Quick Start Guide for the HYPERSIM® Simulator

CONTENTS

1. INTRODUCTION

This document is intended to guide you in your first steps with HYPERSIM® from a Windows-based host PC. All OPAL-RT simulators delivered for HYPERSIM® use a modified Linux kernel optimized for real-time simulation.

1.1. Required material

- OPAL-RT simulator
- Windows-based host PC

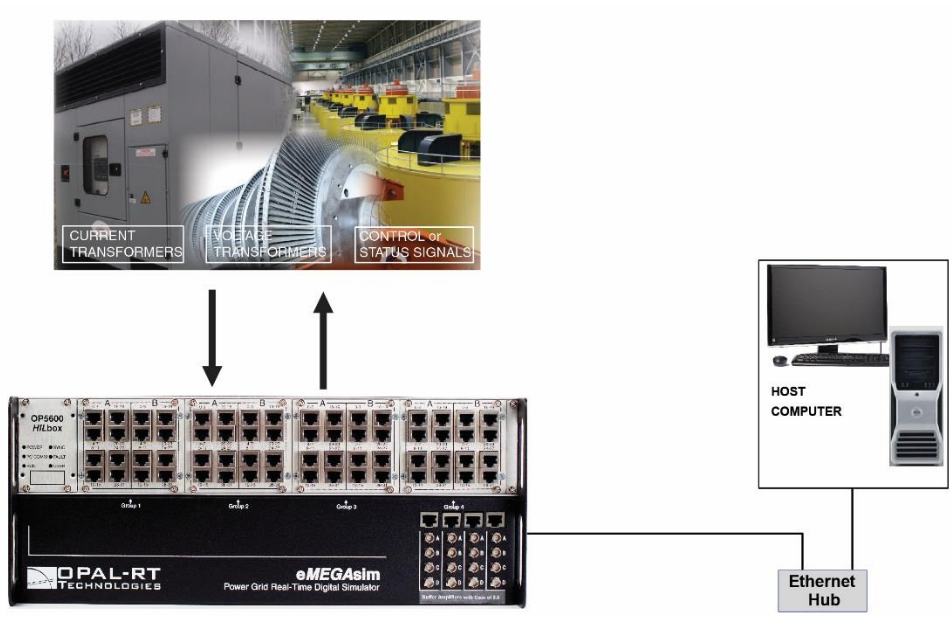


Figure 1: Typical hardware setup

2. FIRST STEPS WITH HYPERSIM®

See **Annex A** for a detailed description of the schematics interface.

2.1. CREATE A SCHEMATIC

1. The first step to running a simulation is to create a schematic. In **File > Options**, click **New Default Document.**

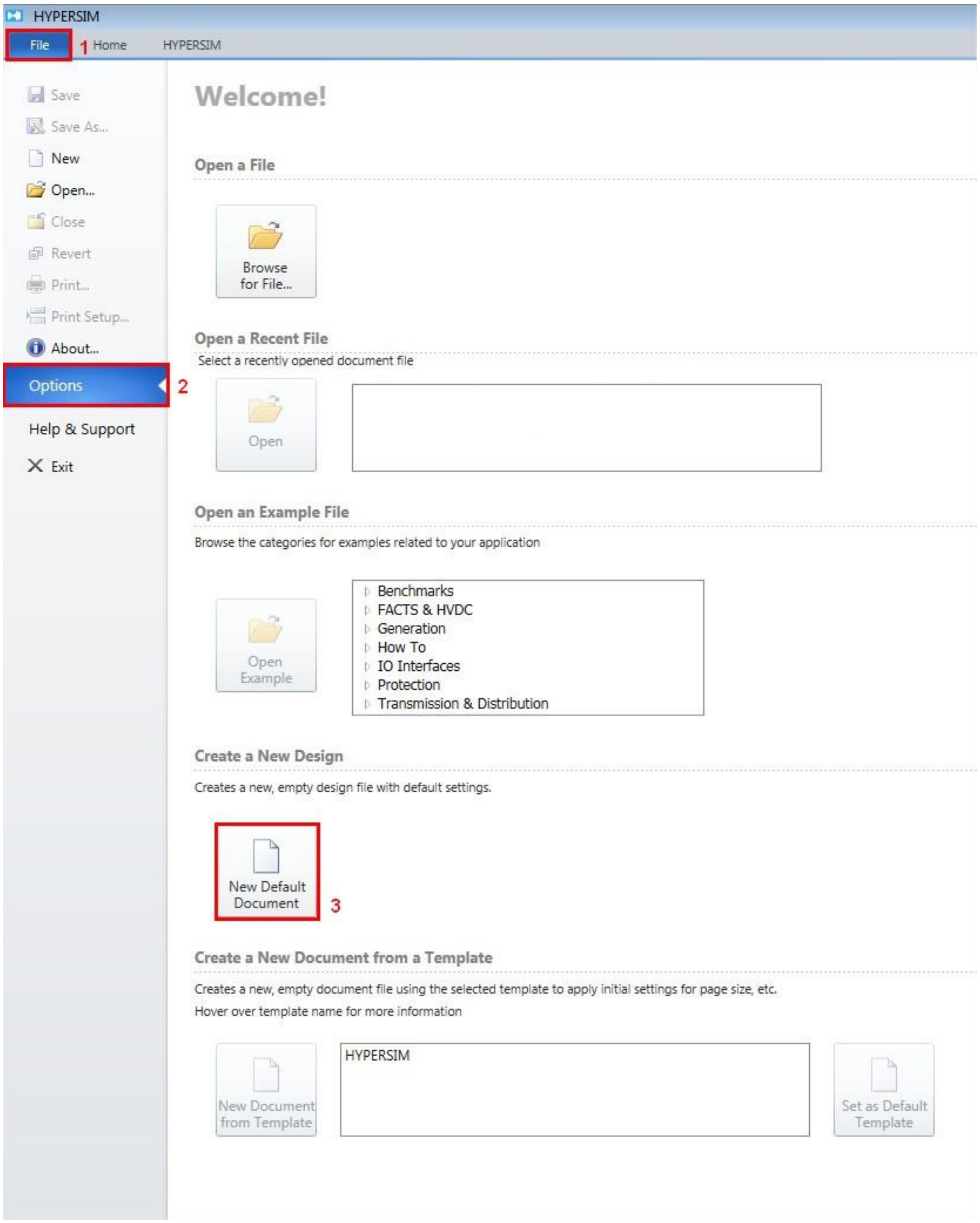


Figure 2: Welcome window

2. A browser opens with the default path: C:\Users**USERNAME**\Documents\HYPERSIM.
3. Either browse to a custom path or save your schematic to the default location.
4. The main modeling and simulation interface appears.

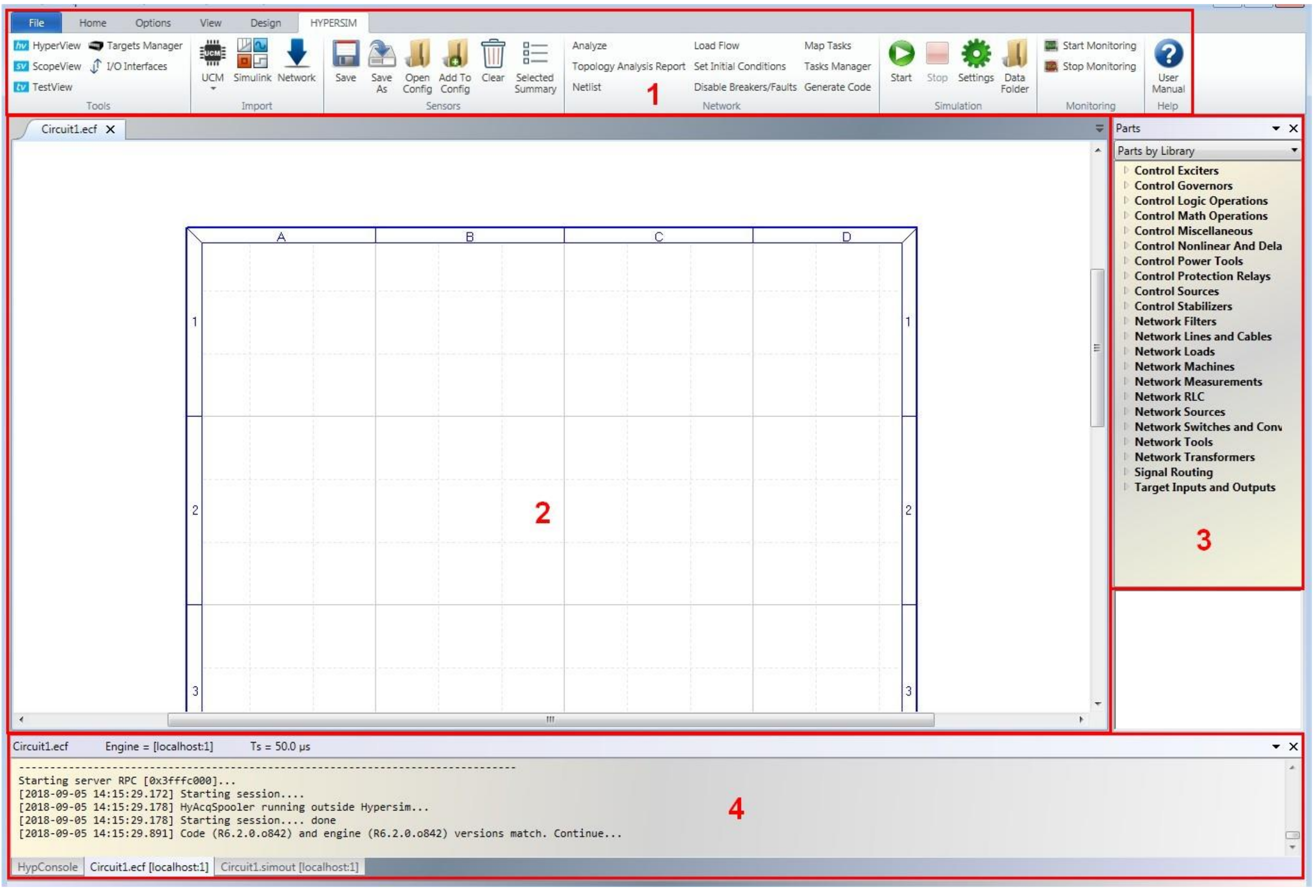


Figure 3: Main page components

1. The tab pane provides access to various menus: **File, Home, Options, View, Design** and **HYPERSIM**®.
2. The **Schematics** window is where schematics are displayed. This includes the **Parts** libraries, with previews of the blocks.

   The ‘Signal Routing’ and ‘Target Inputs and Outputs’ options contain connection options for your model. All other blocks are either power system network components or control blocks. The library can be docked and undocked by clicking the small arrow next to **Parts**.

3. This window also features a view of each open design.

2.2. DRAW A POWER SYSTEM

1. A bus and a **POW** (Point-On-Wave) block are required to start a new model. These can both be found in the **Network Tools** library, along with other components for advanced modeling, such as Simulink-imported models and user-coded models. **The POW is required for acquisition synchronization in ScopeView, and is mandatory in every model with network components.**

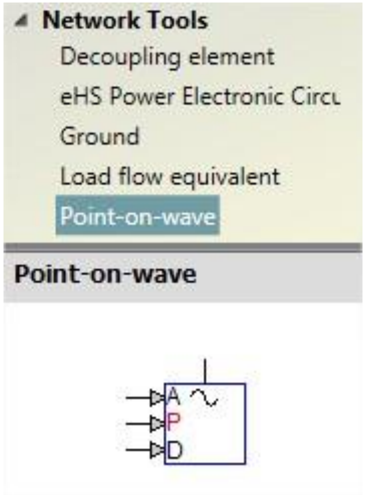


Figure 4: POW block

2. Simply drag and drop a component on the schematic to add it.
3. From the '**Network Sources**' library, add an **AC V source** and connect it to the **POW** previously added.
4. To connect two components, left-click from a connector of the first component to a connector of the second component.
5. Once done, you can click on the signal to see all connected signals (yellow) and connectors (light blue).

