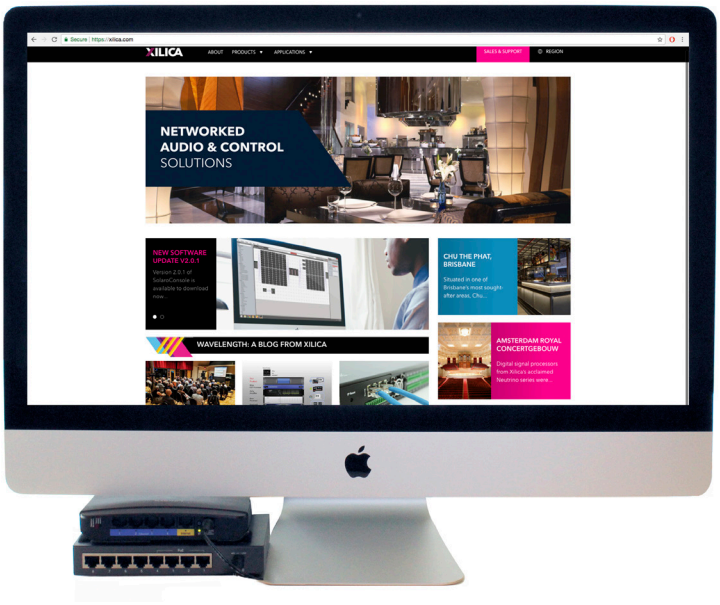
XILICA DESIGNER

User Manual

XILICA

Table of Contents



What you need to Provide

- Computer with a processor 1GHz or higher
- Windows 7 or higher
- Mac OS X 10.8 or higher
- 500 MB of available space
- 1 GB graphics card
- 4 GB of RAM
- Network interface (Router, PoE switch)
  A router is used for IP assignment and easy connectivity to computer and control devices.
  A PoE switch is used for controllers if local power is not used.
- Ethernet cable (Cat5/6)

Getting Help

Additional Help Files and video tutorials are available at our website: www.xilica.com
For further technical support, please email: support@xilica.com and we'll connect you with a solutions engineer. Alternatively, you can call our worldwide offices for immediate assistance:

North America & Rest of World: +1 905-770-0055
Europe: +31 29940-1100
China & Hong Kong SAR: +852 2604-9382

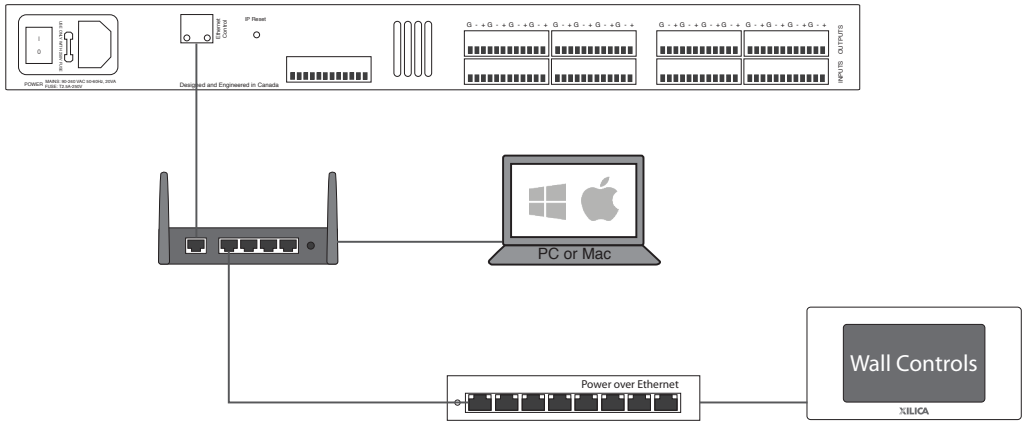
Initial Device Connectivity:

Xilica processors and control devices run on a network based infrastructure and are set up and controlled by a host computer via Ethernet using the Xilica Designer software.

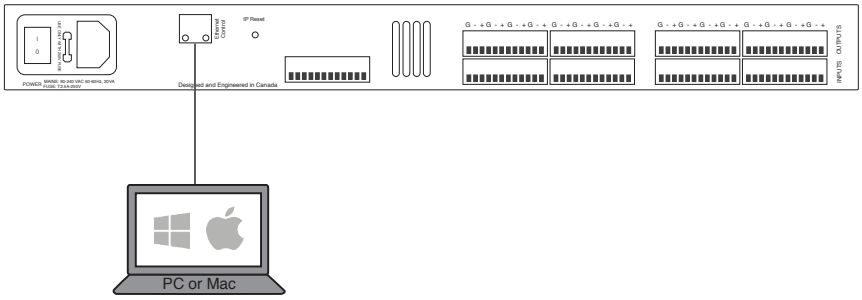
A network connection can be made between the computer and processor using:
a) DHCP enabled Router or Server/Router combination (Recommended)
b) or a non-DHCP direct connection or indirect connection via an Ethernet switch.

Note: DHCP enabled Router/switch gear should be turned on first, with all Ethernet cables connected to the hardware prior to powering on the Hardware. This will allow for proper handling of IP address distribution to the Hardware. The IP address assignment is automatic if connected to a DHCP enabled router.

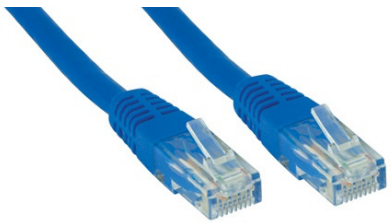
A) DHCP enabled router or server/router combination (Recommended)



B) Non-DHCP direct connection or indirect connection



All wired connections use a standard RJ45 Cat 5/6 (Ethernet) connection.
Xilica Designer and XTouch Applications can also be connected via a Wi-Fi connection, but this is not recommended.



Standard RJ45 cable

A) DHCP enabled router or server/router combination (Recommended)
With DHCP enabled routers and servers, the processor will automatically obtain the IP address upon power up and connection.
When Xilica NeuPanel Series wall controls will also be used, it is recommended to use a router and PoE switch. This combo provides DHCP as well as power to the wall controls. Linksys routers and Netgear switches are recommended.

B) Non-DHCP direct connection or indirect connection
When the processor is connected directly to a computer or indirectly via a switch or hub and DHCP is not avialble to assign IP addresses, the connection process is not automatic.

1. Single processor (Non-DHCP)
Once no DHCP is detected, the processor will either try to connect using the IP address last assigned and stored on the device or attempt to revert to its default IP address of 169.254.128.128.
Under some conditions, the processor may refuse to relinquish its stored IP addresses or revert to its default IP address and thus refuse to connect. We recommend performing an IP Reset (Page 12).

2. Multiple processors (Non-DHCP)
For multiple processor connected to the network with no DHCP available, the user will have to manually assign unique IP addresses to each device. (Page 14-16)

Connect Mains Power
Insert the supplied IEC power cable and connect the AC end of the cord into an AC power source of 90-240 VAC (50/60 Hz).

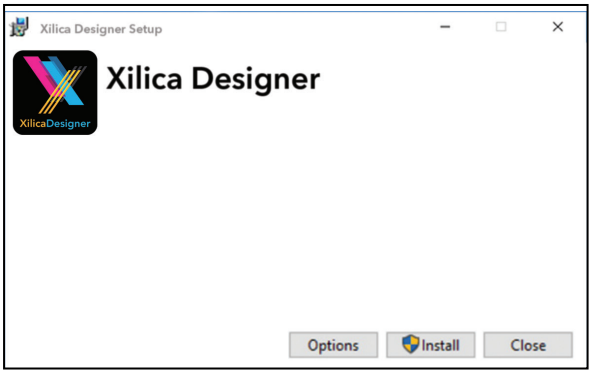
Power On Devices
With your processors and devices connected as a network or directly/indirectly to your computer, power on all devices. On power up, the processors blue power status LED and LCD display will light.
If the processor has an Ethernet cable connected, the orange Network LED will light once the processor initializes. (Please note that this does not mean you have established a network connection but only that an Ethernet cable is connected. Network connection is displayed in Xilica Designer's Network view.

Upon power up, the processor will search for a DHCP router or server to obtain an IP address.
If it locates a DHCP server or router, it will connect quickly. If not, the processor will revert to its default IP address (169.254.128.128). If you are using a non-DHCP direct or indirect connection, please follow 'Manual IP address assignment' on page 13-15.

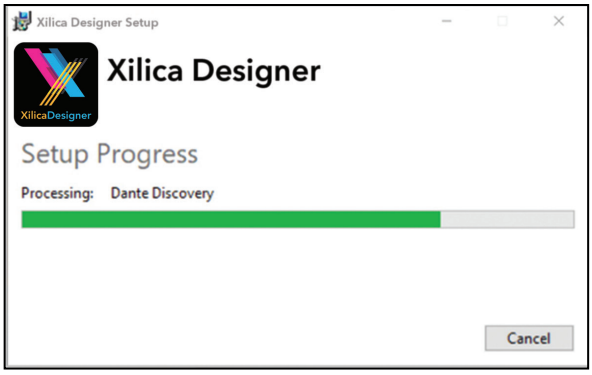
Xilica Designer Software Installation

Windows Platform Installation

1. With the Xilica USB thumb drive included with your Xilica product, transfer the files from the USB to a memorable location on your computer. Alternatively, you can download the latest version of the Xilica Designer software from the Xilica website (www.xilica.com). It is highly recommended that you make sure you are using the latest version.
2. Double click on the 'XilicaDesigner.exe.' file saved on your computer.
3. When asked to install the file, click 'Install' to continue.



4. Allow the program to complete the installation process. This may take a several minutes.



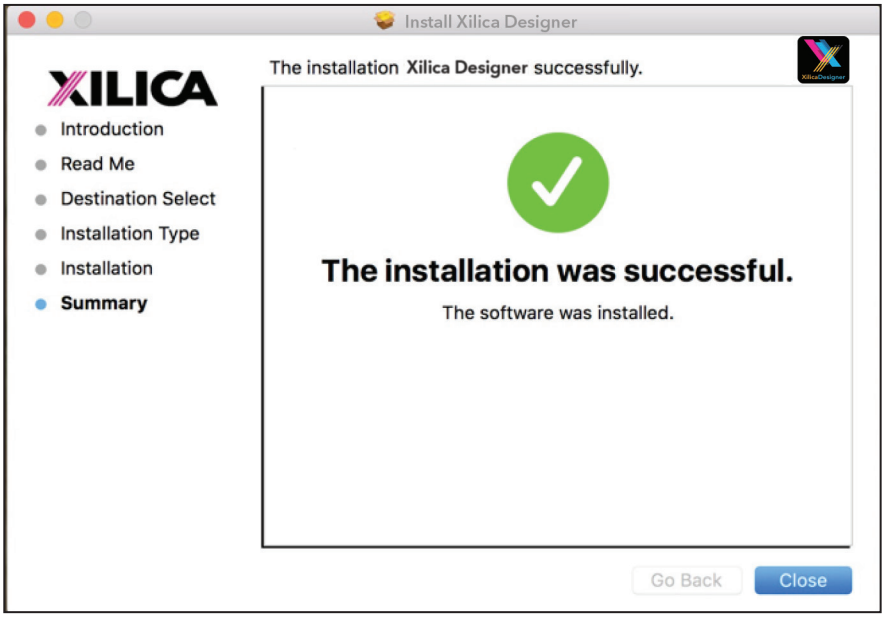
5. When complete, Windows will ask for permission to allow firewall access. The suggested setting is to allow Xilica Designer to communicate in Private networks, such as home or work. Allow access to public networks at your own discretion. Check the appropriate boxes, then click 'Allow Access' to finish.
6. The Xilica Designer software is now installed.

Installation Notes
If a personal Firewall is set up on your computer, a pop-up Firewall window may ask whether users want to "Block" or "Allow" Xilica Designer from accessing the network. Select "Allow" to continue the installation.

Mac OSX Platform Installation

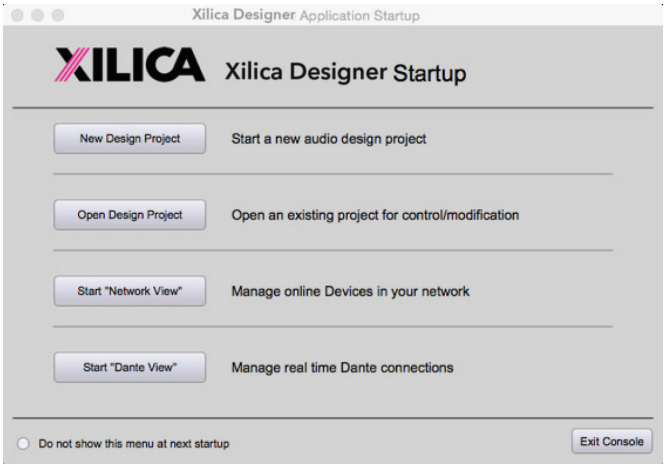
1. With the Xilica USB thumb drive included with your Xilica product, transfer the files from the USB to a memorable location on your computer. Alternatively, you can download the latest version of the Xilica Designer software from the Xilica website (www.xilica.com)
2. Double click on the 'XilicaDesigner.mpkg.' file saved on your computer.
3. OSX will display an installation dialogue. Read and follow each step carefully, then click 'continue' to proceed.

4. When the installation is successful, the following dialogue will be displayed.



Launch the Xilica Designer Software

Upon launching the Xilica Designer software, a start-up window will pop up.



You may select a 'New Design Project', 'Open Design Project', 'Start Network View' or 'Start Dante View'. (Network and Dante View are also available within the Xilica Designer software).

Select 'Start Network View'.

Network view

The Network View displays all processors and control devices on the network. The Network View displays information such as the device model, a network connection indicator, Computer address, IP address, Manufacturer and the firmware version.

In Network View, you should see your processor(s) listed.
At the top left of the device block is a network connection indicator. This indicator displays three colors: red, yellow and green. (Circled in Red)

Network Connection Indicators:

Green: The device is Connected and operational.
Yellow: The device is Connected and online, but not operational. Hovering over the network indicator will display a pop-up message of identified problems. (Normally this would indicate that no device design is loaded.)
Red: The device is not connected and offline. There is no communication between the Xilica Designer software and the device. Please check all cables, modular cards, connections and power. If the processor is busy performing a firmware upgrade or is in the process of rebooting, this may be a temporary offline interruption.

At times you may just see an exclamation mark (!). This indicates that a firmware upgrade is available. Normally this is not an issue unless there are updated modules in the project file that the outdated firmware does not support.

Connection Problems?

Yellow Network indicator
In Xilica Designer's Network View, if there is a Yellow network connection indicator at the top left of the device, the device is connected and online, but Not operational. To assist in identifying the problem, hover your cursor over the device network indicator and a pop-up message will identify the problems it has detected.

Probable causes include:

Non-DHCP Connection
When you are Not connected to a DHCP enabled router or server, the processor will revert to its auto-configured IP Address (169.254.128.128). In Network View, you can view the IP address displayed for your device(s). If the default IP address is shown, the default IP address is in effect.
If the default IP address is incorrect, please follow the 'IP Reset' instructions below.

For multiple processor connected to the network with no DHCP available, the user will have to manually assign unique IP addresses to each device. Please refer to 'Manual IP address assignment' and 'Assigning a Static IP Address to your computer' on Page 13-15.

DHCP Connection
If you are connected to a DHCP enabled router,
1. Reboot the router.
2. Restart the processor.
3. Shut down the Xilica Designer software and restart the program again.
4. The network connection indicator should now be green, indicating that the processor is connected, online and operational.

Note: If you are Not connecting via Wi-Fi, turn off your device's Wi-Fi so the device can connect to the desired network.
If the connection indicator is still Yellow (connected but Not operational) after the reboot procedure, it is possible that the processor is holding onto a previously assigned IP address and is not allowing the processor to revert to its default IP address. To resolve this issue, the processors' network settings and password need to be reset. Please follow the IP reset / Reset processor network settings and password procedure.

IP Reset / Reset Processor Network Settings and Password

1. Shut down the Xilica Designer software.
2. Power Off the processor.
3. At the back of the processor, you will see a small, recessed push button labelled "IP Reset".
4. Push the IP Reset button inward using a small pointed object.
5. While holding the button pushed in, power up the device.
6. Wait 5-10 seconds after power up until the orange "Network" light begins to flash, and then release the IP Reset push button.
7. Wait for the processor to power up completely. This may take up to several minutes.
8. Open the Xilica Designer software and select "Start Network View"
9. In Network View, the network status indicator should now be green (Connected and operational)

If the default IP address of 169.254.128.128 is still shown, you're either using an incorrectly configured Static IP setup, or the DHCP server still isn't available to the device.

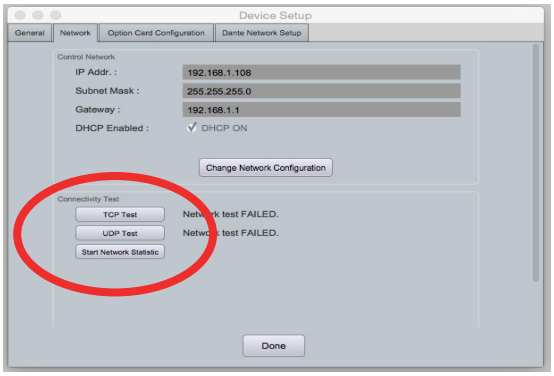
Software network problems continued

Device Not Ready
If the pop-up message shown says Device Not ready, then the processor needs a design loaded to the unit. If the same error message persists, restart the device and reboot the Xilica Designer software.

DSP Processing Error
If the pop-up message shown says DSP Processing Error, this could be a bad pre-designed DSP project.
1. Retry reloading the pre-designed DSP app schematic.
2. If the connection indicator is still yellow, reboot the Xilica Designer software and restart the processor.

Error in Firmware Upgrade
The pop up message shown will print out an error code. Retry the Firmware Upgrade again.

Device can communicate to Xilica Designer with UDP but cannot communicate with TCP
1. Right click the device in Network view and select **"Device Set up"**. Select the **"Network tab"**

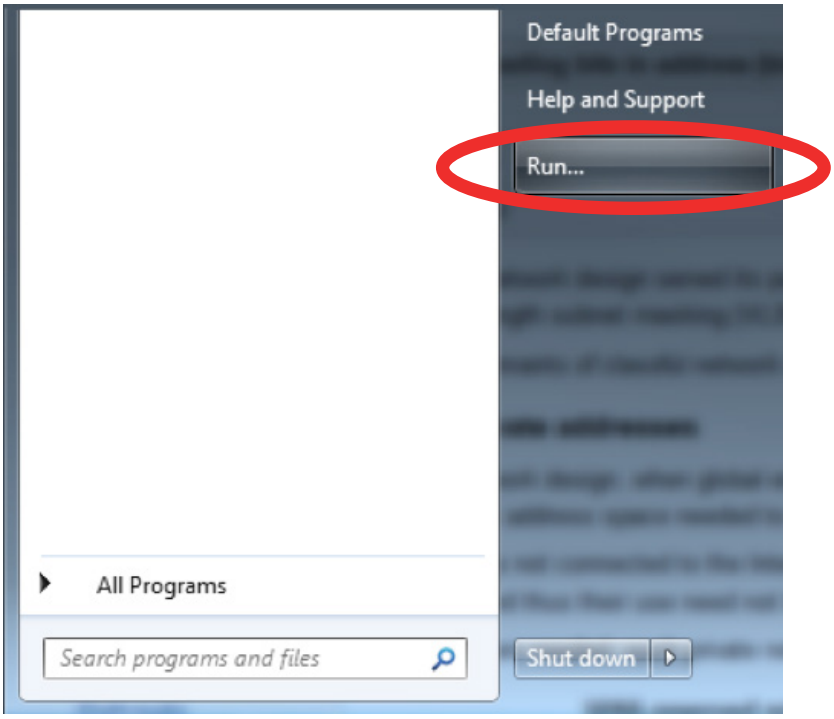


2. Click "**TCP Test" / "UDP Test**" to test TCP / UDP connections.
3. If failed, please check your PC's firewall or router settings. If you are not connecting via Wi-Fi. make sure that Wi-Fi is turned off.
4. Click "**Start Network Statistics**" to see network statistic information.
5. Once a change has been made, restart Xilica Designer and review your device connection.

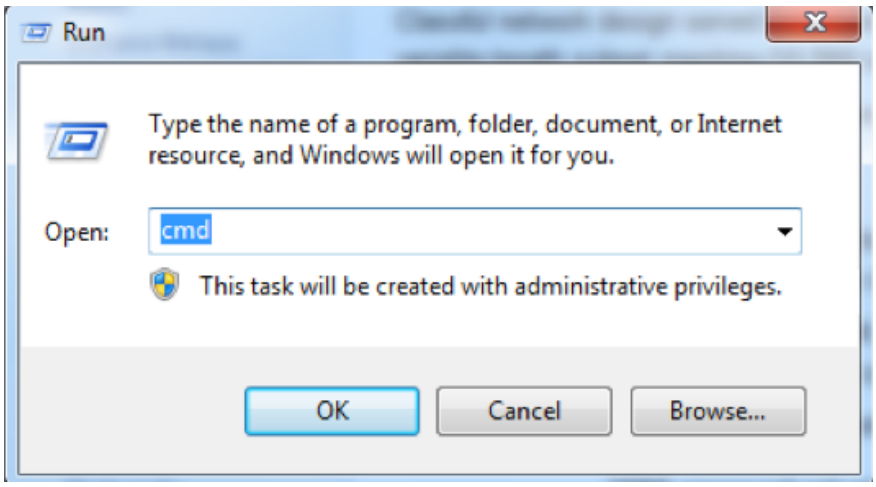
Determining Network information

In this section, we will be navigating through Microsoft Windows to determine your home networking information for the use of programming your processor.

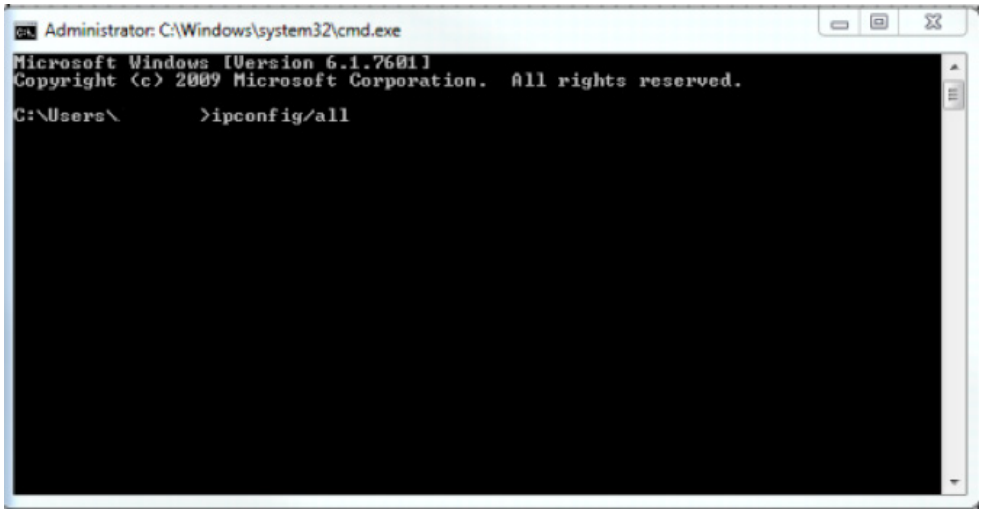
The first step is to open the '**Start Menu**' and select '**Run…**'



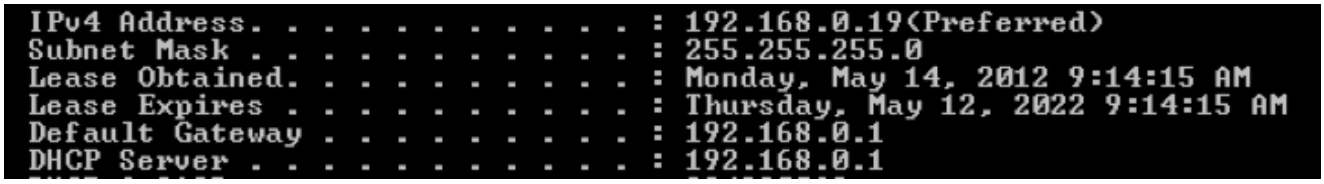
In the newly opened 'Run' window, **type** '**cmd**' into the 'Open' box. Hit enter or click '**OK**'.



You have now opened the command prompt, where we can display the TCP/IP Network Configuration information. Type '**ipconfig/all**' (No Quotes) and hit enter.



From this list of values, you can now locate your network's 'Default Gateway', 'IP (IPv4) Address' and 'Subnet Mask' values.

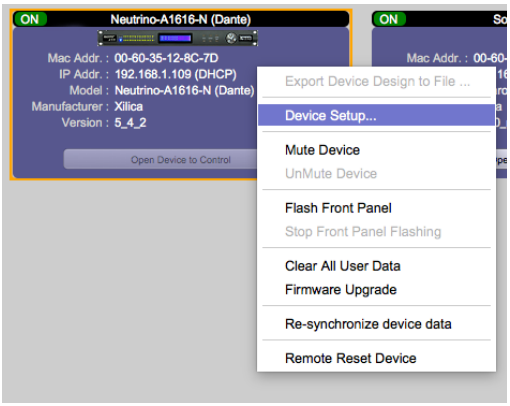


To determine an IP Address for your processor, use the first 3 values (Octets. In our example, 192.168.0) of the 'IPv4 Address' and select the desired value for the fourth Octet (In our case, 19). Ensure that the value you choose is unique to any other system on your network, so we could set ours to 192.168.0.20. Again the 192.168.0.XXX must be unique to each device. Reference the IP address (IPV4) of your PC in the TCP/IP Network Configuration menu for an example of how the IP address should be structured.

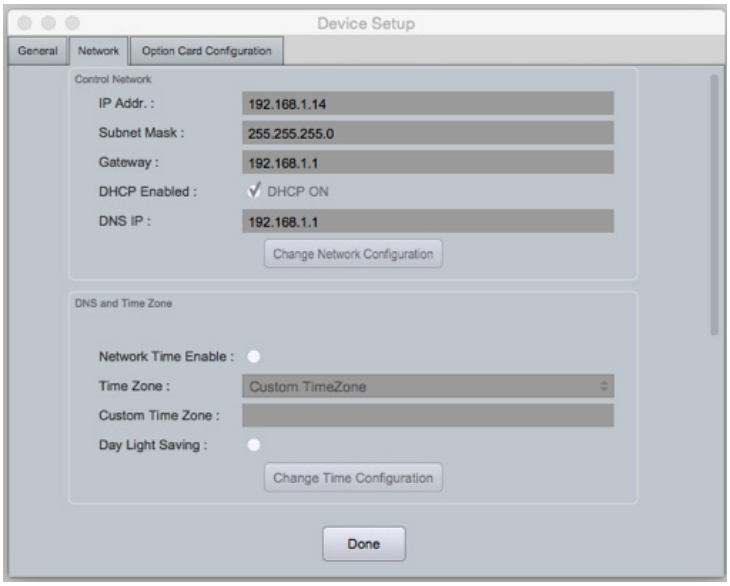
Manual Assignment of IP Addresses for devices

There are applications that require the IP address to be manually assigned (the same solution may apply to some connection issues).
To manually assign IP addresses,

1. In the Network View, right click the device and select '**Device Setup**'.



2. In the '**Network**' tab, select "**Change Network Configuration**" to disable DHCP and to insert IP addresses manually (It also provides two built-in test procedures, device security, and device information along with Dante Configuration, if applicable).



3. When finished, select "**Apply**" to save changes and then "**Done**" to exit.
4. Repeat steps 1 through 3 for each subsequent processor so each processor has its own unique IP address. (For example: 192.168.1.180/181/182...)
The devices will appear Offline in Network view until you are able to assign a static IP address to your computer. (Page 14-15)

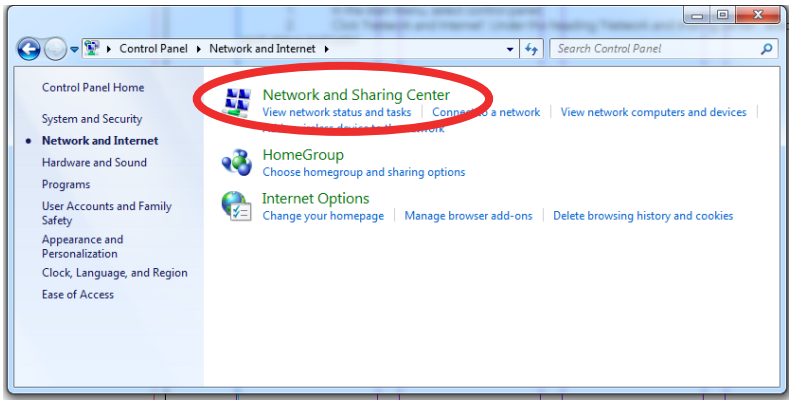
Assigning a Static IP address for your computer

The following process applies to manually assigning a unique static IP address to your computer.

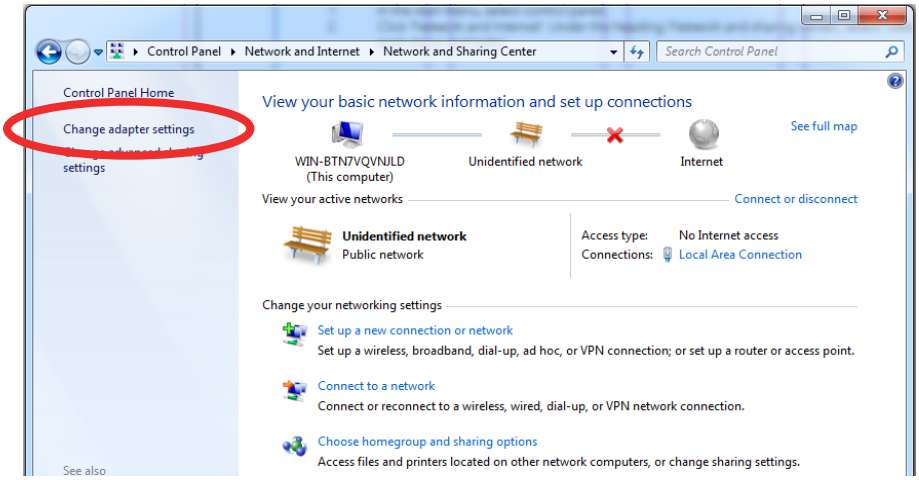
Windows platform

1. In the start menu, select **control panel.**

2. Click **'Network and Internet'.** Under the heading **'Network and sharing center',** select **'View network status and tasks'**



3. Click on '**Change adapter settings**' on the left tab.



4. Select '**Local Area Connection**' and click on the **Properties** button. Select **Internet Protocol Version 4 (TCP/IPv4)** then '**Properties**' to access the manual IP settings.

5. Set up your computer's IP address to be **192.168.1.X** where X can be any value from 0-255, but unique from other manually assigned device IP addresses.

6. Use the following settings for your PC's unique static address:
IP address: 192.168.1.X (X is any value from 0-255 but unique from other device IP addresses)
Subnet mask: 255.255.255.0
Gateway: 192.168.1.1
DNS Servers: 192.168.1.1

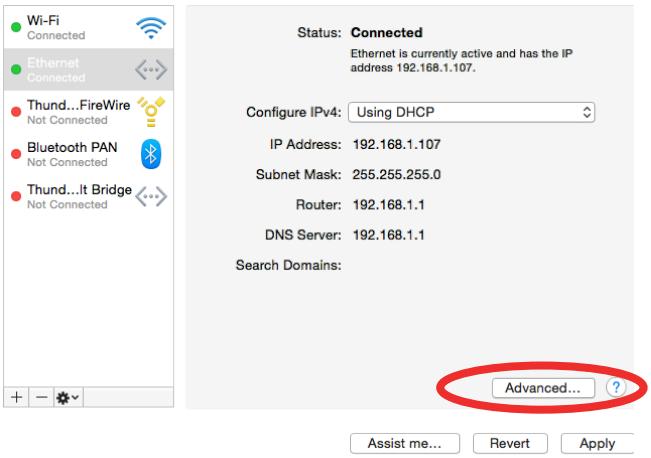
7. If your devices are set up following the 'Manual Device IP Assignment' and 'Assigning a Static IP Address to your computer' sections, the devices will now appear online and connected in Xilica Designer's Network view.

