the IP address of the main Dante module and pressing "Search"):



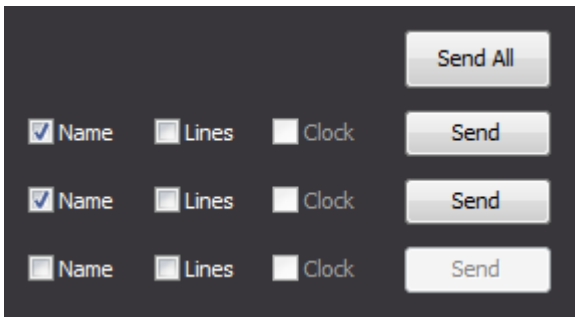
As shown, in this case 3 lines appear that correspond to the 3 Dante modules installed in the unit. In order to access to all information, in the "Dante IP" field of lines 2 and 3 you should configure the IP address of the additional Dante modules and then press the associated "Search" buttons:



The Dante channels associated with each module appear in the right column.

In this case, if you want to modify any Dante parameter, for example the name of the modules, you can do it one by one, by checking the "Name" box and then pressing the "Send" button that gets active:

or change several at once, by checking the "Name" box of all the modules you want to rename and then pressing the new "**Send All**" button above:



6.14. HW Switch.

This option of the "Config" menu allows you to create configurations in order to activate crosspoints to a certain output line according to the status combination of up to 7 physical or virtual GPIs.

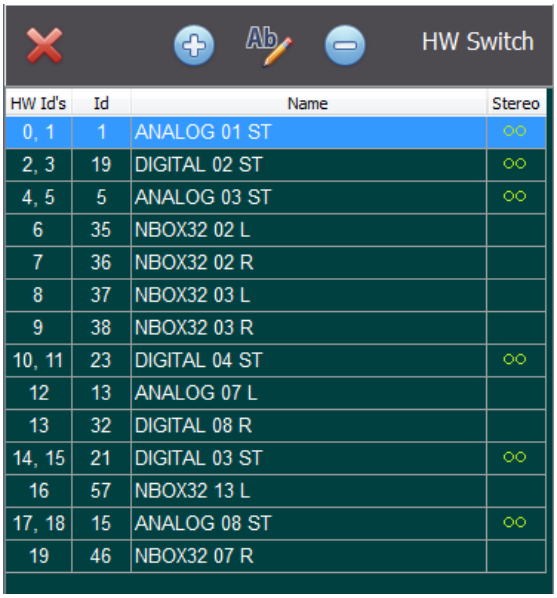
That HW Switch configuration is stored on the NETBOX unit (in its non volatile memory) and no server is needed in order to execute it.

You can create up to 20 HW Switch configurations per NETBOX, but keep in mind that when the output line is stereo the configuration counts double. Different inputs can be routed to each output depending on the switching of the configured GPIs.

In this working mode, the outputs will always act as NON summable, so the last GPIs status will define which input is routed to the output.

When virtual GPIs are used, the GPI number corresponds to the one configured in the " Virtual GPI to Physical GPO association" area of the "Remote IP" section of the "Config" menu: it won't be managed the way explained in the corresponding section of this manual but it will be used as GPI status for HW Switch.

When you click on "**HW Switch**" option in the "Config" drop-down menu, the list of defined HW Switches window will open (each one shows the associated output list):



HW Id's	Id	Name	Stereo
0, 1	1	ANALOG 01 ST	∞
2, 3	19	DIGITAL 02 ST	∞
4, 5	5	ANALOG 03 ST	∞
6	35	NBOX32 02 L	
7	36	NBOX32 02 R	
8	37	NBOX32 03 L	
9	38	NBOX32 03 R	
10, 11	23	DIGITAL 04 ST	∞
12	13	ANALOG 07 L	
13	32	DIGITAL 08 R	
14, 15	21	DIGITAL 03 ST	∞
16	57	NBOX32 13 L	
17, 18	15	ANALOG 08 ST	∞
19	46	NBOX32 07 R	

The buttons available in the upper part of the list are the following:

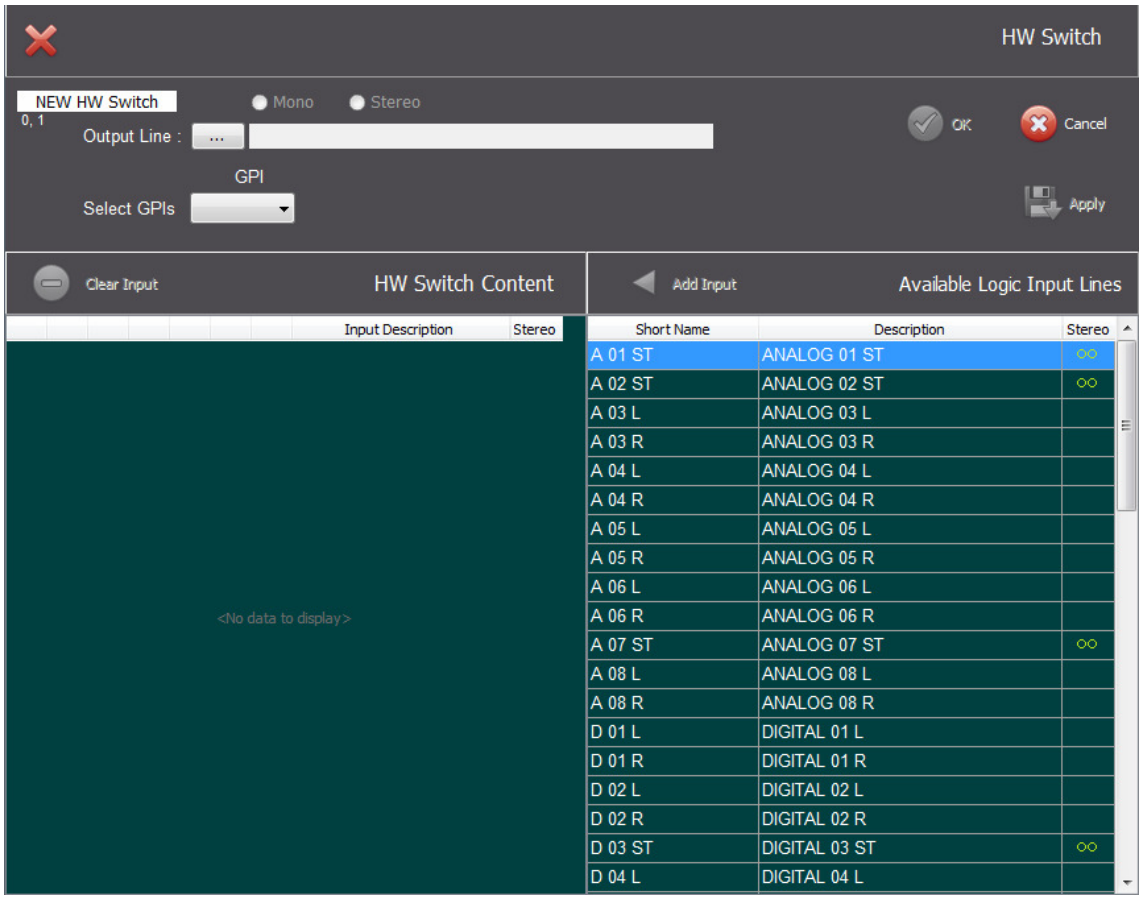
allows you to close the window.

allows you to create a new HW Switch.

allows you to edit an existing HW Switch.

allows you to delete the selected HW Switch from the list. Confirmation is required.