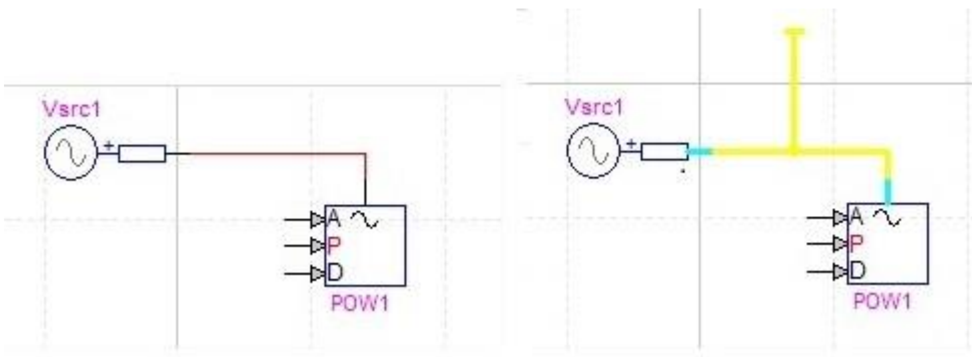


Figure 5: Connecting components

6. To change the signal to 3-phase signal, right-click the signal, then select '**Line Type**', then '**3-Phase Signal**'.

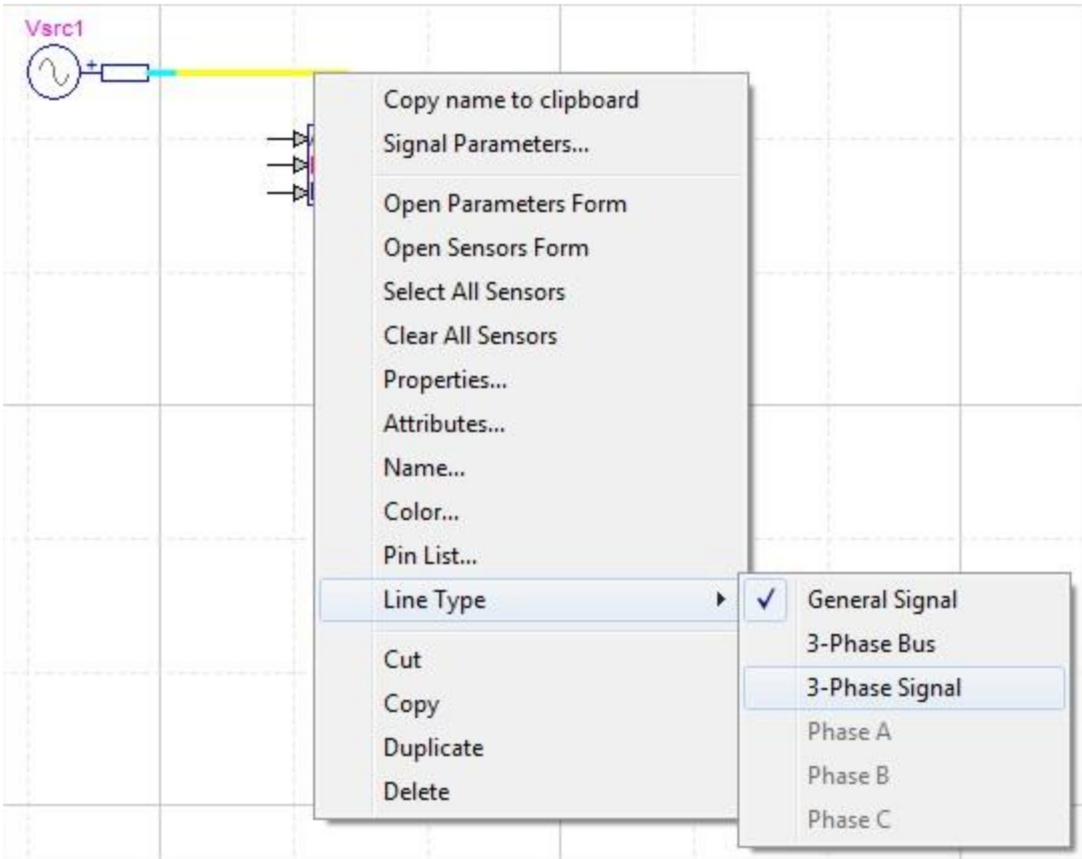


Figure 6: Changing signal types

7. To delete all selected signals (highlighted in yellow), press '**Delete**' on your keyboard or use the right-click menu.
8. To remove only one branch of a signal, use the **Zap** tool in the **Home toolbar**. The **Tools** group includes common operations for signals.

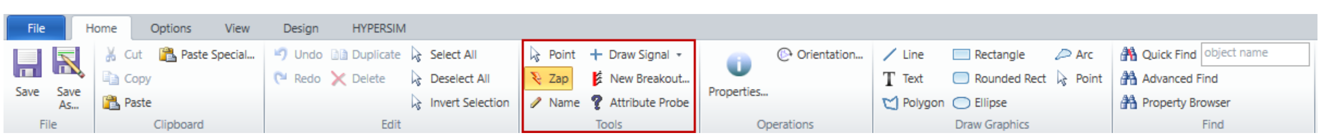


Figure 7: Deleting signals using the toolbar

9. You can now close the model. You'll be prompted to save the model you were working on.
10. This tutorial won't use that model again, so click **No**.

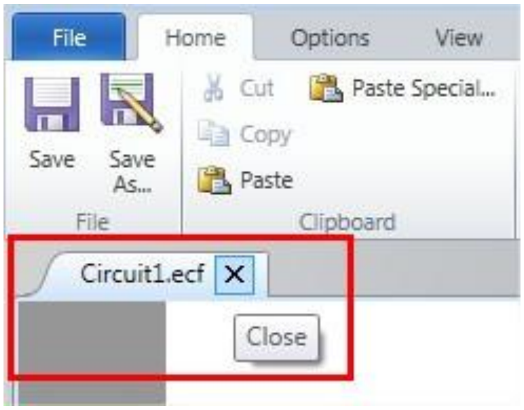


Figure 8: Deleting signals using the toolbar

11. For more on practical keyboard and other shortcuts, see **Annex B**.

That's all you need to know for now. Let's open an existing demo model.

2.3. OPEN A MODEL, LOAD SENSORS AND START A SIMULATION

1. Open the **model HVAC_500kV_6Bus.ecf** from the Example menu located in the **Transmission & Distribution** category, then click '**Open Example**'.

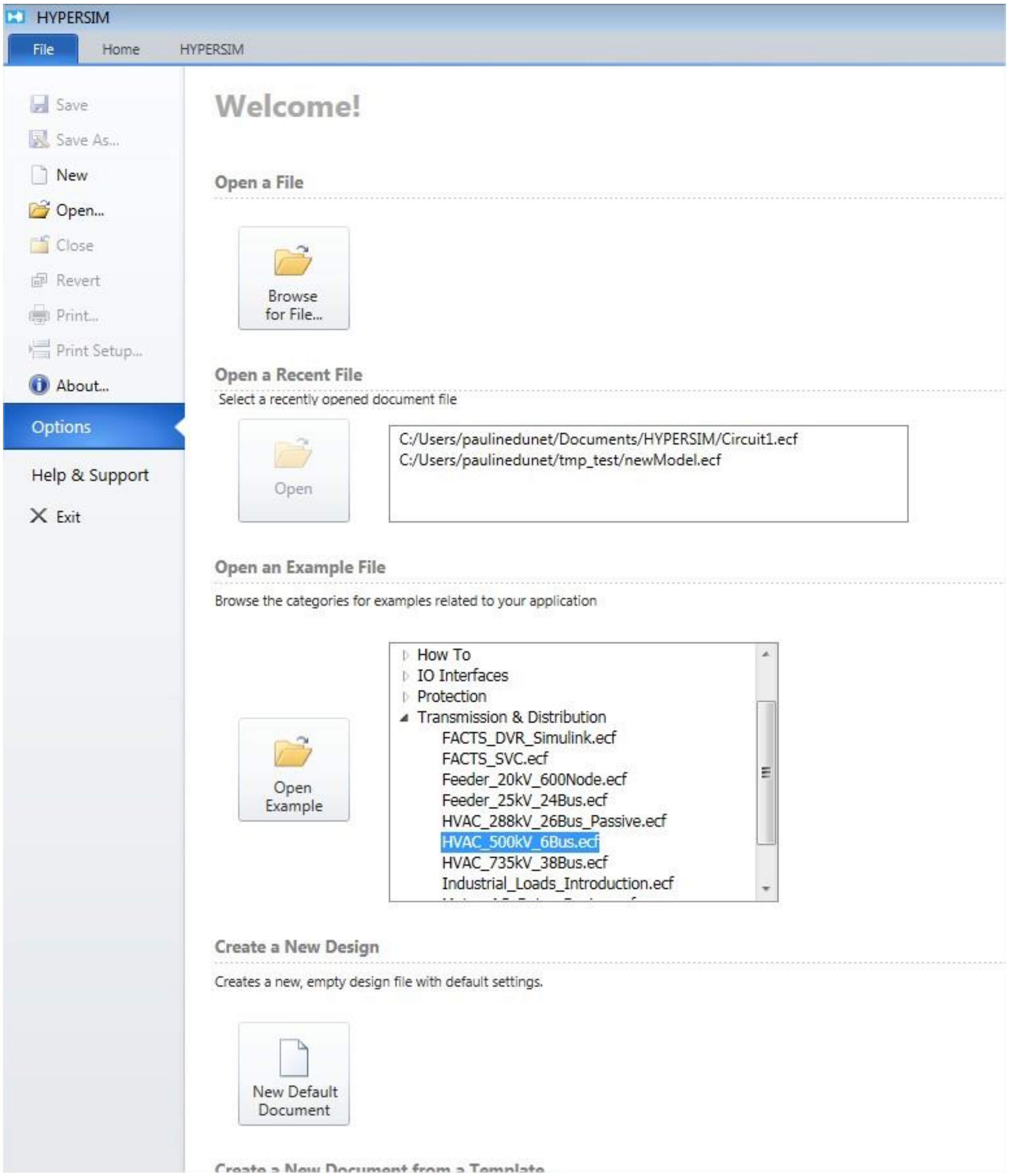


Figure 9: Opening an example file (Welcome page)

2. You are then prompted to choose a destination folder for the model to be imported. (By default, this is: C:\Users***USERNAME***\Documents\HYPERSIM). Click **Select Folder**.

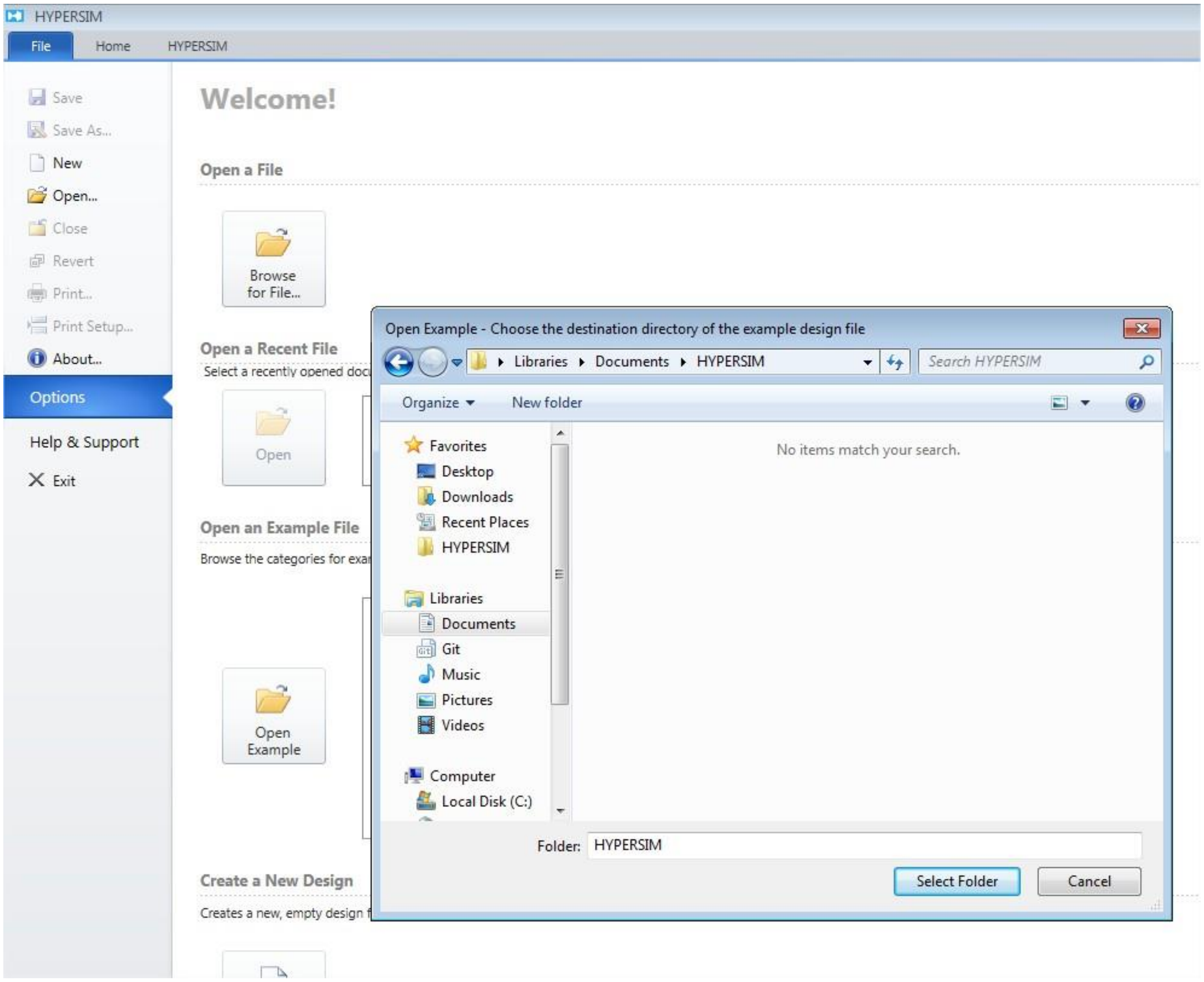


Figure 10: Choosing a directory to save the file

3. Once the folder is selected, HYPERSIM® copies the example files to the selected folder.
4. You can zoom in and out of the design by using CTRL and scrolling the mouse wheel up or down.