Mac platform

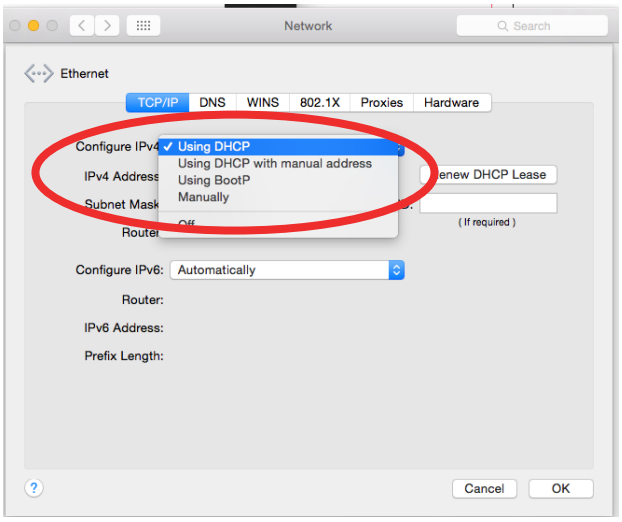
1. From the Apple menu, select **System preferences.**

2. Select **'Network'.** From the sidebar, select the network interface you are using.

3. Then click '**Advanced...**'



4. In the TCP/IP tab, set **Configure IPv4** to '**Manually**' using the drop down menu.



5. Enter a static IP address in the IPv4 Address field. Set up your computer's IP address to be **192.168.1.X** where X can be any value from 0-255, but unique from other device IP addresses.

6. Use the following settings for your computers static address:
IP address: 192.168.1.X (X is any value from 0-255 but unique from other device IP addresses)
Subnet mask: 255.255.255.0
Router: 192.168.1.1

7. Click '**Ok**' and '**Apply**' to apply your changes. Then reboot the Xilica Designer software.

If your devices are set up following the 'Manual Device IP Assignment' and 'Assigning a Static IP Address to your computer' sections, the devices will now appear online and connected in Xilica Designer's Network view.

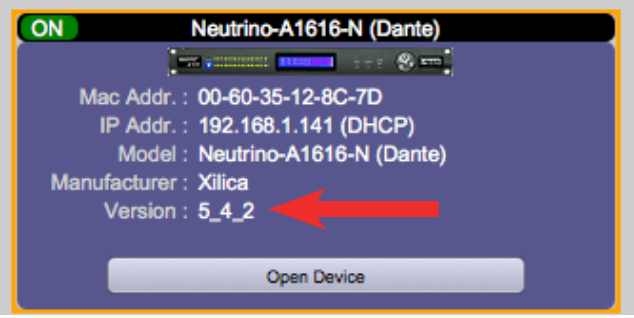
Firmware Upgrade

It is strongly recommended that you check the Xilica website (www.xilica.com) frequently for the latest software and firmware versions, as these updates may contain critical bug fixes and new features.

Note: Using an older version of software with a newer firmware or newer software with an older firmware will work but some of the features may not be available and bugs could exist.

Before you begin, check your software and firmware versions.

1. In Xilica Designer's Network View, select 'Start Network View'.



The Network View shows all processors and devices connected to the network.
The device's current firmware version is displayed here. (Ex. Version 1.0.0)

2. To view the current software version, click on the "About" tab at the top of the software. This window will display your current software version.

Matching the Firmware

To assist you in determining which firmware file is appropriate for your device, refer to the chart below.
Note: The file structure may be different from the date that this list was created. Always check the Xilica website (www.xilica.com) to keep updated.

#_#_# Represents the 3 digit version code of the firmware update.

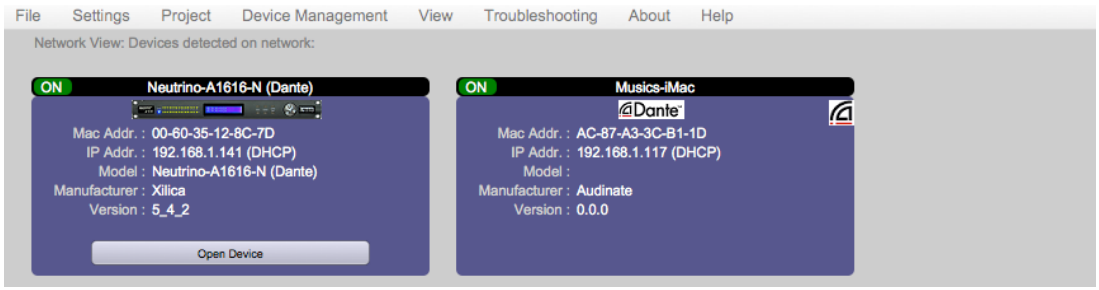
(SOLARO_#_#_#.img)	Solaro series: QR, FR
(NEUTRINO_#_#_#.img)	Neutrino series: A, A-D (AES), A-N (Dante), A-ND (Dante, AES)
(UNO_#_#_#.img)	Uno series: U, U-D (AES), U-N (Dante), U-ND (Dante, AES)
(NEUTRINO-AEC_#_#_#.img)	Neutrino AEC Series
(UNO-AEC_#_#_#.img)	Uno AEC Series
(RIO_#_#_#.img)	Rio Series
(NEUPANEL MINI_#_#_#.img)	NeuPanel Mini Series: K1, K4, S4, S8, S4K1
(NeuPanel Touch_#_#_#.zip)	NeuPanel Touch Device

Step-by-Step Firmware Upgrade Guide

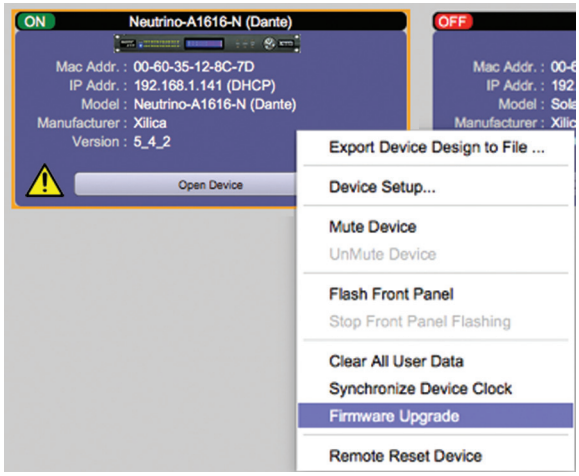
The hardware device must be connected and operational (Green indicator) before upgrading the firmware.

1. Download the latest firmware version for your device from our website. (www.xilica.com)

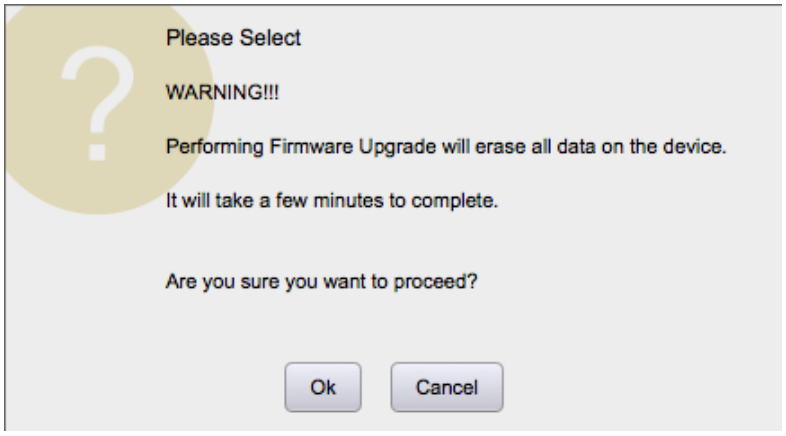
In Network View, all the units on the network are displayed. The network connection indicator is displayed at the top left of each device.



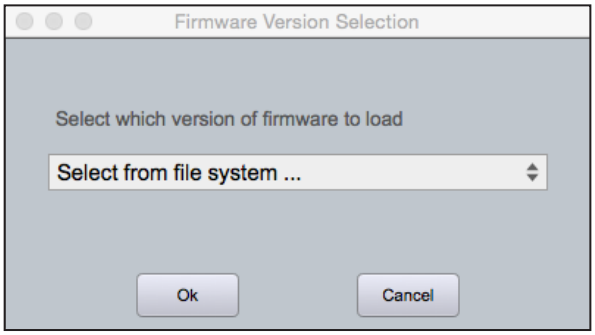
2. Save any design files from the device onto your computer. All programmed data on the device will be erased during the upgrade process. (To save, navigate to the File tab at the top left of the software and click 'Save'.)
   After the firmware upgrade is complete, you may reload saved design files back into the device.
3. Right click the device that you would like to update. Select 'Firmware Upgrade'.



4. A pop up window will ask you if you'd like to proceed with the firmware upgrade. Select "OK" to proceed.



5. Click 'Ok' to select a file from your computer. Then navigate to the appropriate firmware file that you have downloaded from our website. Select the correct file and click "Open".
(Ex. A Neutrino A1616-N is being updated, so the firmware file at the time of this document is Neutrino_5_4_2.img.)



6. A status bar in the device window will monitor the Firmware upgrade progress.

7. When the Firmware has been uploaded to the device, the device will automatically restart and update its internal data. This may take several minutes.
8. During this period, the device network indicator will turn RED and appear offline.
DO NOT POWER OFF THE DEVICE as the device is performing self-initialization.

9. Once the device is initialized, the status indicator will become Yellow. This indicates that the device does not have a design file loaded to it yet.

NOTE: Powering Off your device during a firmware upgrade can result in a complete corruption of the processor. If this happens, please follow the 'Xilica Designer: Firmware Upgrade' guide.

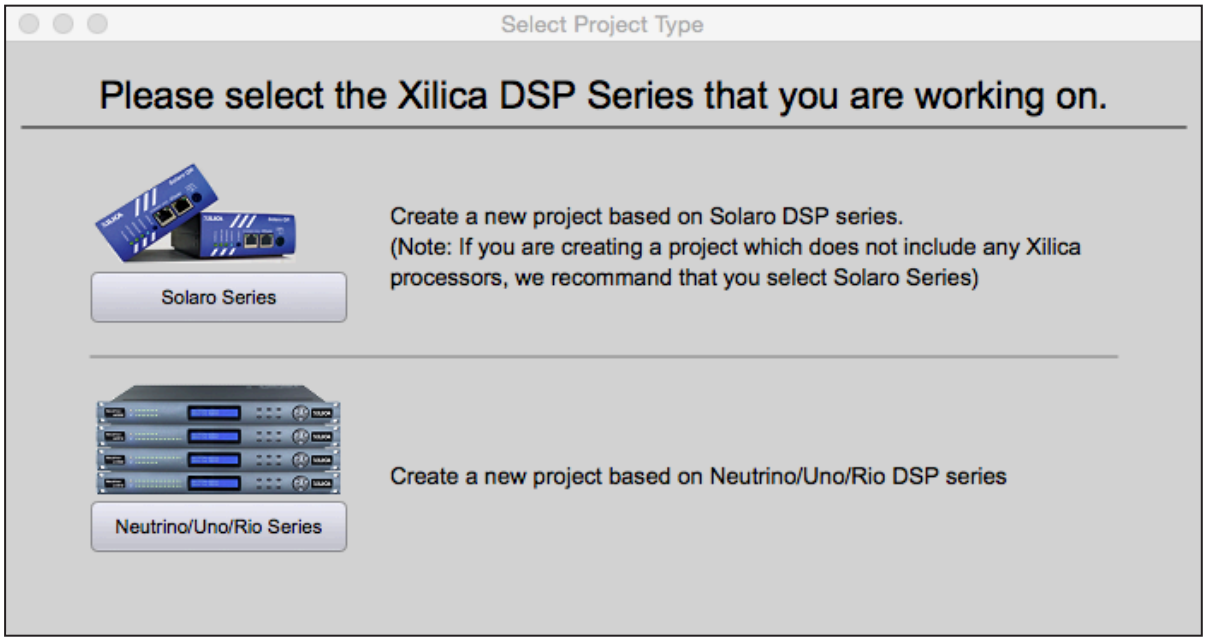
Note: If more than one unit needs a firmware upgrade, you can save time by updating them all at the same time. This can be done as long as they are the same Network and have a green network indicator (connected and operational), as displayed on the Network View page.

10. The network indicator of the device should now be green and the device is ready for use.

Creating a Project

At the top left under the 'File' tab, select 'New Project'.

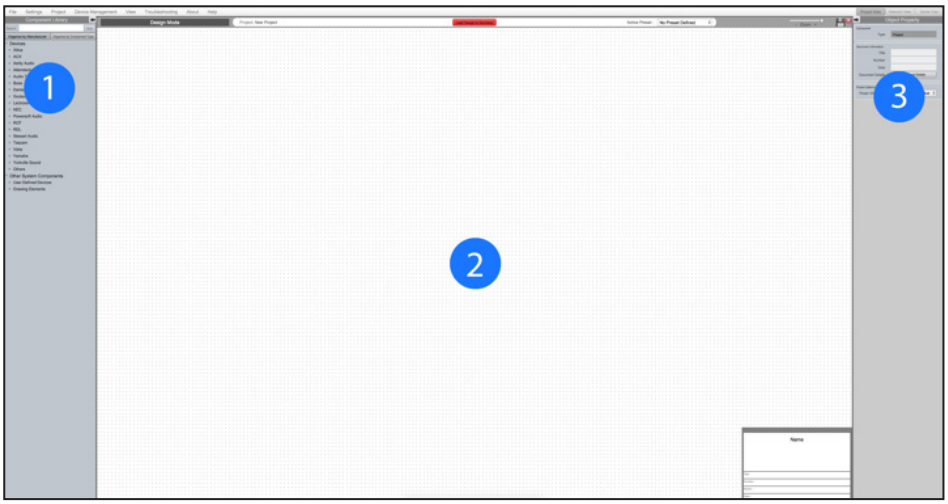
When creating a new project, Xilica Designer will ask you what DSP you are using. Neutrino/Uno series DSP is rather different from Solaro series DSP, therefore the two DSPs cannot be used in the same project file.



For more information, please visit the Xilica Designer section in the Xilica website: www.xilica.com

For the example, a Xilica Neutrino A1616-N DSP will be used.
If you are using a Solaro series DSP, please refer to the Solaro DSP user manual.

Project view



1. To the left of the screen is the Component Libraries Menu.
   This menu displays a list of design modules and devices that you can use in your project.

2. The dotted grid in the centre of the screen is your work area.
   This area allows you to configure, connect and organize your design modules.

3. To the right of the screen is Object Properties.
   This menu allows you to customize your design modules and connections. Simply select your desired module and this menu will display different parameters that you can change in that particular module. (Ex. name, module color, I/O)

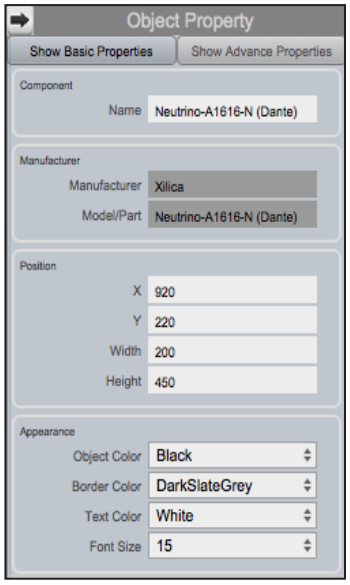
Starting a design

For the example, a single DSP hardware block will be used, but a design can be done with multiple DSP hardware items, including the Dante Digital Audio Transport.

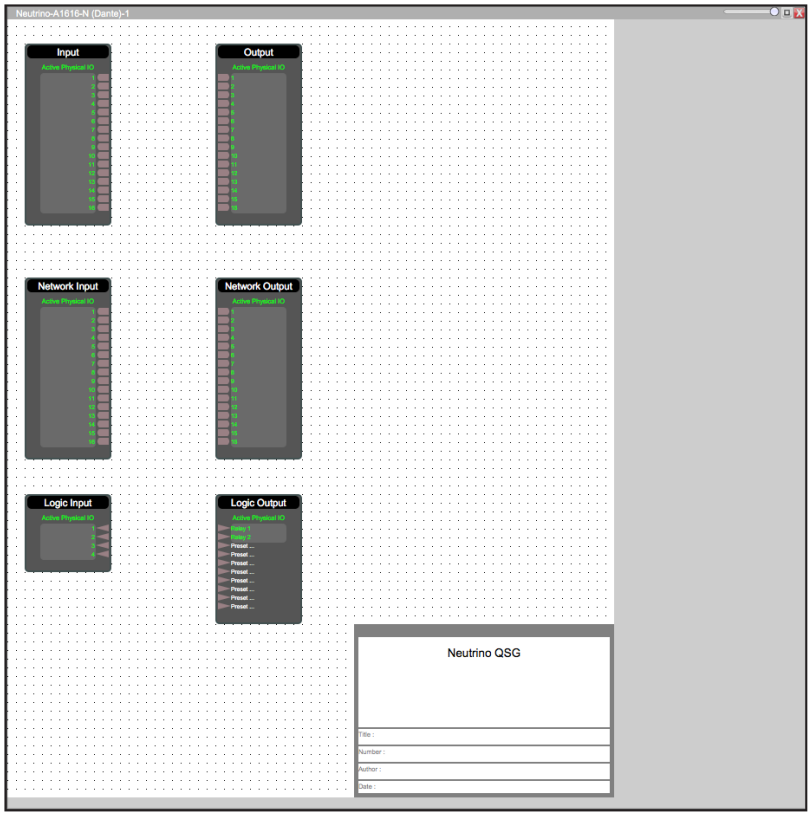
It is a good idea to have an understanding of the system design needed for the project prior to starting. Projects can be designed Offline (no devices connected) and the design can be loaded to your devices once the devices are connected and online.

1. From the Component Library, drag and drop your DSP model to the dotted work area.

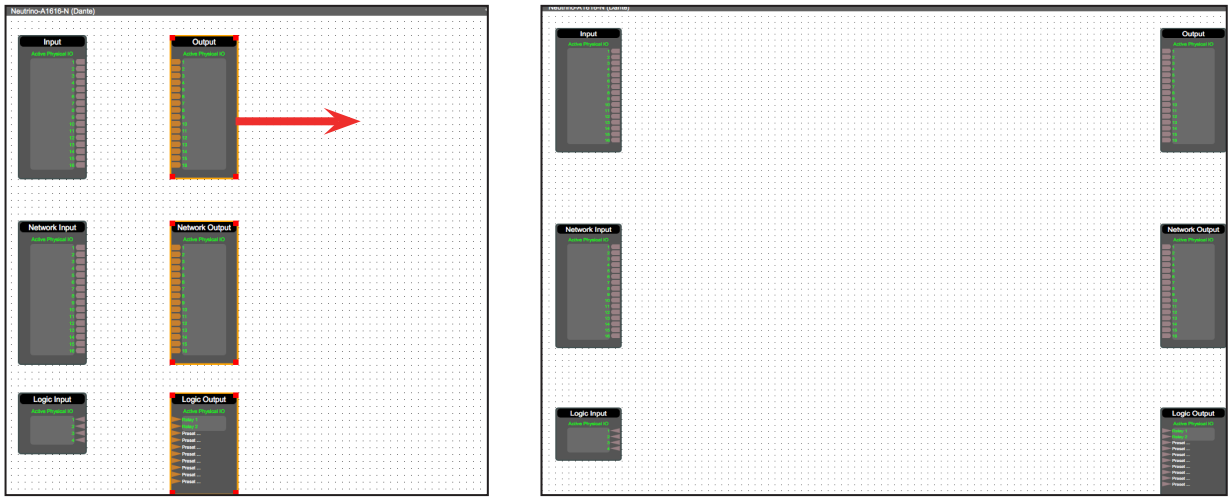
2. With the DSP module highlighted, you may adjust module parameters in the Object Property menu on the right. Object Properties differ for each module selected.



3. Double click the DSP module to open the design schematic.
A new window will appear. Resize the window by clicking and dragging the corner of the window.

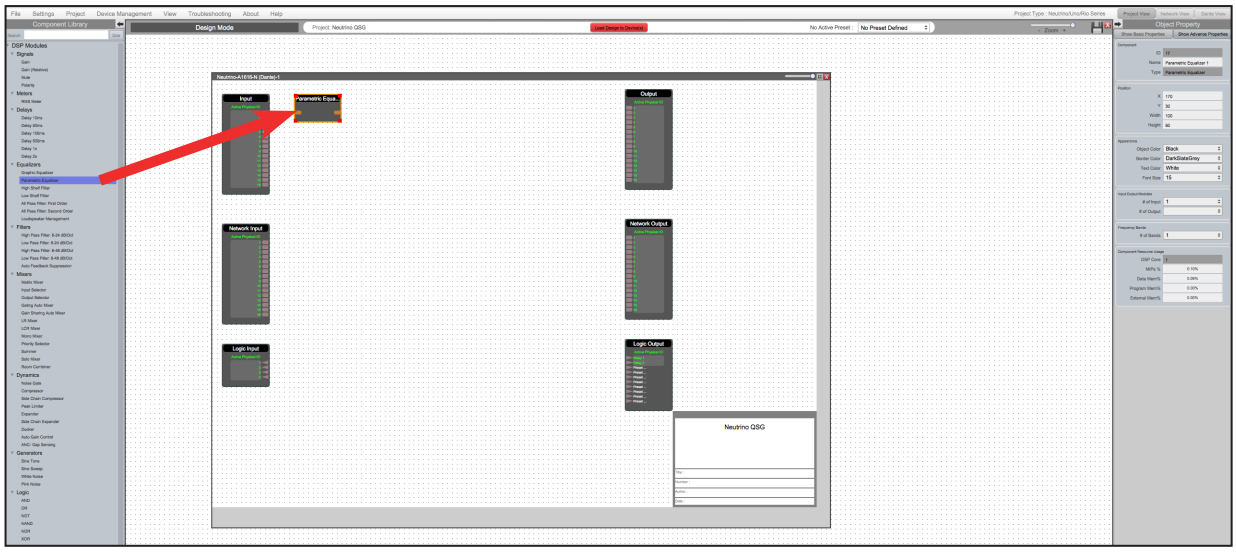


4. To space out your work area, click and drag a selection box around the output modules and drag them to the right. This will extend the work area.



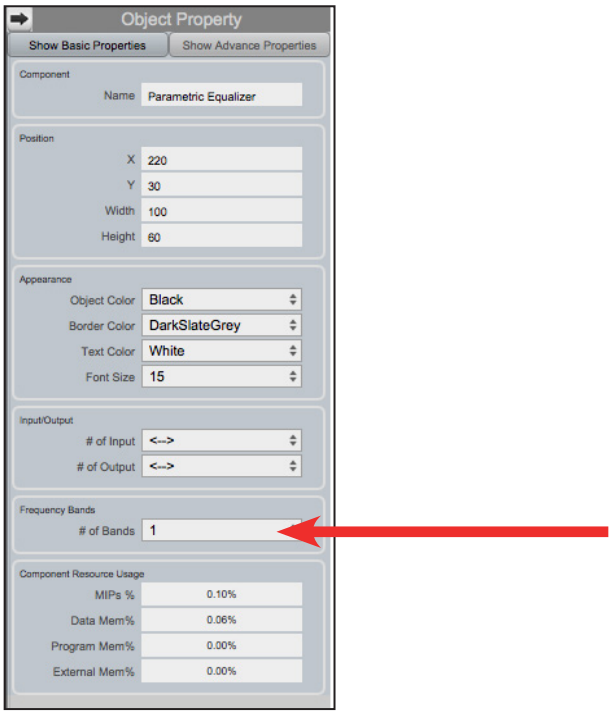
Notice that when this window is selected, the Component Library menu on the left, now displays a variety of DSP modules.

5. Click and drag a DSP module into the device schematic window.

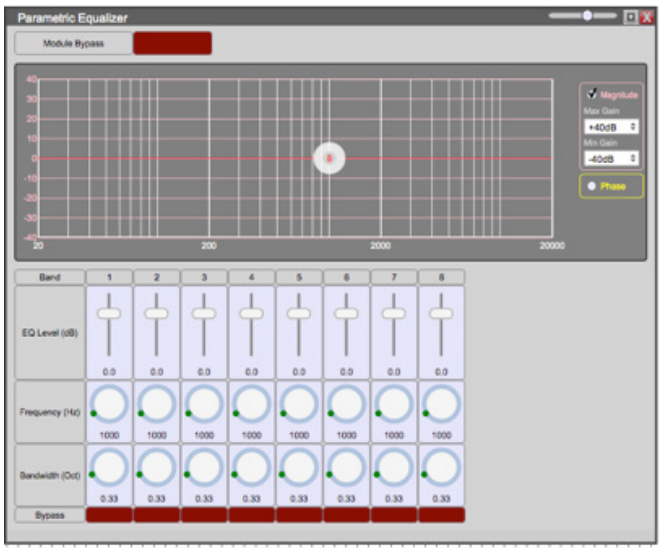


In the example, a PEQ module was added.

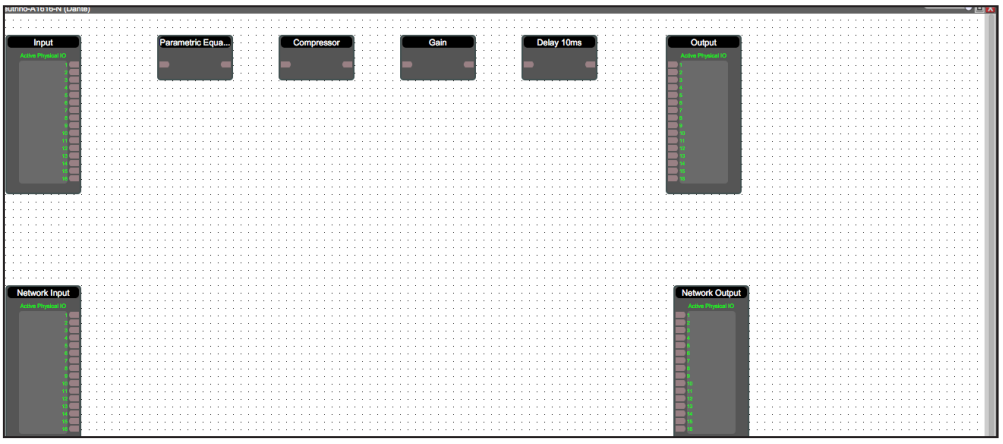
6. In the Object Property menu on the right, you can customize your module.
The number of bands for the PEQ module is determined in the object property menu.



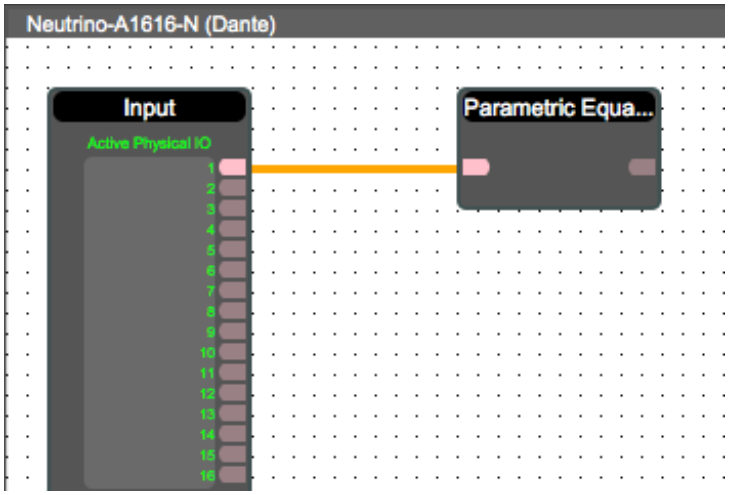
7. Double click the DSP module to open it and view your changes.



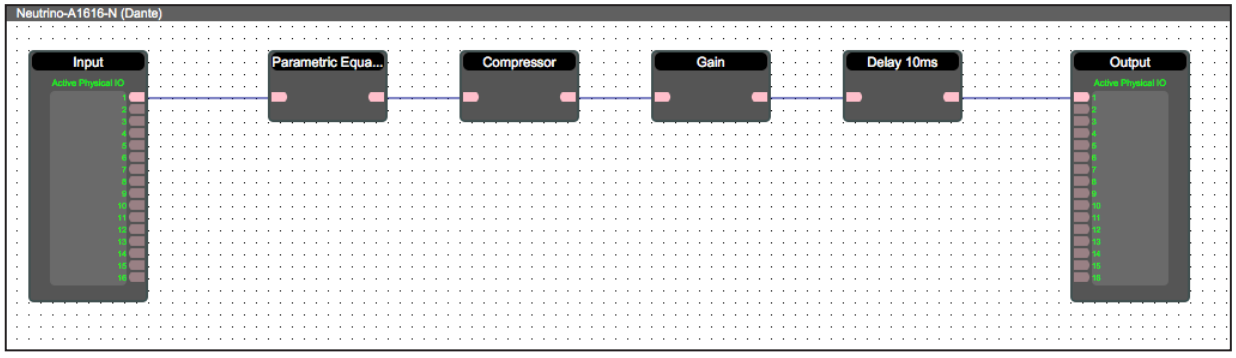
8. Drag and drop other desired modules into the device schematic work area.



9. Click and drag from the first input module node to the PEQ input node. This will create a wire.



10. For this example, we will route the input to the output channel.

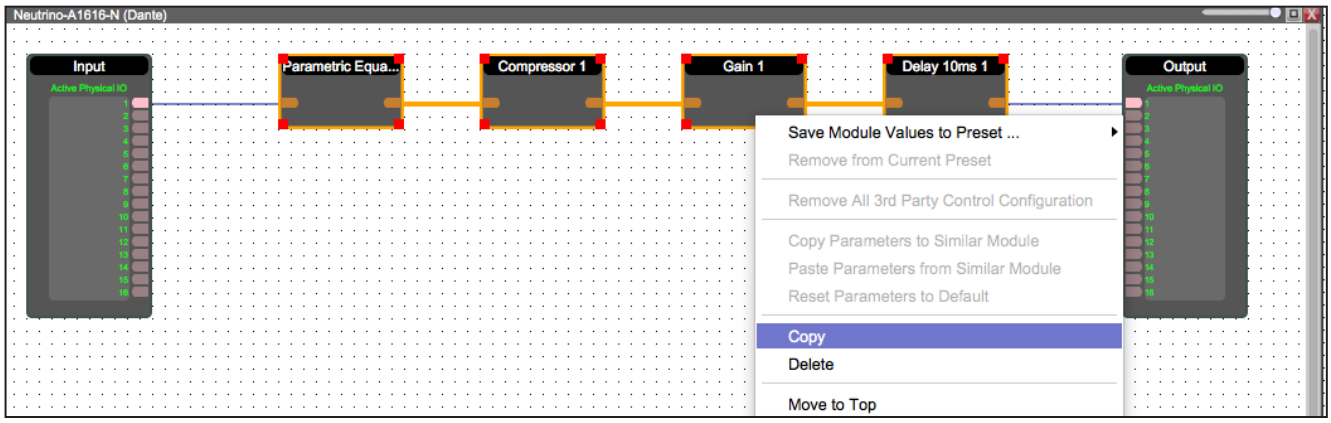


You can add different DSP blocks or duplicate the same processing chain for each channel.

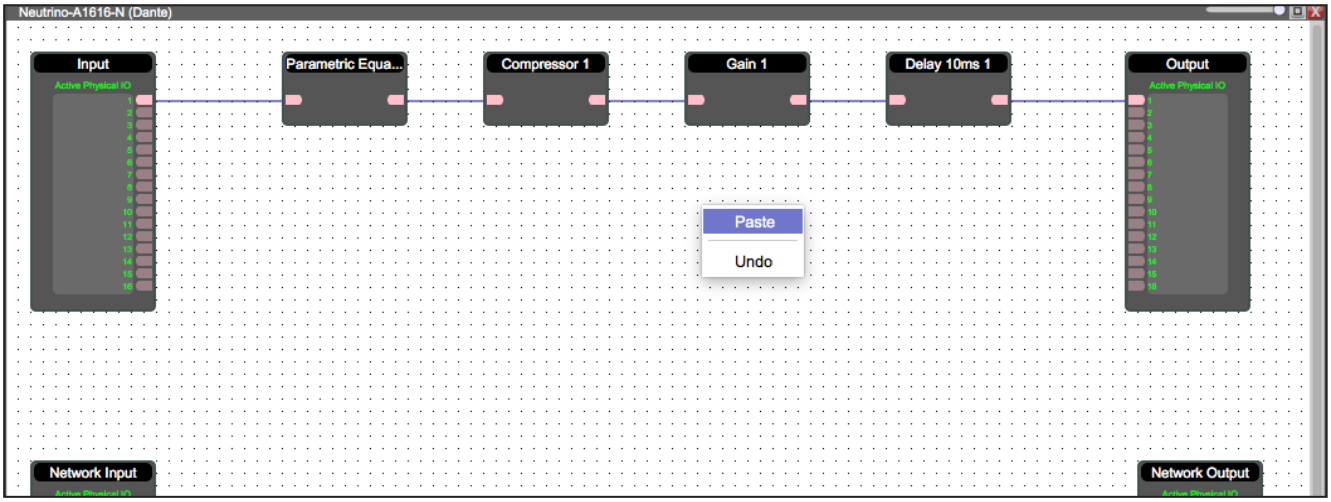
To duplicate the same processing chain,

a. Click and drag a selection box around the DSP modules. Ensure all modules are highlighted.