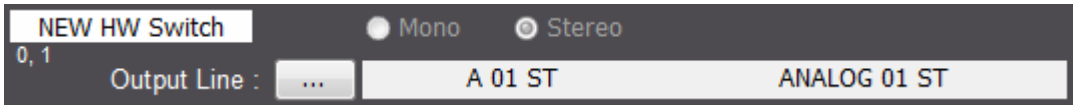
If no HW Switch has been created yet, the list will be empty. Click on the button and the window for creating HW Switch will appear (this is the same window that will allow you to edit an existing HW Switch). From the upper left part of the window from left to right and going downwards, the following fields and controls are available:



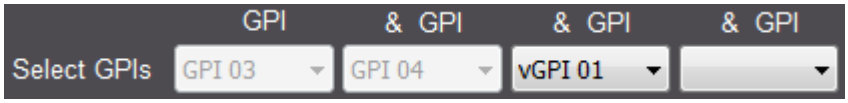
allows you to close the "NEW HW Switch" / "EDIT HW Switch" window.

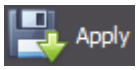
Below this area, the HW Switch information area will appear:

- "**NEW HW Switch**" or "**EDIT HW Switch**" field indicates if we are creating a new HW Switch or editing an existing one.
- "**Output Line":** field allowing us to select the output line associated to HW Switch. The ... button will give access to a list in order to make that selection. Once selected it will be shown whether the output is mono or stereo:



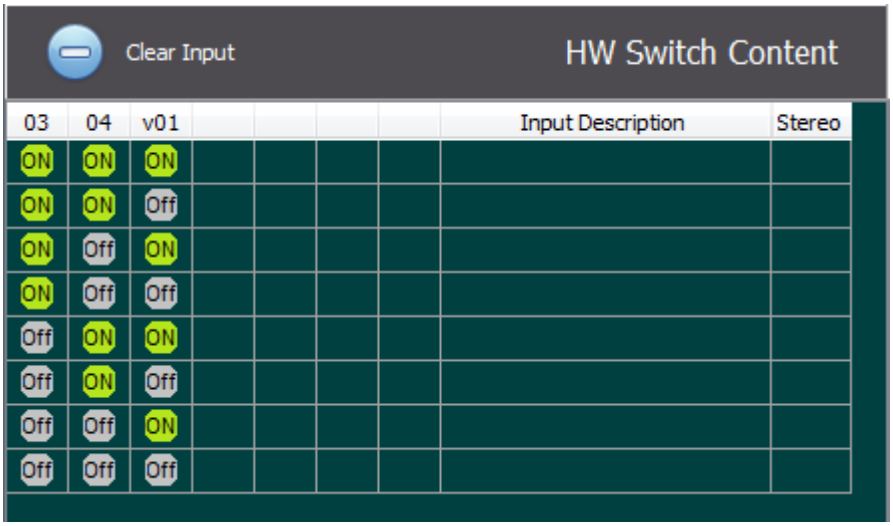
- "**Select GPIs**"**:** field allowing us to select the GPIs (between 1 and 7) that will define the inputs routing according to their status. When the first GPI is selected a new field will appear in order to select the second GPI and so on (up to a maximum of 7).



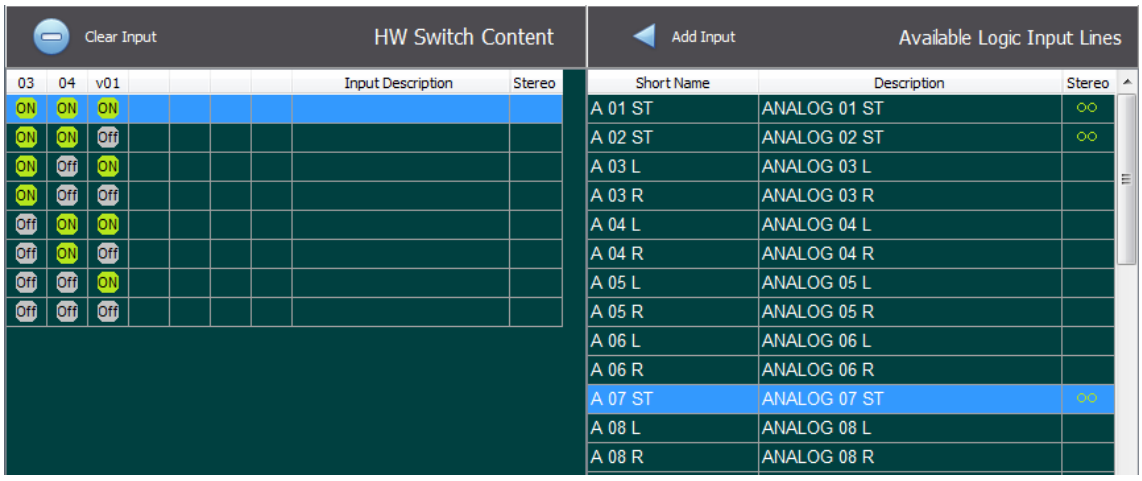
Once selection is finished the 

button should be pressed.

Then the "**HW Switch Content**" are will show a list with the possible combinations of the selected GPIs status (in the example the physical GPIs 3 and 4 and the virtual GPI 1):

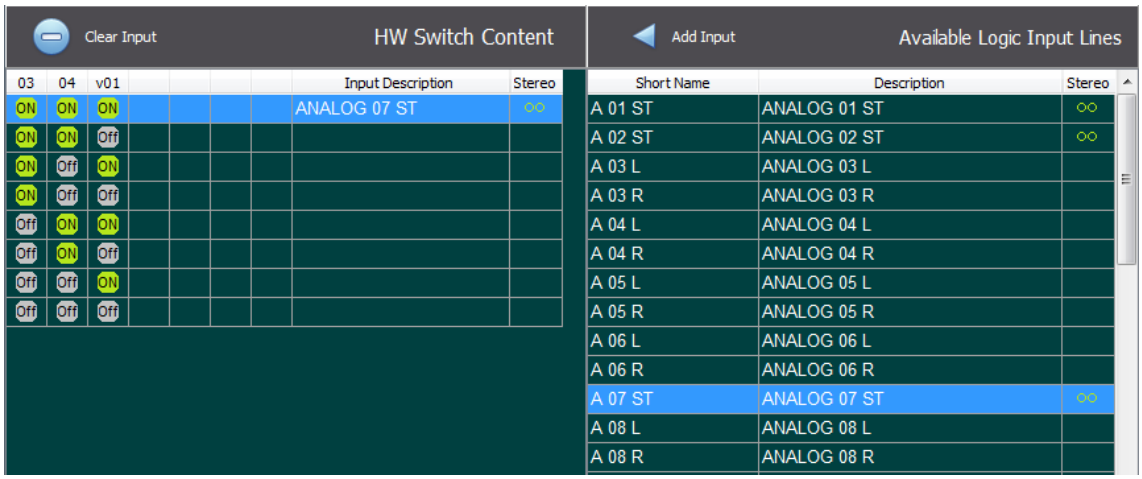


Next one of the combinations should be selected (line is marked blue) and then, in "Available Logic Input Lines" list, the input line that will be routed to the associated output line when that combination of the selected GPIs status is detected.