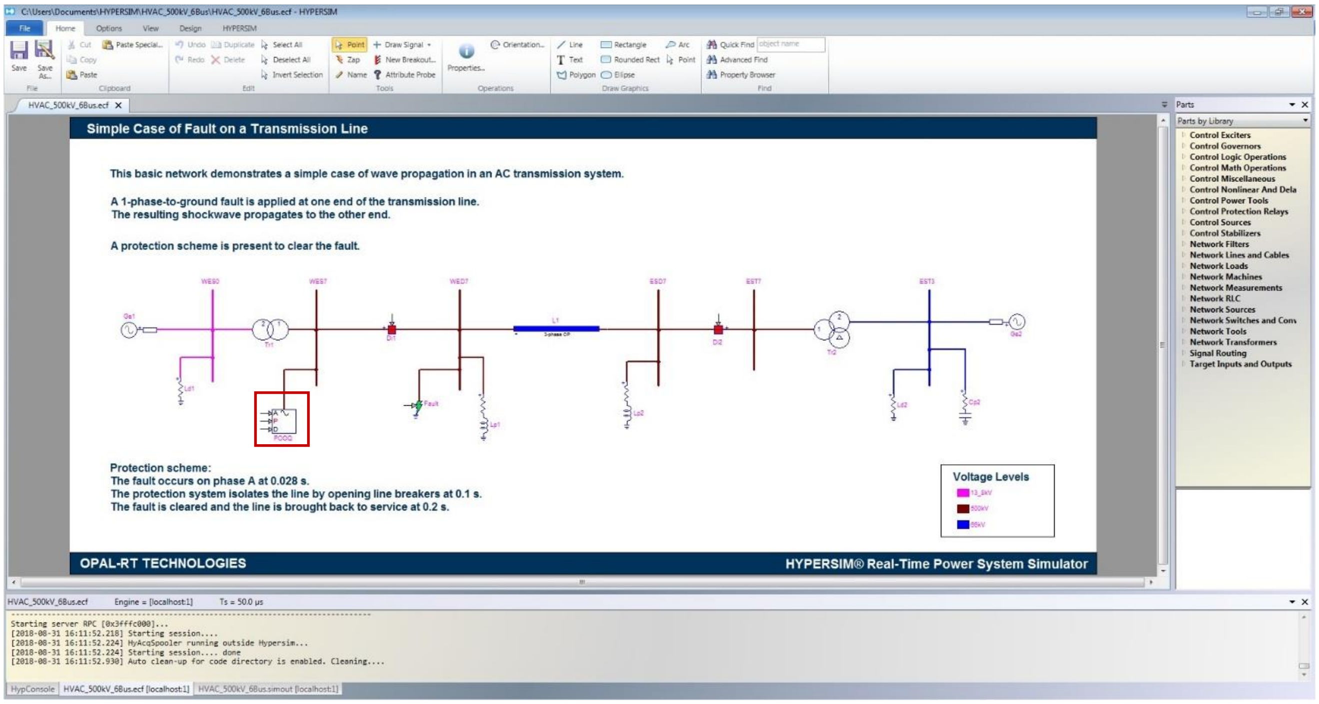


Figure 11: HYPERSIM® schematics page

Note: The block inside the red rectangle is called a Point-On-Wave (POW) and is **mandatory** in all models with network components. It is required for time synchronization of the acquisition. It works by monitoring one phase signal of the bus to which it is attached, so ensure that the system is always stable at that location. Placing it adjacent to a generator helps for stability of the voltage sine wave.

5. To visualize and edit sensors, right-click the block and click **Open Sensors Form**. For this example, let's use Di1.

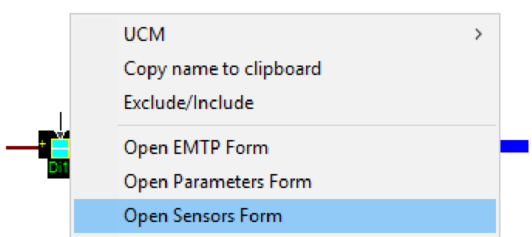
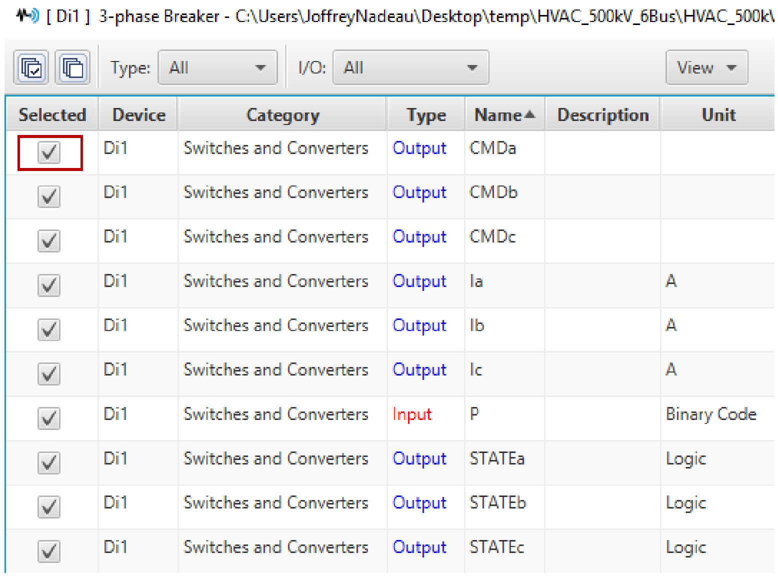


Figure 12: Open Sensors form

Use the **Selected** checkbox to select or de-select specific sensors.


[Di1] 3-phase Breaker - C:\Users\JoffreyNadeau\Desktop\temp\HVAC_500kV_6Bus\HVAC_500k\

Type: All I/O: All View

Selected	Device	Category	Type	Name▲	Description	Unit
☑	Di1	Switches and Converters	Output	CMDa		
☑	Di1	Switches and Converters	Output	CMDb		
☑	Di1	Switches and Converters	Output	CMDc		
☑	Di1	Switches and Converters	Output	Ia		A
☑	Di1	Switches and Converters	Output	Ib		A
☑	Di1	Switches and Converters	Output	Ic		A
☑	Di1	Switches and Converters	Input	P		Binary Code
☑	Di1	Switches and Converters	Output	STATEa		Logic
☑	Di1	Switches and Converters	Output	STATEb		Logic
☑	Di1	Switches and Converters	Output	STATEc		Logic

Figure 13: Sensors form

6. Go to the HYPERSIM® ribbon to access the sensor control.
7. Click **Selected Summary** to see the selected sensors.

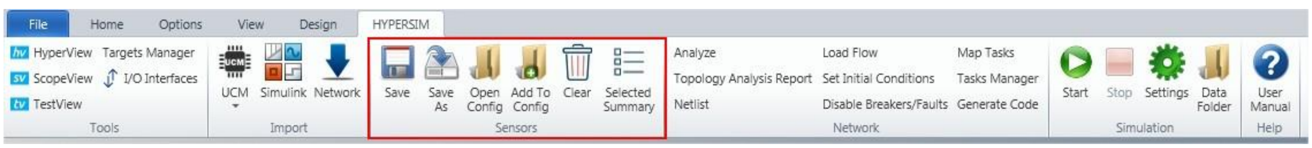
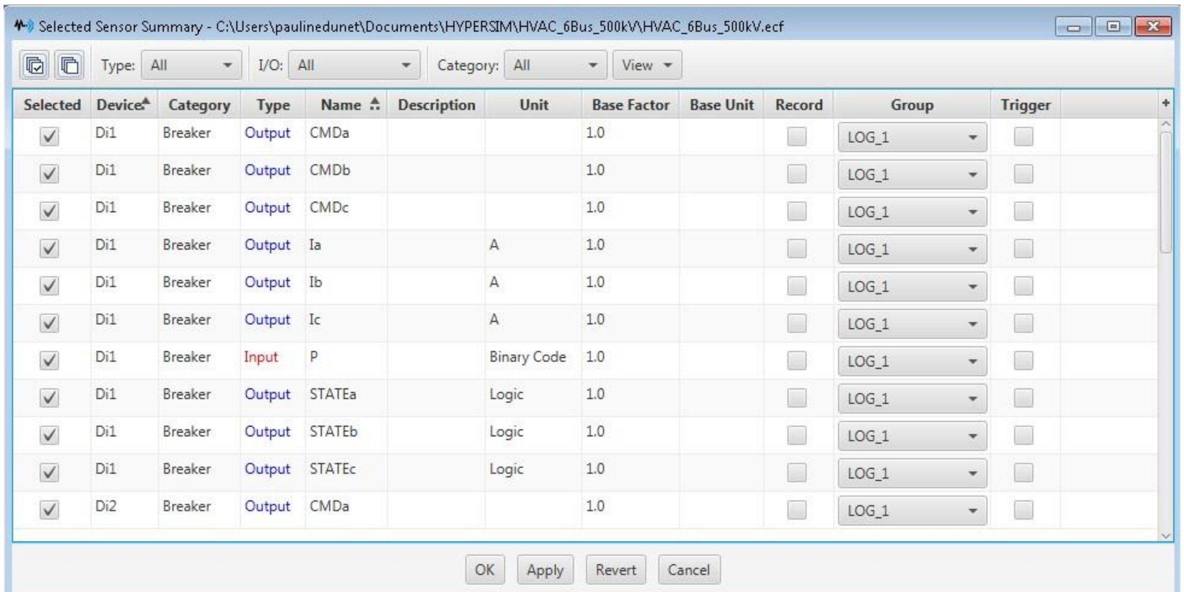


Figure 14: HYPERSIM® ribbon

8. All selected sensors from the design appear in this panel.

Note: In HYPERSIM®, sensors play two roles: first, they allow monitoring of the various signals across the model in ScopeView. Second, they can be used to configure analog and digital I/O as well as communication protocols.