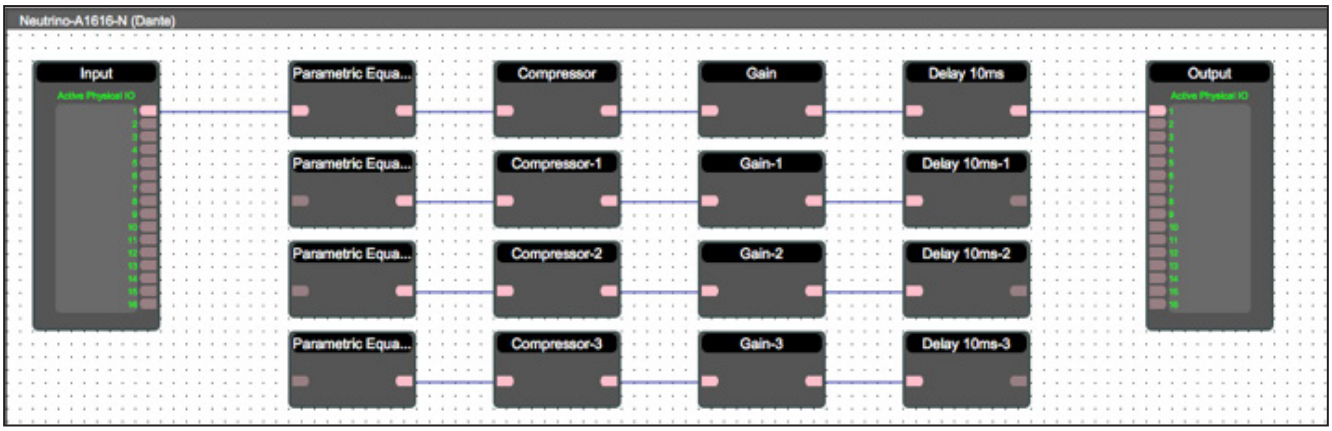
b. Right click the module and select 'Copy'.



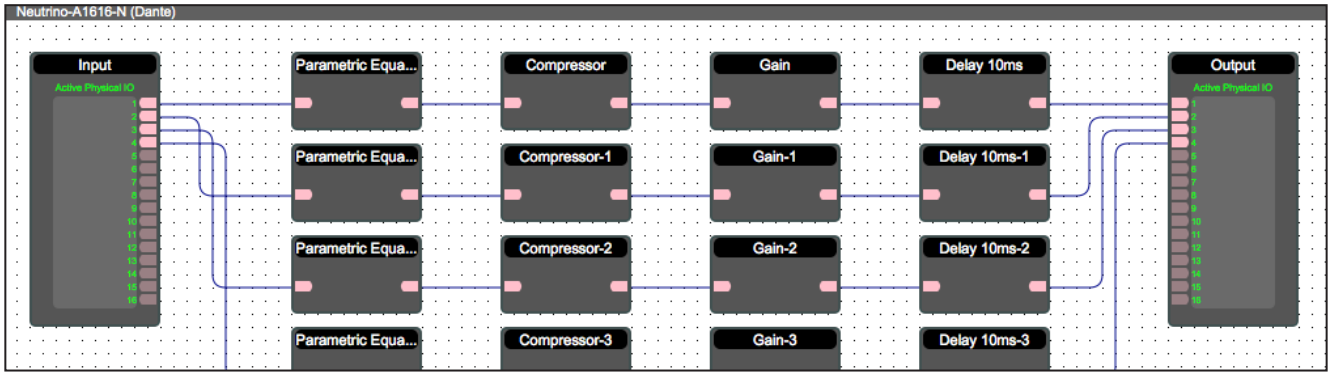
c. Then right click the work area and select 'Paste'.

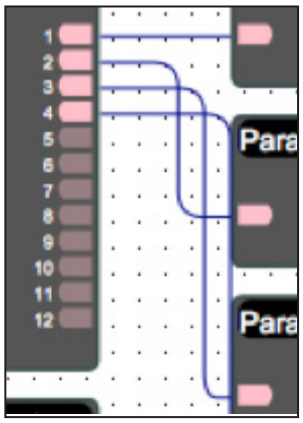


For the example, the chain is duplicated four times.

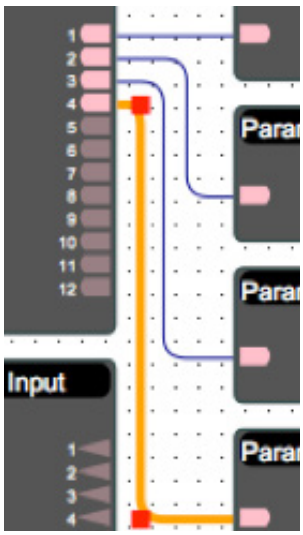


Wire the modules using the same wiring process as above.

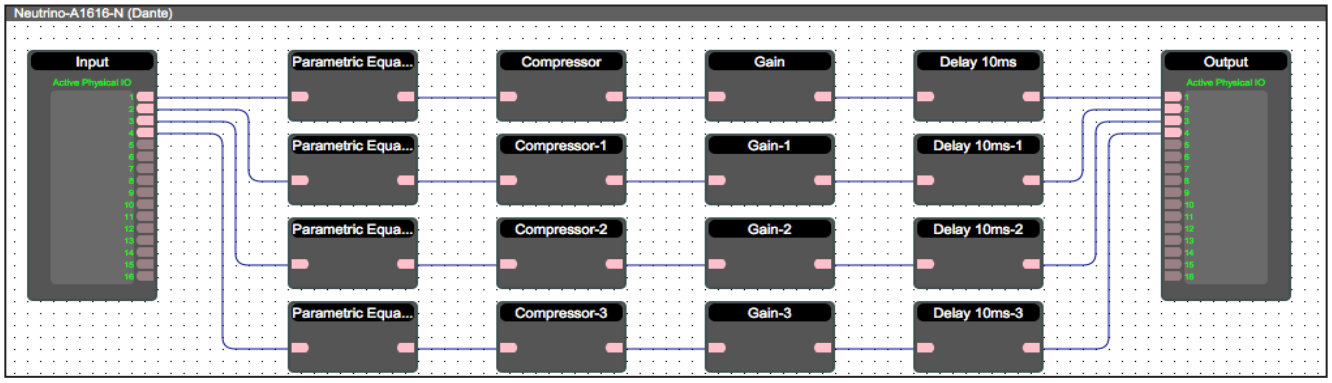




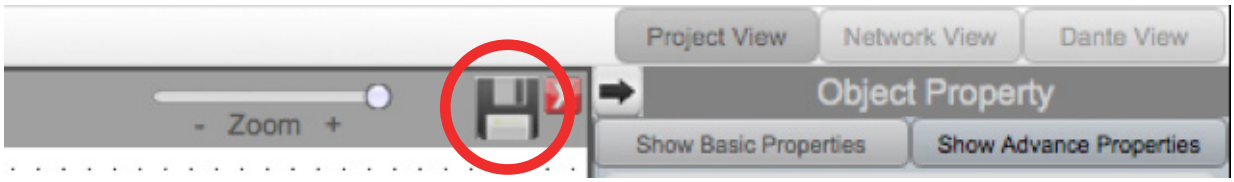
When drawing wires, wires may overlap and be difficult to read.



To move wires, click and drag the corner of a wire
Or highlight the wire and click and drag the red corner nodes.



11. To save your project,
navigate to the top left of the software. Under File, click 'Save As' to save a new project file or
If a project file is already created, click 'Save' to save all changes. You may also use the save icon at the top right of the work area to save changes.



It is recommended to back up your Master project file to an external location.

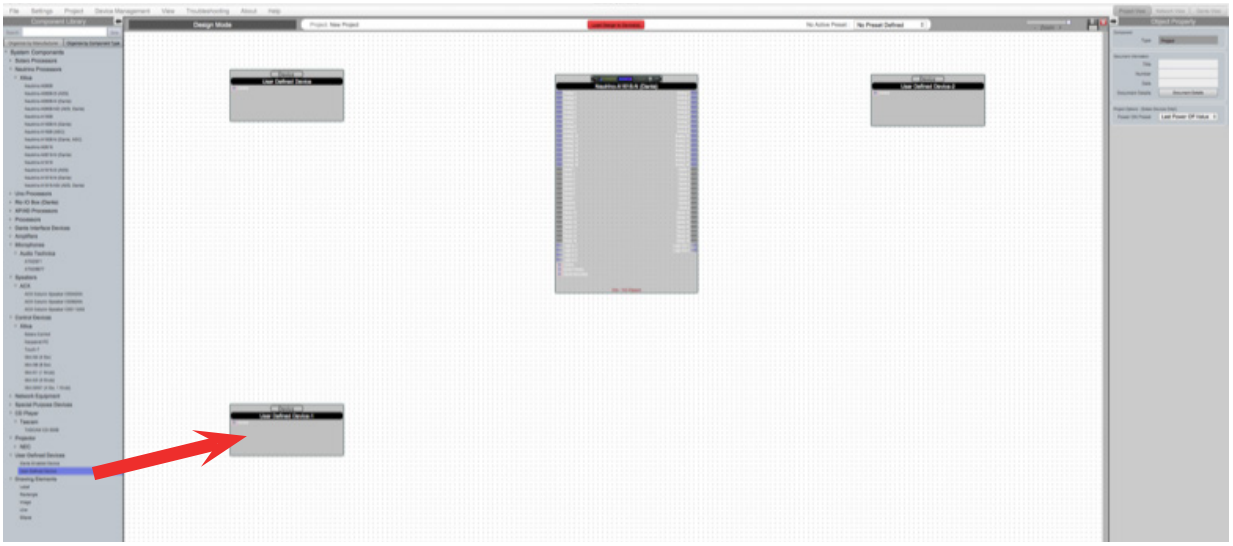
Design Blueprint

Xilica Designer allows the designer to create a Blueprint that can be used for documentation, submittals, and handed to an installation technician for use on the job site. The Blue Print page can include wire identification/type, equipment names and locations, including notes for the job.

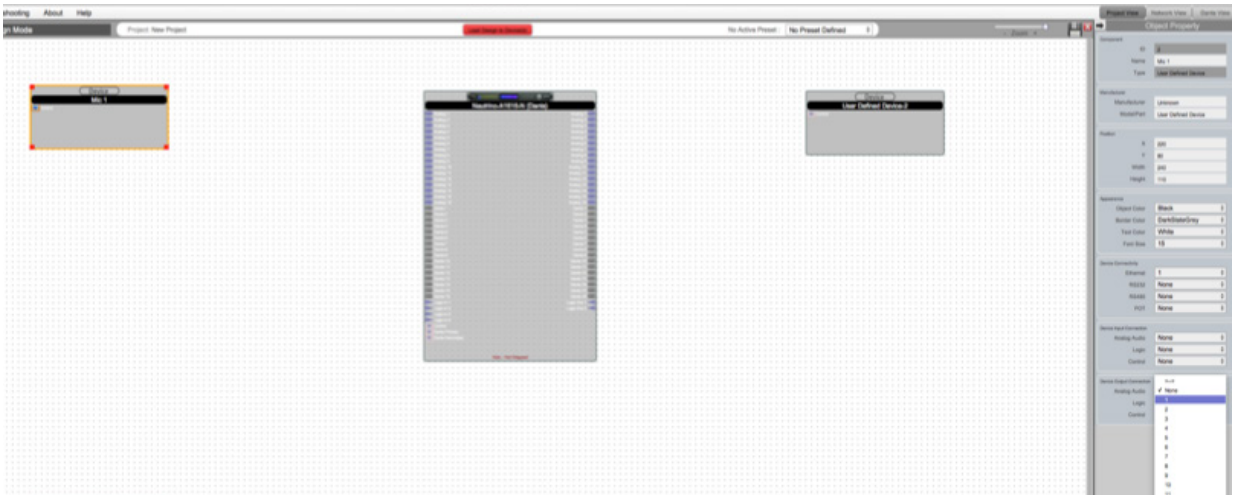
Let's begin by creating a 'Blueprint' for the above example design.

1. From the Component Libraries Menu on the left, click and drag the devices needed for your design.

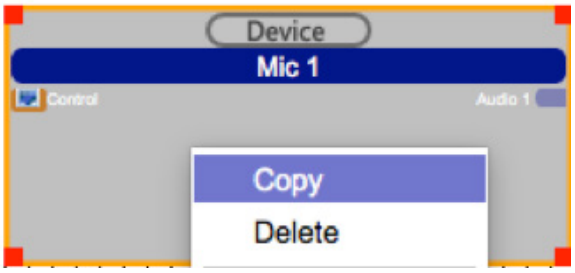
   For our design example, a Neutrino A1616-N and three User Defined Devices were added to the work area. Simply click and drag modules to move them and click and drag the corner of the module to resize the objects. (Alternatively, you may resize objects using the Object Property menu)

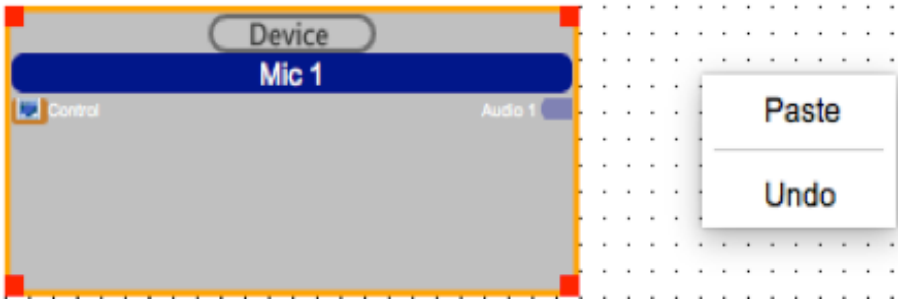


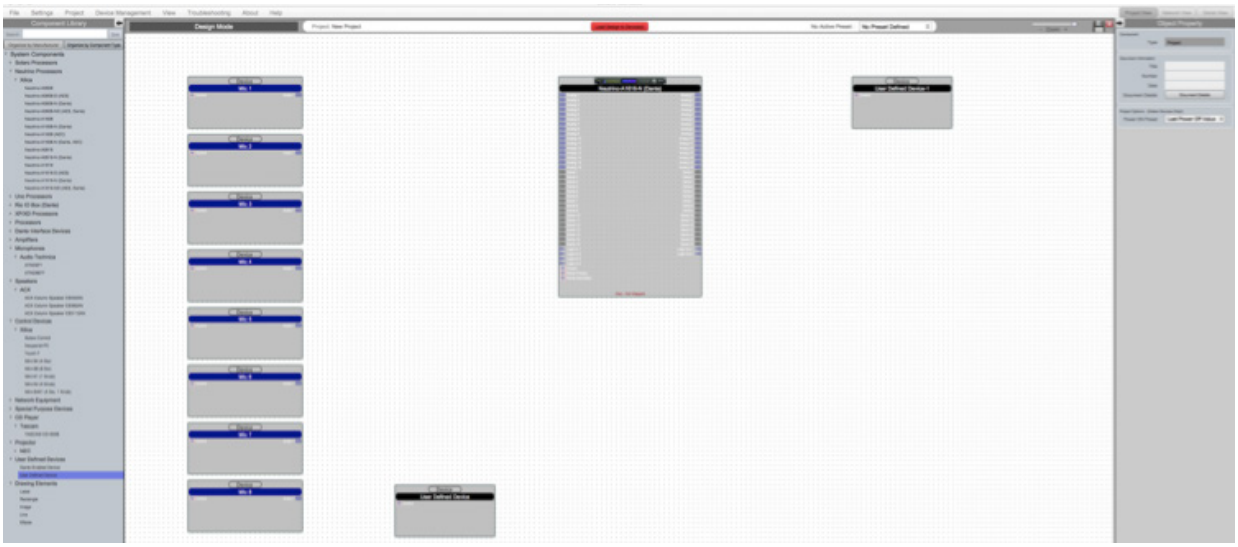
2. Select the first User Defined Device. With the device highlighted, you can change the Object Properties using the menu on the right. (Properties include: Device name, information, color, device connection and I/O options)
   For this module, we will create a microphone.



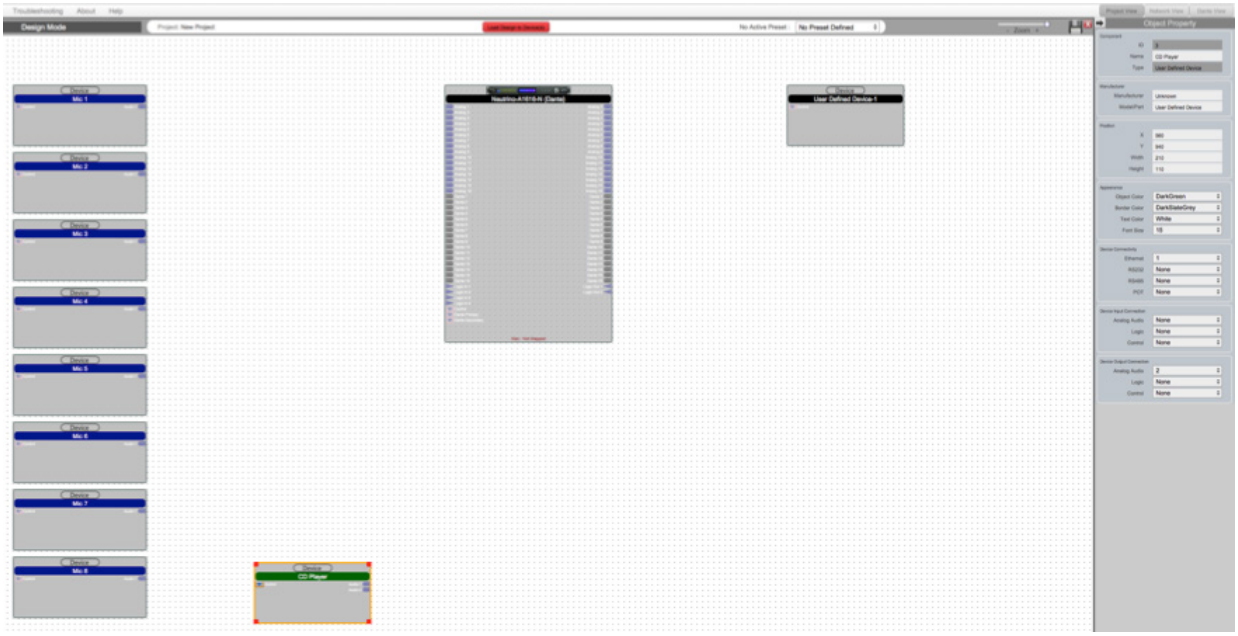
3. Since our example includes eight microphones, we will need to duplicate this module.
To duplicate a device, right click the highlighted module and select 'Copy'. Then select the dotted work area and click 'Paste'. You may also copy and paste multiple modules at once.



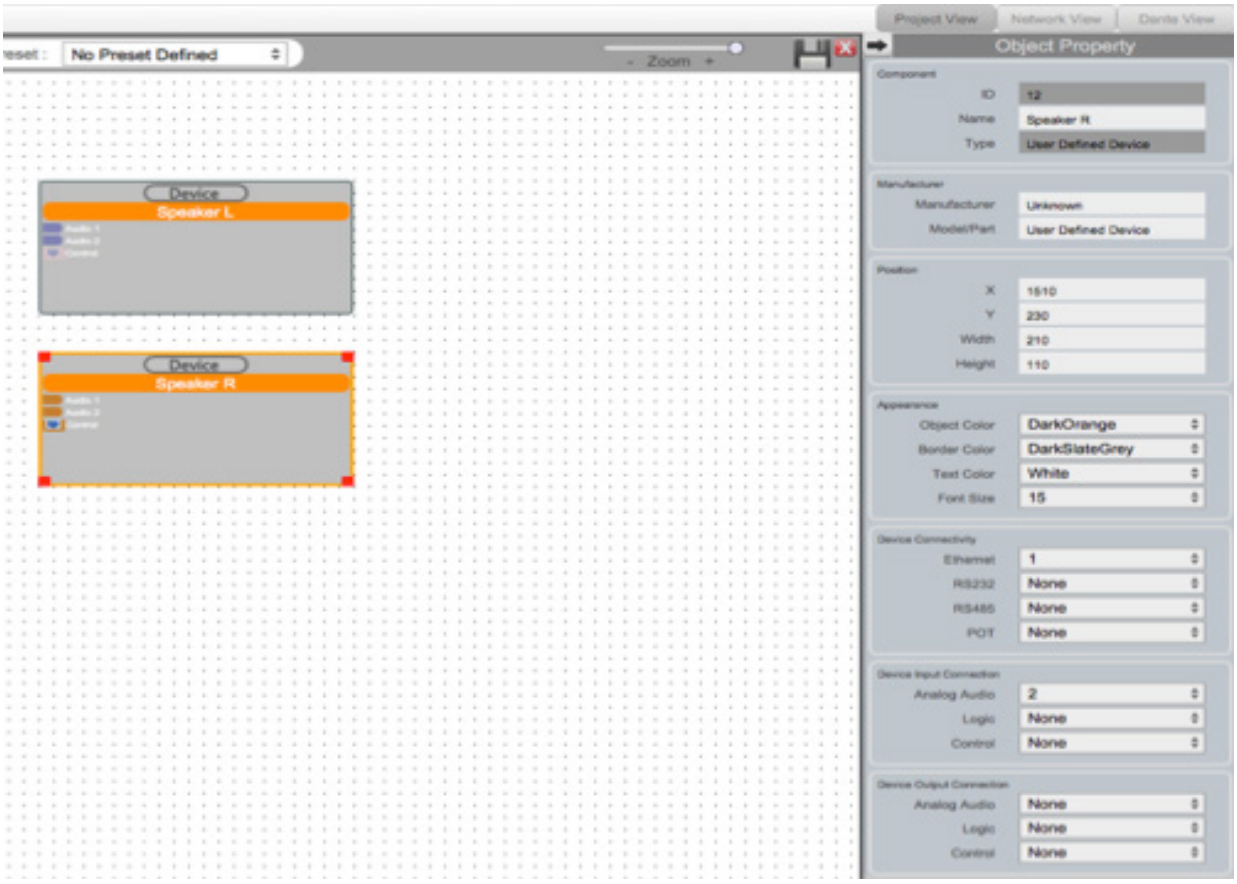




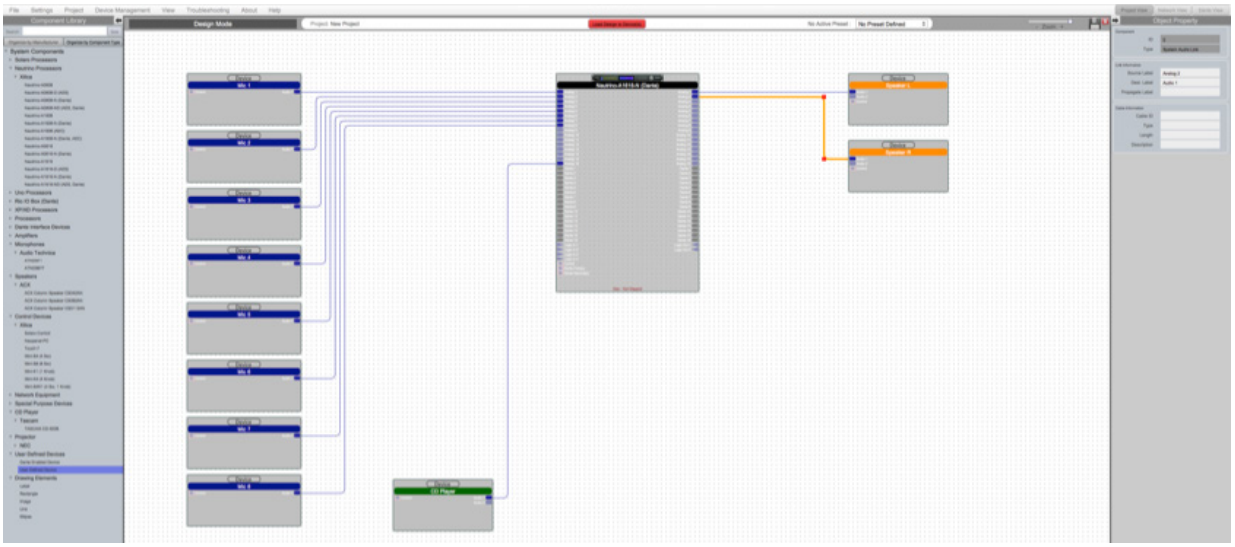
4. For the next blank User Defined Device, we will create a background music source. Similarly, adjust the object properties using the menu on the right.



5. For our third blank User Defined Device, we will create two output speakers. Adjust the object properties on the right and duplicate the device so that there are two speakers.

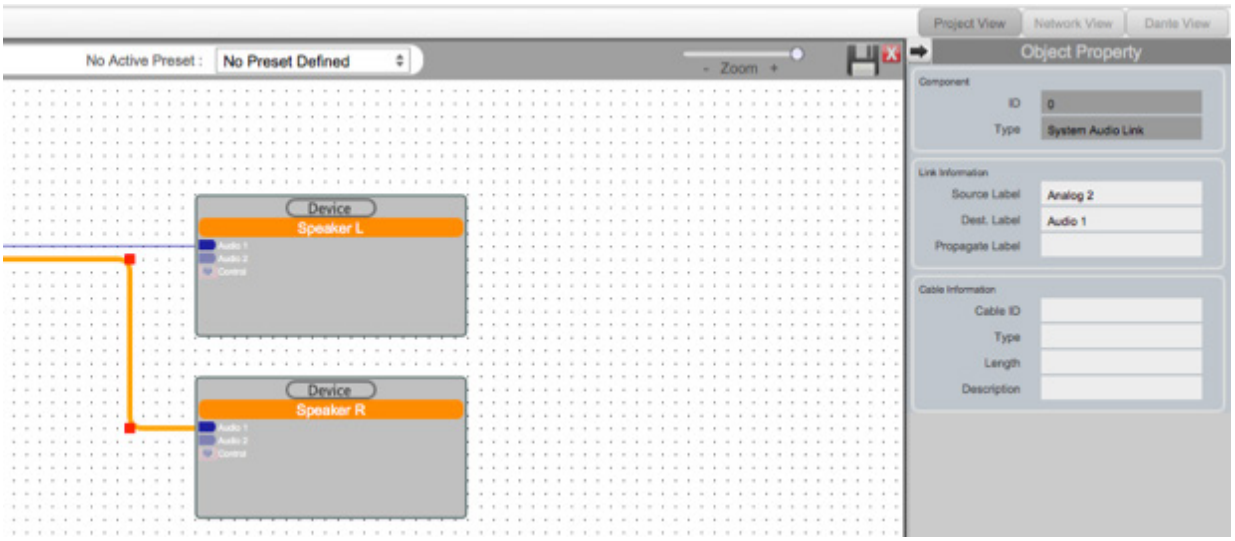


6. To connect your device modules together, simply click and drag from an output node to an input node. This will create a virtual wire.

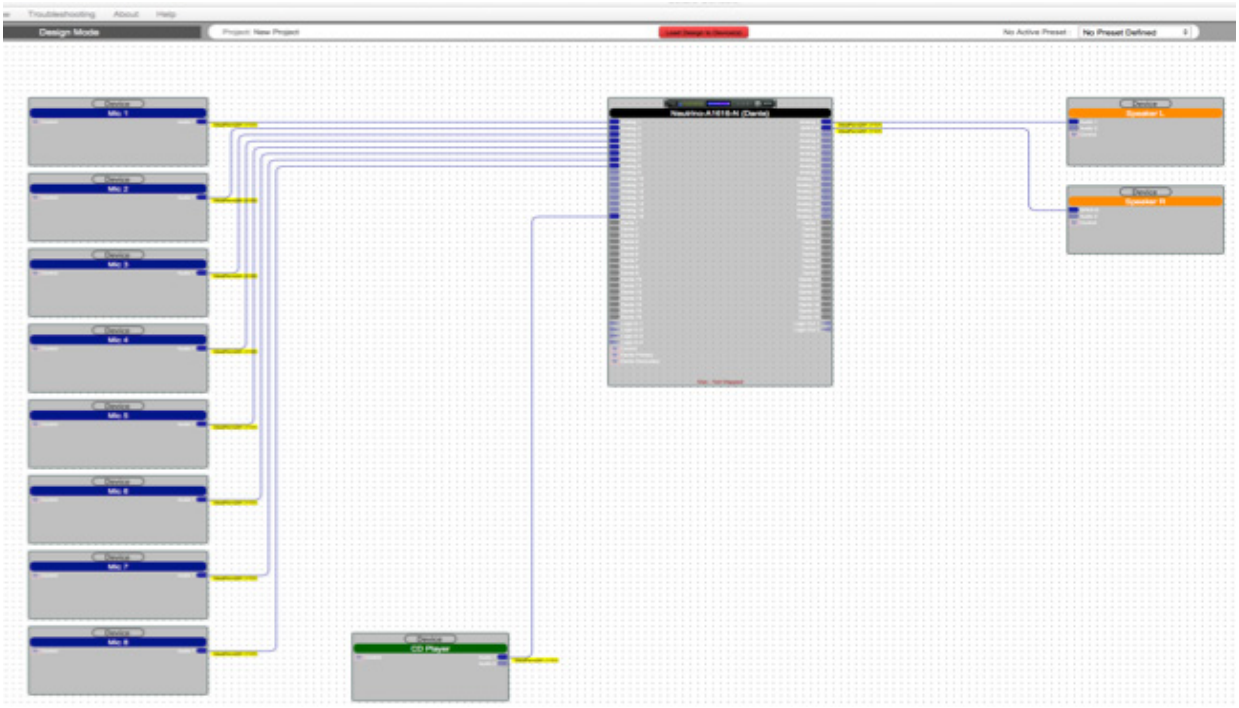


Wire adjustment may be necessary. Select the wire and use the red nodes to adjust the wire path. You may also select multiple wires and adjust them as a group.

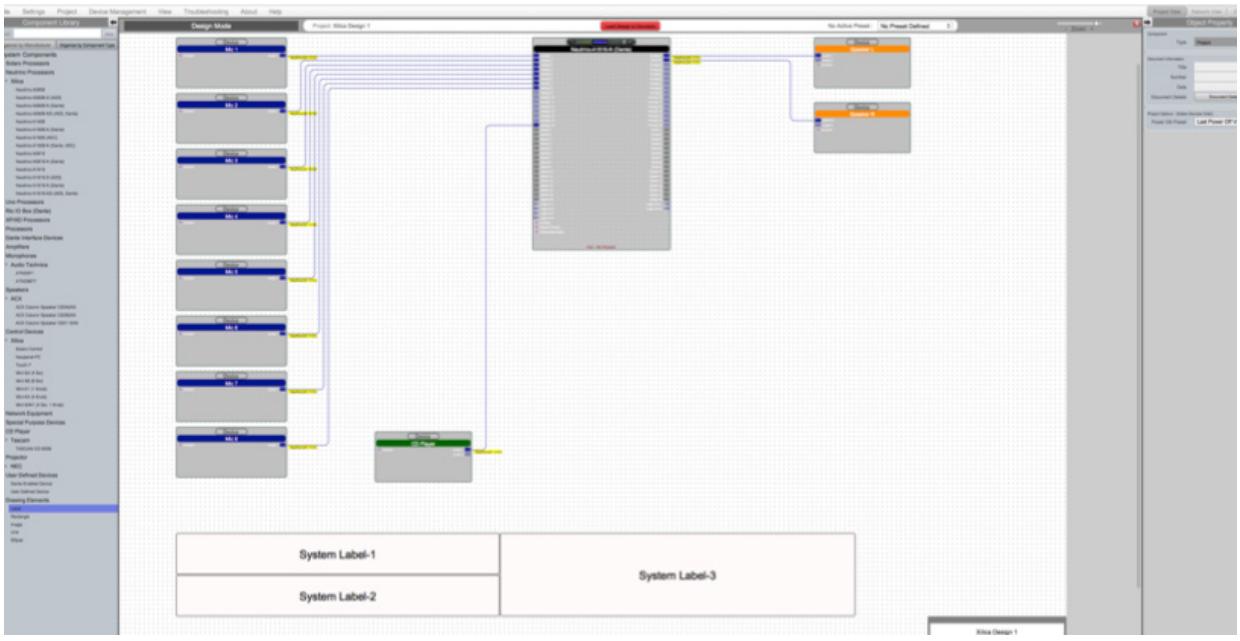
7. Wires can also be named and labelled under the Object Property menu on the right.
Select a wire and change the Cable Information. Labelled wires will be displayed in the work area.