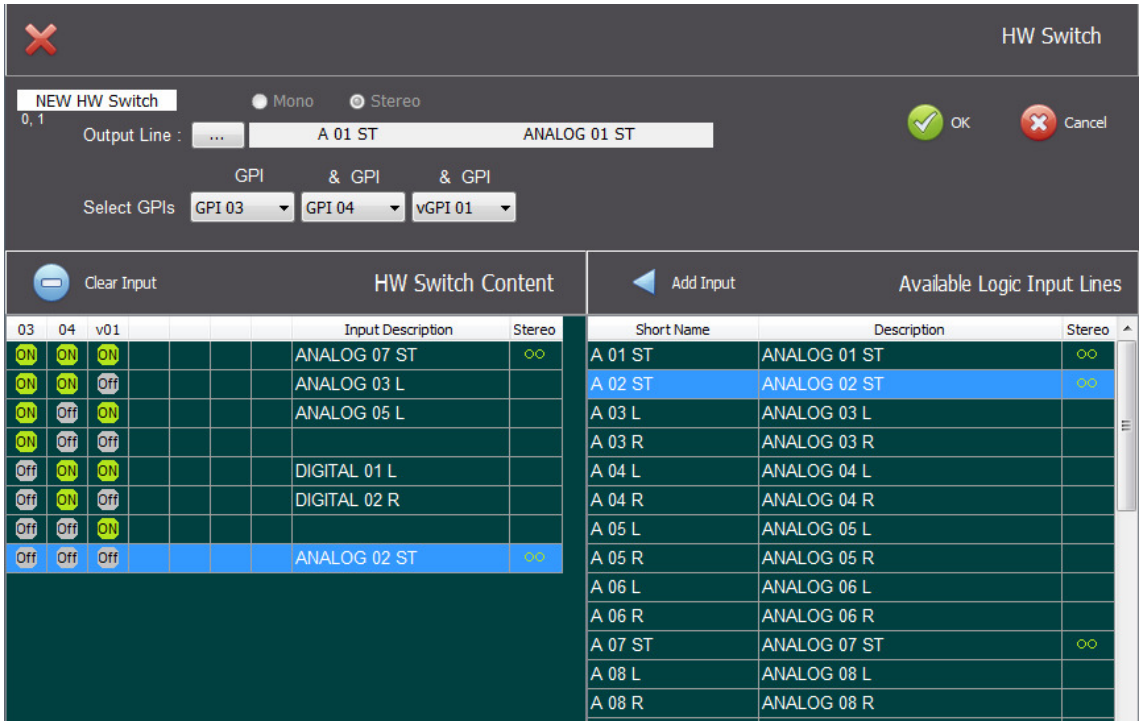


Once selection is done the

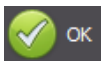
button should be pressed, so the input line will appear in the "HW Switch Content" list, where it will also be shown whether it's stereo or not:



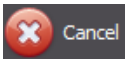
This way you can select the different status combinations and then associate the desired inputs:



In the example, we have defined the routing associated to 6 of the 8 possible status combinations of the 3 selected GPIs.

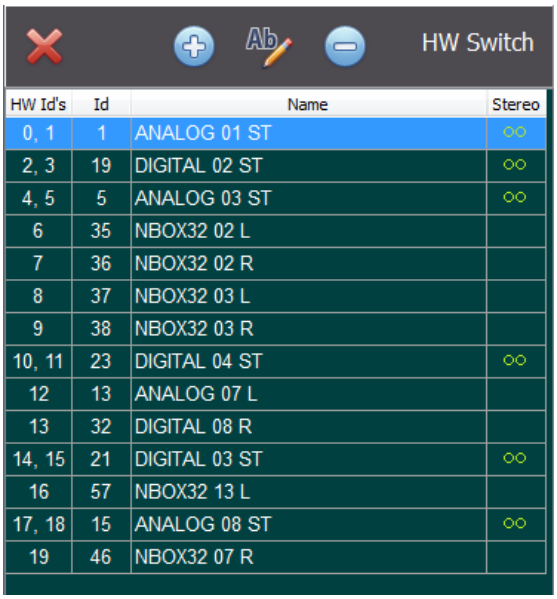
Once selection is finished the 

button should be pressed to confirm the creation or edition of the HW Switch and store it on the NETBOX unit.

The 

button allows you to discard the creation or edition of the HW Switch.

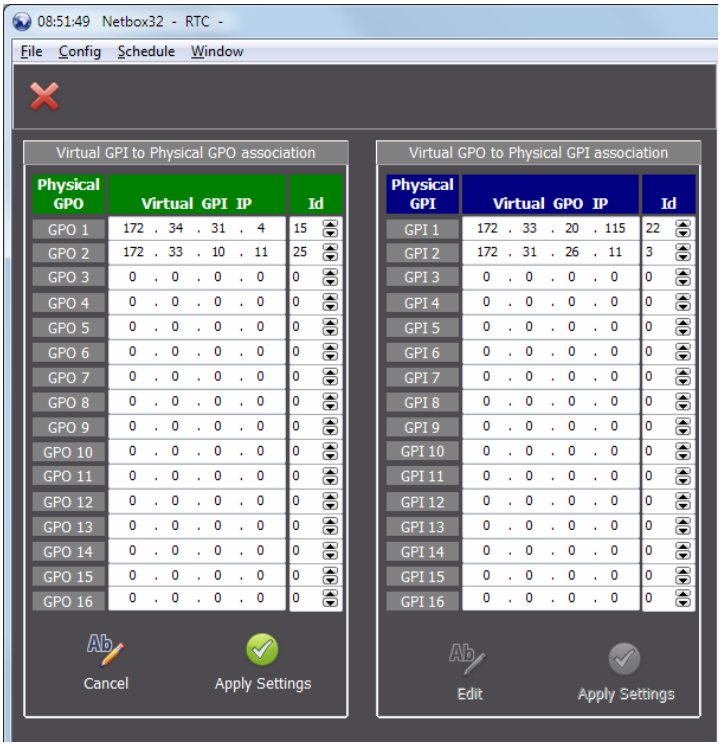
The list of defined HW Switches shows the HW Switch identifier in the first column ("HW Id's"), double when the associated output line is stereo. The following columns shows the identifier number of the output line ("Id"), the line description ("Name") and whether it's stereo o not ("Stereo"):

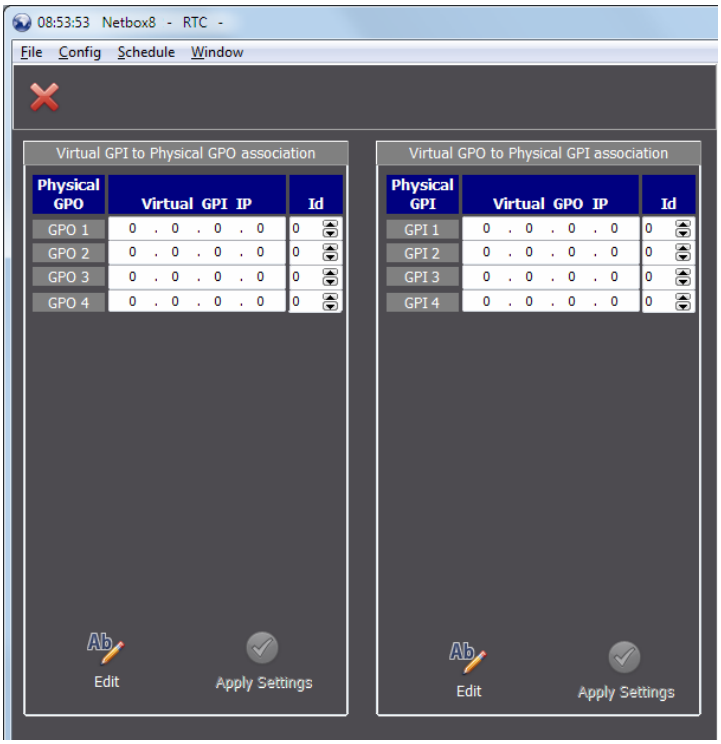


HW Id's	Id	Name	Stereo
0, 1	1	ANALOG 01 ST	∞
2, 3	19	DIGITAL 02 ST	∞
4, 5	5	ANALOG 03 ST	∞
6	35	NBOX32 02 L	
7	36	NBOX32 02 R	
8	37	NBOX32 03 L	
9	38	NBOX32 03 R	
10, 11	23	DIGITAL 04 ST	∞
12	13	ANALOG 07 L	
13	32	DIGITAL 08 R	
14, 15	21	DIGITAL 03 ST	∞
16	57	NBOX32 13 L	
17, 18	15	ANALOG 08 ST	∞
19	46	NBOX32 07 R	