Selected	Device	Category	Type	Name	Description	Unit	Base Factor	Base Unit	Record	Group	Trigger
☑	Di1	Breaker	Output	CMDa			1.0		☐	LOG_1	☐
☑	Di1	Breaker	Output	CMDb			1.0		☐	LOG_1	☐
☑	Di1	Breaker	Output	CMDc			1.0		☐	LOG_1	☐
☑	Di1	Breaker	Output	Ia		A	1.0		☐	LOG_1	☐
☑	Di1	Breaker	Output	Ib		A	1.0		☐	LOG_1	☐
☑	Di1	Breaker	Output	Ic		A	1.0		☐	LOG_1	☐
☑	Di1	Breaker	Input	P		Binary Code	1.0		☐	LOG_1	☐
☑	Di1	Breaker	Output	STATEa		Logic	1.0		☐	LOG_1	☐
☑	Di1	Breaker	Output	STATEb		Logic	1.0		☐	LOG_1	☐
☑	Di1	Breaker	Output	STATEc		Logic	1.0		☐	LOG_1	☐
☑	Di2	Breaker	Output	CMDa			1.0		☐	LOG_1	☐

Figure 15: Sensor summary window

9. Add the OPAL-RT Simulator to the target list by clicking **Tools/Target Manager** > **Target**.

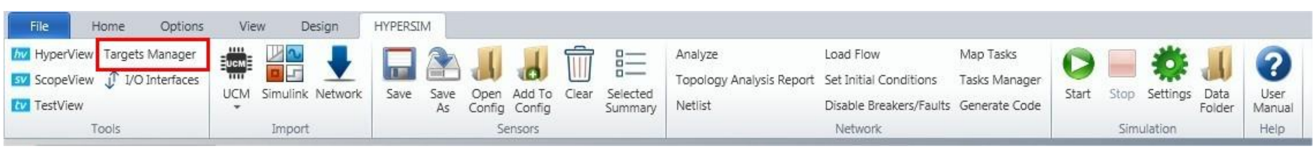


Figure 16: Open Target Manager

10. Click the plus sign **+** (**Add Target**)

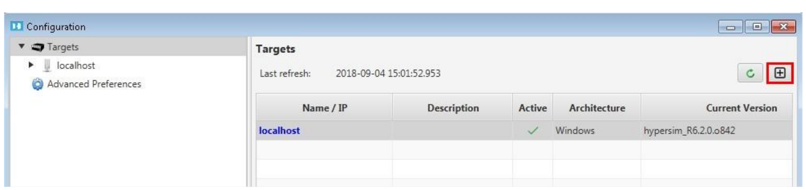


Figure 17: Add target

11. Enter the IP address of your target, then press OK.

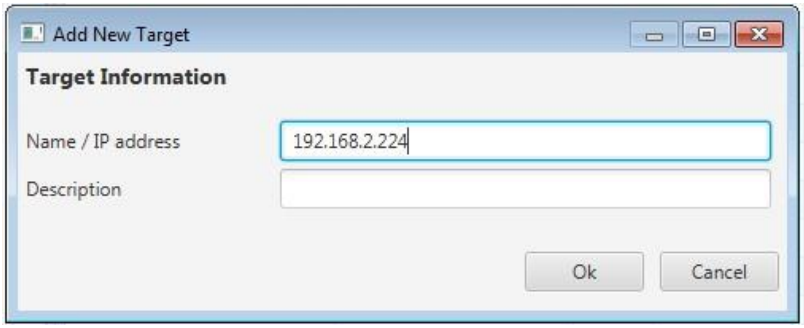


Figure 18: Setting target IP address

12. If the **HyCore version** is set to 'Unavailable', refer to the document 'How to install HYPERSIM® on a target'.

13. Click **Simulation/Settings** on the HYPERSIM® ribbon.

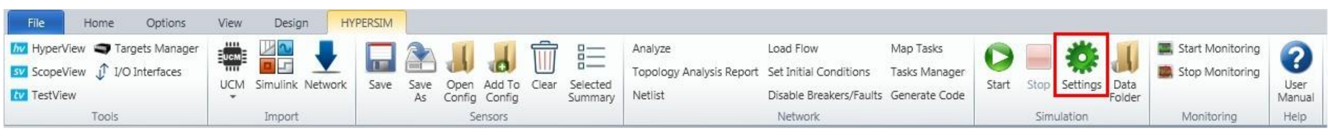


Figure 19: Opening Simulation options

14. Under **Target**, choose the target IP previously entered in **Target Manager**. Under **Simulation mode**, choose **Real-time** then click **OK**.

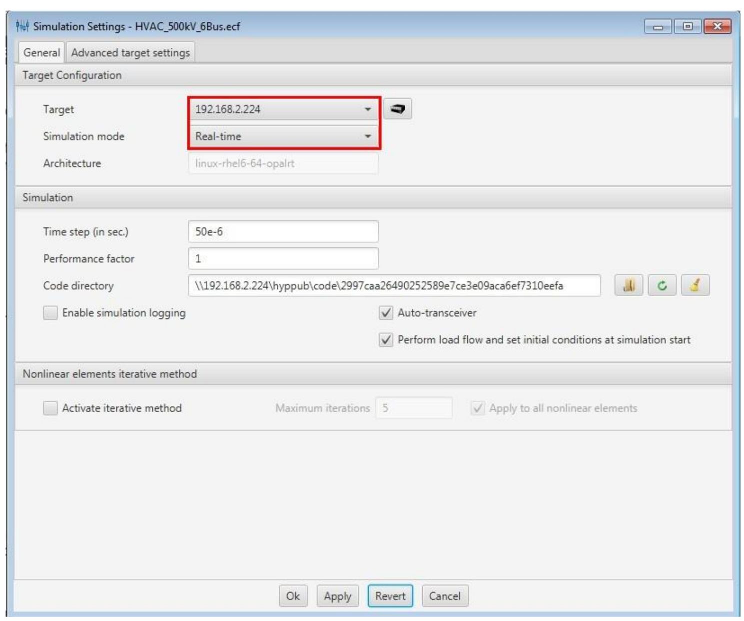


Figure 20: Configuring the target

15. By clicking **Simulation/Start** in the HYPERSIM® toolbar, the tasks are mapped automatically to the various cores, the code is compiled, and the simulation starts running. This can take anywhere from a few seconds to a few minutes, depending on the size of the model. Once it is started, the **Start** button is greyed out, and the **Stop** button is available to stop the simulation.

2.4. MONITOR SIMULATION RESULTS WITH SCOPEVIEW

1. In the HYPERSIM toolbar click **ScopeView**.

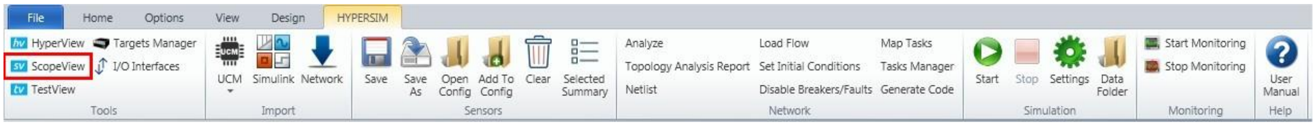


Figure 21: HYPERSIM® ScopeView tool

2. See **Annex D** for a detailed description of ScopeView options. For this tutorial, open a pre-prepared template. Click the folder icon with a **T** in the upper right corner and browse to the directory *C:\Users**USERNAME**\Documents\HYPERSIM,* and select the xml template file *HVAC_500kV_6Bus.xml*.

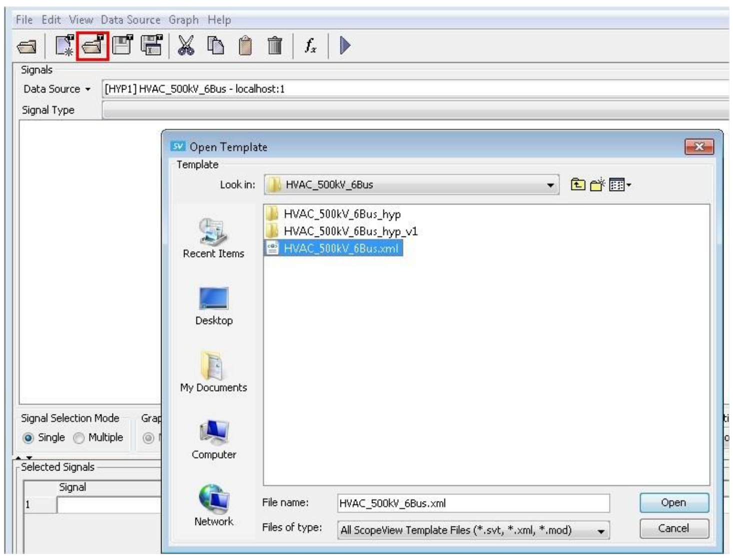


Figure 22: Opening a ScopeView template

3. The acquisition parameters are saved along with the template, so there is no need to apply changes for this tutorial. However, here are the basics for the acquisition parameters:

You can bring up the **Acquisition Parameters** by pressing the designated button on the tool bar.

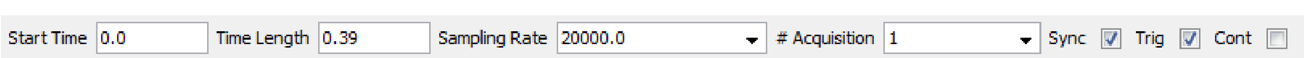


Figure 23: Acquisition parameters

- **Start Time:** Time at which the acquisition starts, after pressing the '**Play**' button, in seconds.
- **Time Length:** Duration of the acquisition, in seconds.
- **Sampling Rate:** Number of samples per second used to represent a signal. This number must be an integer multiple of the step size. Otherwise, a linear interpolation of the data points is processed by the acquisition spooler before sending the data to **ScopeView** (i.e., before the graphs appear).

$$Maximum\_sampling\_rate = \frac{1}{Step\_size}$$

- **# Acquisition:** Number of consecutive acquisitions to process. This mode is used specifically with events programmed to change over multiple acquisitions, such as incremental, Gaussian or uniform Gaussian (in the breaker model).
- **Sync:** If checked, as soon as you press **Play**, the acquisition spooler waits for the voltage (at the bus to which the POW is attached) to cross a specific level (by default, 0) on a specific phase (by default, A), with a specific slope (by default, positive). Only then will the acquisition start, as well as the timing for the various events programmed.

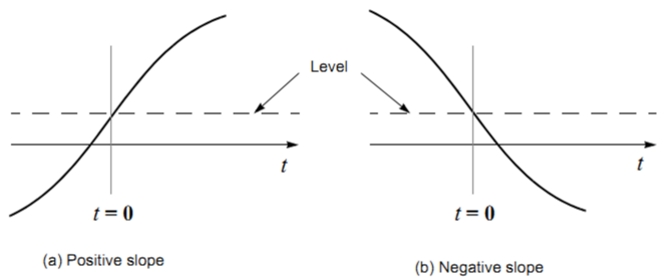


Figure 24: Point-on-wave synchronization

- **Trigger:** If checked, the various events programmed occur.
- **Cont:** If checked, acquisitions are performed continuously.

4. After setting the appropriate acquisition parameters, start an acquisition. The results should appear in the scope window.

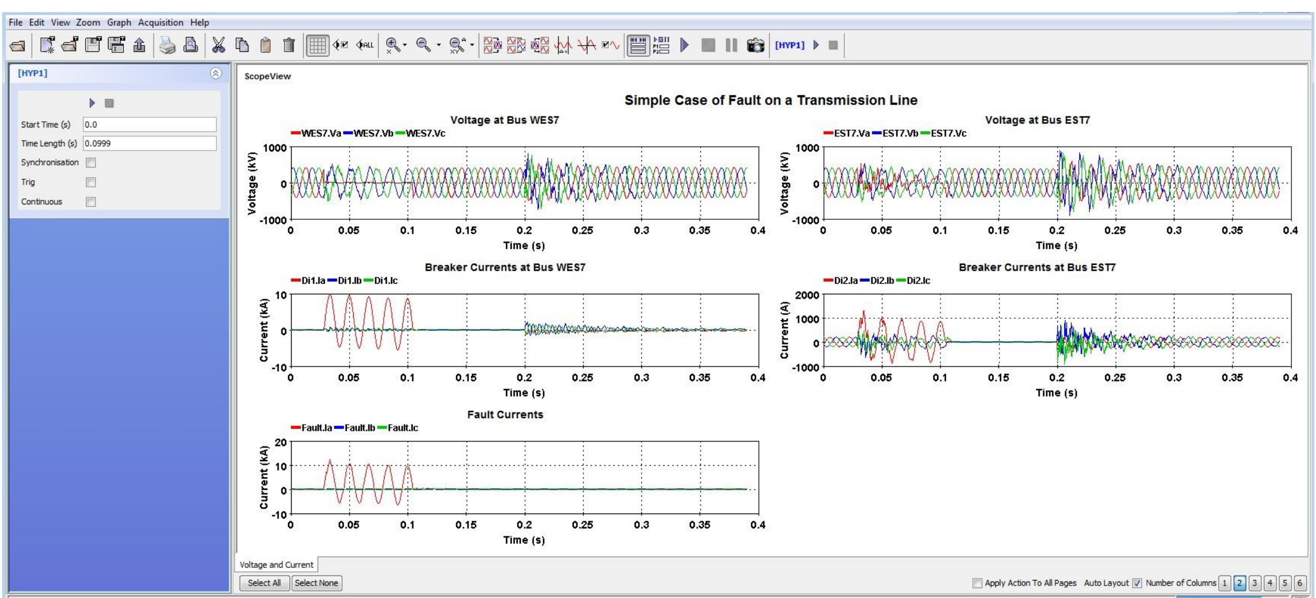


Figure 25: Acquisition results

A fault occurs on Bus *WED7* on phase *A* at 0.028 second. The protection system isolates the line by opening line breakers at 0.1 second. The fault is cleared and the line is brought back to service at 0.2

seconds. Fault currents as well as bus voltages and breaker currents (for both sides of the line) can be monitored in **ScopeView**.