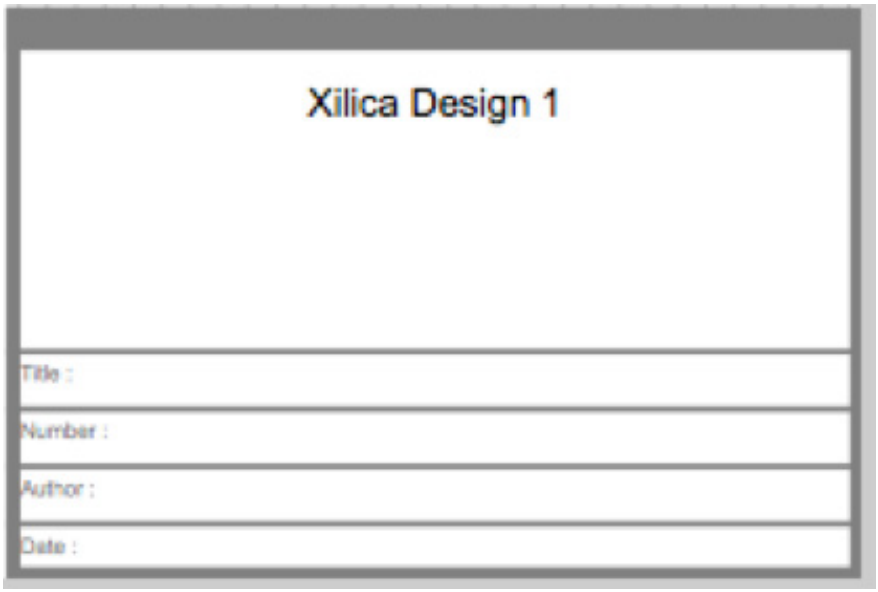
'Source Label' labels the input of the wire. 'Dest. Label' labels the wire destination.



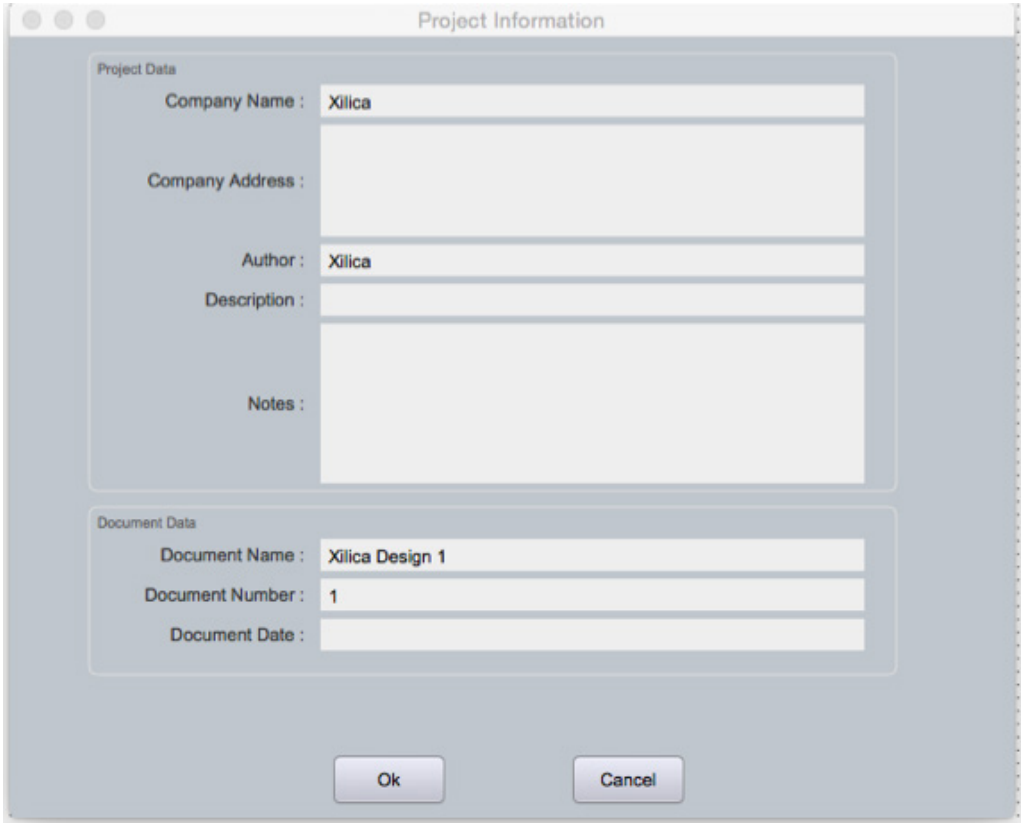
8. Under the ‘Drawing Elements’ in the Component Library menu, labels, shapes and lines can be added to the project work area for the finished look of the Blueprint.



9. At the bottom right of the work area displays a Project information box.



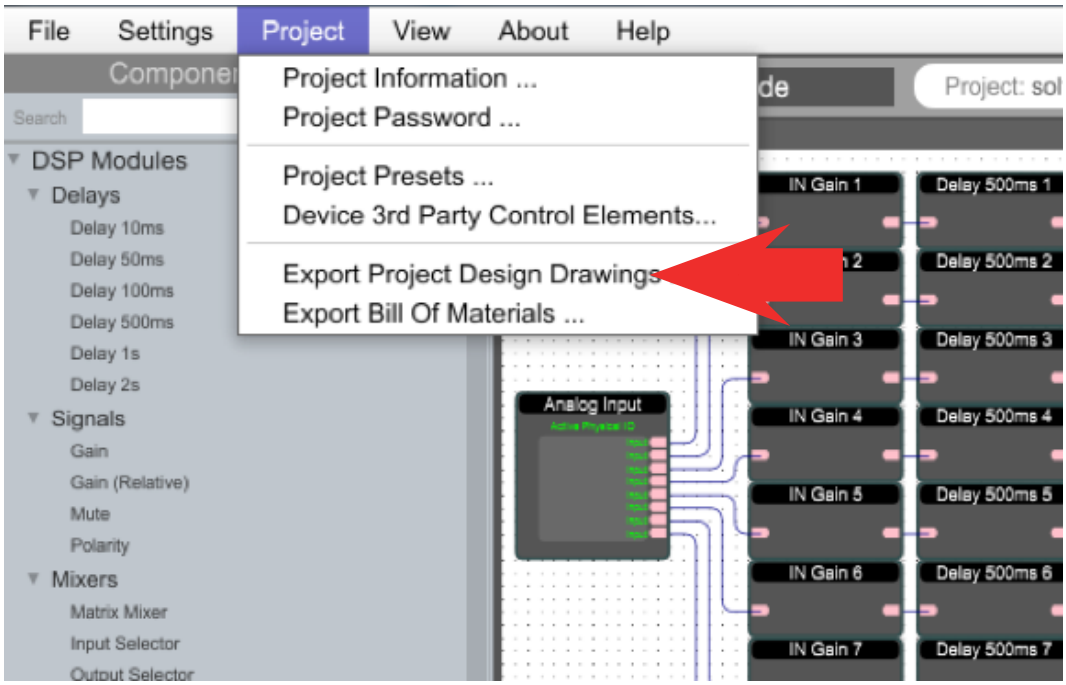
Simply edit the project information by double clicking this box. Then click 'Done' to save your changes.



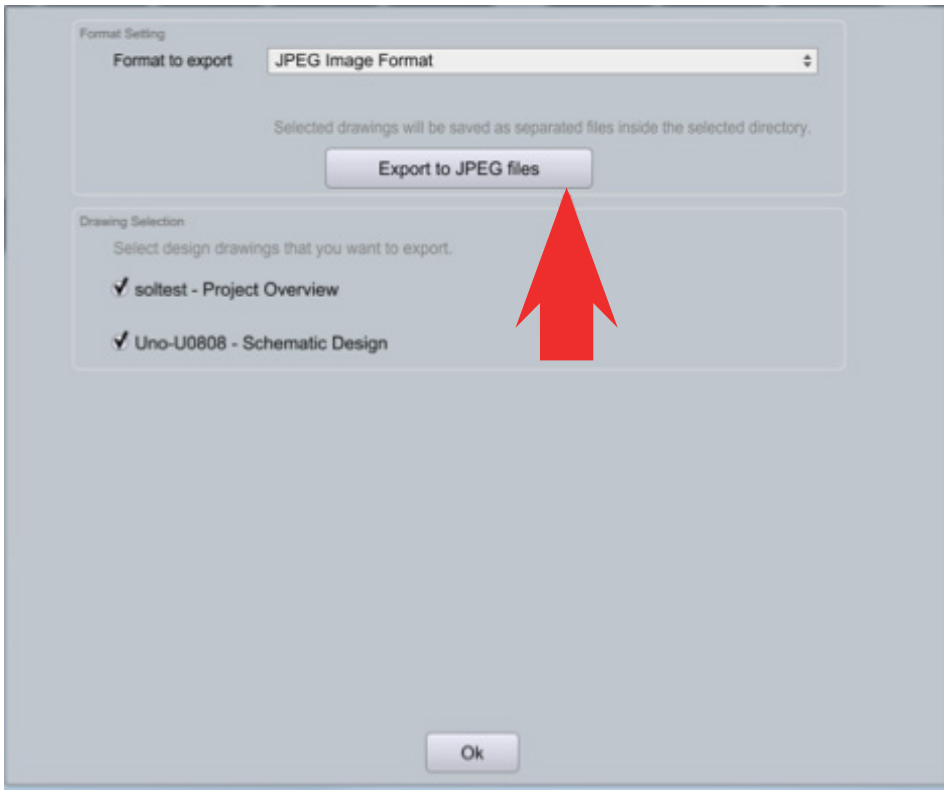
To this point only the documentation has been created for the design.

Export design to Jpeg or AutoCAD

At any point in the design process, you have the ability to print a copy of your project as a .jpg or .dfx image file.



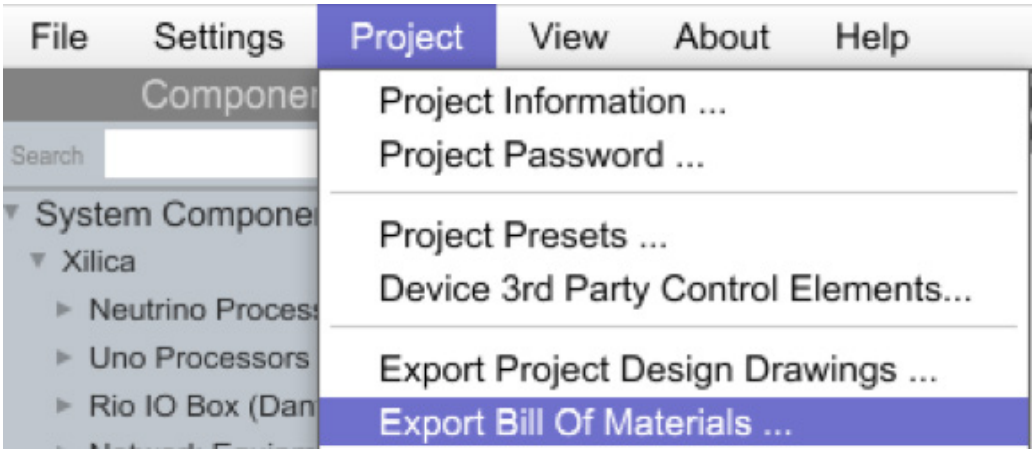
Under the 'Project' tab at the top of the software, select 'Export Project Design Drawings…'
This will print a jpeg image of the project view at a resolution of 1800 x 1200 pixels.



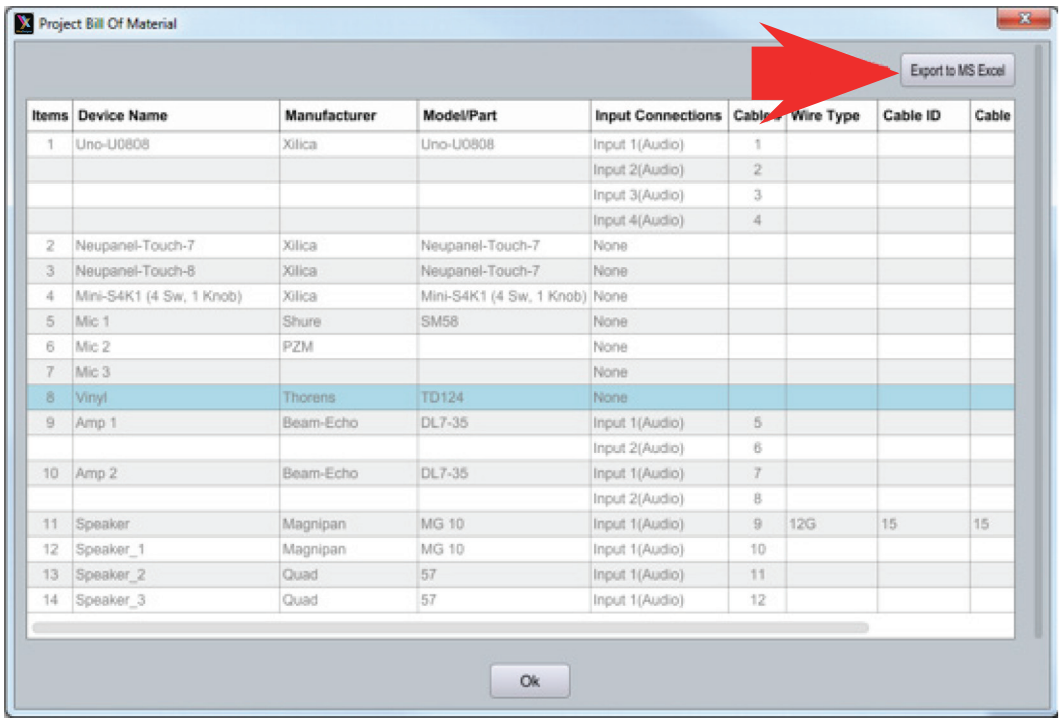
Export Bill of Materials

Export Bill of Materials generates an organized document listing all physical cabling and hardware description required for your project. Elements such as CD Players, microphones, amplifiers, and speakers, all play an important part in the compilation of the bill of materials.

Under the 'Project' tab, select 'Export Bill of Materials'.



You may need to save your project first. At the top left of the software, click 'File' and 'Save Project As'. Navigate to your destination folder, type in the file name as you wish, and then click 'Save'.



Items	Device Name	Manufacturer	Model/Part	Input Connections	Cable	Wire Type	Cable ID	Cable
1	Uno-U0808	Xilica	Uno-U0808	Input 1(Audio)	1			
				Input 2(Audio)	2			
				Input 3(Audio)	3			
				Input 4(Audio)	4			
2	Neupanel-Touch-7	Xilica	Neupanel-Touch-7	None				
3	Neupanel-Touch-8	Xilica	Neupanel-Touch-7	None				
4	Mini-S4K1 (4 Sw, 1 Knob)	Xilica	Mini-S4K1 (4 Sw, 1 Knob)	None				
5	Mic 1	Shure	SM58	None				
6	Mic 2	PZM		None				
7	Mic 3			None				
8	Vinyl	Thorens	TD124	None				
9	Amp 1	Beam-Echo	DL7-35	Input 1(Audio)	5			
				Input 2(Audio)	6			
10	Amp 2	Beam-Echo	DL7-35	Input 1(Audio)	7			
				Input 2(Audio)	8			
11	Speaker	Magnipan	MG 10	Input 1(Audio)	9	12G	15	15
12	Speaker_1	Magnipan	MG 10	Input 1(Audio)	10			
13	Speaker_2	Quad	57	Input 1(Audio)	11			
14	Speaker_3	Quad	57	Input 1(Audio)	12			

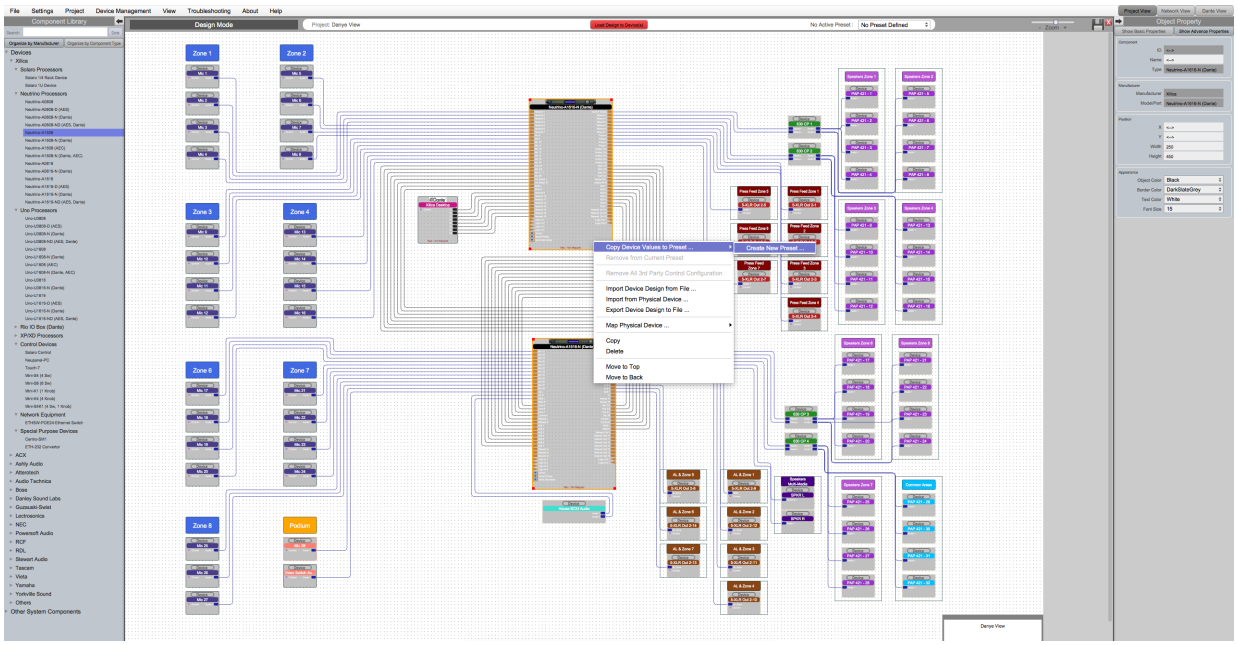
Clicking on "Export to MS Excel" will generate an XLS file with the information entered. This can be saved as desired.

Presets

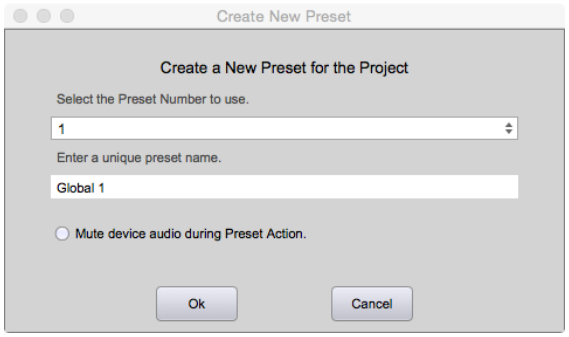
Global presets

Global presets allow the user to recall a saved setting for all devices in the system.

1. Make sure all devices in the system are set to their desired settings. Click and drag a selection box around all devices to highlight them. Then, right click on a DSP hardware module blocks.
2. Select 'Copy Device Values to Preset'.
   Then 'Create New Preset'.



3. When creating a new preset, the preset will automatically be saved in the next available preset slot. You can rename the preset and also choose to 'Mute Device during Preset Action'.

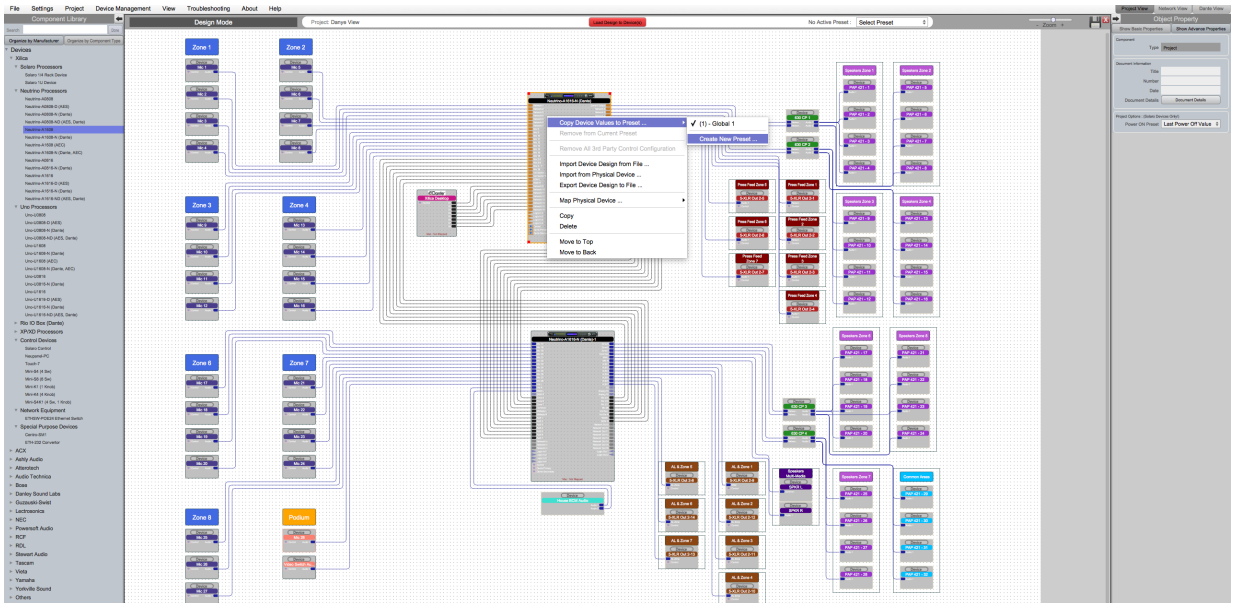


4. Click 'Ok' to save your settings.

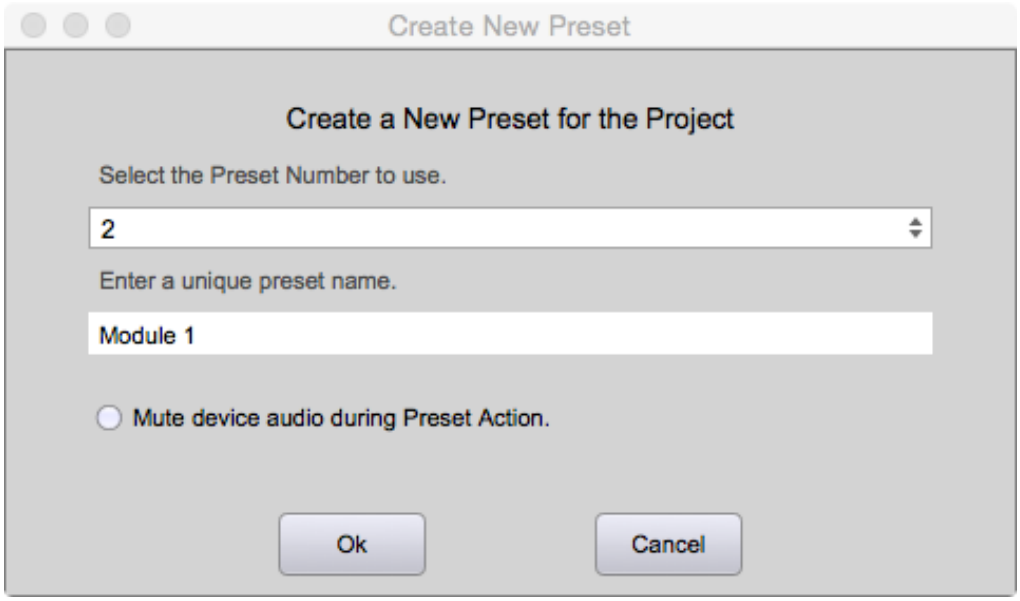
Module presets

Module presets allow the user to recall a saved setting for a device module.

1. Make sure that the device is set to the desired settings. Then right click on the device.
2. Select ‘Copy Device Values to Preset’.
   Then ‘Create New Preset’.



3. When creating a new preset, the preset will automatically be saved in the next available preset slot. You can rename the preset and also choose to ‘Mute Device during Preset Action’.

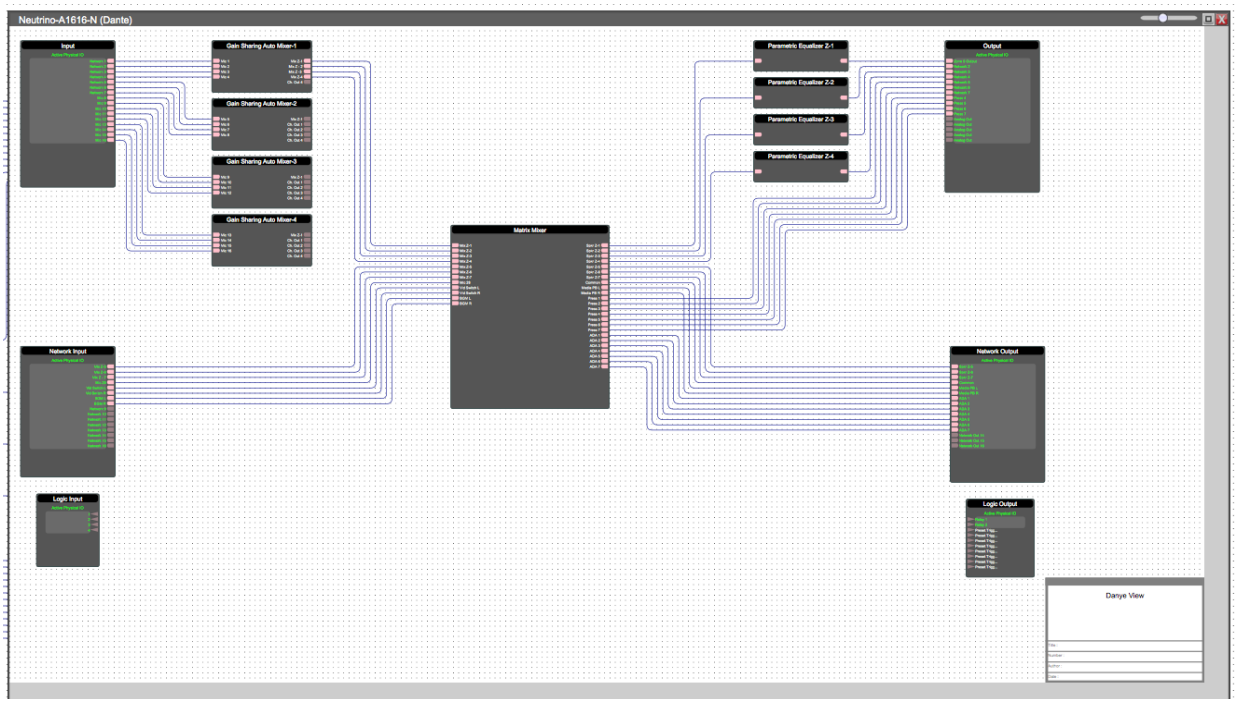


4. Click ‘Ok’ to save your settings.

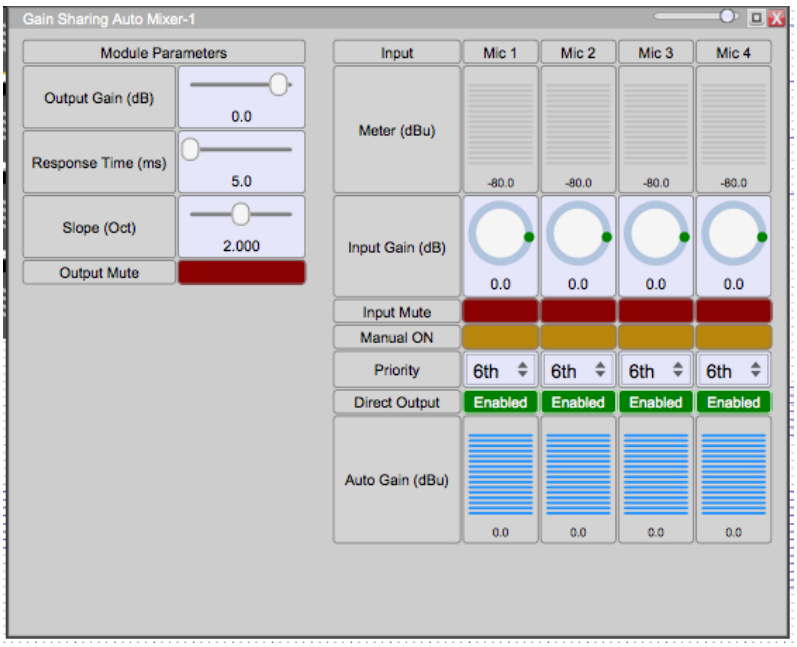
Individual DSP module preset

To create a DSP module preset,

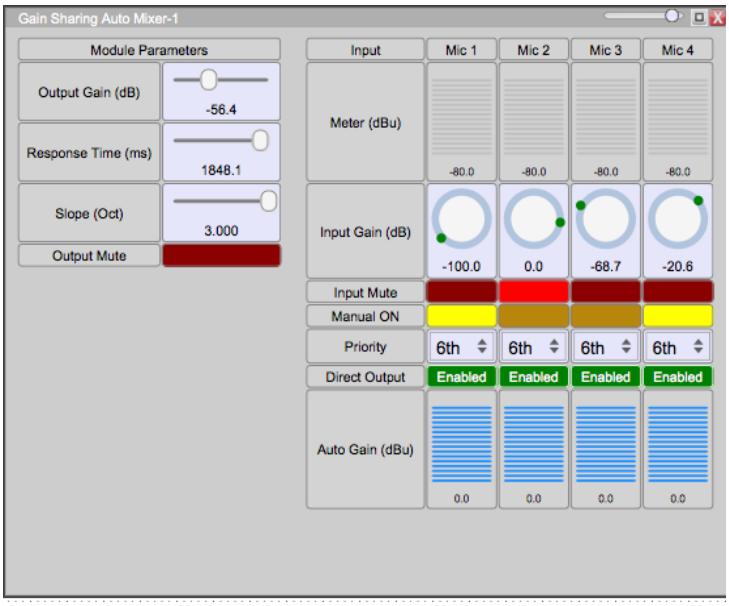
1. Double click a device in your work area to open the device schematic.