6.15. Remote IP.

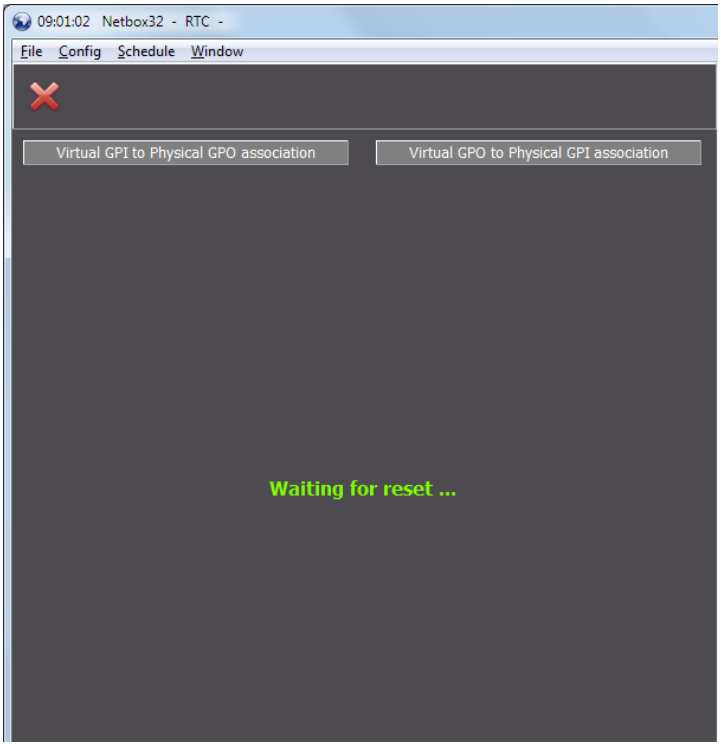
NETBOX 32 AD MX and NETBOX DSP have 16 physical opto-coupled GPI/O's, while NETBOX 8 AD MX has 4 physical opto-coupled GPI/O's. This section allows the user of "NetboxRTC" to define what physical GPI or GPO corresponds to what virtual GPO or GPI, i.e. a sort of translation table for physical to virtual GPI/O's. Any modification made in this section will be stored in the non volatile memory of the NETBOX and will reset the communications between the SW and the unit.





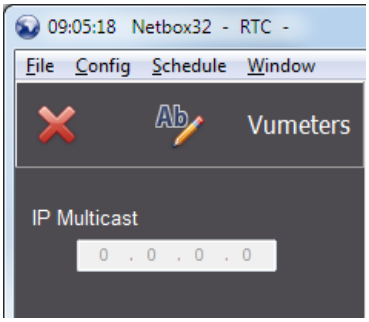
To edit the virtual GPI to physical GPO association, click on the "Edit" button. The table header will turn green and allow the user to edit the IP address and the Id number for the virtual GPI expected and that should correspond to a GPO that would be physically connected to the NETBOX AD unit. The process is the same for the association of a virtual GPO to a physical GPI, a device that is connected to the NETBOX unit could send a command to GPI 1 that in this case translates to IP address 172.33.20.115 and Id number 22.

When finished click on the "Apply Settings" button and the configuration will be stored in the non-volatile memory on the NETBOX. A message will indicate that the communications between the unit and the control application software is resetting:



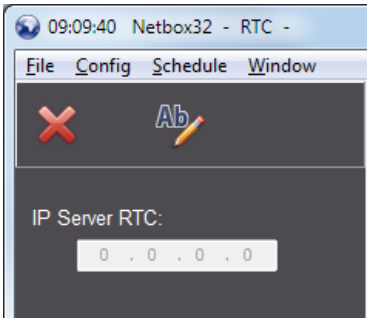
6.16. Vumeters.

This option of the "Config" menu allows you to add or edit a Multicast IP address for Vumeters. This configuration is stored in the non volatile memory of the NETBOX. When there is more than one NETBOX device in our network a different IP Multicast should be associated to each device.



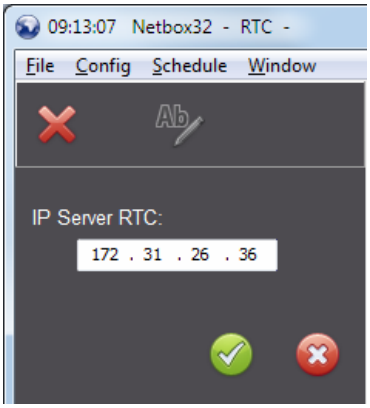
6.17. Netbox - IP Server RTC.

The last option in the "Config" menu is the "Netbox - IP Server RTC". This is where you need to indicate or modify the IP address where the Service and Server application of the "NetboxRTC" application is hosted and running. This is usually also the location for the database system of the "NetBoxRTC".



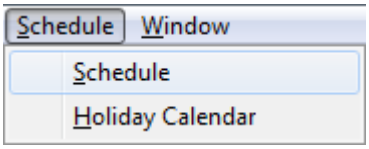
The virtual GPI's sent from external equipment are point to the IP address of the NETBOX and not to the IP address of the "NetboxRTC" Server application. On the other hand, the NETBOX does not have sufficient system memory capacity to keep such vast and complex information, so you are relying on the Server application and the associated database to accomplish the configuring of the NETBOX matrix, according to salvos and macros. Therefore, you need to indicate to the NETBOX where this service is hosted so that when a virtual GPI and Id Number is received by the NETBOX (triggered remotely) it basically "relays" the received information to this IP address that is hosting the Server application. The Server application and database may handle several NETBOX devices. At the footer of the "NetboxRTC" application window this IP Address is visible:

To edit the IP address, simply click on the button. Now you can edit the IP address where the RTC server application and database are stored. This can actually be the same computer where the client application "NetboxRTC" is installed. Once the IP address has been completed, either click on the button to confirm the address or the button to cancel. This configuration is stored in the non volatile memory of the NETBOX.

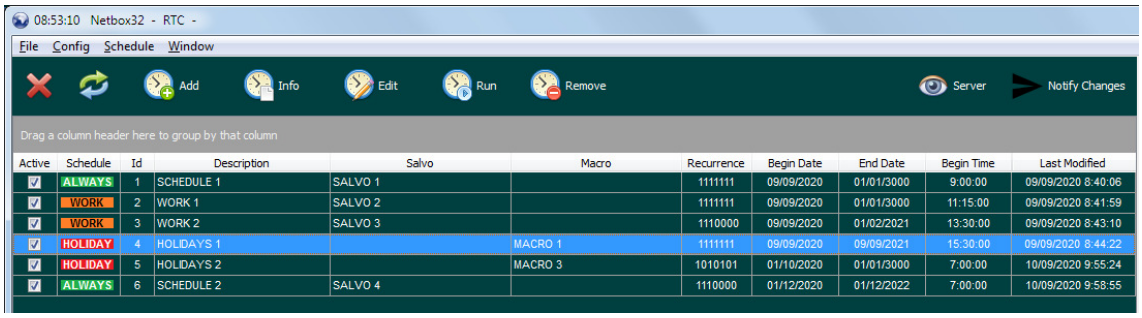


IMPORTANTE NOTE: In case the Server IP address is not defined or there is no connection to Server, no defined schedule could be executed and alarms or GPIOs won't work (although the physical/virtual GPIOs association described in section 6.11 of this manual would be operative). In that case the RTC application could be used to connect/disconnect crosspoints, protect/unprotect crosspoints or execute salvos and macros, but always manually.