3. USING ANALOG AND DIGITAL I/O IN HYPERSIM®

3.1. HARDWARE SETUP

This tutorial is written for an OP5600 simulator with a configuration 9 bitstream, but may be used with other types of simulators or firmware by substituting the correct I/O groups and configuration files.

3.1.1. ANALOG LOOPBACK

1. Loopback components have been provided with your simulator. Assemble them as shown here to complete the analog loopback cable. Analog loopback cables are useful to test inputs and outputs before connecting the simulation to an external device. This helps validate the voltage levels programmed and the integrity of the various signals by directly reading the signals sent.

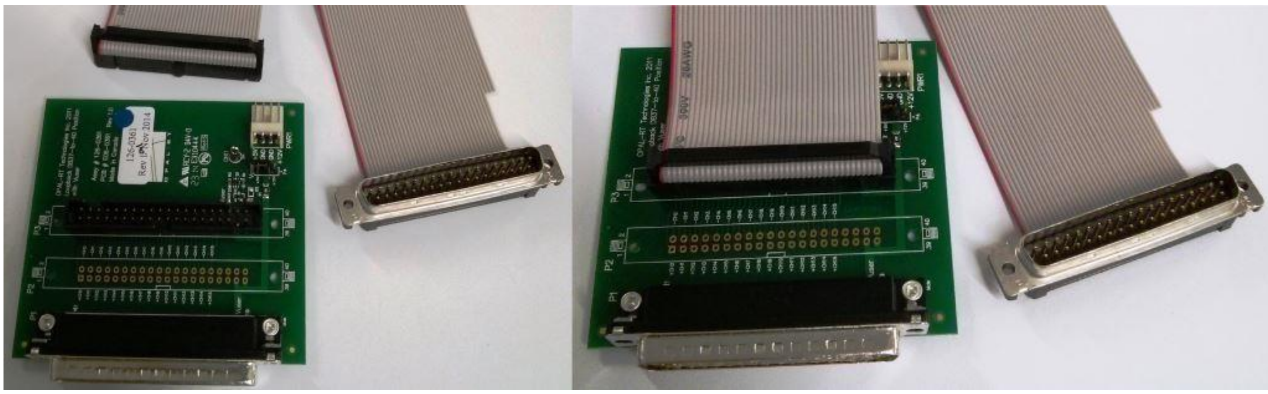


Figure 26: Loopback kit

2. At the back of the simulator, connect both ends of this cable to group 1, slot A, connector P1 (analog output) and group 1, slot B, connector P1 (analog input) as shown here:

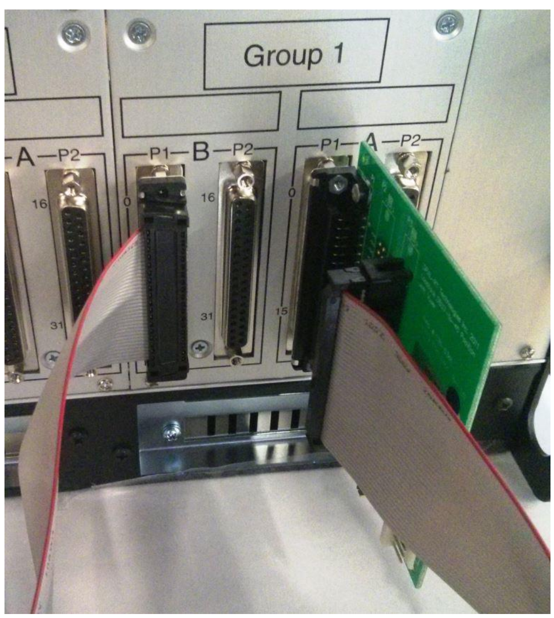


Figure 27: Connecting the loopback kit for analog

3.1.2. DIGITAL LOOPBACK

1. The digital loopback requires one more component, a Molex 4-pin cable connected to a 4-30 VDC power supply.

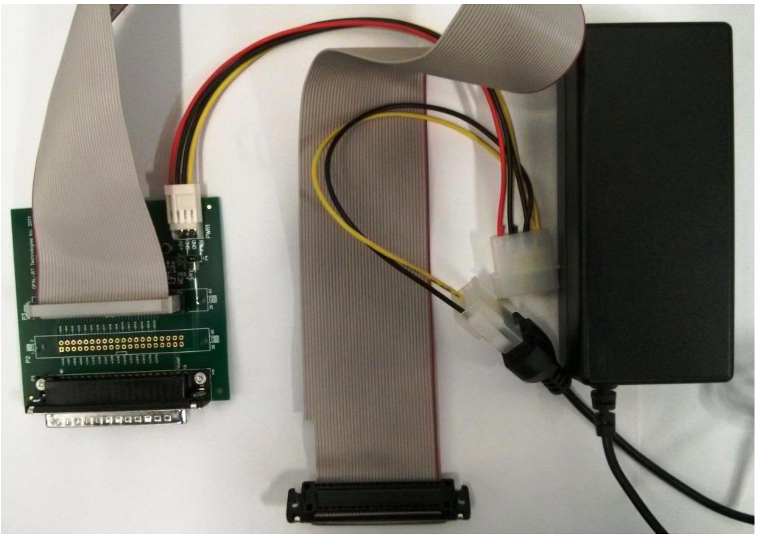


Figure 28: Loopback kit with external power for digital signals

2. Connect the end with the Molex cable to group 4, slot B, connector P1 (digital output) and the other end to group 4, slot A, connector P1 (digital input). Then connect the power supply to the wall outlet, as shown in the figure below. **The DC voltage output of the power supply is the voltage level to be measured when the digital signal is high.**

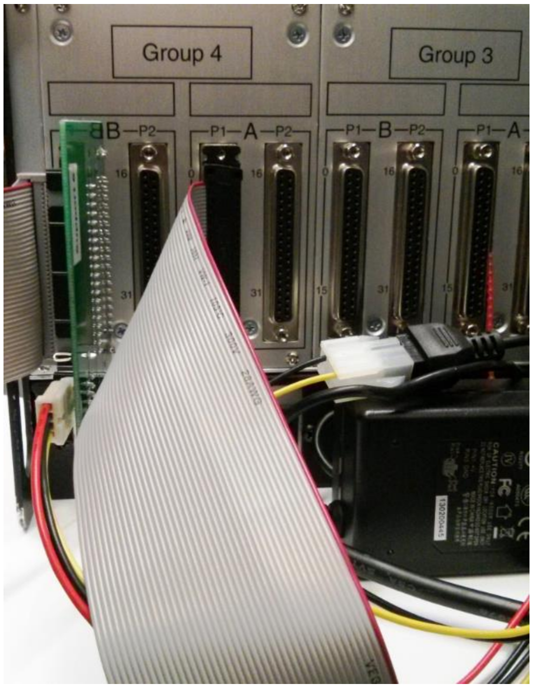


Figure 29: Connecting the loopback kit for digital signals

3.2. INTERFACING INPUTS AND OUTPUTS TO THE SIMULATION

1. Click the **HyperView** button in the HYPERSIM ribbon. For a detailed description of HyperView options, see **Annex C**.

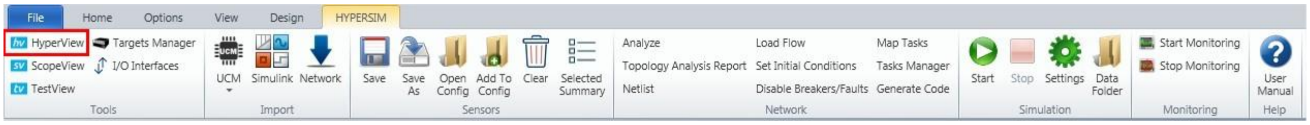


Figure 30: HYPERSIM® HyperView tool

2. The information required to map signals to I/O channels is found, in HyperView, in the **Real-Time** tab. On the left-hand side of this page, you find all processors available for simulation. On at least one of the icons, you should see a green square in the upper right corner labeled **I/O**. This is the core that processes tasks linked to I/O signals. Expand it to find a series of I/O types and numbers that you'll use in the sensor configuration to output signals.

3.

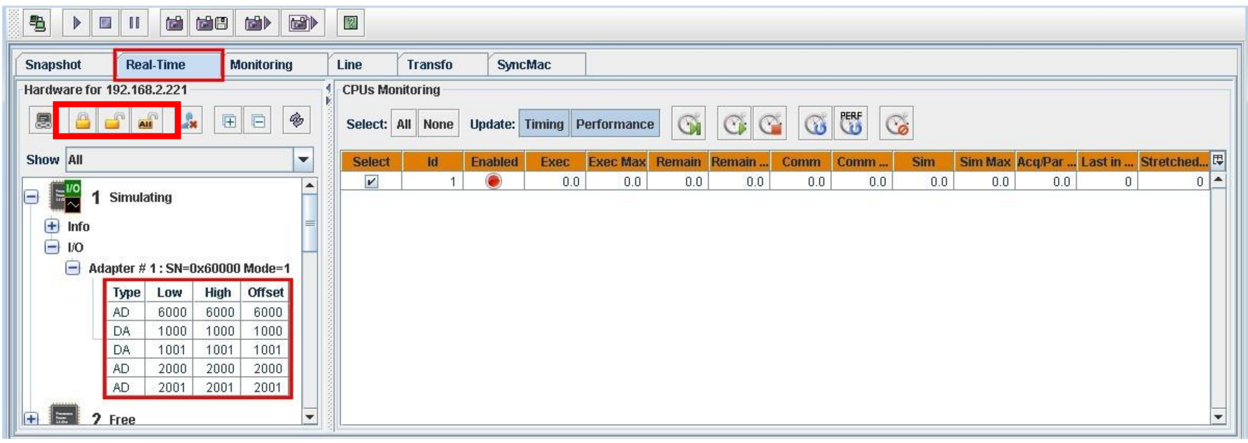


Figure 31: HyperView window

3.2.1. ANALOG OUTPUT SENSORS

Ensure that the bitstream is correctly flashed before using the simulator: if it's not done properly, I/Os won't display in HYPERSIM®.

1. Go back to the HYPERSIM® schematics window and stop the simulation (if it was running). Right-click the bus named **WES7**, and select **Open Sensors Form** to open a window where available signals are listed for that specific block. Note that you must right-click the component directly, and not on the block name.

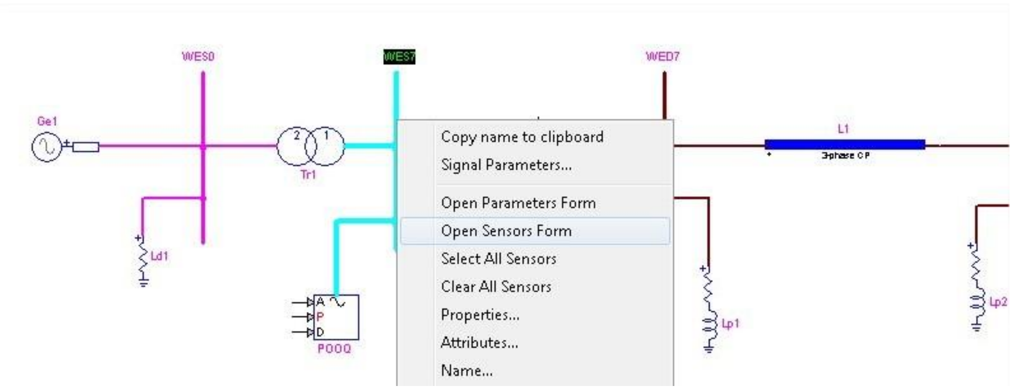


Figure 32: Opening HYPERSIM® Sensors Form

2. To configure a specific I/O, click the '**…**' (ellipses) button under the **I/O Type** column.

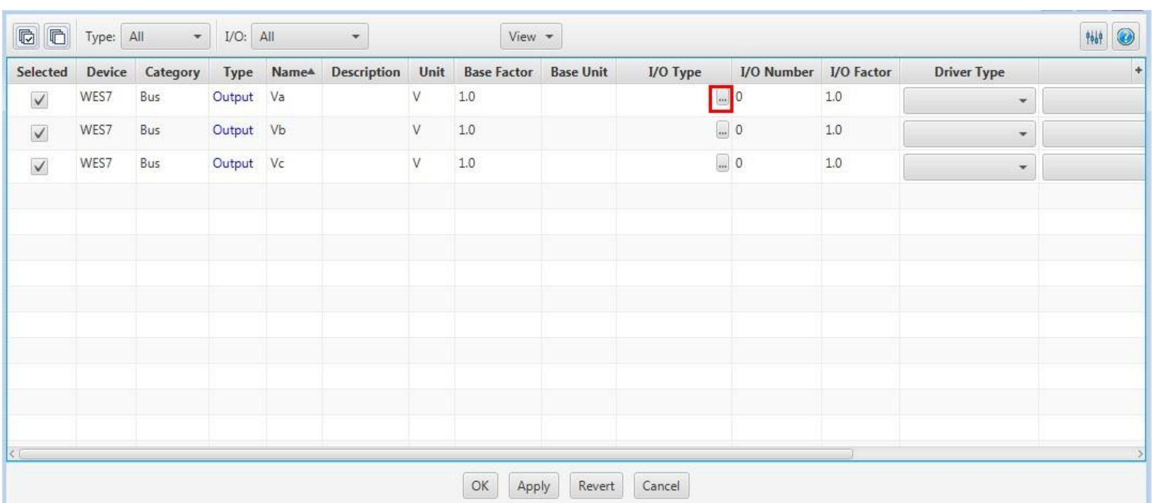


Figure 33: Configuring I/O

The I/O Selector window lets you choose from all available I/O on the selected simulator. Selection of the I/O DA 1000, for example, results in the following dialog.