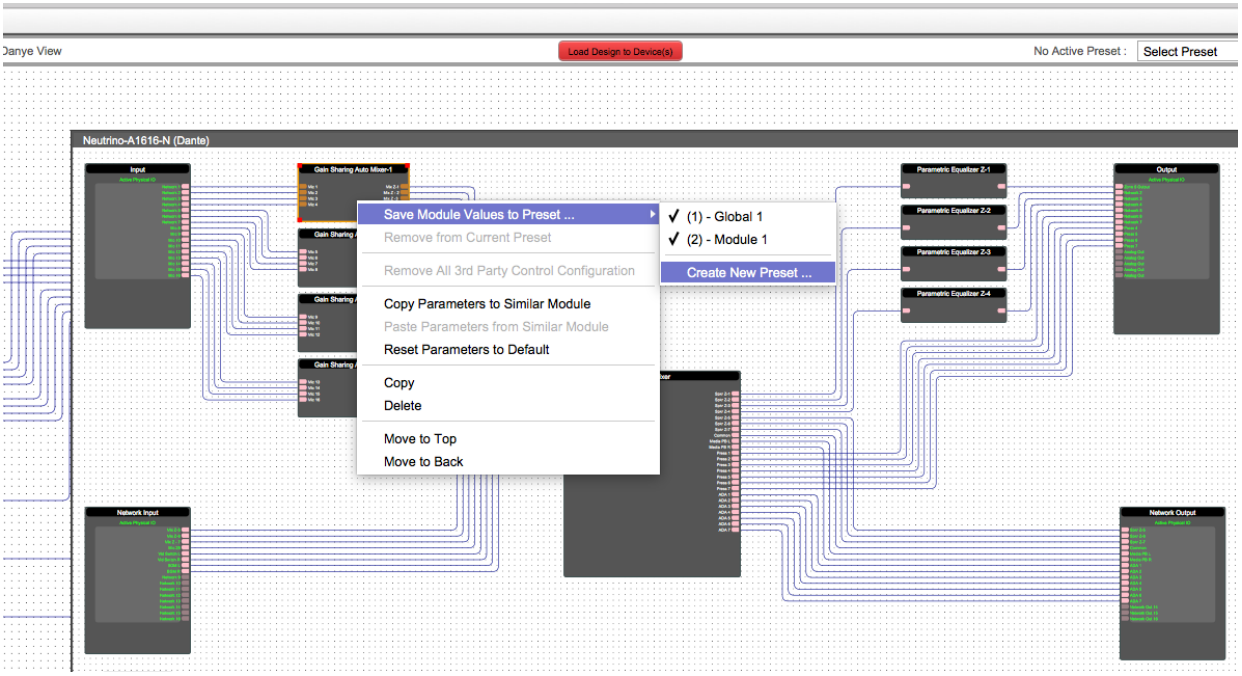
2. Double click a processing block to open it.



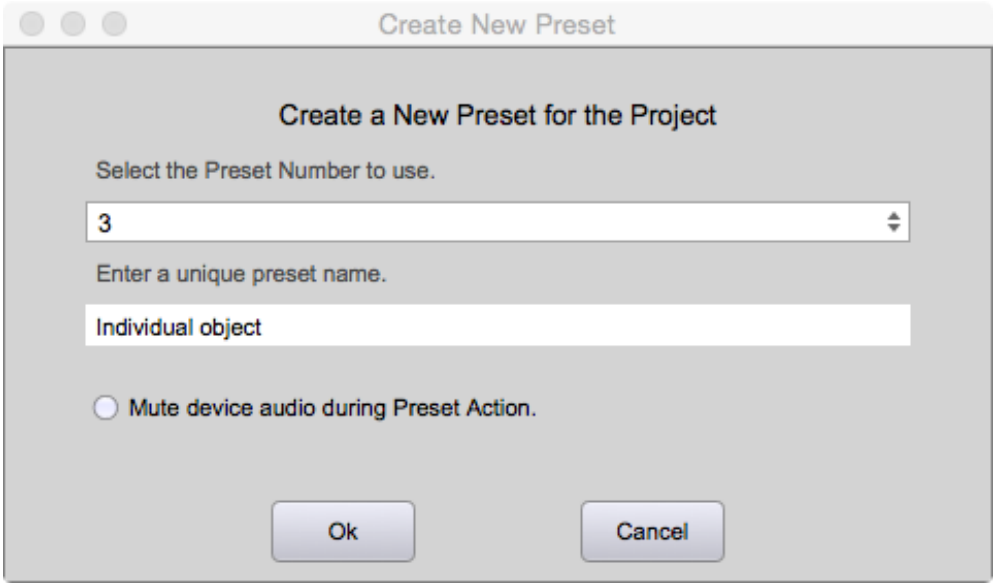
3. Adjust the processing module parameters to the settings that you'd like to save.



4. Close the module parameters. Right click the DSP module block and select 'Copy Device Values to Preset'. Then 'Create New Preset'.



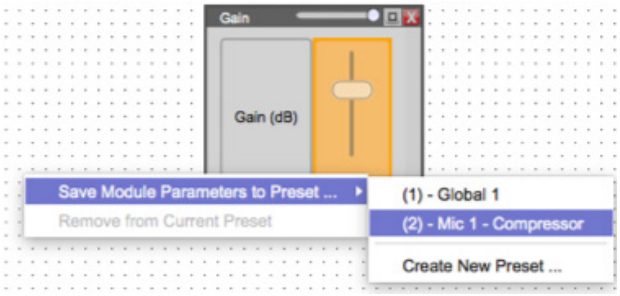
5. When creating a new preset, the preset will automatically be saved in the next available preset slot. You can rename the preset and also choose to 'Mute Device during Preset Action'.



6. Click 'Ok' to save your settings.

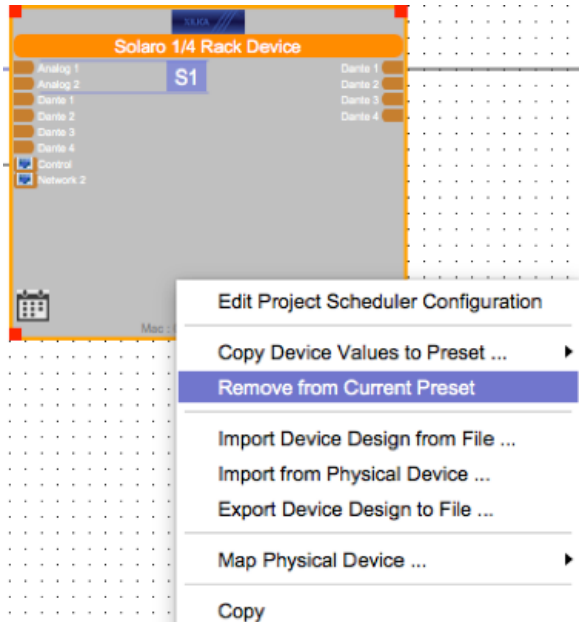
Adding new parameters/modules/devices to an existing preset

1. Click and drag the selection box to highlight the desired parameters/modules/devices Or Hold Cmd + Click (Mac), or Ctrl + Click (PC) to select individual objects.
2. With the desired objects highlighted, right click and select 'Save Module Value to Preset'.
3. Select the preset number/name that you would like to add the new selection to.



Removing parameters/modules/devices from an existing preset

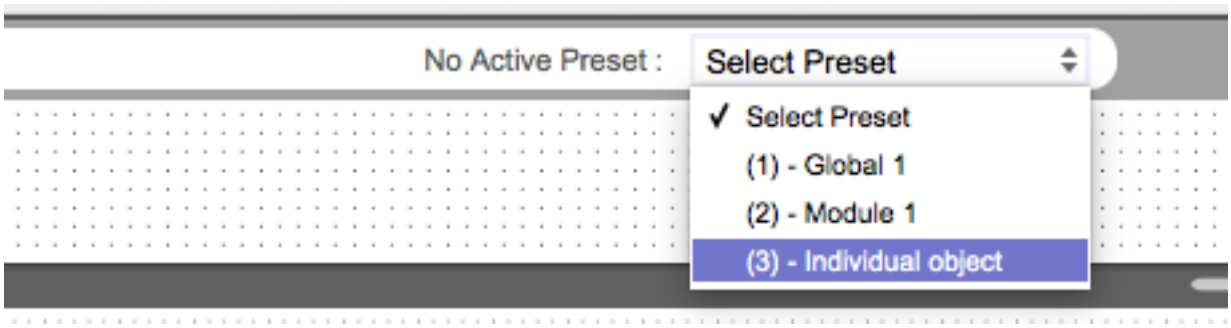
1. With the preset activated, select the object(s) you would like to remove.
2. Right click and select 'Remove from current preset'



Activate presets

To activate saved presets,

1. At the top right of the work area beside the heading "Active preset", select the drop down menu.
2. This will list all of your saved project presets. Select your desired preset to activate the preset.

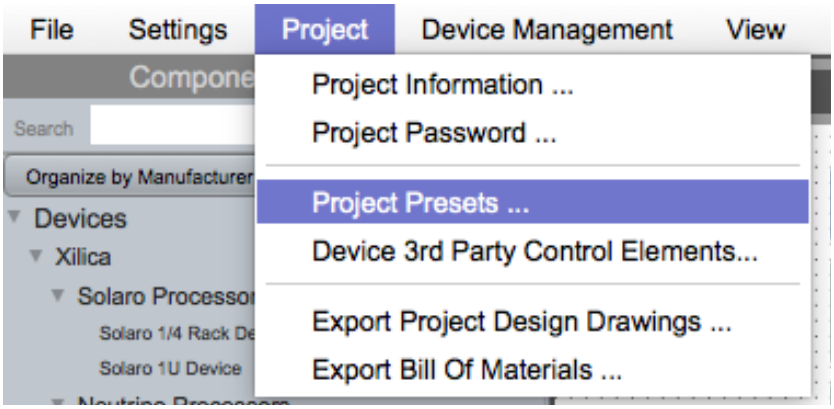


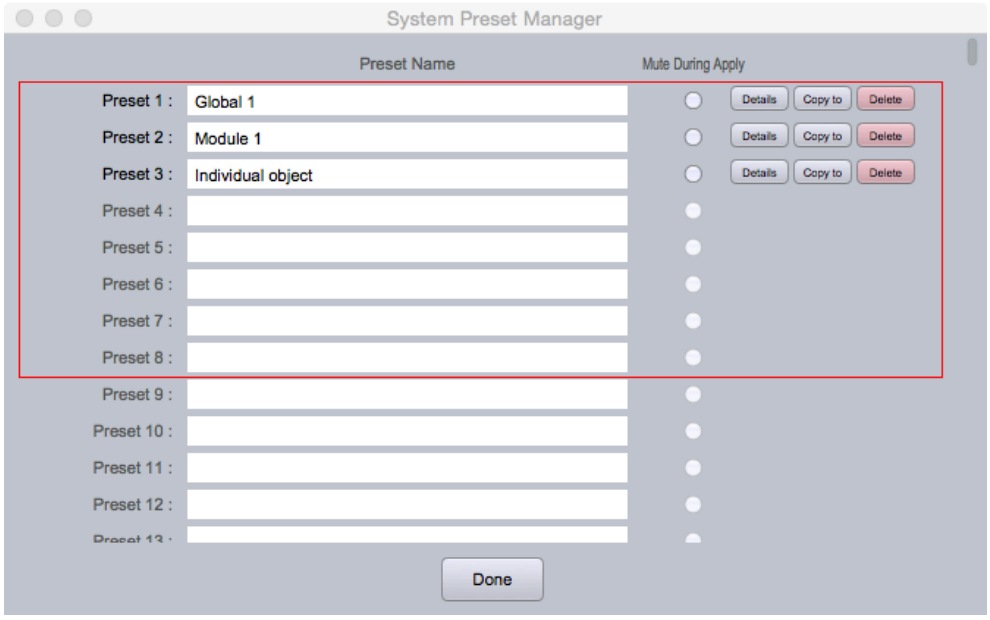
3. After applying the preset, any Device, Module and Parameter saved in the preset will be outlined in Red.

Preset management

The Preset Management window displays all of your saved presets and can be used to rename, copy and remove presets from the project file.

1. At the top of the software, under the Project tab, select “Project Presets”.
Project Presets lists all presets in the project.





In this window you can view the 200 available presets.
The red box highlights presets that can be controlled by GPIO inputs. This is done by wiring the GPIO circuit into the GPIO Output DSP Block. (Please refer to the Xilica Designer: GPIO guide)

Remove presets
Presets can be removed by checking the 'Delete' button next to the desired preset.

Rename presets
Presets can be renamed by typing in the 'Preset Name' text box next to the corresponding preset number.

Each preset includes a check box for 'Mute During Apply'. When changes are applied and saved, the Audio System will mute for a short period of time. Mute time is dependent upon how many parameters are being changed within the selected preset. Uncheck this box for a seamless preset change (Best for changing a small number of values).

Duplicating presets
Presets can be duplicated to and saved to other preset numbers/names by selecting 'Copy to'.

Save your changes by clicking '**Done**' at the bottom of the window.

Saving changes from Online mode

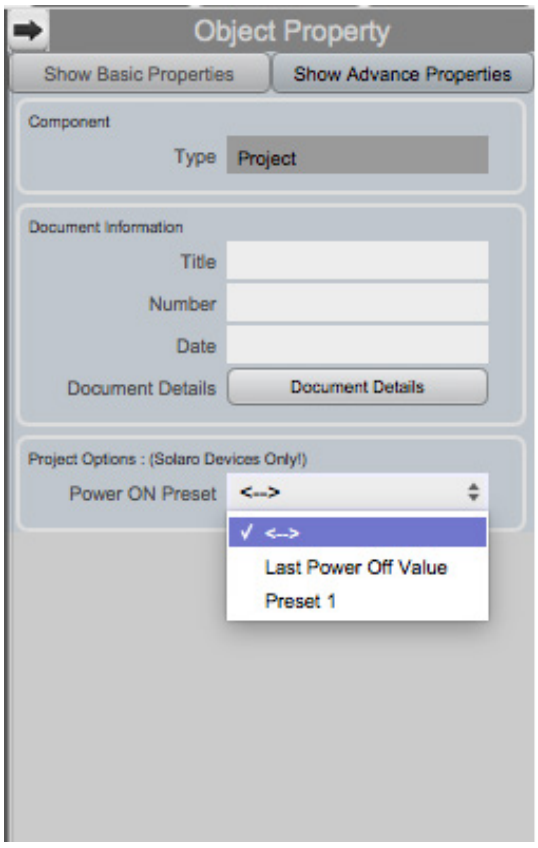
1. If you are online and want to save your changes back into the design file, select 'Yes' when prompted to copy device parameters when switching back to design mode.
2. The changes will be applied to your design. You may add the new parameter/module/device settings to an existing preset or create new presets.

Working with power ON preset

The power ON preset applies to your hardware devices automatically once you turn your hardware device ON. Please note this is only available for Solaro devices.

To enable the Power ON preset,

1. Click on the dotted work area.
2. In the Object Property menu beside 'Power ON Preset', click the drop down menu.
3. Select the desired preset that you would like to recall when your devices are powered ON.

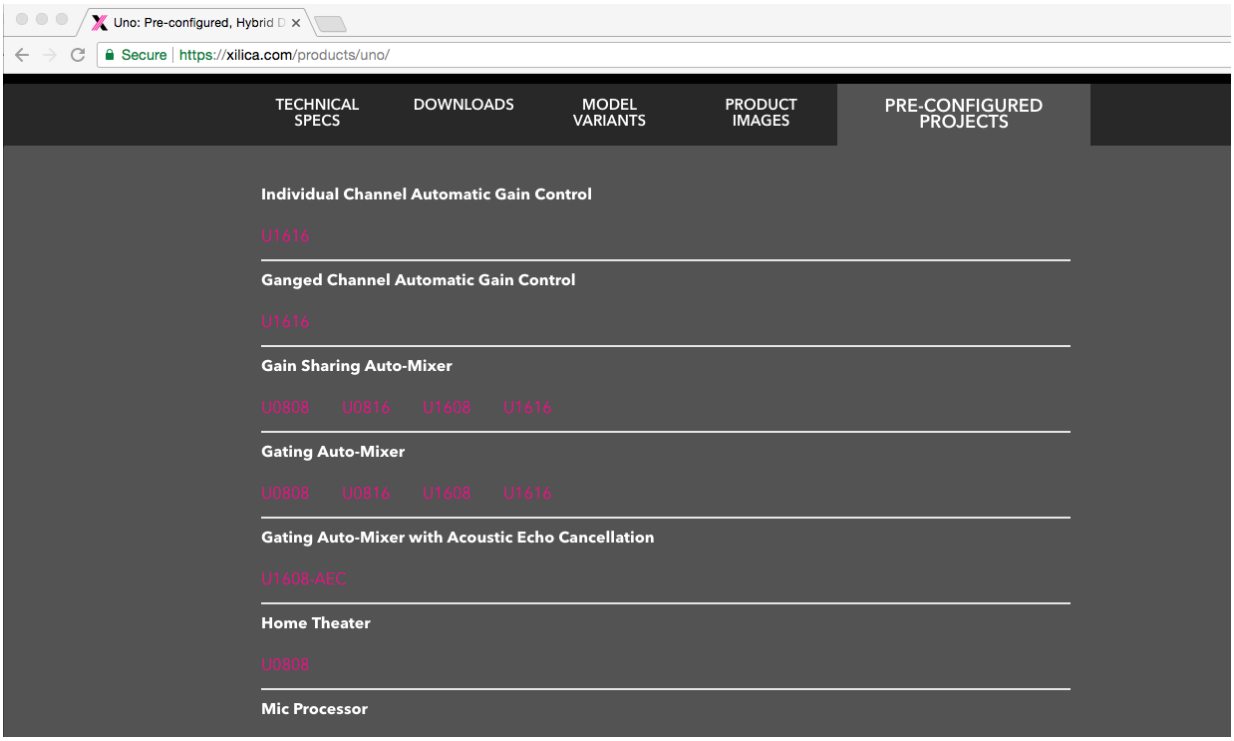


Importing a pre-configured project design into an Uno device

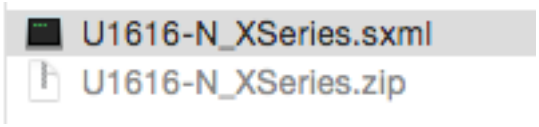
To browse pre-designed Uno projects, visit the Xilica website. (www.xilica.com)
For free, custom Uno projects, please contact Xilica's technical support team at: support@xilica.com.

Note: the Uno project you choose must match the processor module you are using. For example, you cannot load an Uno U1616 project into an Uno U0808 processor.

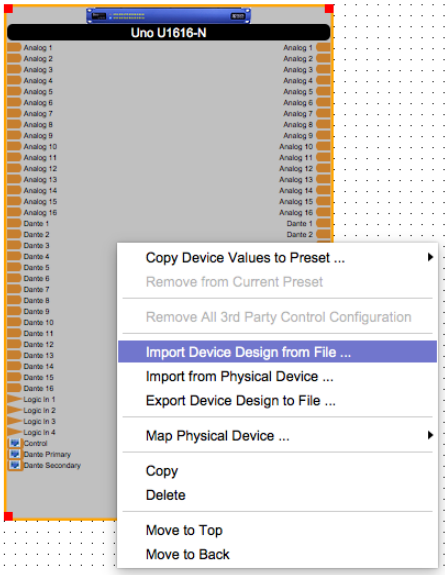
1. Navigate to the Xilica website (www.xilica.com). At the bottom of the Uno product page, select 'Pre-designed projects' tab and choose a project to download and apply to your Uno device. Save the project file (.zip) to a memorable location on your computer.



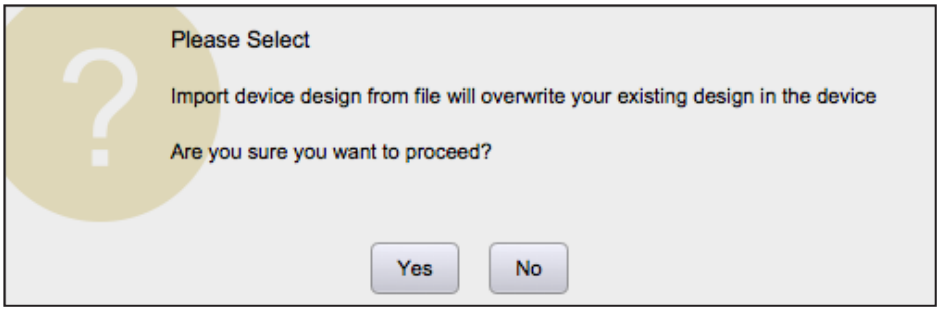
2. Double click the downloaded .zip project file to open it. An .sxml project file will appear. Uno projects are .sxml files (New Project). Saved DSP design projects are .pjxml files (Open Project).



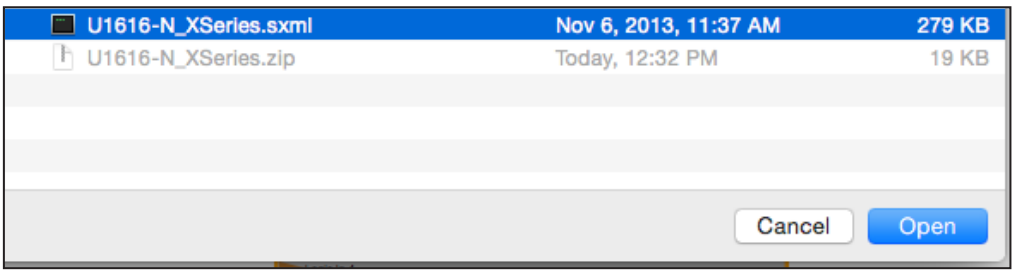
3. Navigate back to Xilica Designer's project view.
Right click the Uno DSP modules and select 'Import Device Design to File'.



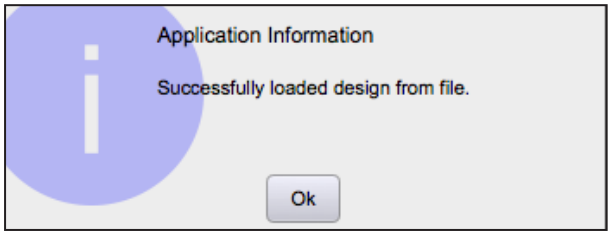
4. Please note that loading a project design into the Uno device will override all settings on the Uno. Click 'Ok' to proceed.



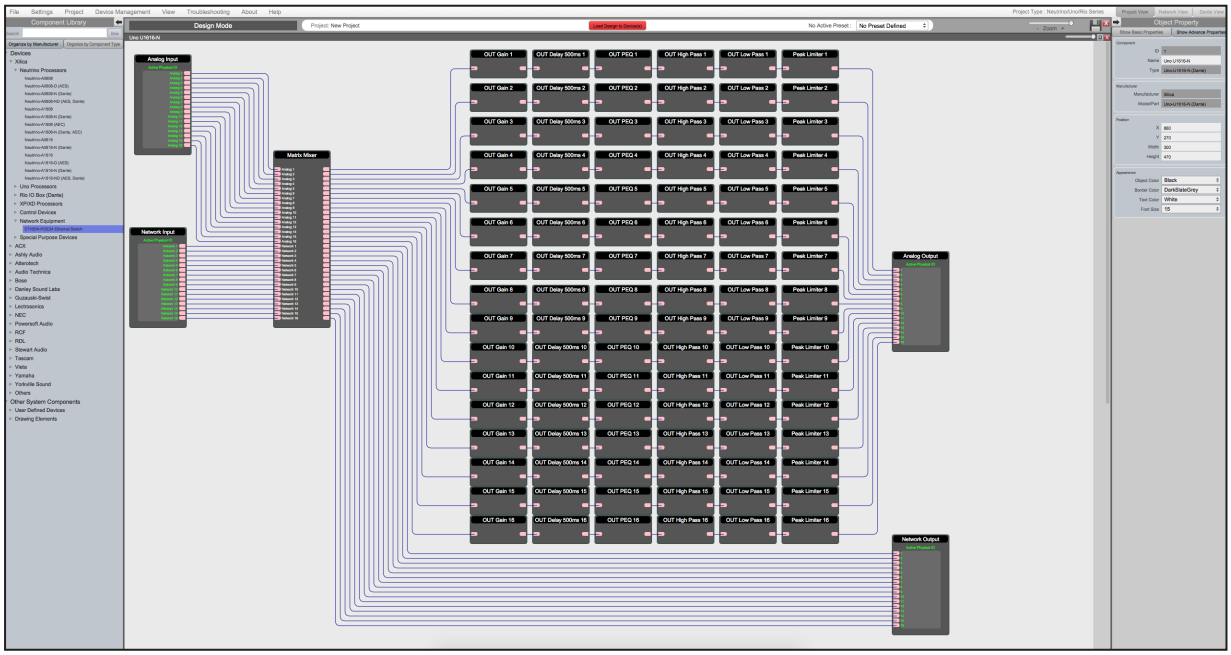
5. Locate and select the Uno project file (.sxml) that you have downloaded from the Xilica website. Click 'Open' to import the file to your Uno device.



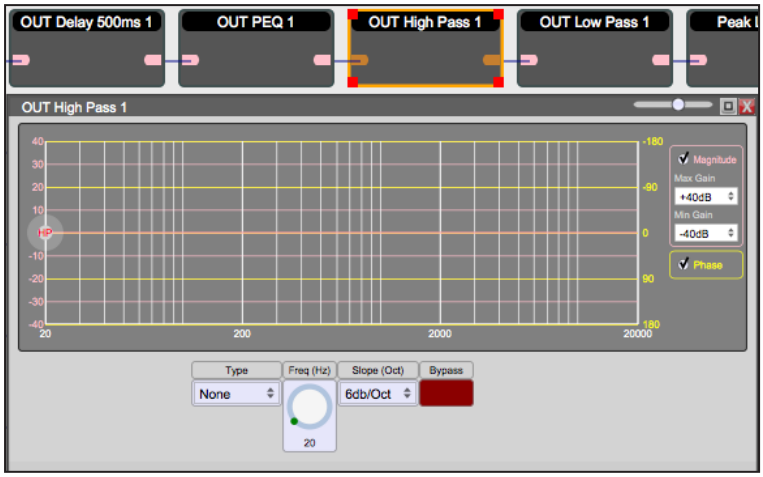
6. When the file has successfully imported into the Uno module, the following message will appear. Click 'Ok'.



7. Double click the Uno DSP module to view the imported project file.
Click and drag the corner of the new window to resize it. You may also use the zoom slider at the top right of the window to adjust the view of the work area.

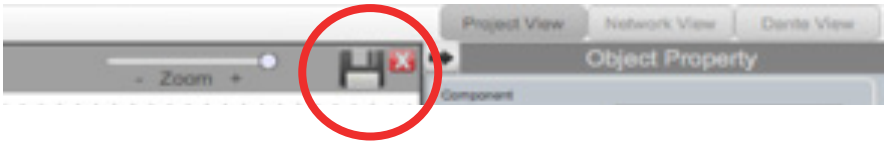


8. Double click on any parameter module to open the module and make parameter adjustments as required. In the example, a High Pass filter was opened.



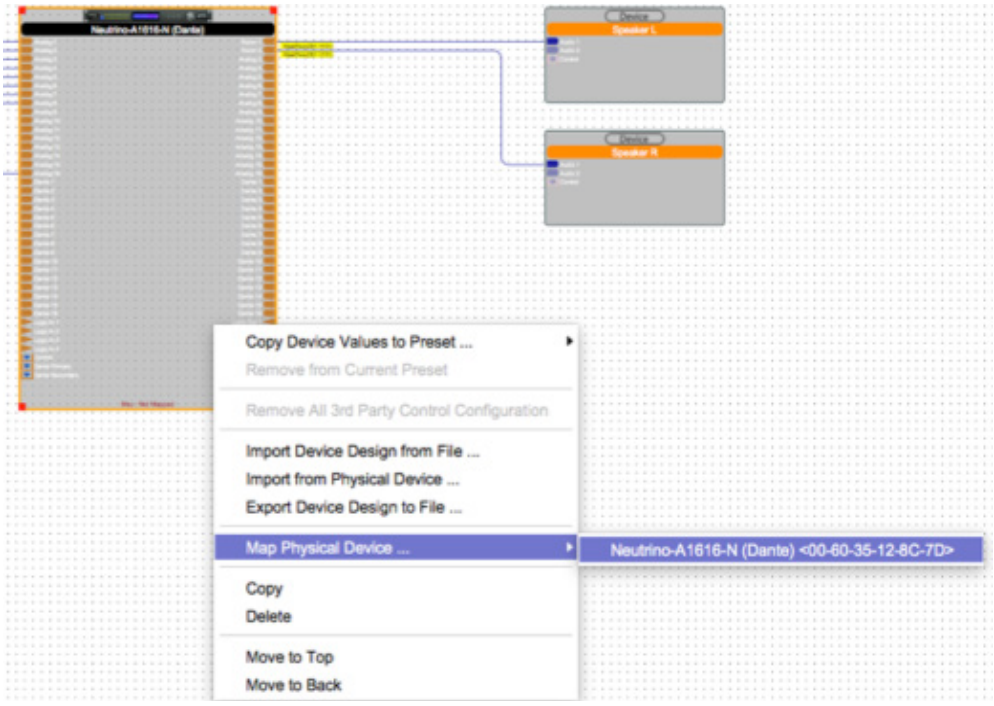
Map device(s)/Online mode

Before going online, you will be prompted to save the project file.
Simply save by using the top 'File' tab and 'save' or click on the disk icon at the top right of the work area.



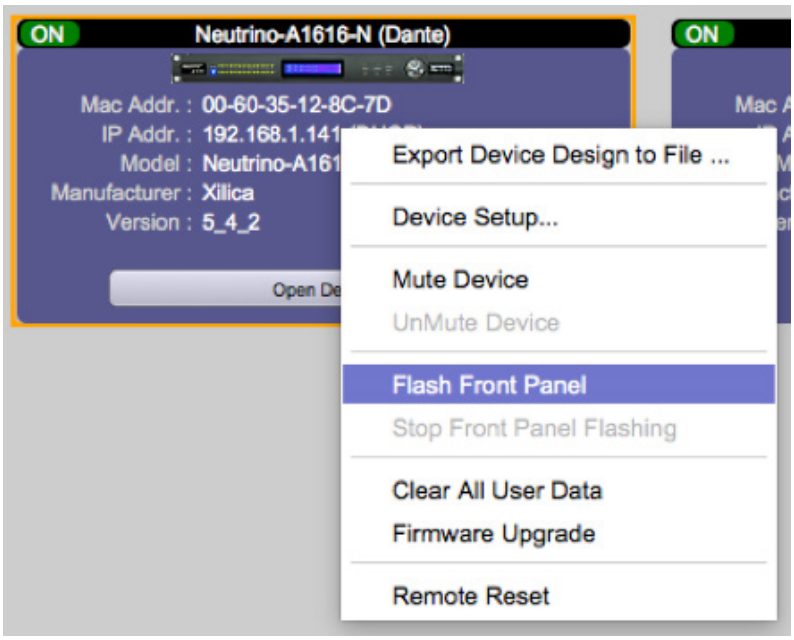
Please note that in order to go Online, all devices must be connected and online. (Displayed in Network view with a green indicator).

1. Right click the DSP device that you would like to map.
2. Select 'Map to Physical Device'.
3. A drop down menu will appear. Select the correct device.

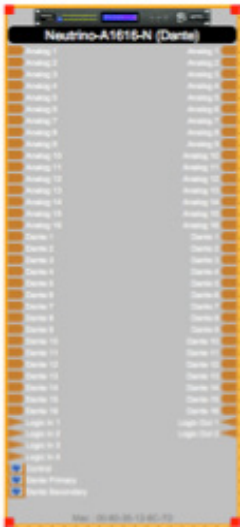


Note: If there is more than one of the same devices in the network, match the device Mac address in Network view with the device listed.

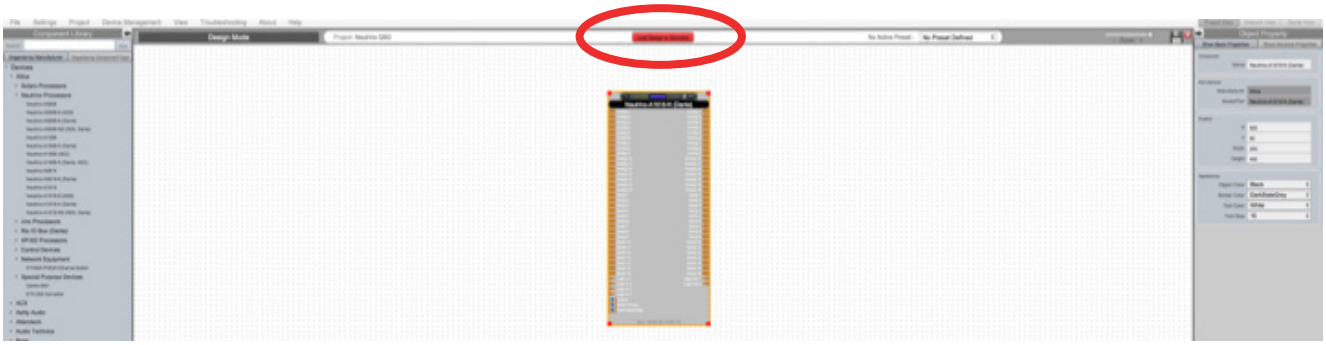
You can also flash the front panel lights on the correct device by right clicking on the device in Network View and selecting 'Flash Front Panel'.