7. DROP-DOWN MENU: Schedule.



A schedule is a way to automate the execution of salvos and macros. What becomes important when using schedules is that the system time is synched correctly for example to an NTP or a station Masterclock. When you open the "**Schedule**" option from the "Schedule" drop-down menu, the schedule menu and list will appear:

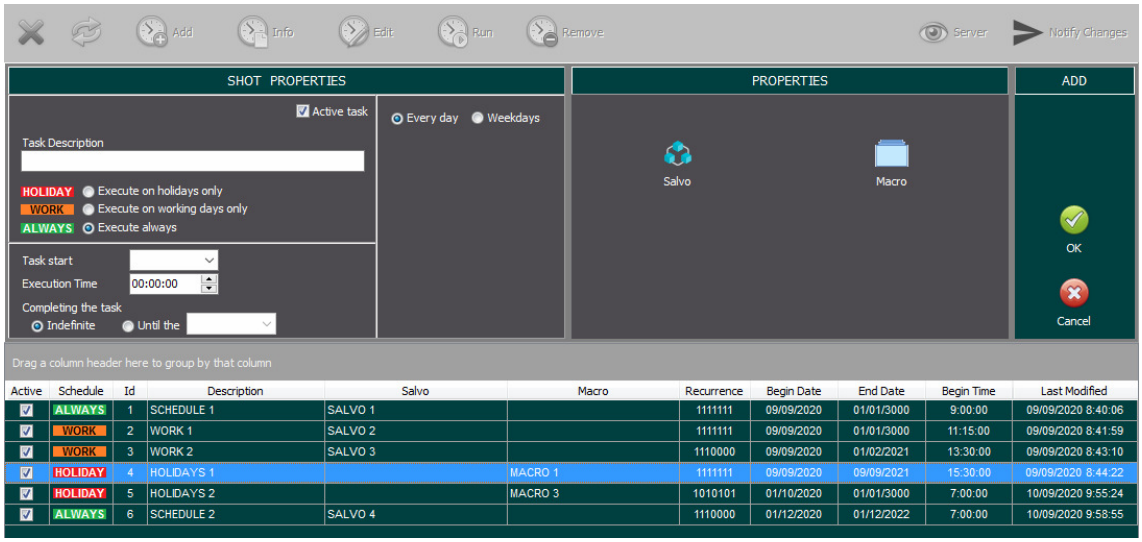


The buttons that will appear immediately beneath the application drop-down menus are:

Allows to exit the schedule menu.

Allows to refresh the schedule list. This is useful when several users are operating the "NetboxRTC" from different positions.

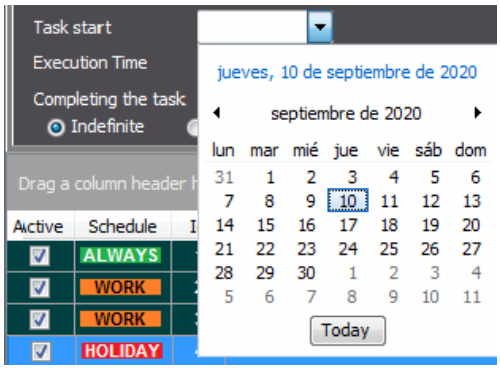
Allows to open the "Add Schedule" dialogue and menu:



From left to right, the first section is "**SHOT PROPERTIES**". This section defines all the basic properties in regards to time and dates of the scheduled task. Here you can activate the task or schedule by clicking or selecting the "Active task" tic box. When a task or schedule is active this means that it will be executed as programmed. If not active, the task is programmed but will not be executed.

The "Task Description" field allows the user to edit a description for the scheduled task. Right beneath this field, you can define if the scheduled task should be executed always, only on holidays or only on working days, according to the calendar that you have defined in the "**Holiday Calendar**" option of the "Schedule" drop-down menu (more information at the end of this section).

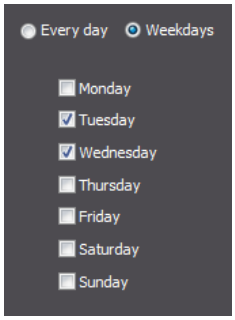
You can define a date to start the task and the exact time that you want the task to be executed. You can also decide if this task should be executed as indefinitely or with a specific end date. The "Task start" field is a date select field and will return the users computer date and calendar; here you will select the date that the scheduled task should start being executed from.



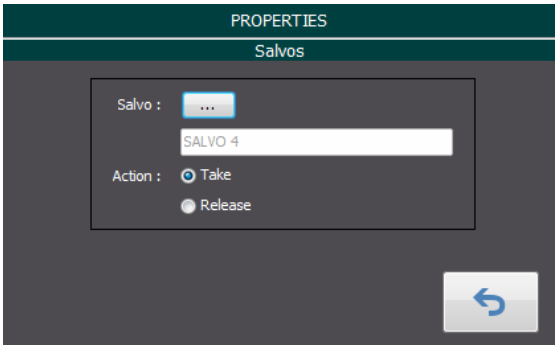
The "Execution Time" field is the time that you define that the scheduled task should be triggered or "shot".

"Completing the task" area has two options: "Indefinite" and "Until the" date. "Indefinite" option will begin executing the scheduled task until you either make it inactive or you delete the schedule. The "Until the" field will define a date limitation or time window, i.e. from the task start until the last date of execution of the scheduled task. Even if the schedule remains active after the last date of execution defined, the scheduled task will not be executed again after this date.

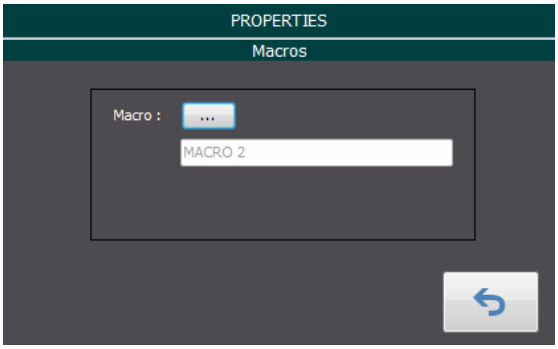
Immediately to the right and still in the section of "SHOT PROPERTIES" there are two options: "Every day" and "Weekdays". If you want to define specific weekdays for the scheduled task to be executed, you select the "Weekdays" option. This will provide access to the selection of the specific weekdays – Monday through Sunday. Simply select what weekdays the scheduled task should be executed on within the date and time limitations previously defined.



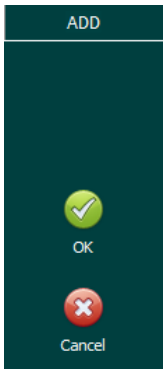
The next section to the Right is "**PROPERTIES**". It has two buttons available: "Salvo" and "Macro". The "Salvo" button will open a window where the ... button will give access to a list of the existing salvos in order to choose one of them. Once the salvo has been selected it will be shown in the field immediately below the button. Below this field there are two "Action" options that can be selected – "Take" or "Release", meaning that you either choose to "Take" the salvo or to "Release" the salvo. The last button in the lower right corner will return the "PROPERTIES" section to present the "Salvo" / "Macro" buttons and without further action, i.e. no changes will be made to the scheduled task.



The "Macro" button will open a window where the ... button will give access to a list of the existing macros in order to choose one of them. Once the macro has been selected it will be shown in the field immediately below the button. The last button in the lower right corner will return the "PROPERTIES" section to present the "Salvo" / "Macro" buttons and without further action, i.e. no changes will be made to the scheduled task.