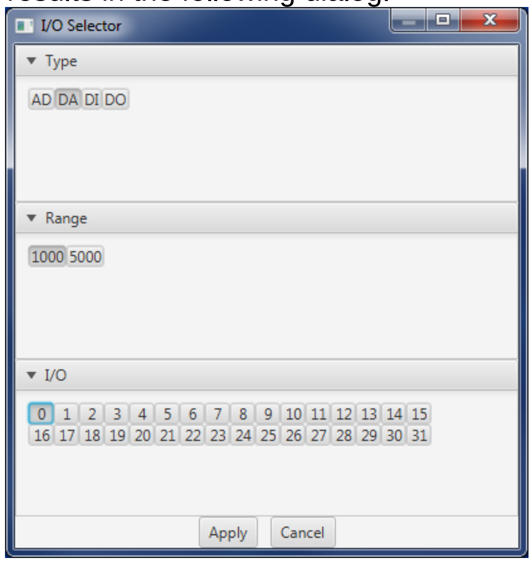


Figure 34: Selecting I/O

3. You can also directly enter the value in each field of the **Sensors Form**. Use right-click to copy or paste values. You can also enter a sequence of I/O numbers by selecting multiple lines.

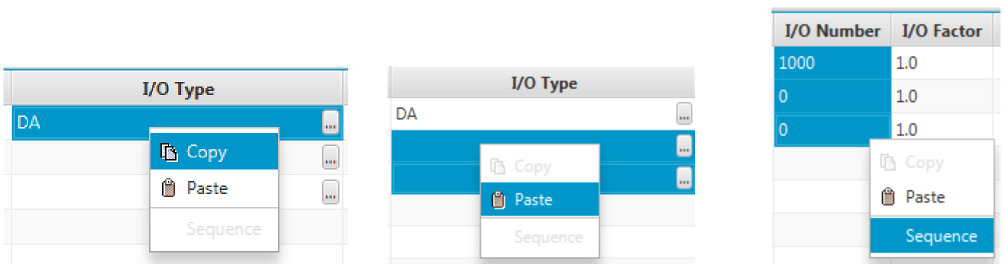


Figure 35: Selecting multiple I/O sequences

4. First select all three checkboxes on the left-hand side and set the values, as follows:

To display additional parameters use the **+** (plus) button on the column description bar.

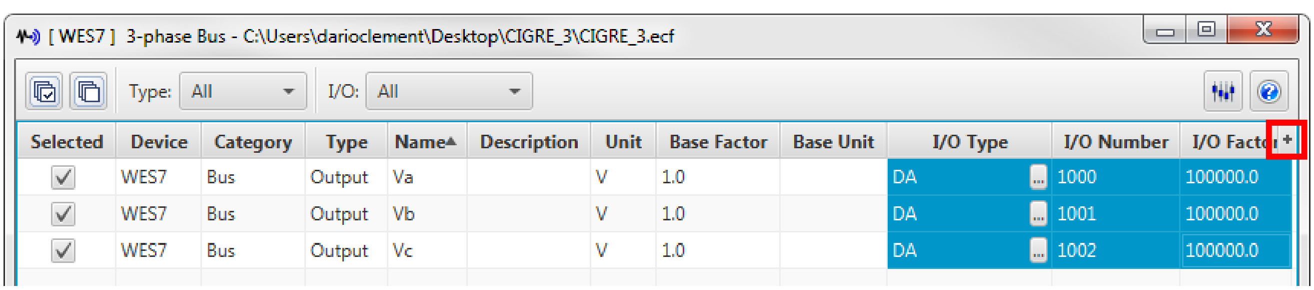


Figure 36: Selecting additional parameters to be displayed

4.1. The **I/O-Type** column can take 4 different values: DA for analog output, AD for analog input, DO for digital output and DI for digital input. (DA stands for digital-to-analog converter and is often used to refer to the process by which binary values are converted to a measurable analog signal. AD is thus the inverse analog-to-digital conversion.) Let's use DA for now.

4.2. The **I/O-Number** column contains the channel number from which the signal should be sent. These are the numbers from step 2 in the **Low** to **High** range.

4.3. HYPERSIM® knows where the hardware is located; it scans through all mezzanines in the following order: group 1 slot A, 1B, 2A, 2B, 3A, 3B, 4A and 4B. Say that you have an OP5600 simulator with analog output boards in slots 1A and 2A (as in bitstream configuration 9). If you use I/O-Number 1026, it refers to the 11th channel of the 2A board, because numbers 1000 to 1015 correspond to board 1A (see figure below). **For this tutorial, use I/O-Numbers 1000 to 1002 for the bus three-phase voltage values.**

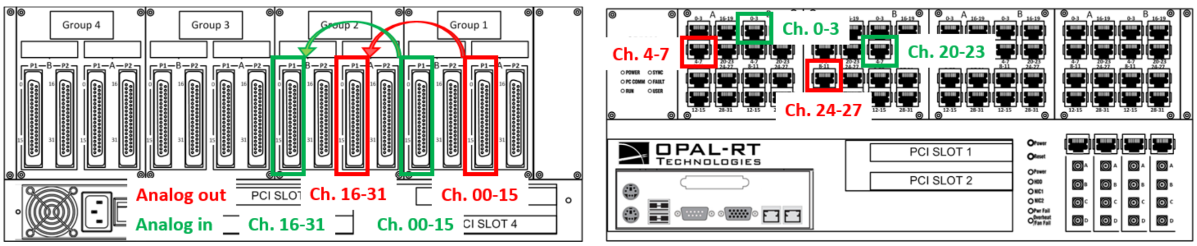


Figure 37: Example of signal routing

If you wish to use external monitoring with an oscilloscope:

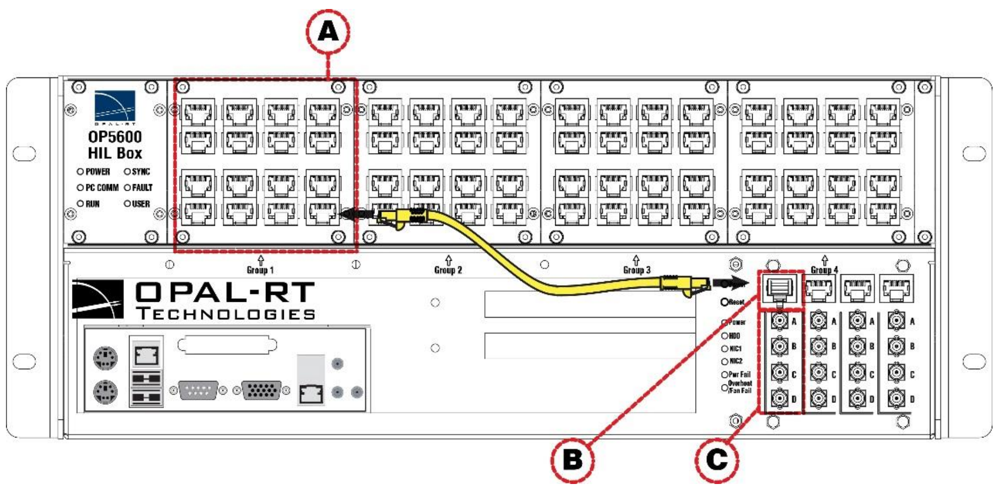


Figure 38: Monitoring connections

A. Connect one end of an RJ45 cable between the first connector of interface 1A.
B. Connect the other end of the RJ45 cable to the first BNC splitting interface, as seen above.
C. Connect the mini-BNC to BNC cable from any of the A, B or C connectors to the oscilloscope. **Note that a gain of 0.1 is applied to that interface.**

4.4. The I/O factor column is a scaling factor used to adapt to the analog output board's range, typically ±16 V. **It must be used as a divider.** Here, the bus voltage is around 400 kV peak-to-peak, so using 100 000 as a divider means that about 4 V should be measured on the simulator analog output.

3.2.2. ANALOG INPUT SENSORS

1. Analog inputs require a slight modification to the model because input sensors cannot be set directly on network components such as a 3-phase bus. You will need to open the **Target Inputs and Outputs** library and add the **Target input, analog** block to your network.

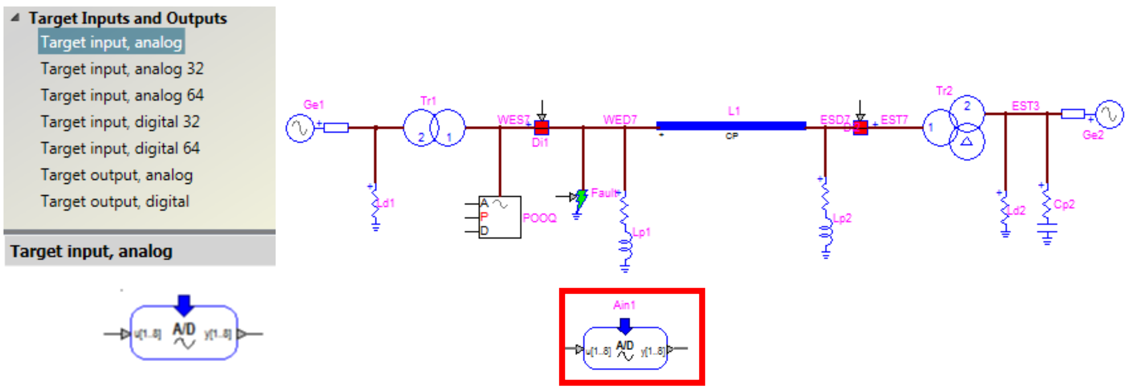


Figure 39: Modifying analog inputs

2. Right-click the **Target input, analog** block and select **Open Sensors Form** to edit the sensors.

   Input the values as below.

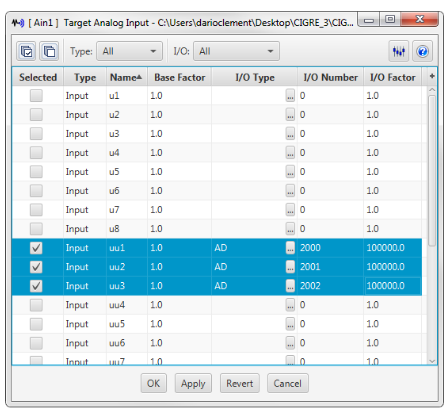


Figure 40: Inputting I/O values

Since the voltage values were scaled down to adapt to the analog output limitations, it is necessary to multiply these values again once they are received on the analog input side.

3.2.3. DIGITAL OUTPUT SENSORS

1. To interface a square wave signal with the digital output, you need to convert the data format from float to integer, because float values cannot be output to the digital interface.
2. You can distinguish both data types on the various blocks by the color of the signal name: black corresponds to float and red corresponds to integer. You will find the square wave generator in **Control Sources** and the **Data type, float to integer** conversion block in the **Signal Routing** library.