Once mapped, the module will become a solid grey colour.

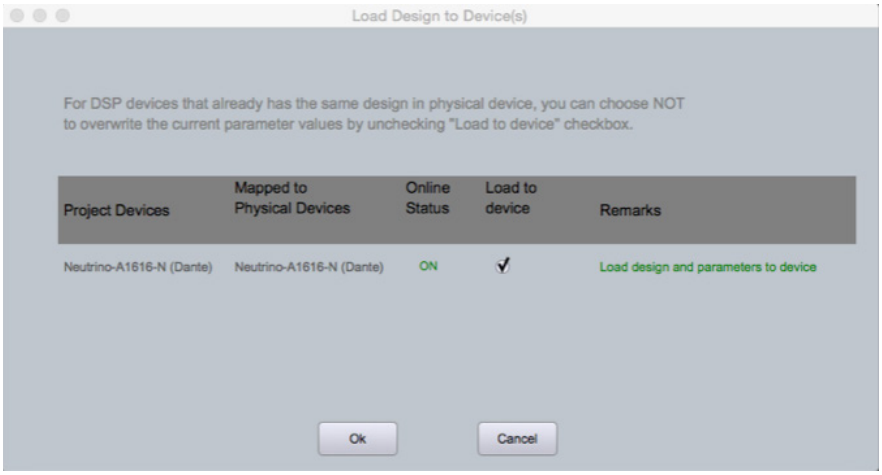


4. Then click the red 'Load Design to Device(s)' button at the top of the work area.

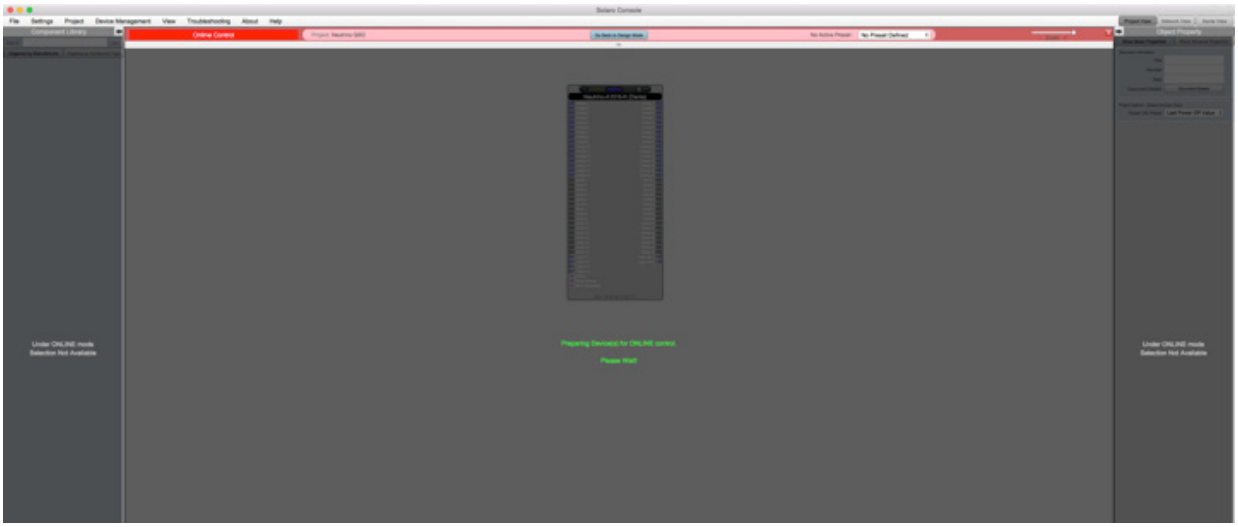


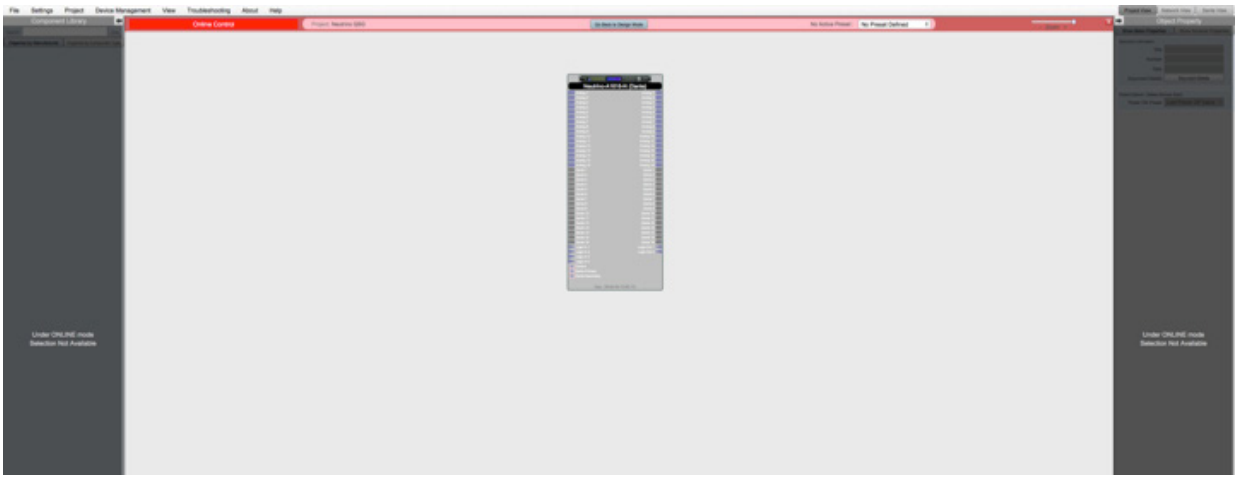
Note: It is very important to make sure the name of the DSP block in the design file matches exactly to the unit in the Network View. Otherwise you will not be able to load the design to the physical device.
The name can be changed either in the design file to match the DSP device on the network
OR the DSP unit's name can be changed in Network view by right clicking the device and selecting Device Setup.

5. A window will pop up. Check the connected devices that you would like to load your design to. Then click 'Ok'.



6. Going online may take up to several minutes. Please do not disrupt the process.
The progress bar at the top will display the overall progress percentage.

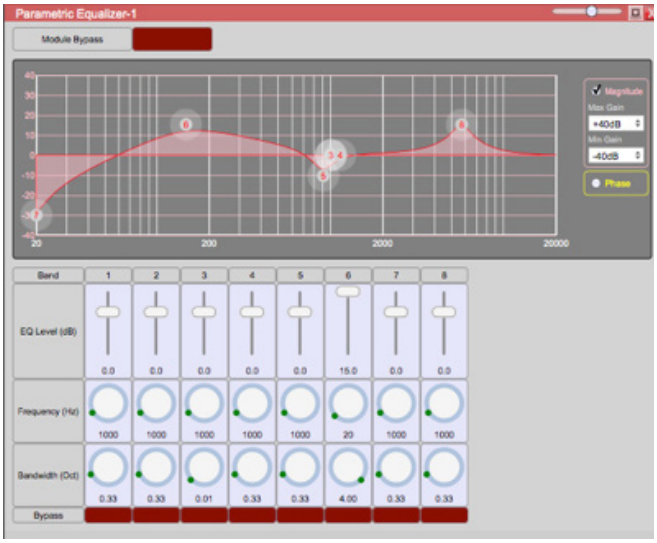




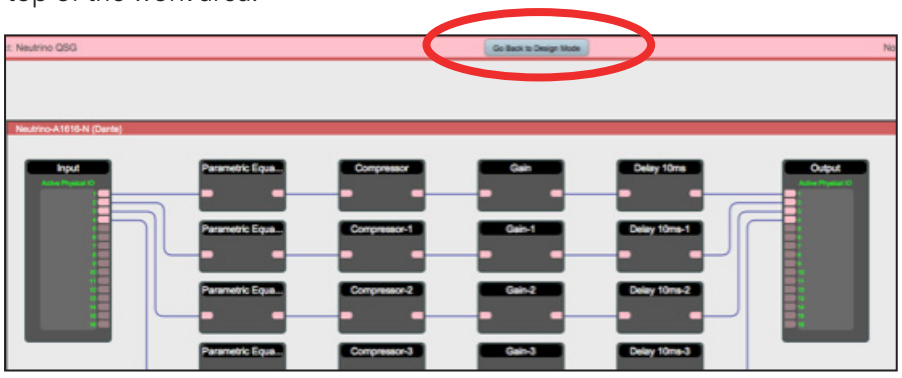
Once online, notice that the work area has become a solid color and the design menus are no longer available.

7. Double click the DSP module to open up the device schematic.

You can adjust the DSP module parameters in real-time.

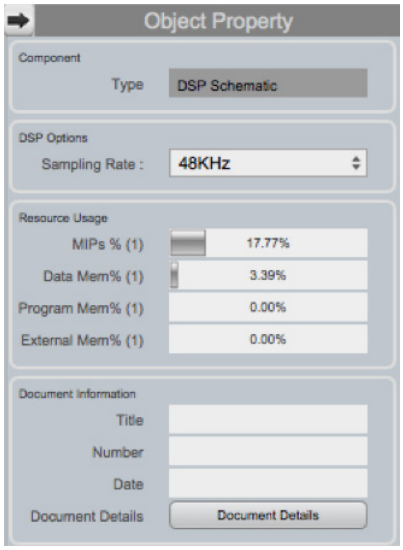


8. You can go back to design mode at any time by clicking the 'Go Back to Design Mode' button at the top of the work area.



Device resources

Device Resources can be viewed by clicking on the dotted work area within a DSP module.
When in Design mode, the device resource is an estimated DSP resource.
When in Online mode, the device resource is actual DSP resource.



Dante devices

To create a Dante' network exclusively using Solaro/Uno/Neutrino Series DSP hardware, no other software is needed to control network audio signal flow. However it is recommended that Dante' Controller Software is downloaded and available for the project as the Dante' Controller software adds additional tools for completing a successful project.

Important things to consider concerning Dante' use:

When designing a network of Dante-enabled devices,
Wherever possible, use gigabit links and always use gigabit links between switches. If your Dante-enabled devices have gigabit-capable interfaces then connect them using gigabit links. Enable QoS on your Ethernet switches when using 100Mbit/s devices.

Switches that should be used when using Dante enabled hardware:
Dante provides high performance audio networking on off-the-shelf Ethernet switches. Here is a non-exhaustive list of switches that have been used successfully with Dante:

- Linksys: SRW224G4, SRW2024, SRW2016, SRW2008, SRW208G, SLM2008, SLM2024
- Dell: PowerConnect 2708, PowerConnect 5324
- HP ProCurve: Various models including ProCurve 3500 series, ProCurve 2600 series
- Cisco: Various models including Catalyst 3750 series, Catalyst 3560 Series

Important features when purchasing a switch:
Dante makes use of standard Voice over IP (VoIP) Quality of Service (QoS) switch features, to prioritize clock sync and audio traffic over other network traffic. VoIP QoS features are available in a variety of inexpensive and enterprise Ethernet switches. Any switches with the following features should be appropriate for use with Dante:

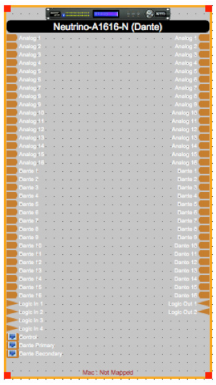
- Gigabit ports for inter-switch connections
- Quality of Service (QoS) with 4 queues
- Diffserv (DSCP) QoS, with strict priority

A managed switch is also recommended, to provide detailed information about the operation of each network link: port speed, error counters, bandwidth used, etc.

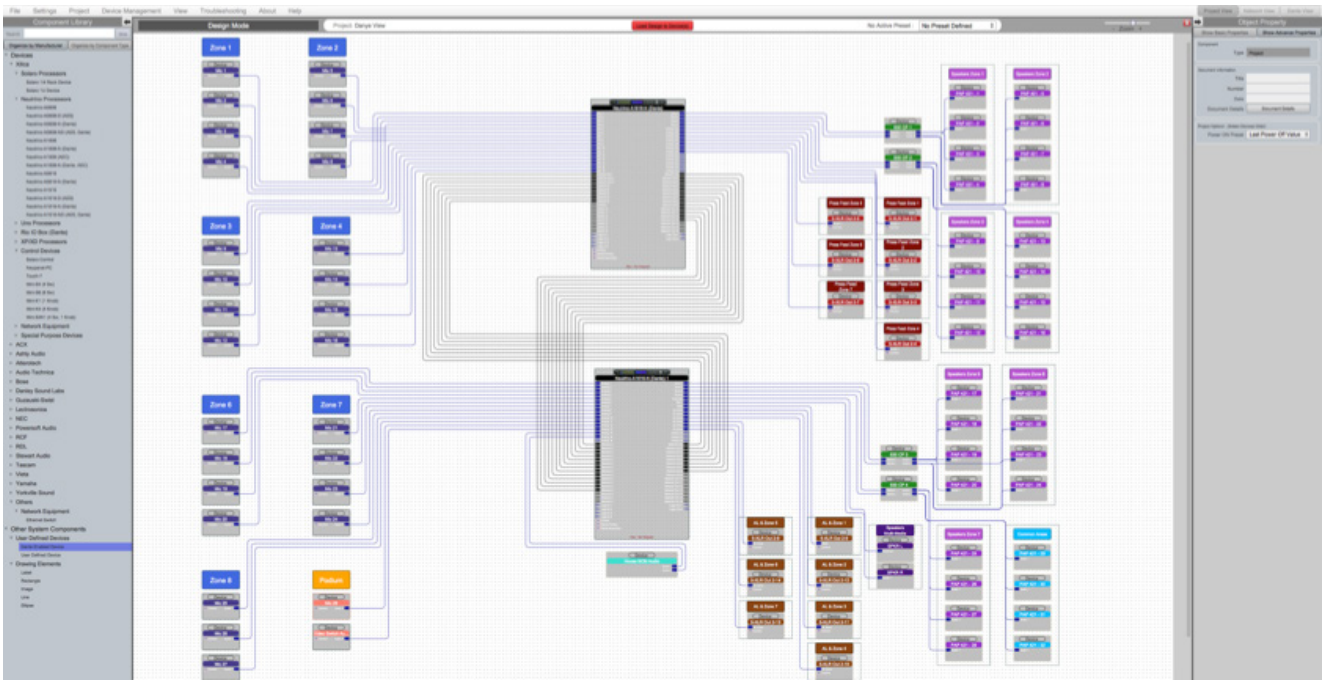
All Dante connections are made in the Xilica Designer software and there is no need to use the Audinate "Dante' Controller" software. However the Dante controller can still be used, if needed.
Dante controller is equipped with many useful functions such as: an Event Log, Clock Status to set the Master clock, Device Status including IP addresses and Routing. The Routing function allows for cross point connecting of the entire Dante audio network.

Note: If routing changes are made on the Dante Controller software, the Xilica Designer network will revert back to the original settings if there is a power cycle event. In other words, if there are routing changes that need to be permanent, the changes need to be reflected in the wiring of the Blueprint.
For a complete understanding of the Audinate Dante Platform refer to: www.audinate.com

Xilica 'N' model processor versions provide 16x16 I/O channels of Dante network audio bi-directionally over Ethernet. Xilica Designer also provides a list of available Dante enabled devices from other manufacturers to use within your project. You may also create your own user defined Dante device.



Below is an example of a typical conference centre design utilizing two Neutrino A1616-N DSP devices on an exclusive Dante' network. Dante' wiring is represented in black wiring. Analog wiring is in blue.



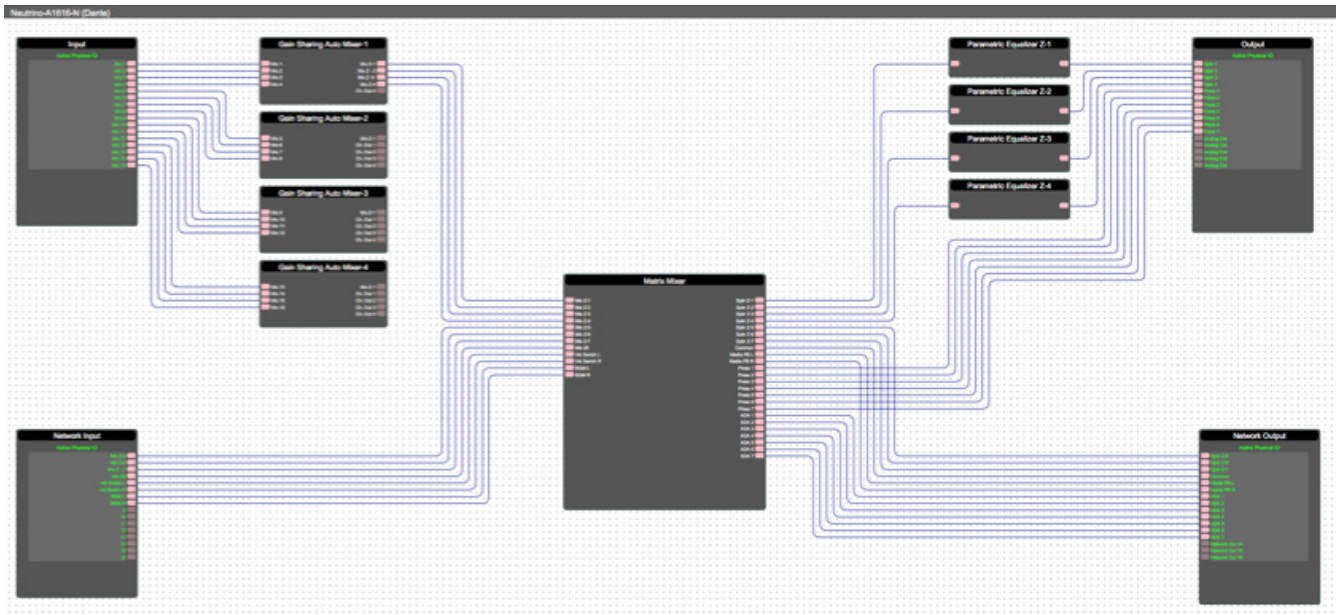
A brief Description of this audio design Blueprint:

Inputs: 27 conference table microphones broken out into 7 zones (mix-minus), 1 podium microphone, stereo audio from video switcher in podium and background music from facility.

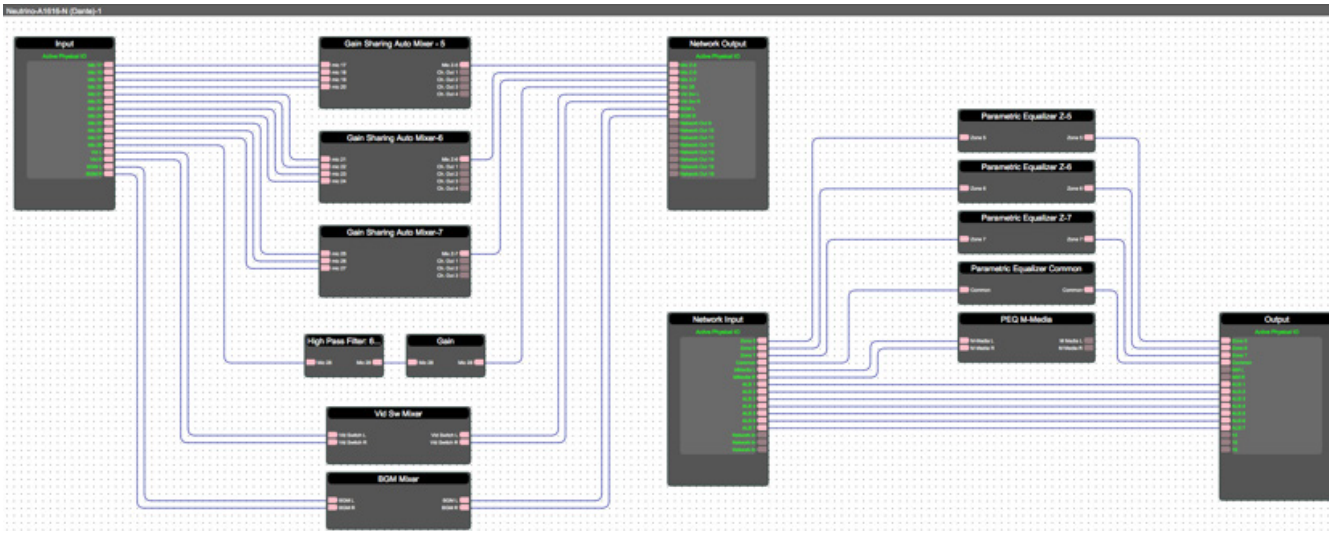
Outputs: 7 zones of distributed ceiling speakers, stereo main speakers for media playback, 7 zones of press feeds and 7 zones of assisted listening/translation. Dante network will provide audio transport function between the two Neutrino DSP units.

Follow the signal flow on the Network I/O blocks: The Network Output shown in the Unit 1 design flows through the Unit 2 Network Input, while the Network Output of the Unit 2 design flows through the Unit 1 Network Input.
It is very helpful to label the Network I/O to keep track of the signal flow in the design.

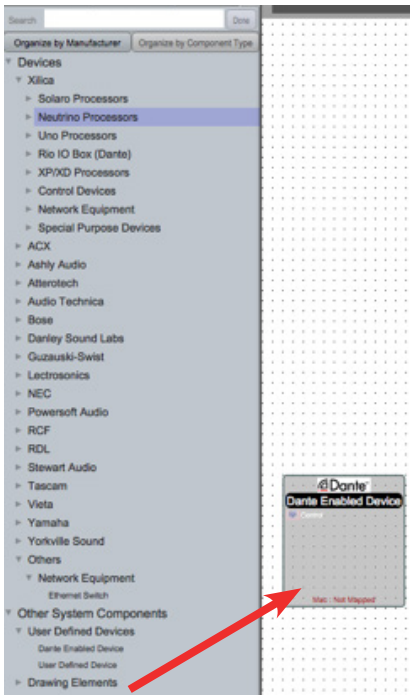
Example of the internal configuration inside the first Neutrino A1616-N DSP or “Unit 1”:



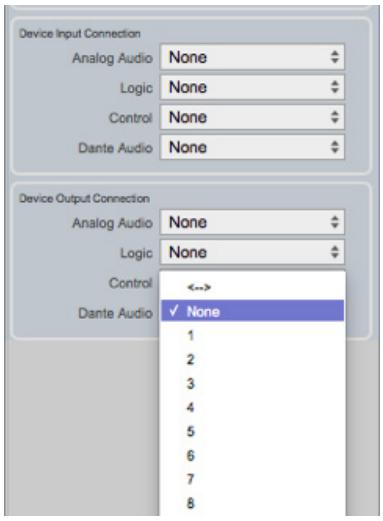
Example of the internal configuration inside the second Neutrino A1616-N DSP or “Unit 2”:



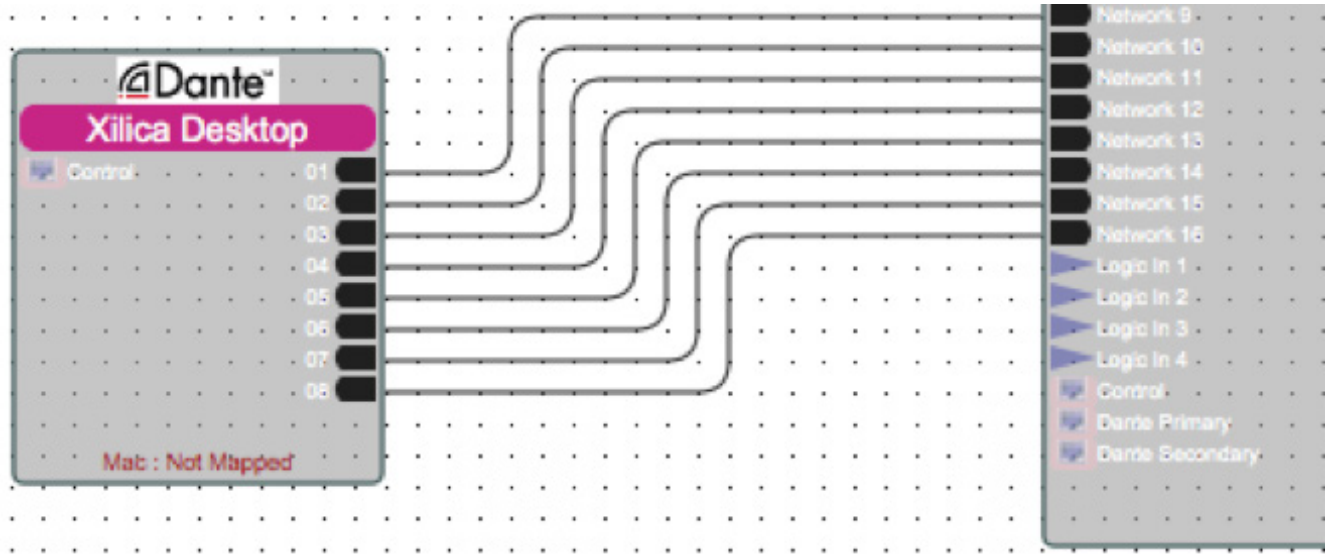
Creating a design with other Dante hardware can be found in the Project Design Element of the Component Libraries. You can select your Dante device from the list of manufacturers or you can create your own Dante device by adding a 'Dante Enabled Device'.



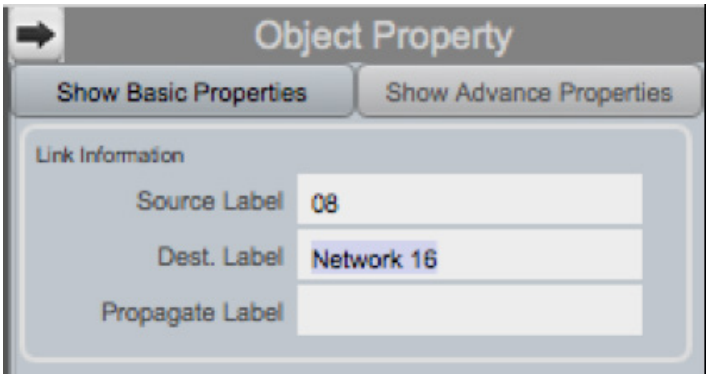
Adjust the number of Network Inputs or Outputs needed on the Dante Enabled Device to represent the third party hardware's function in the design. This is done in the "Input/Output" section of the Object Property menu on the right.



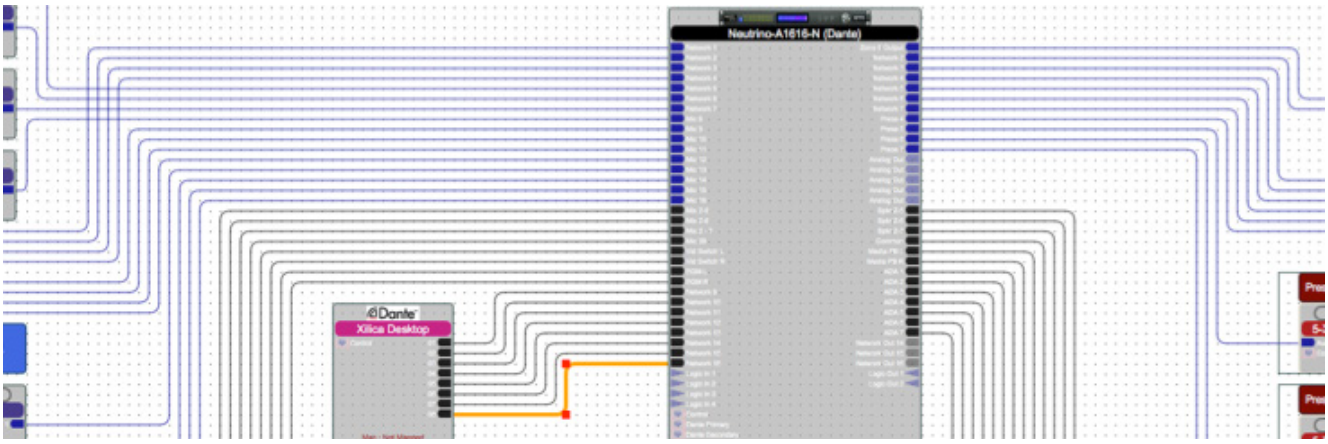
The 'Dante Enabled Device' must be properly named and the channels must be named.
Naming is done in the Component Properties pane, the same as Naming any other Processing Block.



To name the Channel,
select the wire and change the name in the Object Property menu.
"Source Label" labels the wire input and 'Dest. Label' labels the wire destination.

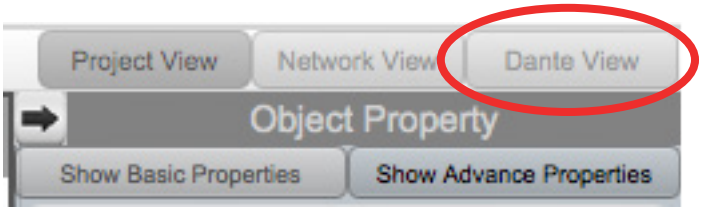


Notice the Identification of the hardware on the network, the channel names, and the connection points. Pay attention to the destination of your Dante devices. This is where most connection mistakes are made due to confusion in signal flow.

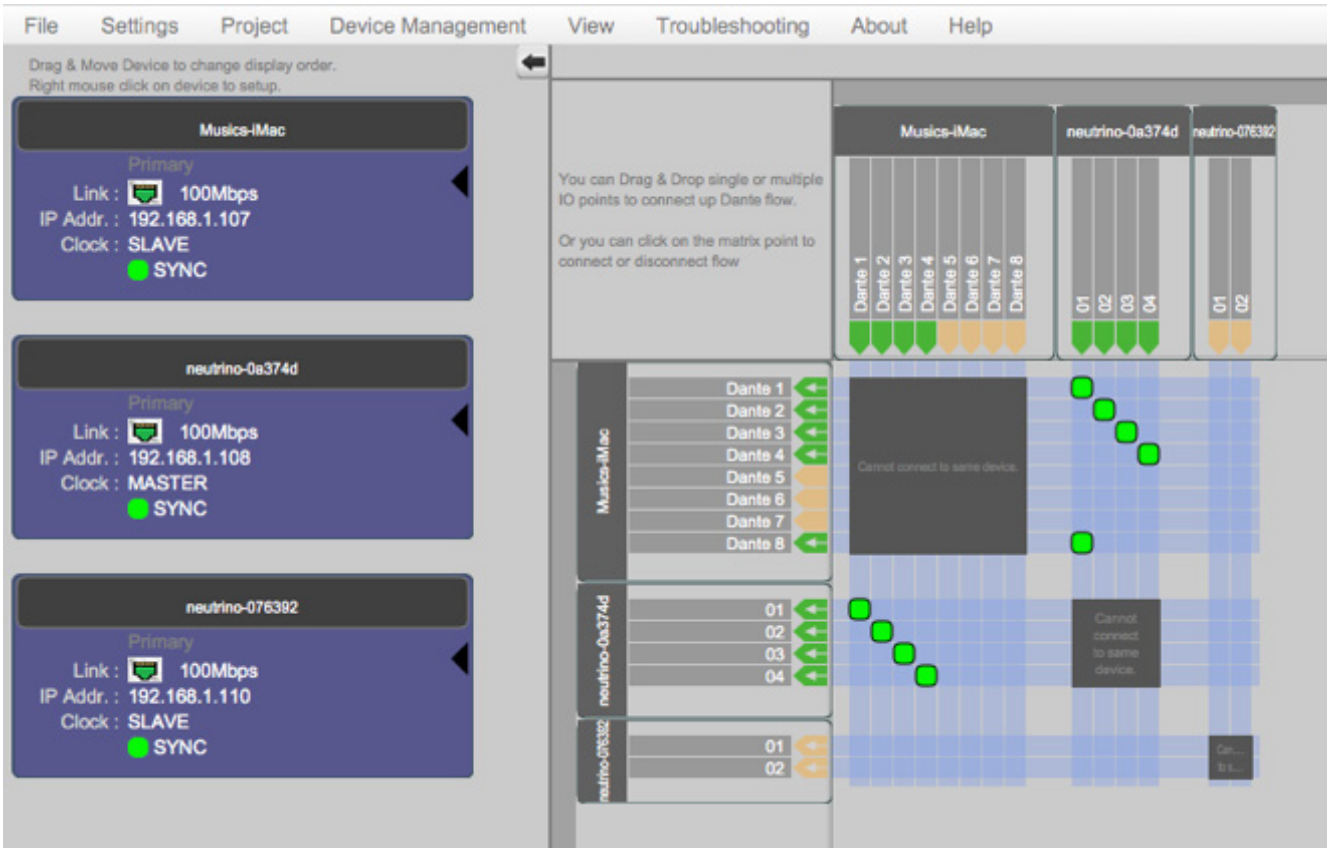


Dante view

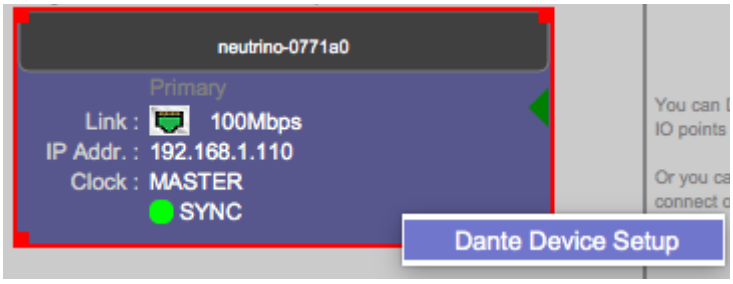
At the top right of the software, select 'Dante view'.