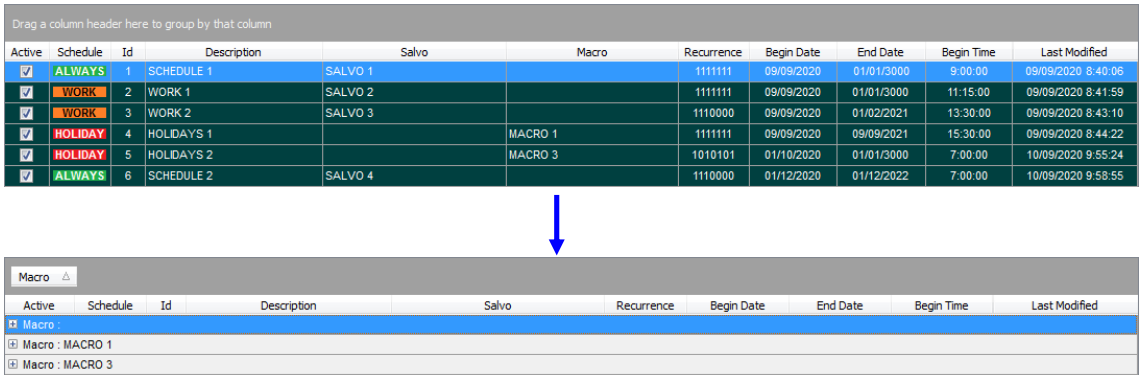


The last section in the add and edit schedules window is the "**ADD**" section. This section only provides two buttons. The "OK" button will confirm all the changes and edits to the scheduled task and add the task to the database if you are in the add schedule mode. If you are editing an existing schedule (the section is now called "EDIT") it will overwrite this schedule with the changes that you have introduced.

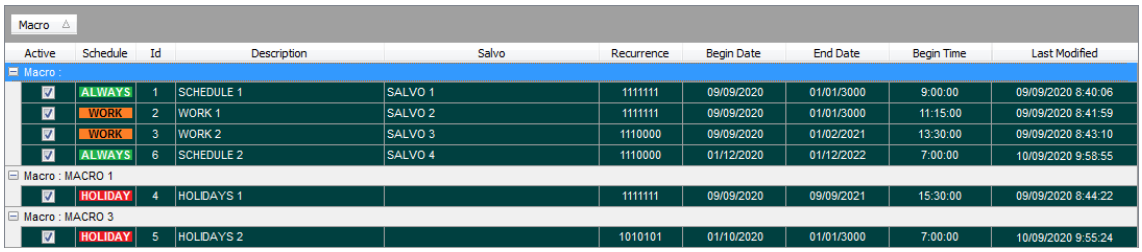


"Cancel" will cancel all edits to the current schedule. If you are adding a scheduled task to the list of schedules this will be ignored and no schedule added. If you are editing an existing schedule, all edits to this schedule will be ignored and discarded.

Immediately below the scheduled task sections there is a grey field holding the text "Drag a column header here to group by that column". This becomes very useful when you have a long list of scheduled tasks as it allows us to group the tasks. For example if you drag the "Macro" header into the grey area the list view would change to something similar to this example. The tasks are now grouped by macros.



If you expand the view for each of the macros in this new list, you will obtain a view as follows:



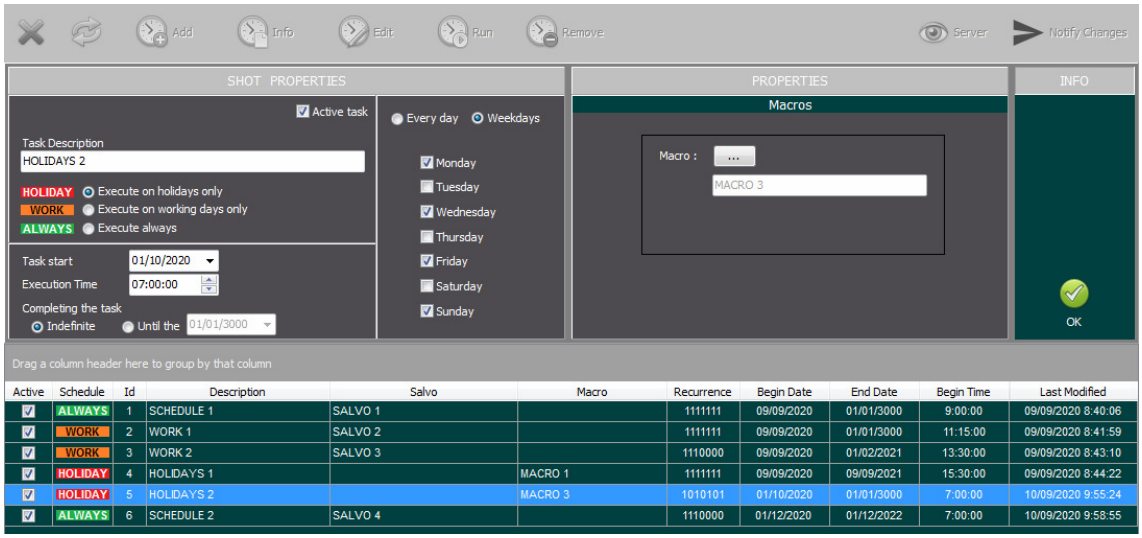
The column headers are the following:

- "**Active**": if the tic box is marked or selected, this means that the schedule is active, i.e. has been set to active and will be executed when the moment programmed to trigger the task arrives. If inactive or the tic box is not marked or selected, the schedule or the task has not been marked active and will therefore not be executed.
- "**Schedule**": indicates when the task will be executed according to calendar. You can choose to execute this a working day ("WORK"), a holiday ("HOLIDAY") or any day of the week whether it's a working day or not ("ALWAYS").
- "**Id**": this is an automatic counter and is just showing the number (order of creation) of the schedule in the database. The numbering is automatic and a number order will never be re-used, i.e. if you create schedule 20 and at a point in time delete this schedule, no schedule will receive number 20 as Id.
- "**Description**": allows the user to provide a brief description of the schedule.

- "**Salvo":** indicates what salvo is used in the scheduled task. If this field is empty this means that no salvo has been defined to be part of the schedule.
- "**Macro":** indicates what macro is used in the scheduled task. If this field is empty this means that no macro has been defined to be part of the schedule.
- "**Recurrence":** indicates in binary the recurrence in a representation of seven days and with the first day being Monday. For instance, 0101011 would indicate that the schedule will be executed on Tuesday, Thursday, Saturday and Sunday and would not be executed on Monday, Wednesday and Friday.
- "**Begin Date":** indicates the date from when the schedule will start to be executed.
- "**End Date":** indicates the last date that the schedule will be executed on the time that has been configured.
- "**Begin Time":** indicates the exact time when the schedule will be executed on the "Begin Date" and with the "Recurrence" that has been configured.
- "**Last Modified":** shows the last date and time that the schedule was modified by any of the users with access to the "NetboxRTC" application.