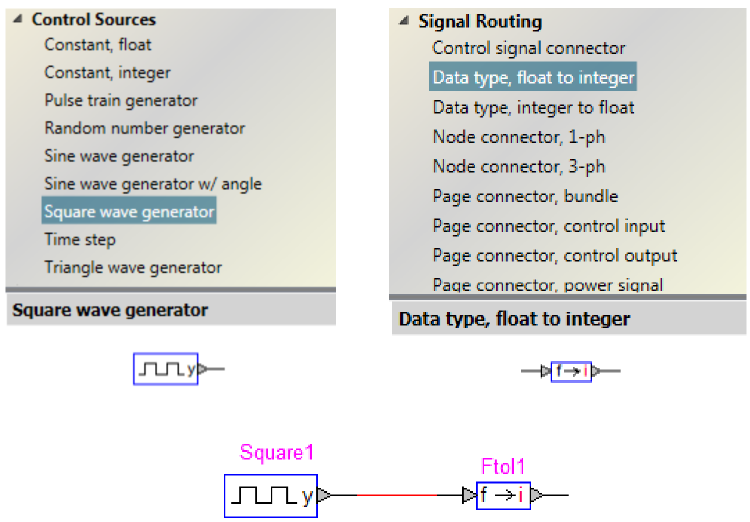


Figure 41: Selecting Control Sources and Signal Routing

3. Before setting the sensors, look at the square wave by double-clicking the block. This opens a mask that presents the various parameters to be modified. For this tutorial, we can leave it as is: a 100 Hz pulsed signal with a duty cycle of 50% is easily recognizable on an oscilloscope.

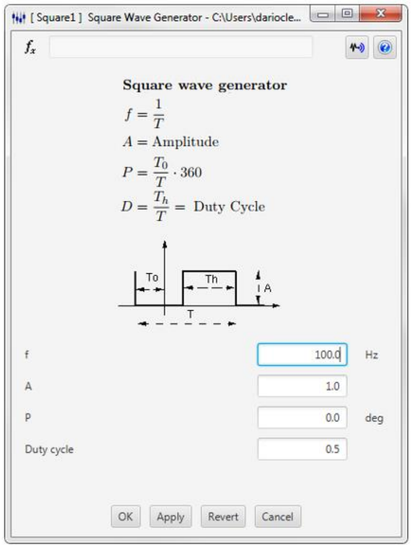
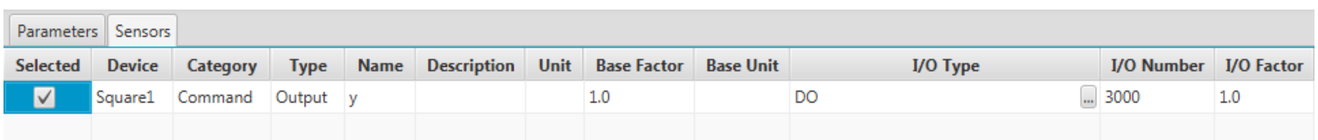


Figure 42: Changing Square wave parameter values

4. Taking into account the data format, we'll set the DO sensors on the **Data type, float to integer** block on the **y** signal. Since the square wave has an amplitude of one, no scaling is required. The voltage measured at the output of the simulator is always the supply voltage (5 V or 12 V with a loopback) when the value is 1.



Parameters | Sensors

Selected	Device	Category	Type	Name	Description	Unit	Base Factor	Base Unit	I/O Type	I/O Number	I/O Factor
✓	Square1	Command	Output	y			1.0		DO	3000	1.0

Figure 43: Sensors toolbar

3.2.4. DIGITAL INPUT SENSORS

As with the analog input, a dedicated block is needed for the digital input. The **Target input, digital** network component can be found in the **Target Inputs and Outputs** library. Connect it to a bus for task mapping purposes.

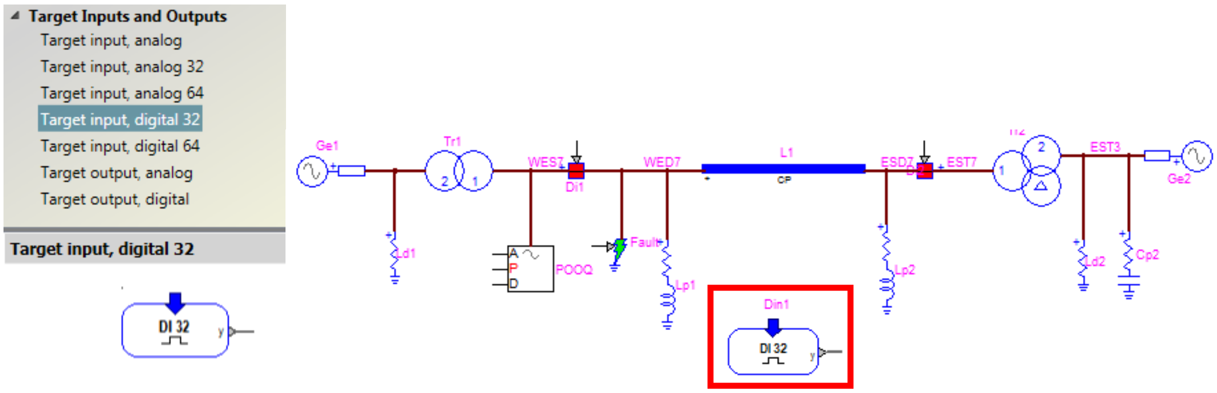


Figure 44: Adding a digital input block

1. Right-click the digital input block and select **Open Sensors From** to open the sensor parameters, and set the values as shown.

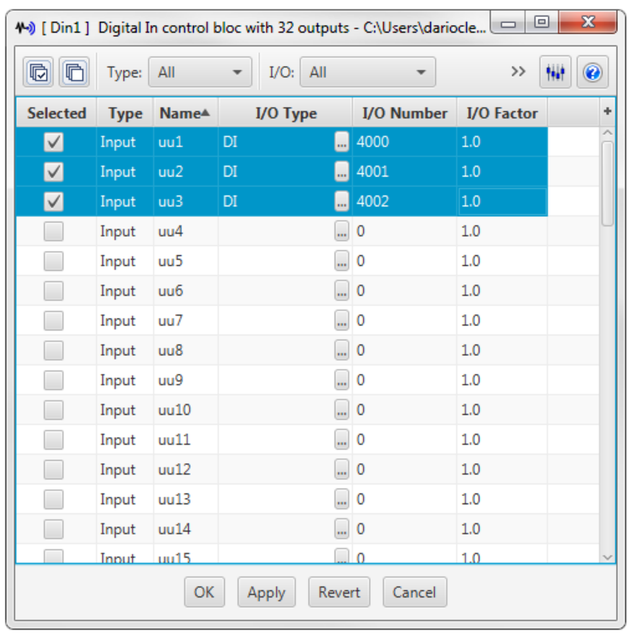


Figure 45: Inputting I/O values

3.2.5. TASK MAPPING AND SIMULATION START

1. Now that all sensors have been set, in the HYPERSIM tab under **Network** click **Map tasks**.

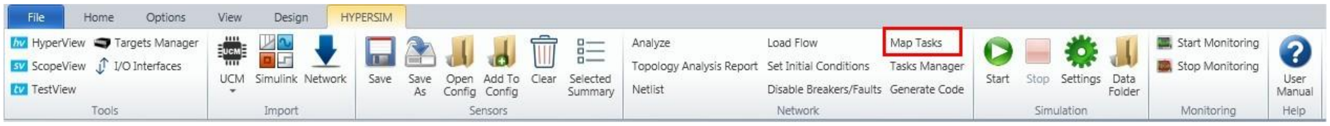


Figure 46: HYPERSIM toolbar, Map Tasks

2. Then look at the log window for the current network. The log window is very important; it provides a lot of information on the model, I/O, error messages, etc. There are two consoles for a given model: i) **Model**.ecf console gives information about operations in the model, and ii) **Model**.simout console gives information from the simulation. **When something does not work as expected, always look at these windows first.**

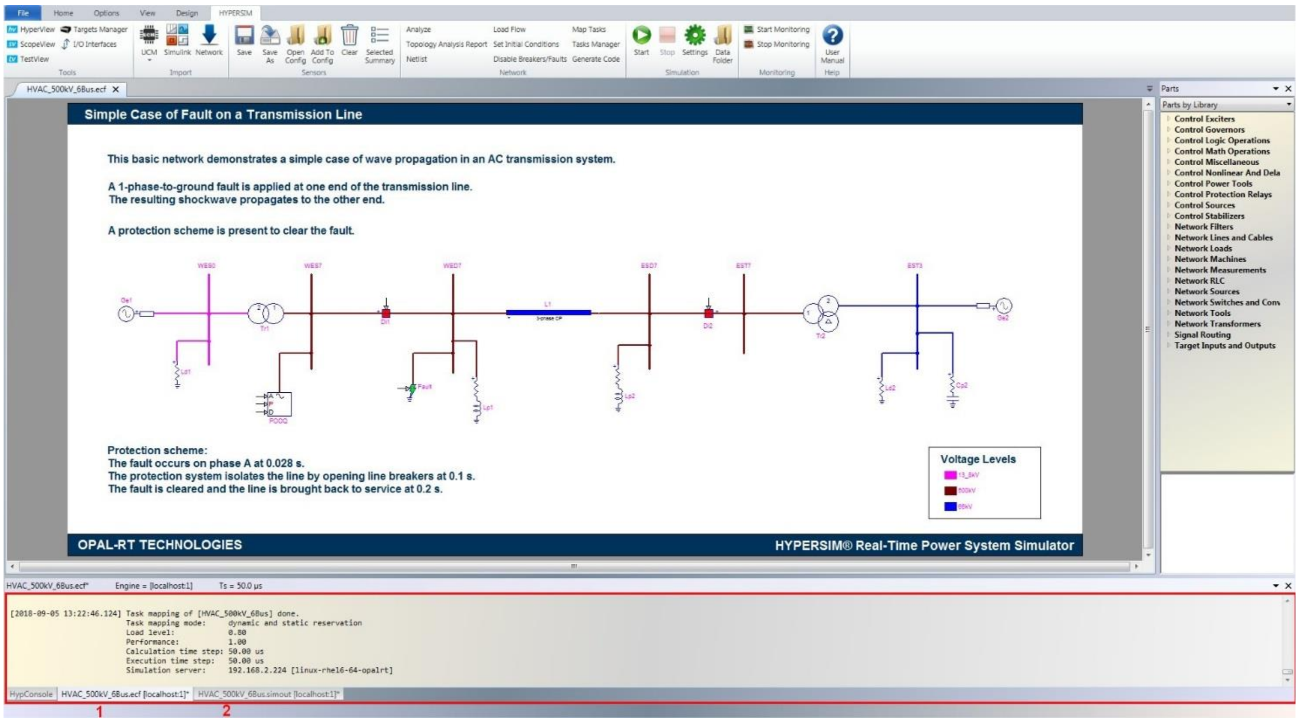
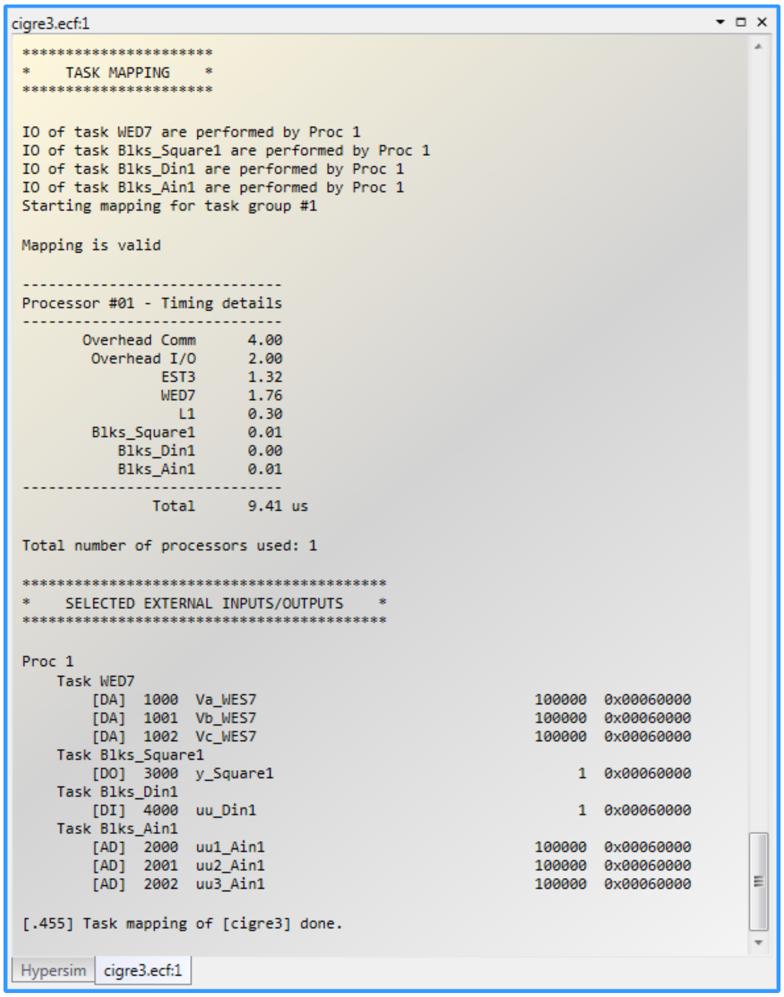