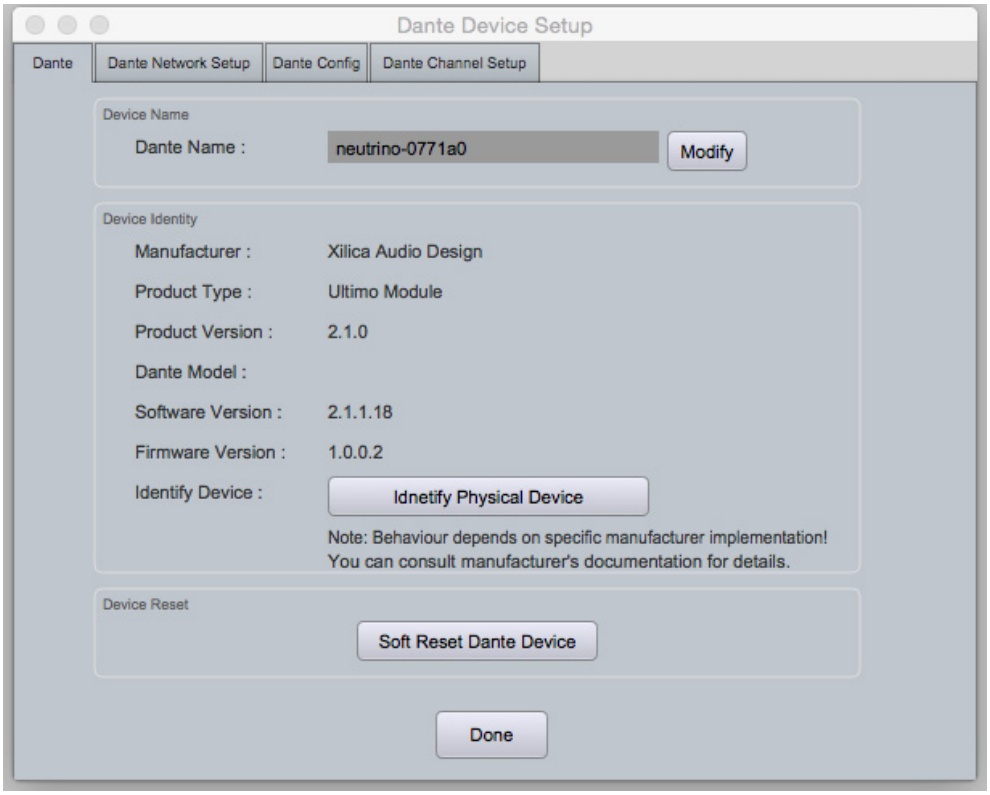
Dante view is where you can configure and view connected Dante devices.



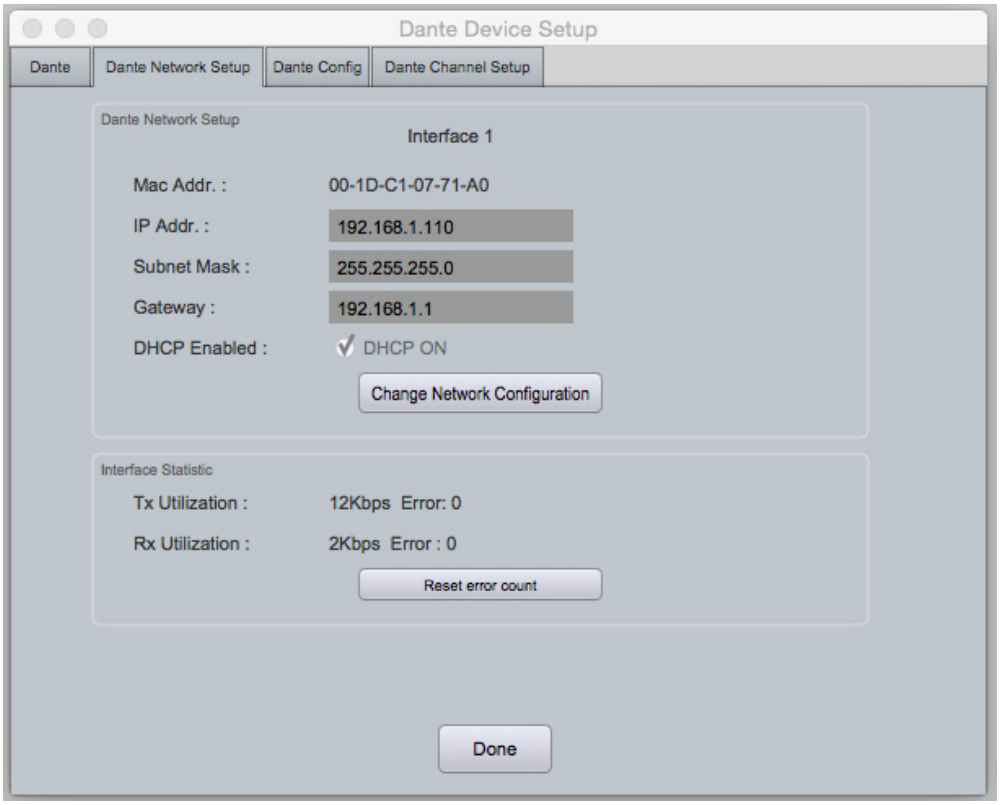
Connected Dante devices are listed on the left. Click and drag devices to rearrange the order of devices.
To view device settings, right click the desired device block, and select 'Dante Device Set up'.



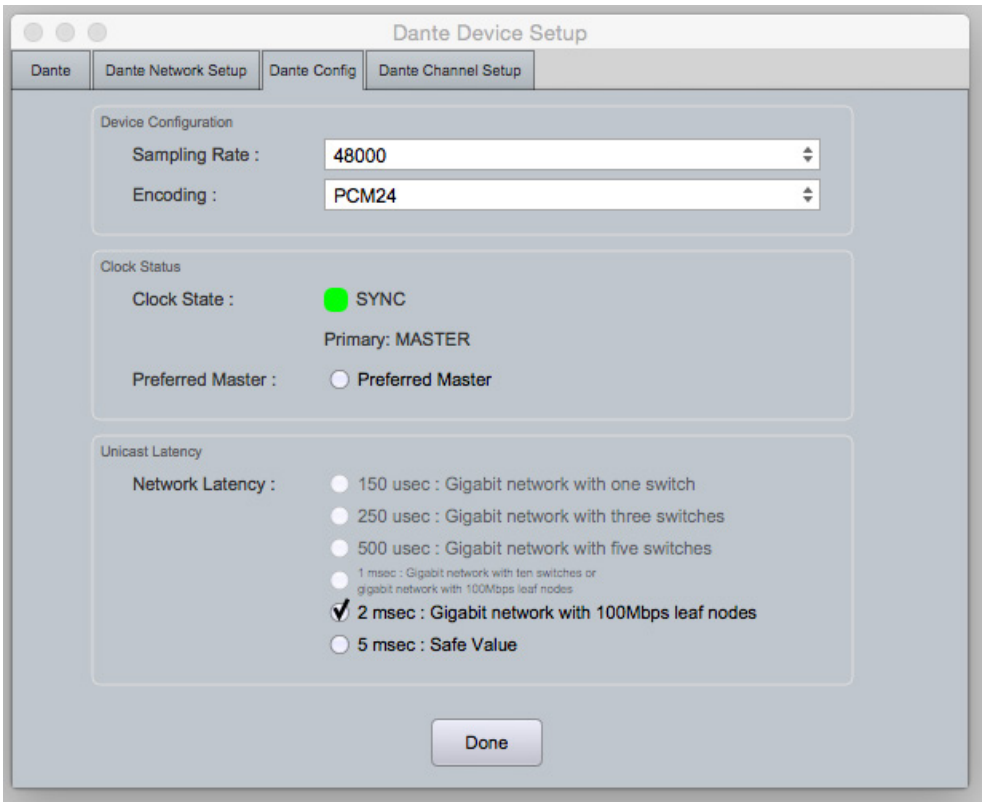
The first Dante tab displays product model information. You may also rename devices.



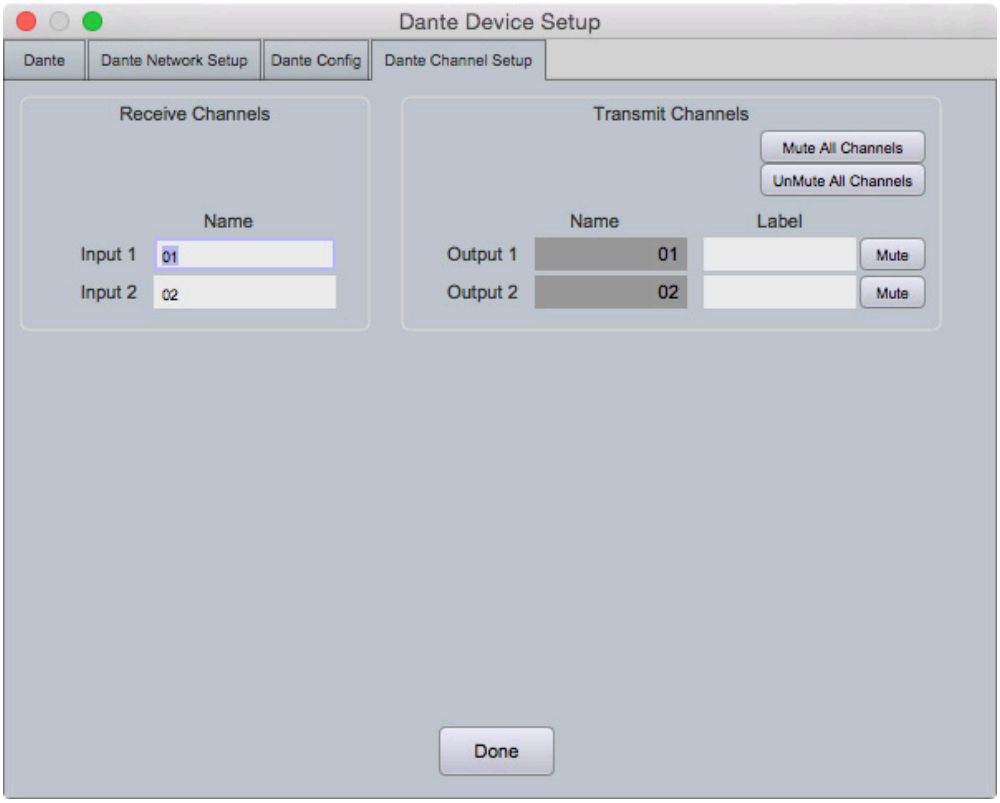
The next Dante Network Setup tab displays your Dante device network information.

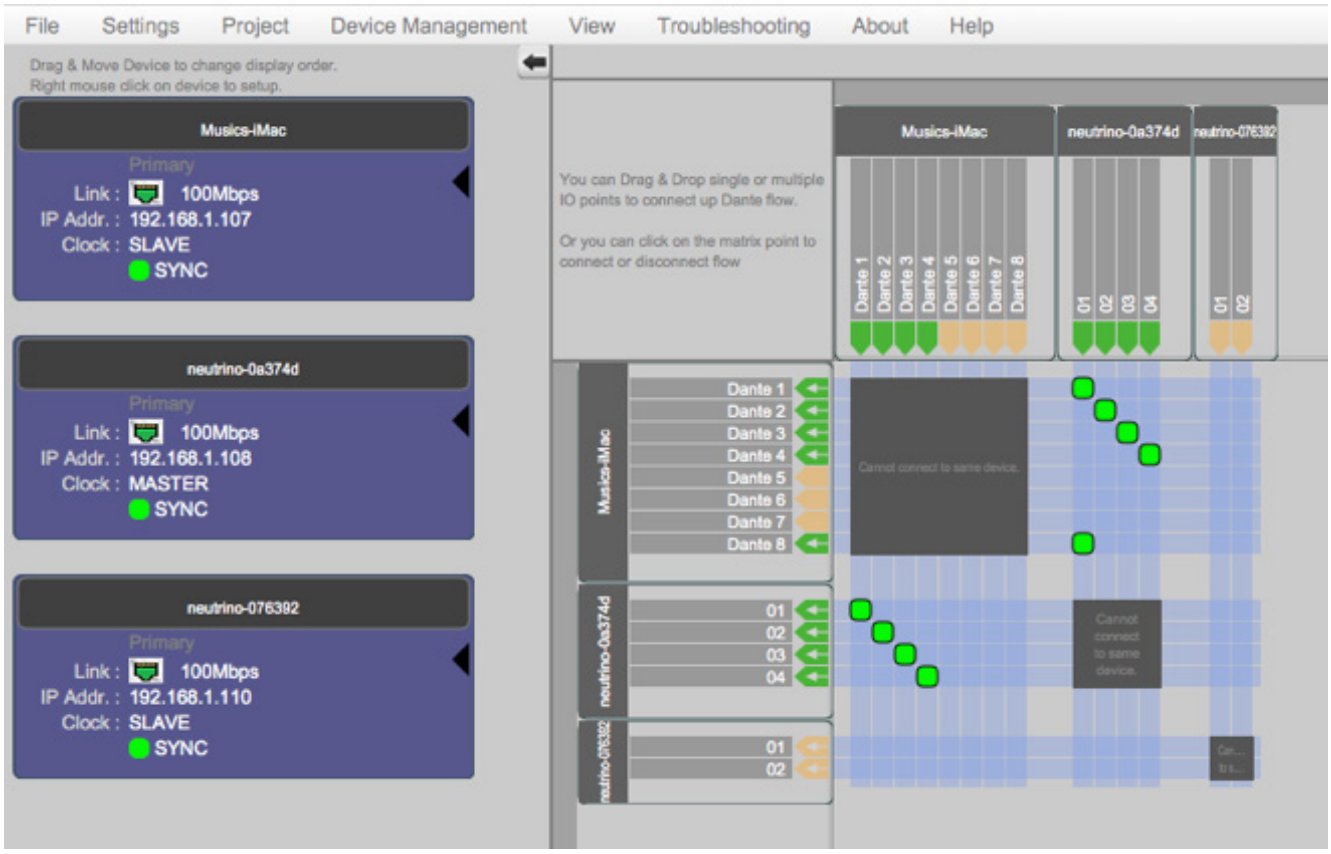


Dante Config tab allows you to configure your Dante device.



Dante Channel set up allows you to rename input and output channels. You may also mute channels. Once done reviewing you device settings, click 'Done'.





The diagram on the right is where you may route your Dante enabled devices.
Drag & drop single or multiple I/O points to connect up Dante flow. Or you can click on the matrix point to connect or disconnect flow.

GPIO Guide

What is GPIO?

The term 'GPIO' refers to the sequence of operations and conditioning required for a specific action to be performed. Within Xilica Designer, this sequence is to be carried out on a physical electronic signal in order to perform programmed actions such as triggering presets or muting an audio channel.

Where does this electronic signal originate?

The electronic signal in question (referred to as the GPIO input) is generated by creating a wired connection between the physical GPIO input ports (1 through 4, found on the back of the Neutrino/Uno DSP) and the corresponding ground pin (G).

Operations and conditions (Modules)

GPIO operations are programmed within Xilica Designer through the use of wiring and modules, much like that of an audio signal. Each function, operation, or condition has its own module block that can be inserted and wired into your design.

Features and Typical uses

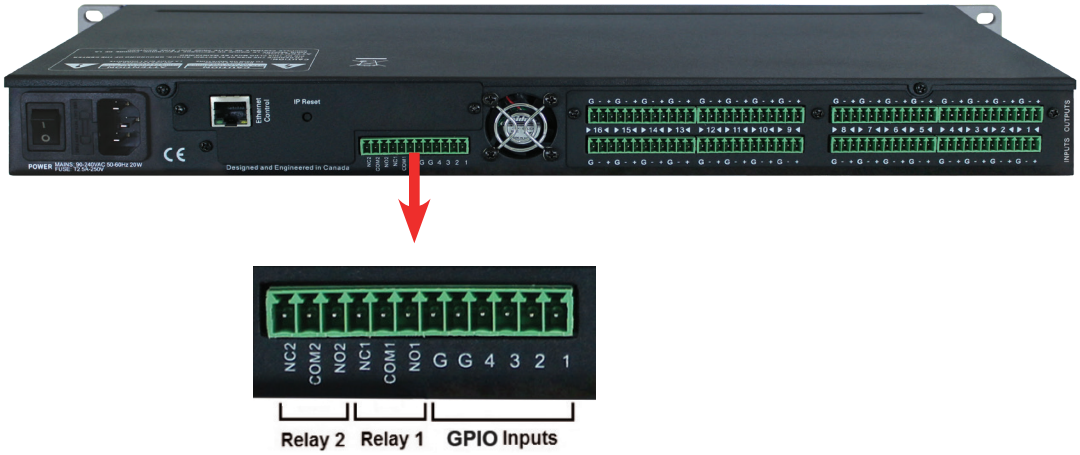
The GPIO Output module allows for two basic functions: Triggering the physical relays and Triggering preset recalls. This basic approach is very open ended in the fact that presets can perform any change of settings to the device that you wish, and the external relays can complete any possible electronic circuit you require. The physical output contacts do not provide any specific voltages, meaning that powered circuit of any voltage or power requirements will be compatible with you processor. Xilica Designer processors have physical GPIO I/O capabilities allowing for added functionality such as:

- Emergency Mute
- Contact Switch Input
- LED Display
- Preset Triggering
- GPIO Controlled Mute
- And more…

Solaro series GPIO cards

For Solaro series GPIO cards, the same GPIO card can act as logic (On/Off) or control/voltage (0-100%) input value. The same hardware card can perform both functionalities. It all depends on the design software and which card configuration you select. If you select the card as GPIO, it will have On/Off logic behaviour. But if you select it as GPIO control, it will have 0-100% behaviour.

Hardware

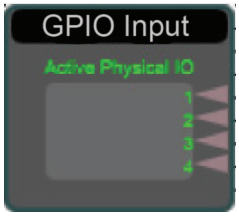


The GPIO I/O structure can be broken down into three basic groups:

1. GPIO Inputs
2. Output Relay 1
3. Output Relay 2

GPIO Inputs

There are six contact dedicated GPIO input signals. Four are input signal contacts and two are dedicated as ground.
GPIO input signals are momentary unless otherwise programmed in Xilica Designer.
Creating a physical connection between any input channel and Ground (G) will send a GPIO HIGH signal to the processor through the corresponding channel.

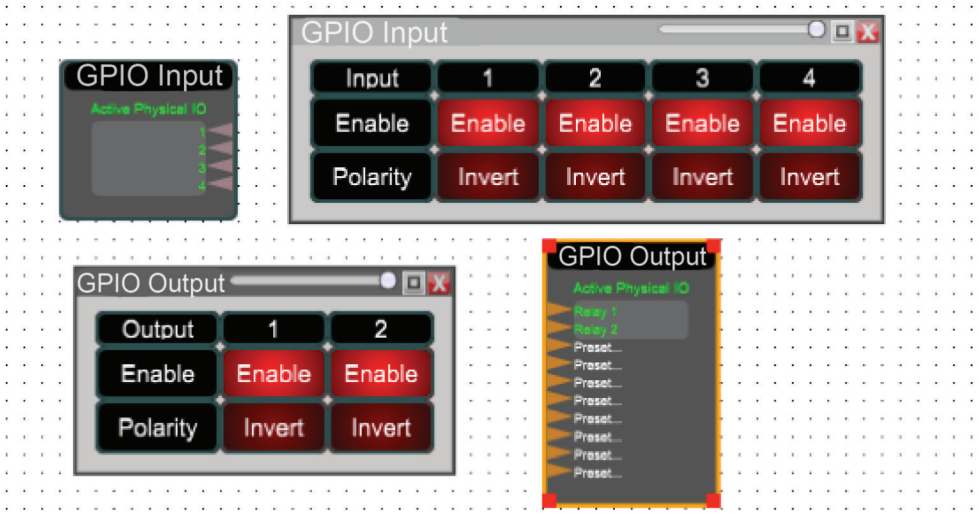


Relays 1&2

Each relay has a normally open (NO) contact, a normally closed (NC) contact, and a common.
When a HIGH signal reaches the Relay contact in the GPIO Output module, both the NO and NC physical contacts will invert, either closing the external circuit (NO) or opening it (NC).

Design introduction

Designing GPIO circuits with a Neutrino DSP is very similar to that of designing an audio schematic. The initial blank palette features both an input and output module, and requires the virtual wiring of GPIO modules to complete the I/O circuit.

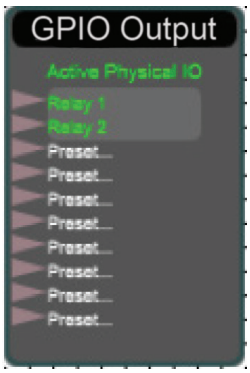


GPIO input

The GPIO input is the first point that the physical GPIO signals enter the DSP schematic. This module has 4 nodes labelled 1-4 by default; any and all channels can be either inverted or disabled. These nodes represent the physical GPIO input contacts on the hardware.

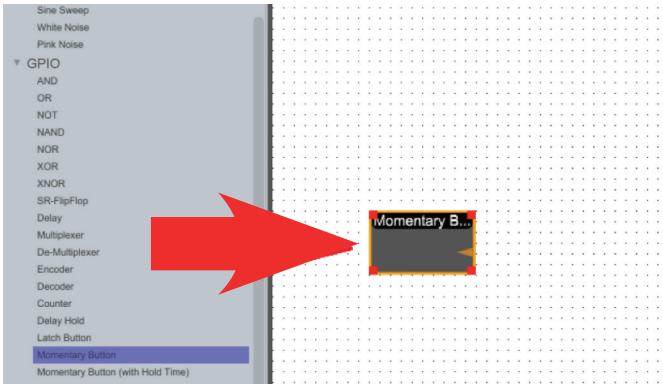
GPIO output

The GPIO output module is the final destination of the GPIO signal from where you can decide to either trigger the internal physical relays, or trigger presets 1 through 8. Each relay has a normally open (NO) contact, a normally closed (NC) contact. When a HIGH signal reaches the Relay contact in the GPIO Output Module, both the NO and NC physical contacts will invert, either closing the external circuit (NO) or opening it (NC). Open the GPIO Output module to find that the output relays can be enabled/disabled or inverted if need be. The Preset Trigger feature is simple in functionality. Send a HIGH signal to the desired channel in order to recall the corresponding preset.



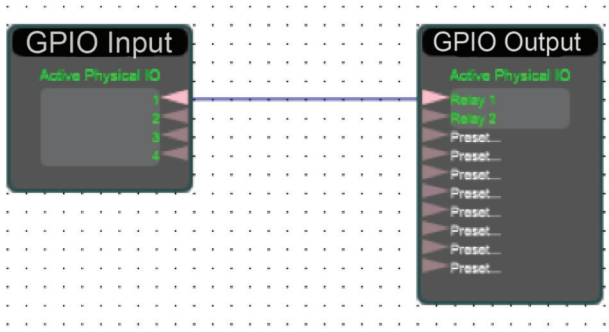
Design basics

To add any component that suits your design, simply click and drag the desired module from the Component Library to the work area.

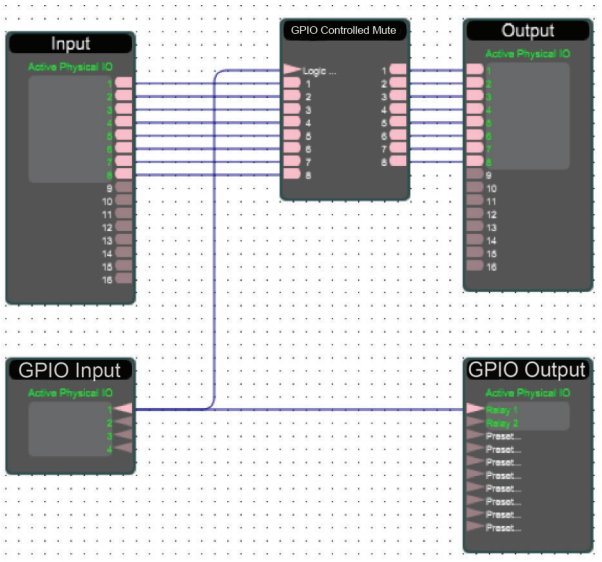


Wiring

Connect modules together by creating a wire. Simply click and drag from the output node of one module to an input node.

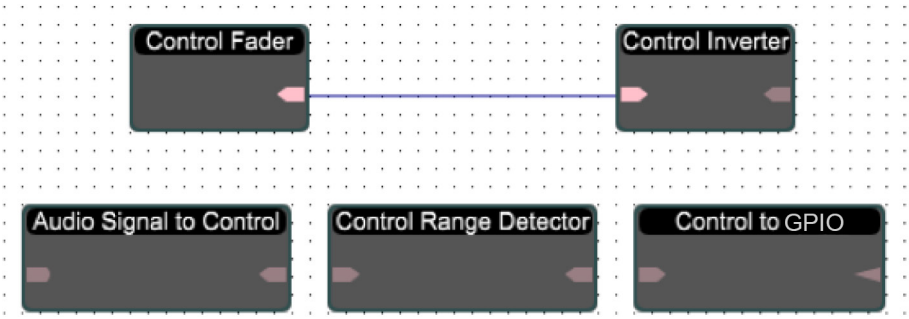


Depending on the modules being wired, you can link multiple wires to single nodes. This will send the signal through both wires simultaneously as a parallel connection.

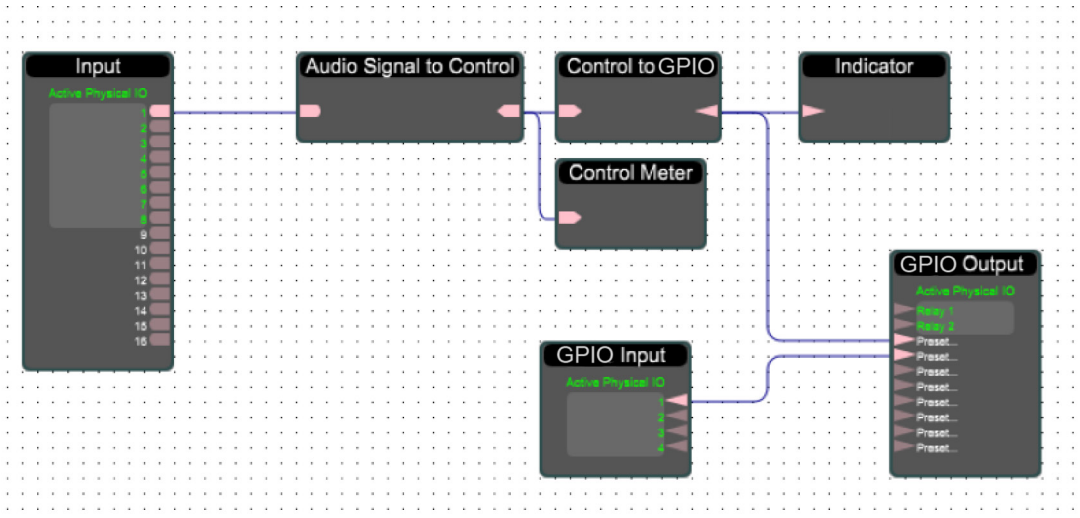


In the example above, notice that the input signal and Relay 1 is being routed to the GPIO Controlled Mute. One practical use for this configuration is having an external LED light up when the emergency mute button is active.

Control objects



Like GPIO, control signals are data signals that are used as a utility for functionality, however control values are represented as a range. This range can be manipulated and detected via various threshold tools to allow for intricate control over various applications.
Some modules are dedicated to the conversion of signals, be it from Audio to Control, Control to GPIO, or GPIO to Control. These tools can be used in conjunction with each other for such purposes as using a line level signal to trigger a preset, which in turn could turn on an emergency mute, for example.



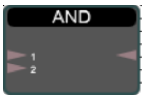
In the above example, the audio signal is converted to Control, and the Control is then converted to GPIO.

Uno series

GPIO designs are embedded in Uno design Apps.
To browse the Uno app library, visit the Xilica website. (www.xilica.com)
For custom Uno apps, please contact Xilica's technical support team at: support@xilica.com.

GPIO Modules

AND



All inputs GPIO high for GPIO high output.

OR

Any input(s) GPIO high for GPIO high output.

NOT

Output is opposite GPIO signal of input.

NAND

GPIO high output unless all inputs are GPIO high.

NOR

GPIO high output if all inputs are GPIO low.

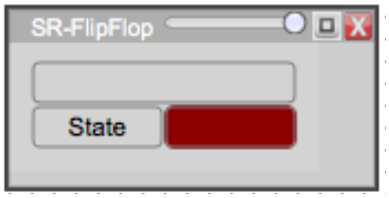
XOR

Output GPIO low If all inputs are the same output is GPIO low, otherwise GPIO high.

XNOR

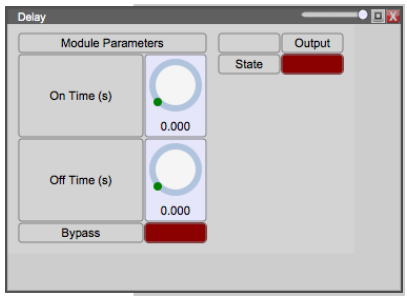
If all inputs are the same output is GPIO high.

SR-Flip Flop



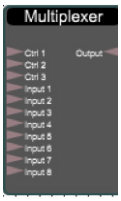
For each trigger (low-to-high signal) at the Trigger pin, Output Q will switch GPIO values. The module displays the current State of the Q pin. Output /Q is always the opposite of Output Q. While the Set pin is GPIO high Output Q will be GPIO high and While the Reset I/O is GPIO high Output Q will be GPIO low.

Delay



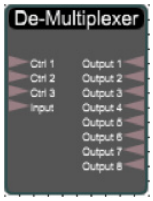
When the input is GPIO high for at least the On Time (0 to 60s, 1ms steps) the output will be GPIO high for the on time and GPIO low for the Off Time (0 to 60s, 1ms steps). Bypass and output State indicator features present.

Multiplexer



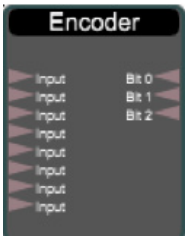
Route one of eight inputs to the output using the three Ctrl (Control) I/O's. The module shows the channel selected.

De-Multiplexer



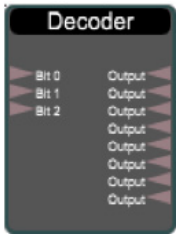
Route the input to one of the eight outputs using the three Ctrl (Control) I/O's. The
module shows the channel selected.

Encoder



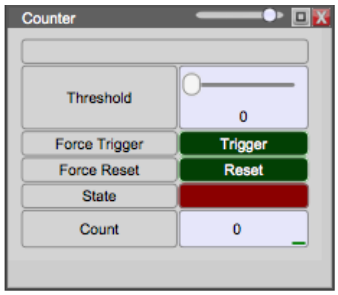
Convert eight GPIO inputs to three binary inputs. The module shows encoded Channels.