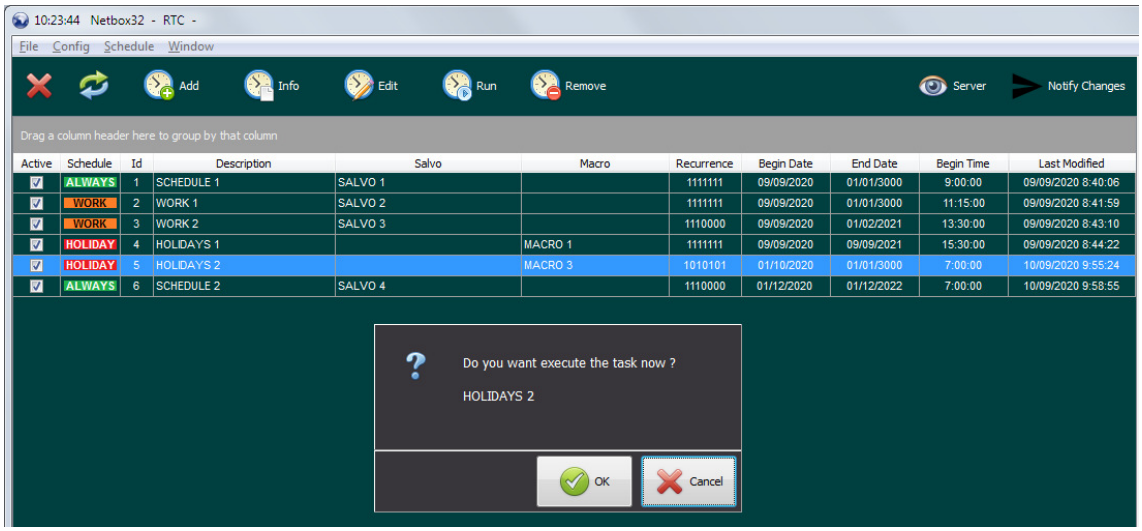
The "Info" button will open a similar view of the scheduled task as the add or edit schedules window.

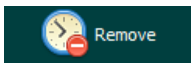
This is however merely an informative view of the schedule. To obtain information about the scheduled task you need to select the schedule form the list below and then click the" Info" button:



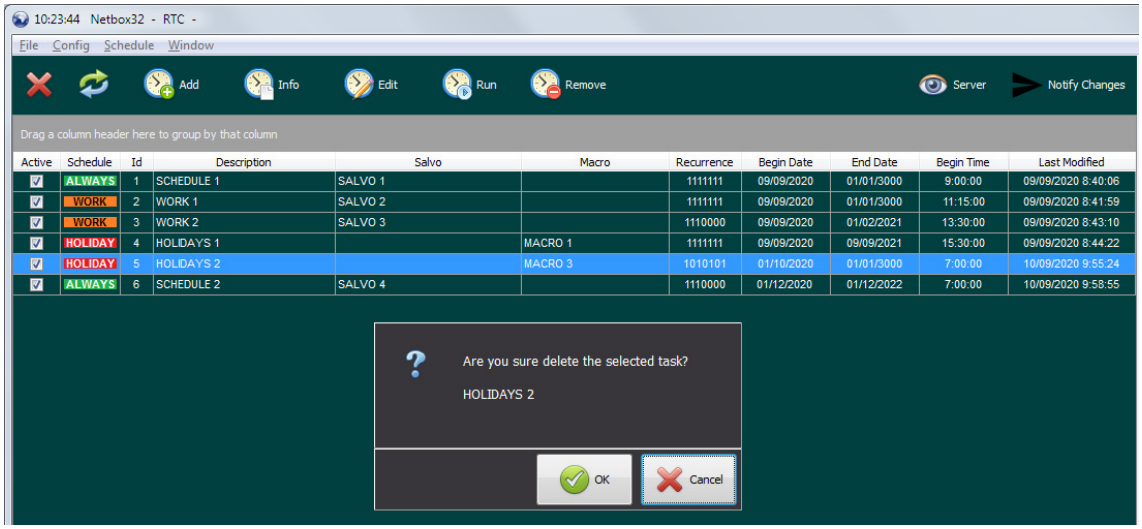
The "Edit" button will open the selected schedule and allow the user to modify the specifics of the scheduled task. Select the scheduled task to be edited from the list and then click on the "Edit" button. The procedure is exactly the same as the one described previously to add a schedule.

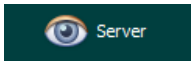
The "Run" button allows the user to run a schedule manually and at will, for example outside the established dates and time or an inactive schedule. Select the scheduled task form the schedule list and click on the "Run" button. The user will be prompted to confirm the execution of the scheduled task.



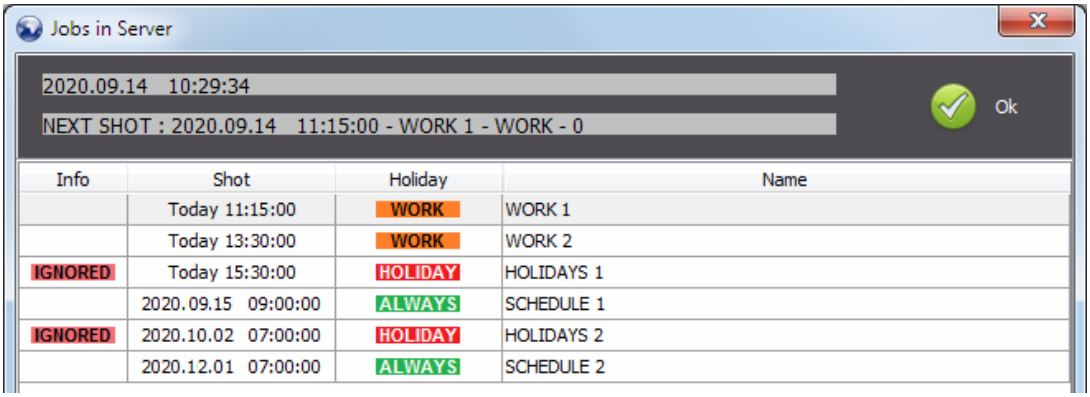


The "Remove" button allows the user to remove or delete a schedule from the list of scheduled tasks. Select the schedule to be deleted and click on the "Remove" button. The user will be prompted to confirm the deletion of the scheduled task.

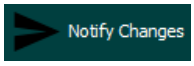




The "Server" button allows the user to consult the scheduled tasks that are programmed to be executed by the "NetboxRTC" Server. When the user clicks on this button the following window will appear:

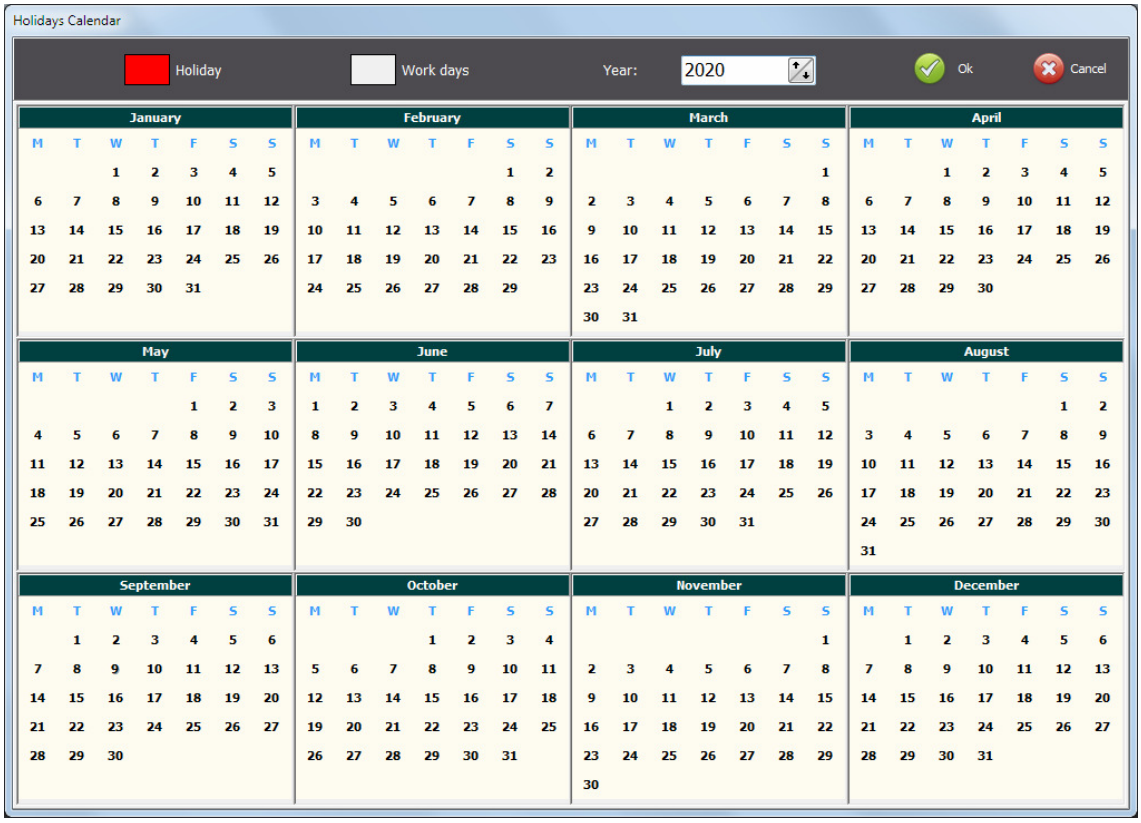


This windows provides information concerning the scheduled tasks or jobs that are pending for the Server application. It will provide information concerning the date and time that the query was sent to the Server application, the date and time for the next shot or task scheduled to be executed by the Server and its property "On" or "Off". Below this information the window will show the scheduled tasks, sorted in ascending order by date and time.

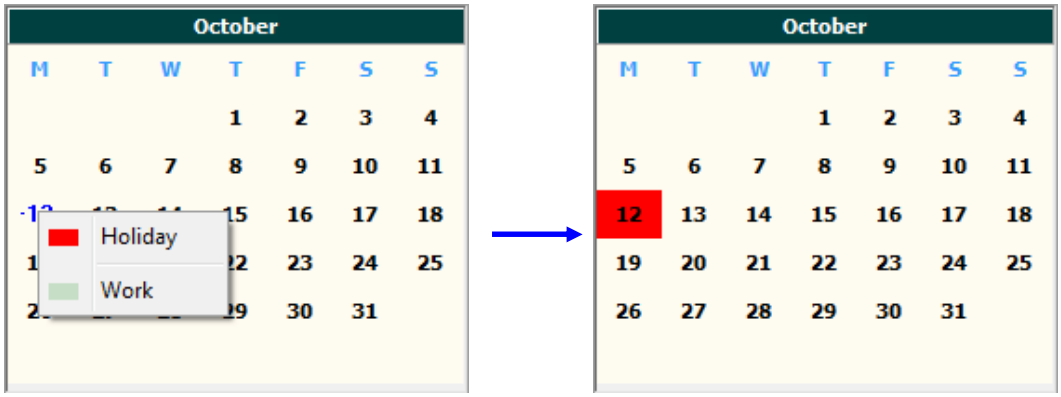


The last button in the schedules section is "Notify Changes". When clicking on this button the "NetboxRTC" application will request the "NetboxRTC" Server application to notify the Client application about any changes in the database and update the information concerning the scheduled tasks.

Finally, the "**Holiday Calendar**" option of the "Schedule" drop-down menu allows you to define a calendar of holidays and working days in order to use it to define when should be executed the different scheduled tasks. It's possible to configure different years.

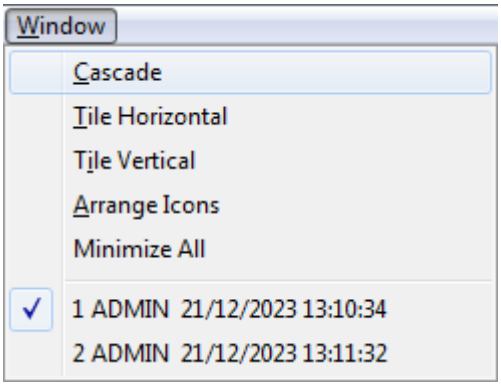


In order to do that, simply click on the desired day and select on the drop-down menu that appears if that day will be a holiday or a working day.

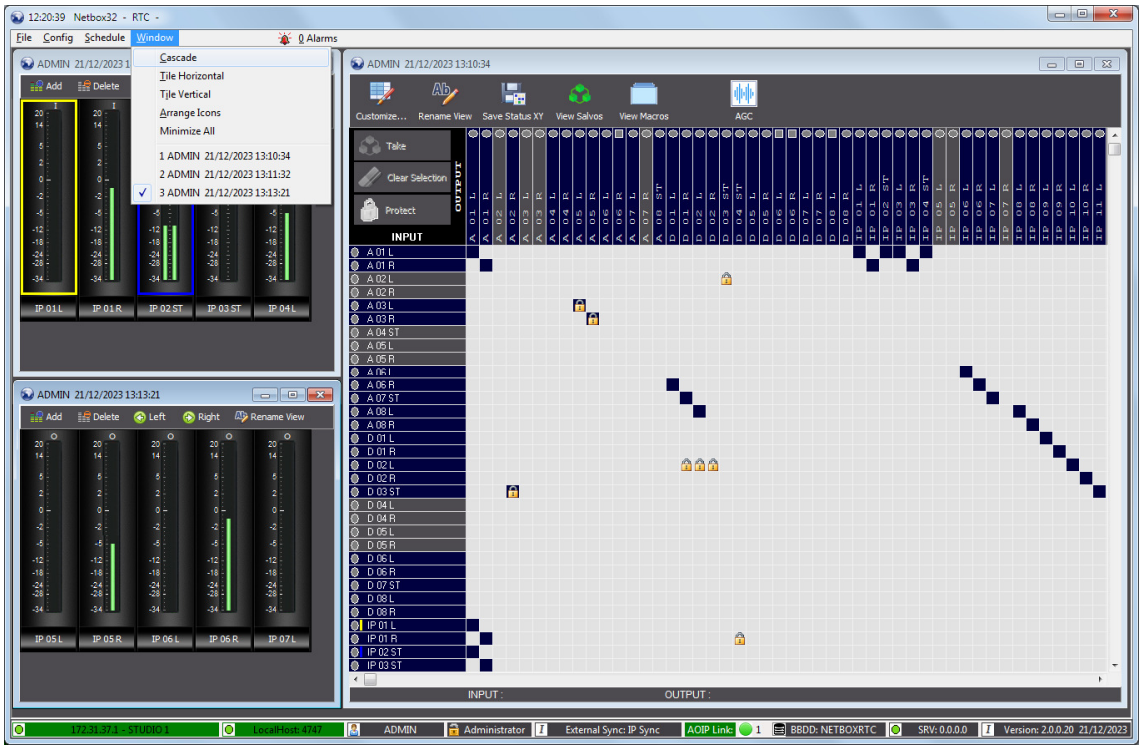


8. DROP-DOWN MENU: Window.

The last drop-down menu of the "NetboxRTC" application is "**Window**". This allows the user to organize the resulting workspace including its XY and vumeters views, as well as the precision vumeters, by re-sizing and placing these as per the user's requirements.



The available options are:

- "**Cascade":** this option will cascade all the views on the monitor.
- "**Tile Horizontal":** this option will tile all the views horizontally on the monitor.
- "**Tile Vertical":** this option will tile all the views vertically on the monitor.
- "**Arrange Icons":** if you have a large number of minimized views on our workspace and in different locations, this option will arrange all these icons. The icons representing the minimized views will be organized at the bottom of the main Window of the workspace.
- "**Minimize All":** this option will minimize all the open views and arrange them as icons at the bottom of the main Window of the workspace.

In the lower part of this menu the open views of the workspace are shown in order to provide a quick access to any of them.