Decoder



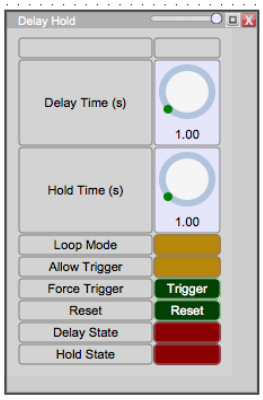
Convert three binary inputs to eight GPIO inputs. The module shows decoded Channels.

Counter



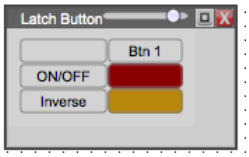
Each trigger (low-to-high signal) at the Trigger I/O adds one to the Count (0 to 99999999); at Threshold (0 to 99999999) the output state becomes high. State indicates the current output condition. Use Force Trigger to add one to the count total. Force Reset and a trigger (low-to-high signal) at the Reset I/O will reset the Count value to zero.

Delay hold



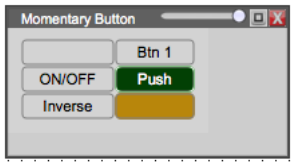
When the input is triggered (low-to-high signal) or the Force Trigger button is used the Delay I/O will activate for the Delay Time (0.01 to 5000s) as can be seen by the Delay State. After the Delay Time, The Hold I/O will activate for the Hold Time (0.01 to 5000s) depicted by the Hold State. Loop Mode will continuously cycle the Delay and Hold for one input trigger (low-to-high signal). Use the Reset button to reset and Force trigger to activate the module. Allow Trigger sets whether a new trigger can occur during an event (Delay and Hold active cycle).

Latch button



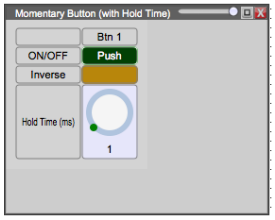
Use the Latch button to output a GPIO high or GPIO low signal with ON/OFF. Reverse the GPIO output with the Inverse button.

Momentary button



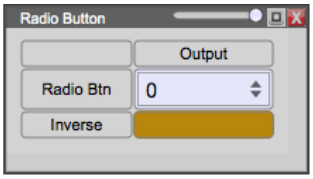
Creates a trigger (low-to-high, or high-to-low signal) event at the output with the ON/OFF key press. Reverse the GPIO output with the inverse button.

Momentary button with hold time



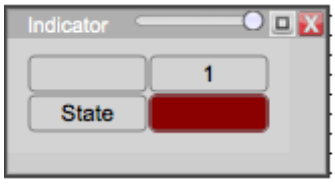
Creates a trigger (low-to-high, or high-to-low signal) event at the output with the ON/OFF key push. Hold down push button to enable ON/OFF function. Reverse the GPIO output with the inverse button.

Radio button



Output a GPIO high from the output selected with Radio Button, keeping all other outputs are GPIO low. Reverse all GPIO outputs on the module with the Inverse button.

Indicator



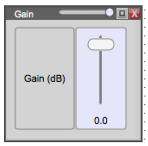
Displays the current State of each input.

DSP Module Descriptions

In project view, double click a processor/device in your work area to open the device. In the Component Library menu on the left, there are many DSP modules available to use in your project design.

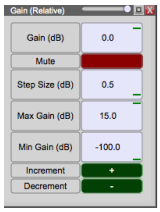
Signal

Gain:



Signal level control (-100dB to +15, 0.1dB steps)

Gain (Relative):

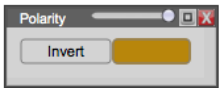


Signal level control that adds or subtracts a set amount from a signal. Includes Mute, step control and min/max gain

Mute:

Nullifies the signal in order to eliminate audio

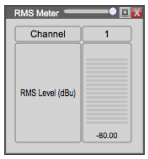
Polarity:



Used to reverse the phase of the signal

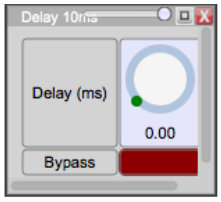
Meters

RMS meter:



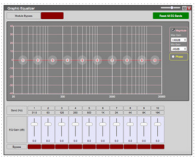
Root Mean Square (RMS) signal meter displayed in bar graph format
RMS level (-80dB to +40dB, 0.1 steps)

Delay 10ms: (0 to 10ms, 0.01ms steps)
Delay 50ms: (0 to 50ms, 0.05ms steps)
Delay 100ms: (0 to 100ms, 0.1ms steps)
Delay 500ms: (0 to 500ms, 0.5ms steps)
Delay 1s: (0 to 1s, 1ms steps)
Delay 2s: (0 to 2s, 2ms steps)



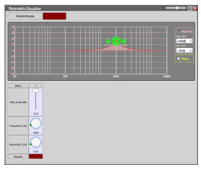
Equalizers

Graphic EQ:



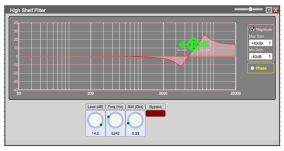
Increase/decrease the EQ Level (-30dB to +15dB, 0.1dB steps) within 10, 15 or 31 bands across the frequency range. Set the number of bands in Component Properties. Bypass feature included for each band

Parametric EQ:



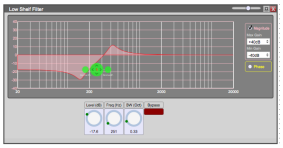
Increase/decrease the EQ Level (-30dB to +15dB, 0.1dB steps) amount at the target Frequency (20Hz to 20kHz, 1 Hz steps) with roll off controlled by the Bandwidth (0.02 to 4oct, 0.01oct steps) setting. Up to 8 bands per channel, set the number of bands in Component Properties. Includes Bypass feature for each band

High Shelf Filter:



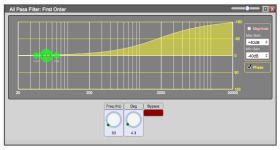
Increase/decrease frequencies above target Frequency (20Hz to 20kHz, 1 Hz steps) by the EQ Level (-30 to +15 dB, 0.1dB steps). Roll off controlled by the Bandwidth (0.01 to 4.00oct, 0.01 steps) setting. Bypass feature included

Low Shelf Filter: :



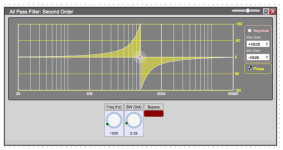
Increase/decrease frequencies below target Frequency (20Hz to 20kHz, 1 Hz steps) by the EQ Level (-30 to +15 dB, 0.1dB steps). Roll off controlled by the Bandwidth (0.01 to 4.00oct, 0.01 steps) setting. Bypass feature included

All Pass Filter 1st Order:



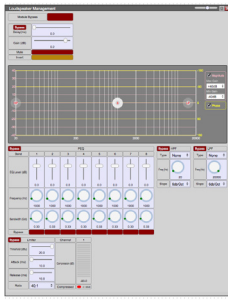
Module parameters can be saved for a schematic design into one of 70 preset slots. Presets can only be created in design mode and can be saved at three different tiers: Device, Module, and Parameter

All Pass Filter 2nd Order:



Unity gain filter, 180 deg. phase shift at target Frequency (20Hz-20kHz, 1 Hz steps) with roll off controlled by the Bandwidth (0.01to 4.00oct, 0.01 steps). Bypass feature included

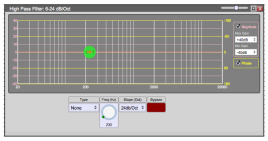
Loudspeaker Management:



Combines all the normally used modules for Loudspeaker Management in one convenient module. Select as many inputs/outputs as needed (In the Component Properties Menu) and alter their parameters in one easy place. Includes module bypass. Modules include: Delay, HPF, LPF, PEQ, Limiter, Gain, Mute and Polarity

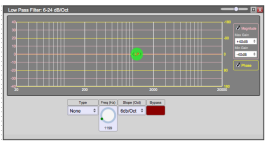
Filters

High Pass Filter 6-24dB/Oct:



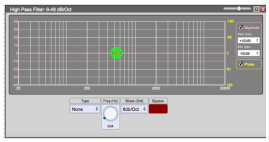
Attenuates frequencies below target Frequency (20Hz to 20kHz, 1 Hz steps and with filter Type (Butterworth, Linkwitz-Riley and Bessel) and Slope (6 to 24dB/oct, 6dB/oct steps). Bypass feature included

Low Pass Filter 6-24dB/Oct:



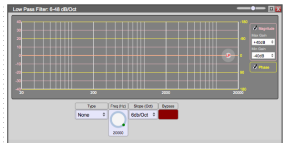
Attenuates frequencies above target Frequency (20Hz to 20kHz, 1 Hz steps) with filter Type (Butterworth, Linkwitz-Riley and Bessel) and Slope (6 to 24dB/oct, 6dB/oct steps). Bypass feature included

High Pass Filter 6-48dB/Oct:



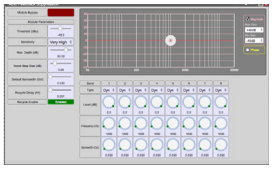
Attenuates frequencies below targetFrequency (20Hz to 20kHz, 1 Hz steps and with filter Type (Butterworth, Linkwitz-Riley and Bessel) and Slope (6 to 48dB/oct, 6dB/oct steps.) Bypass feature included

Low Pass Filter 6-48dB/Oct:



Attenuates frequencies above target Frequency (20Hz to 20kHz, 1 Hz steps) with filter Type (Butterworth, Linkwitz-Riley and Bessel) and Slope (6 to 48dB/oct, 6dB/oct steps). Bypass feature included

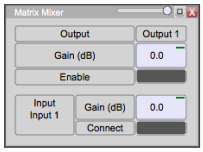
Auto Feedback Suppression:



Creates a notch filter around loud frequencies to eliminate Feedback. Includes eight bands that automatically adjust to create the filters as necessary
Threshold (-100dB to 10 dB, 0.1dB steps) is the minimum level before suppression will occur.
Sensitivity (very low, low, medium, high very high) the higher the level of the offending tone needs to be before suppression
Maximum Depth (0dB to 40 dB, 0.01dB steps) sets the largest amount of attenuation applied to the offending tone
Notch Step Size (0.5dB to 3 dB, 0.01dB steps) sets the rate Maximum Depth is reached.
Recycle Delay (0.1hr to 100hr dB, 0.01dB steps) is the minimum length of time a notch filter is held before the attenuation is reduced by the Notch Step Size. Use Recycle Enable to activate Recycle Delay
Level, Frequency and Bandwidth for each band are displayed in the module. Use Type to switch between Dynamic and Fixed filter types

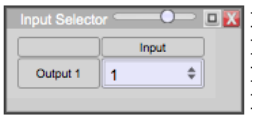
Mixer

Matrix Mixer:



Analog combining Input to Output (-100dB to 0dB, 0.01dB steps)
Includes channels with connect (On/Off), Master Gain (-100dB to 0dB, 0.01dB steps) and Mute control

Input selector

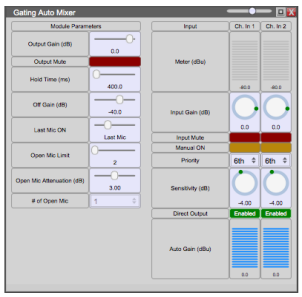


Allows up to 64 channels to be output to 64 output channels. This allows for pull-down channel assignment in XTouch or as an alternative to the Matrix Mixer when level adjustment is not required

Output selector

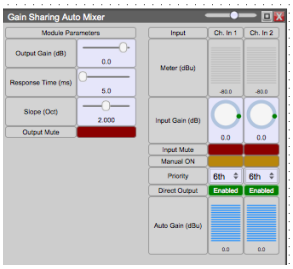
Allows routing from 64 inputs to 64 outputs. This allows for pull-down channel assignment in XTouch or as an alternative to the Matrix Mixer when level adjustment is not required

Gating Auto Mixer:



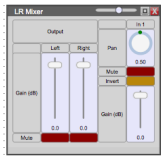
Typically used in a conference setting where multiple MICs are in use but only one (or a few) should be on at any time. This module turns on MICs based on their input signal as compared to the level of the other MICs. Once a channel is activated it typically stays on until another signal is larger. Includes: Mute and Gain (-100 to 16 dB, 0.01dB steps), RMS Meter (-80 to 40 dB, 0.1 dB steps) and Auto Gain meter (-100 to 0 dB, 0.1 dB steps)
The Priority parameter ranges from 0 to 10 (with 0 being the highest and 10 the lowest). An input channel with a higher priority needs less level to be gated on by increasing the input by 2 dB/ priority step
The Sensitivity parameter (-16 to 12dB in 0.01dB steps) determines at what level the channel is gated on compared to the automatically determined threshold
(Ex. If the sensitivity is set to -1dB, an input signal above -1dB of the threshold will be gated on.)
The output includes Gain (-100 to 16 dB, 0.01 dB steps) and Mute control
The time for a MIC to be gated off is set by the Hold Time (50 to 6000 ms, 0.001ms steps).
The Off Gain (-90 to -10 dB, 0.1dB steps) determines the gain for an off channel
Last MIC On mode determines which MIC stays on when no one is talking. (This can be set to none, last MIC used or a specific input)
The maximum number of open (on) MICs is set by the # of Open Mic. The Open MIC Attenuation (0 to 6dB, 0.01 dB steps) attenuates the output by the set amount whenever the number of open MICs is doubled

Gain Sharing Auto Mixer:



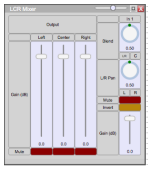
Allows the automatic mixing of input channels to one output channel based on input signal levels. The louder the particular input channel, the louder it will be at the output channel. Each input includes Mute, Gain (-100 to 16 dB, 0.01dB steps), RMS Meter (-80 to 40 dB, 0.1 dB steps) and Auto Gain meter (-100 to 0 dB, 0.1 dB steps) (Shows calculated gain for each channel)
Manual On changes the gain from automatic (off) to fixed (on). While Manual On, the gain for the channel is fixed and will not affect other channels
The Priority parameter ranges from 0 to 10 (with 0 being the highest and 10 the lowest). An input channel with a higher priority will have a larger gain applied dependent on the Slope value and difference in priority between channels
Outputs include Gain (-100 to 16 dB, 0.01 dB steps) and Mute control
The time for gain to occur is set by the Response Time (0.2 to 2000 ms, 0.1 ms steps)
The Slope (1 to 3, 0.001 steps) determines the gain difference between priorities. With a Slope of 1 there is no gain. With a Slope of 2, 2dB per point of priority difference between channels. A Slope of 3 creates a 4dB gain per point of priority

LR Mixer:



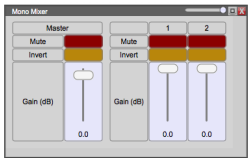
Mixes inputs before splitting into Left and Right outputs based on the Right Ratio for each channel. Includes Input / Output Gain, (-100dB to 15dB, 0.1dB steps), Mute control and input Polarity control

LCR Mixer:



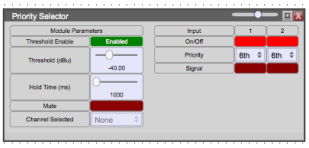
Mixes inputs before splitting into Left, Right and Centre outputs based on the Right Ratio. Includes Input / Output Gain, (-100dB to 15dB, 0.1dB steps), Mute control and input Polarity control

Mono Mixer:



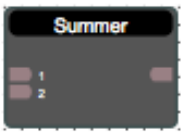
Signal level control that adds or subtracts a set amount from a signal. Includes Mute, step control and min/max gain

Priority Selector:



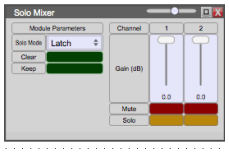
Multiple channel input, single output. Output is the input On channel with highest Priority (First-Tenth) channel above the Threshold (-60dB to 0dB) value when Threshold Enable is activated. Includes output Mute, Hold Time (10ms to 30s), signal (above threshold) and Channel Selection indicators

Summer:



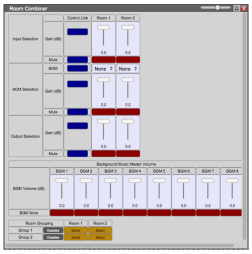
Summed multiple input channel audio is released as a single channel output

Solo Mixer:



Allows quick soloing of up to 64 channels that are passing through the solo mixer. Has both latching and exclusive modes

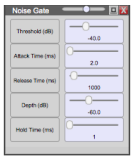
Room Combiner:



Allows for easy reassignment of wall controls to coincide with movable wall rooms creating new zone controls as configured

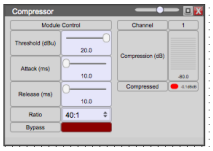
Dynamics

Noise gate:



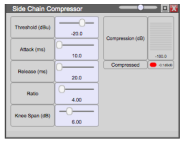
Allows for muting of low-level input signals

Compressor:



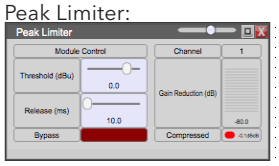
Reduces the signal level above the Threshold (-60 to 20dB, 0.5 dB steps) value by the Ratio (1:1, 40:1, 1:1 steps) set. Attack (0.1 to 2000ms, 0.1ms steps) adjusts the time to reduce the signal once the signal has been exceeded. Bypass feature and RMS meter included

Sidechain Compressor:



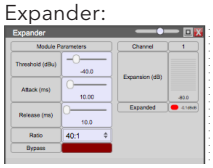
Compressor with side chaining and knee control. This allows multiple compressors to be tied together with a reference side chained line. This allows the multiple compressors to be controlled from a single source for the control of dynamics rather than just the input. This prevents one channel from being greatly compressed and other channels to not be. In addition, the “knee span” will adjust the rate that the ratio is applied after passing over the threshold. IE, if set to a ratio of 40:1, with a knee span of 5db, it will hit the threshold, then take 5db before it hits the full ratio

Peak Limiter:



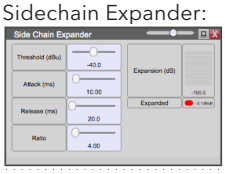
Reduces the signal level above the Threshold (-60 to 20dB, 0.5 dB steps) value by 40:1. Bypass feature and RMS meter included

Expander:



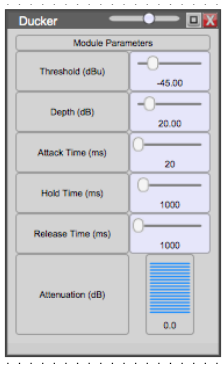
Reduces the signal level below the Threshold (-60 to 20dB, 0.1 dB steps) value by the Ratio (1:1, 40:1, 1:1 steps) set. Attack (0.1 to 100ms, 0.01ms steps) adjusts the time to reduce the signal once the signal has dropped below the threshold. Once the signal has risen above the threshold the Release (0ms to 10s, 1ms) time is the time to stop compressing the signal. Bypass feature and RMS meter included

Sidechain Expander:



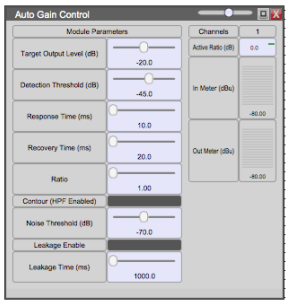
Expander with the addition of Side Chaining similar to Sidechain compressor

Ducker:



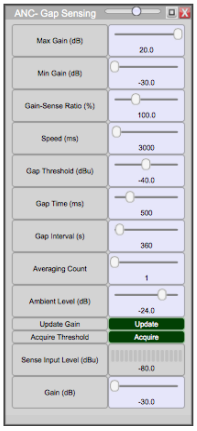
Attenuates channel 1 input by the Depth (0dB to 100dB, 0.01 dB steps) value when channel 2 surpasses the Threshold (-60dB to 0dB, 0.01 dB steps) value
Attack (10 to 500ms, 1ms steps) adjusts the time to reduce the signal once the signal has been exceeded. Once the signal in channel 2 has fallen below the threshold the Release (10ms to 60s, 1ms steps) time is the time to stop attenuating the signal. The Hold Time (10ms to 10s, 1 ms steps) sets the period before the signal is released after channel 2 drops below threshold. Attenuation is shown with an RMS meter

Automatic Gain Control:



Keeps the volume at a set level. When the input is below a threshold, it will amplify to bring the level up to the Target Output Level, and when over will reduce the gain to bring it back below the Target output Level.
Target Output Level (-40 to 0dB)
Detection Threshold (-80 to -20 dB) is the point at which the AGC will start to raise the gain of the signal.
Response Time (1ms to 40,000ms) is the period of time before the AGC begins to act when the level is over or under the Target output Level; Similar to expansion/compression
Recovery Time (1ms to 100,000ms) is the amount of time after the Target
Output level is below the set level before the AGC begins to increase the gain
Ratio (1:1 to 5:1) is how much the change can increase/decrease the gain between samples.
Contour (HPF On/Off) allows higher frequencies to pass regardless of level. All AGC functions act on the lower frequency components
Noise Threshold (-100 to -40Db) is where the noise floor can be set
Leakage Enable (On/Off) enables leakage on the ALC so that short-term instances of over/under the Target Output Level are ignored for this amount of time
Leakage Time (100ms to 100,000ms) is how long the leakage is allowed before the gain starts to compensate

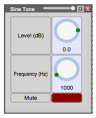
ANC-Gap Sensing:



Used for speech & paging applications. The output level is adjusted automatically in response to variations in the ambient noise level. The ambient noise is measured in the output gaps, when no signal to the outputs is present, or drops to below a pre-defined threshold
Max Gain (-30 to +20db) is the limit that the ANC will raise the signal by
Min Gain (-30 to +20Db) is the lowest level that the signal will lower itself to
Speed (1-60 Seconds) is the amount of time the volume change occurs over
Gap Threshold (-60 to -20db) is the level that the module sees as the point where it will take a sample from the mic on the modules background input
Gap Time (1-2000ms) is the time sampled for the reference ambient noise level
Gap Interval (60-3600 Sec) is the period between the samples of background noise
Averaging Count (1-10) is how many samples are averaged to determine the background noise
Ambient Noise (-60 to -12dB) is the reference background noise level with no program material
Pressing the 'Update Gain' button causes the ANC to immediately update its gain prior to the next scheduled reading. Pressing the "Acquire Threshold" button with no program source input calibrates the normal ambient level

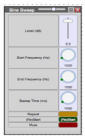
Generators

Sine Tone:



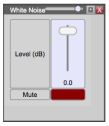
Creates a sine wave with Level (-100 to 20dB, 0.1 dB steps) and Frequency (20Hz to 20kHz, 1Hz steps). Includes mute control

Sine Sweep:



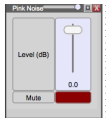
Creates a frequency changing sine wave with Level (-100 to 20dB, 0.1 dB steps), Start Frequency (20Hz to 20kHz, 1Hz steps), End Frequency (20Hz to20kHz, 1Hz steps), Sweep Time (1ms to 60s, 1ms steps) and Mute control. Repeat continuously cycles and starts the cycle from the beginning

White Noise:



Creates white noise with Level (-100 to 20dB, 0.1 dB steps) and Mute control

Pink Noise:

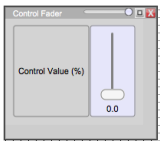


Creates pink noise with Level (-100 to 20dB, 0.1 dB steps) and Mute control

Control

Control fader:

A generic fader, which is used as a reference point for control modules

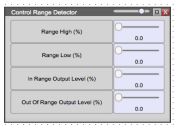


Control Inverter:

Inverts the control signal polarity

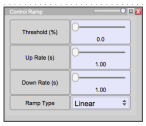
Control Range Detector:

This module outputs 2 values, one for 'in range' and the other for 'out of range'. The user edits both of these output values



Control ramp:

When a control signal goes over a set threshold, it will raise a control level at a set rate on the ouptut. When the input control signal is over the threshold, it will lower the control output by the set rate. Selectable options on this ramp are: audio, log and linear

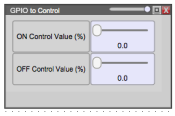


Control to GPIO:

User defined ON and OFF threshold values, which converts the control signal to GPIO format. The user determines the output polarity as well

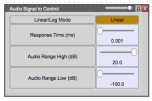
GPIO to Control:

User defined ON and OFF threshold values, which converts the control signal to GPIO format. The user determines the output polarity as well



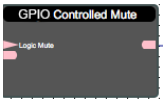
Audio Signal to Control:

Converts an audio signal that is within the High and Low limits (dB) to a control signal after duration of the response time. This conversion can be done in either a linear or logarithmic mode



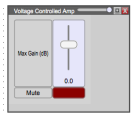
GPIO Controlled Mute:

Mutes the connected audio signal when a GPIO HIGH value is present on the control pin



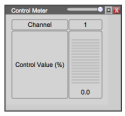
Voltage Controlled Amp:

Adjusts the output gain of an audio signal relative to the control value input



Control Meter:

A simple metering tool that displays the level of the control value input signal



Acoustic Echo Cancellation (AEC):

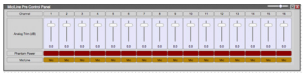
Acoustic Echo Cancellation used for VoIP/Conference telephony to reduce the echo effect when dealing with higher latency audio lines.

I/O information

Mic/Line Gain:

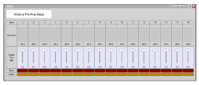
Input signal Mic/Line Gain (-40 to 25 dB in 0.01dB steps)

Phantom Power:



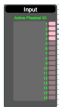
48V Phantom power on/off for each MIC input

Mic/Line:



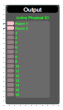
Mic/Line selection for each input. Provides hardware +40dB gain

Analog Input:



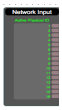
Analog input entering the DSP. Includes RMS level Meter (-80 to 40dB, 0.1dB steps), Gain (-100 to +15 dB, 0.1dB steps), Mute and Polarity

Analog Output:



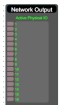
Analog output exiting the DSP. Includes RMS level Meter (-80 to 40dB, 0.1dB steps), Gain (-100 to +15 dB, 0.1dB steps), Mute and Polarity. Green indicates active physical I/O for your processor model

Network Input:



Network input entering the DSP. Includes RMS level Meter (-80 to 40dB, 0.1dB steps), Gain (-100 to +15 dB, 0.1dB steps), Mute and Polarity

Network Output:



Network output exiting the DSP. Includes RMS level Meter (-80 to 40dB, 0.1dB steps), Gain (-100 to +15 dB, 0.1dB steps), Mute and Polarity

GPIO Input:

GPIO input entering the DSP. Includes Enable and Inverse control

GPIO Output:

GPIO output exiting the DSP. Includes Enable and Inverse control

Glossary

Xilica Designer:	A software program created by Xilica used to design, configure and control devices
Hardware Module (Block):	The virtual representation of hardware within the Xilica Designer interface
Online Mode:	The state when the design file is successfully loaded into the designated hardware and ready for use. At this point the unit is "Live" and all changes can be heard in real time
Component Library:	This navigation panel within Xilica Designer has 2 modes. In Project View, it will show the available System Components and Hardware Modules that can be used in the project. Once a DSP module has been placed in the design and opened, the Component Library changes to show the available DSP Modules that can be placed within a Device (Not available for Uno)
Component Properties:	This navigation panel within Xilica Designer allows the user to make changes to the elements in the design. These elements can include Component Name, Position, Appearance, and Number of Inputs/Outputs etc. This panel will also track the Component Resource Usage of the DSP Device
Design Template:	A pre-configured design that can be used as template to create a new custom design. Note: Design Templates are not to be confused with the term APPs that are used in other DSPs. An APP is a fixed designed that cannot be reconfigured
Design App:	A pre-configured design that is loaded as a fixed audio schematic into your hardware device. This is the primary function of the Uno series, our App-based processors
Project View:	This page shows the current project design layout
Network View:	This page shows all of the DSP hardware and control devices that have been found on the Network
Dante View:	This page shows all Dante enabled devices
Control Page:	A window or several windows with control elements, i.e.: faders, buttons, meters, etc. used to create a custom user interface experience
Control Device:	Hardware control devices used to control system settings configured with control pages with the Xilica Designer software
Device Resource:	Indicates the amount of DSP resources used for the selected DSP hardware or individual processing blocks. This can be found in the Component Properties navigation panel. This can be viewed while Online or Offline for Hardware resources, and Offline for the processing blocks. There is a resource button on the top of the window of an open hardware block when in Online Mode
System Presets:	Preset values for audio routes, volumes, EQ, etc. can be created and stored in the system design for recall when needed i.e. Scenes
Dante	Digital Audio Transport System designed by Audinate to transport uncompressed, full bandwidth audio over standard gigabit network hardware
Digital Trim:	Fine adjustment of the input signal Post (after) the Analog Input stage
Analog Trim:	Fine Adjustment of the analog Mic/Line preamp (before) the A/D converter
Auto Wire:	The method used to wire Hardware and Processing Blocks without having to navigate a "point to point" wiring system. Wires can be adjusted and labeled if desired after blocks are connected
Wire label:	A way of identifying wiring for an audio system design. Can be done in the Component Properties when selecting a wire that has been brought into a design

XILICA

Customer Support

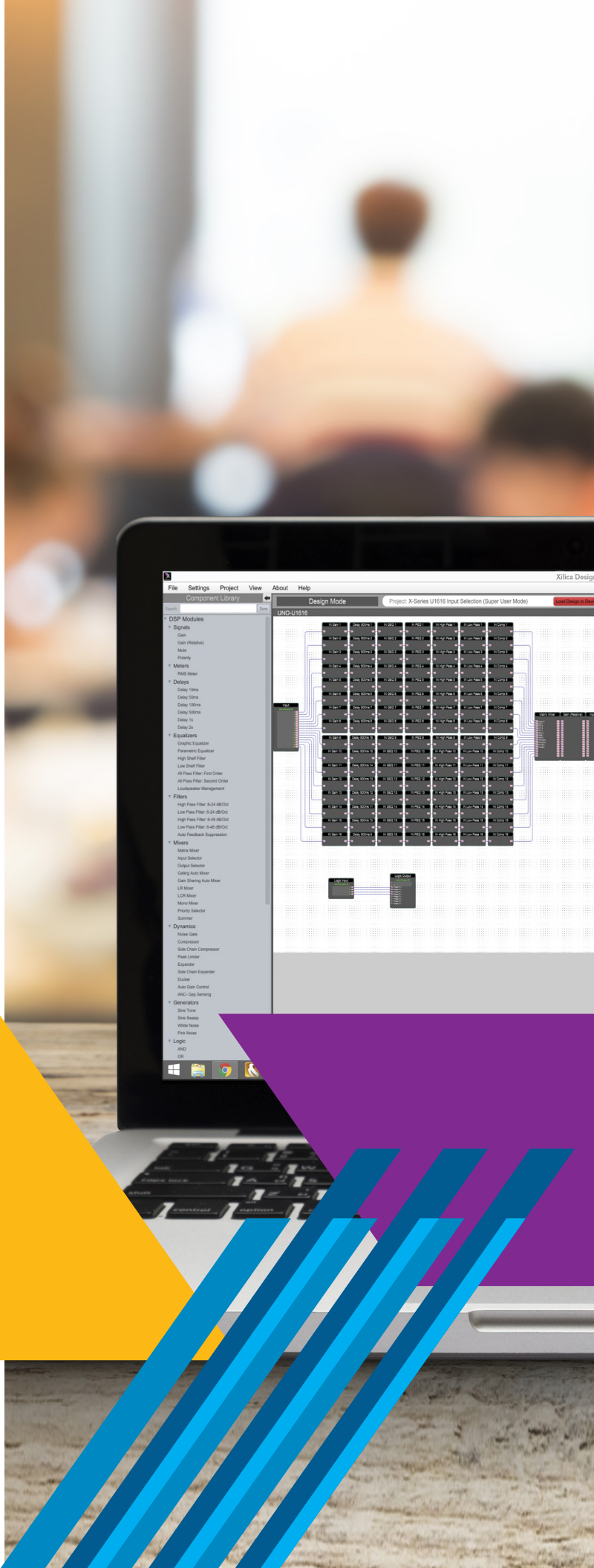
If you'd like to contact us regarding product support or technical designs, email support@xilica.com and we'll connect you with a solutions engineer Alternatively, if you'd like to speak to someone, you can call the following numbers for immediate assistance:

North America & Rest of world: +1 905-770-0055

Europe: +31 29940-1100

China & Hong Kong SAR: +852 2604-9382

www.xilica.com



Version: 2.0