Figure 47: Network log window

Here, all the tasks are listed under **Processor #1** because the model can run on only one core. The task mapper also estimates the approximate time required to compute all tasks. The following section of the **Network Log Window** lists all I/O sensors that have been configured. It's a great way to verify that no sensor has been forgotten. If anything went wrong in the configuration, an error message is displayed.



```
cigre3.ecf:1

**********************
*    TASK MAPPING    *
**********************

IO of task WED7 are performed by Proc 1
IO of task Blks_Square1 are performed by Proc 1
IO of task Blks_Din1 are performed by Proc 1
IO of task Blks_Ain1 are performed by Proc 1
Starting mapping for task group #1

Mapping is valid

------------------------------
Processor #01 - Timing details
------------------------------
       Overhead Comm      4.00
        Overhead I/O      2.00
                EST3      1.32
                WED7      1.76
                  L1      0.30
        Blks_Square1      0.01
           Blks_Din1      0.00
           Blks_Ain1      0.01
------------------------------
               Total      9.41 us

Total number of processors used: 1

******************************************
*    SELECTED EXTERNAL INPUTS/OUTPUTS    *
******************************************

Proc 1
    Task WED7
        [DA]  1000  Va_WES7                            100000  0x00060000
        [DA]  1001  Vb_WES7                            100000  0x00060000
        [DA]  1002  Vc_WES7                            100000  0x00060000
    Task Blks_Square1
        [DO]  3000  y_Square1                               1  0x00060000
    Task Blks_Din1
        [DI]  4000  uu_Din1                                 1  0x00060000
    Task Blks_Ain1
        [AD]  2000  uu1_Ain1                           100000  0x00060000
        [AD]  2001  uu2_Ain1                           100000  0x00060000
        [AD]  2002  uu3_Ain1                           100000  0x00060000

[.455] Task mapping of [cigre3] done.

Hypersim   cigre3.ecf:1
```

Figure 48: Tasks and corresponding I/O for Processor #1

3. If your terminal output corresponds to the one from the last step, left-click to start your simulation.

3.2.6. ACQUISITION AND GRAPH MODIFICATION

1. Go to **ScopeView**, click **Data Source** and then **Load…** to open an interface.

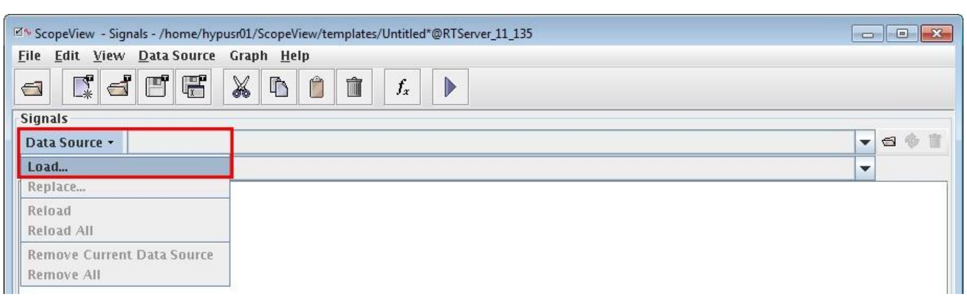


Figure 49: Loading Data Source in ScopeView

2. Go to the **HYPERSIM** tab and click **Load**. All parameters should already be correct.

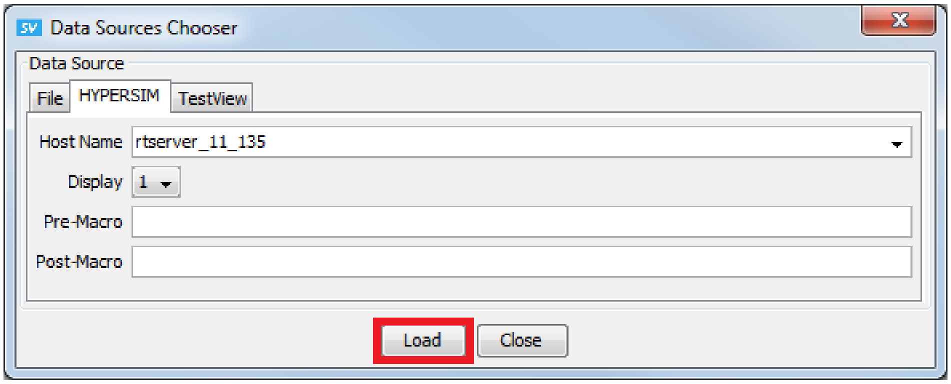


Figure 50: Choosing data source

3. All sensors configured are now available across various categories. Navigate the different signal types and add the analog and digital signals we created in HYPERSIM®.

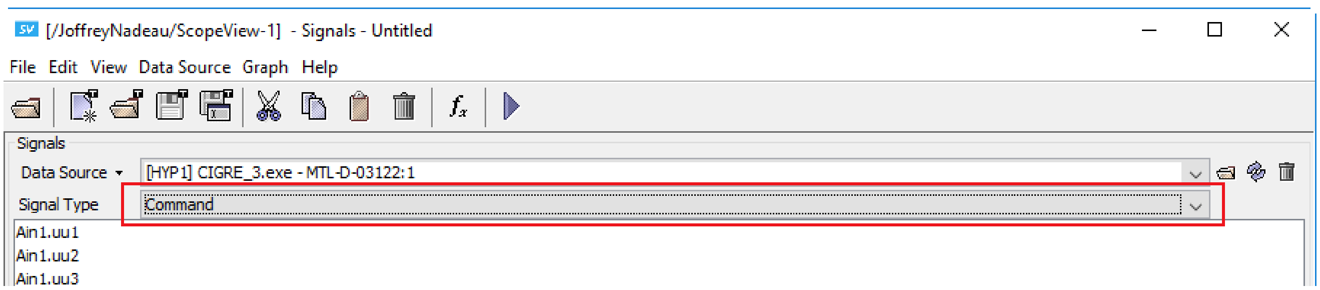


Figure 51: Adding signals

4. You should then see all signals in the list in the lower part of that window.

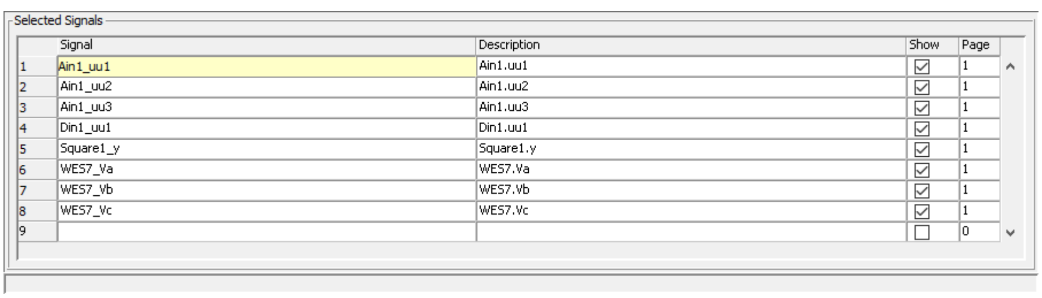


Figure 52: Viewing signals list

5. Go to the scope window of **ScopeView** and click **Play**. You should get the results below.

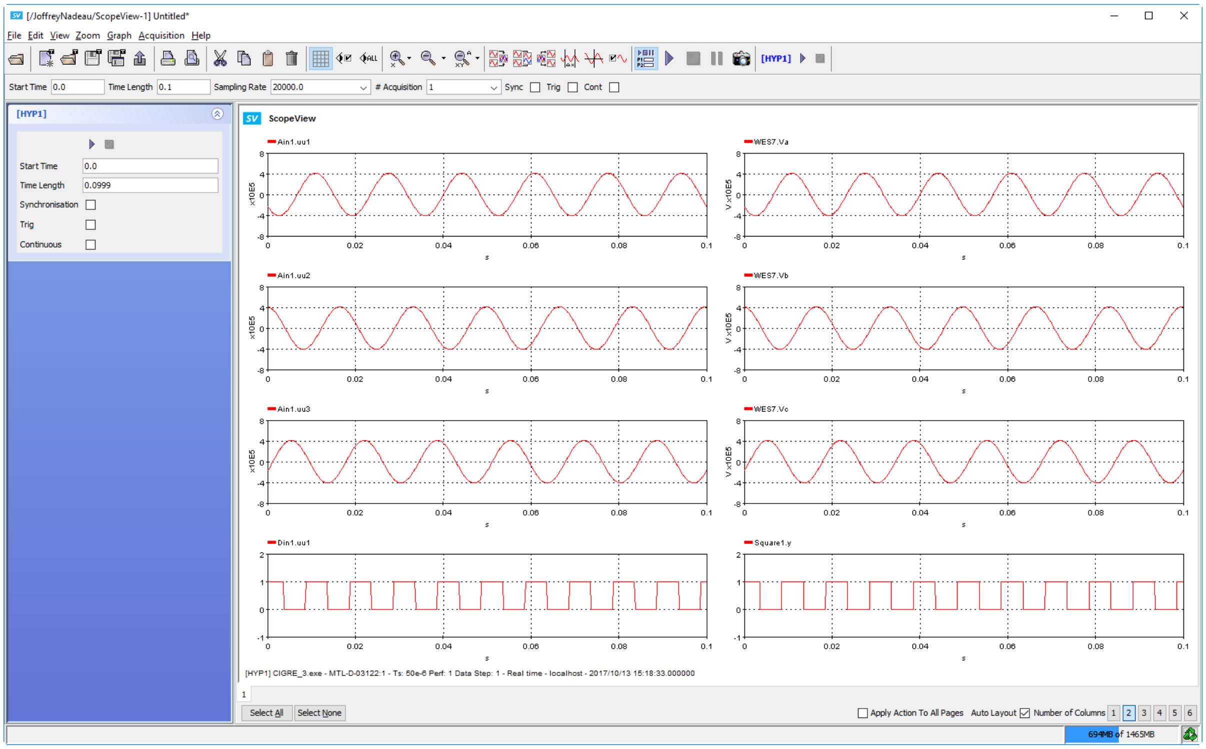


Figure 53: Viewing signals in ScopeView

6. Selecting 2 of the 8 graphs and then clicking the **Superimpose** button highlighted below allows you to compare the signal sent to the signal received. When selecting the graphs, the first one you select is the first to be listed in the multi-signal graph.

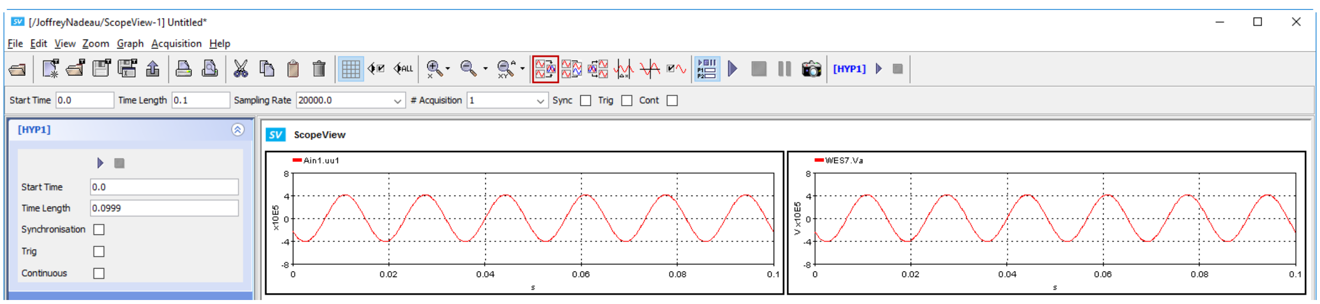


Figure 54: Superimposing signals for comparison

7. Repeat this for all 4 signals, and you'll get the results below. All signals overlay correctly, however you may notice a slight time shift between signals. This is normal when using I/O, as time step delays are required to output the signal and read it on the input. Depending on the synchronization mode, the delay can range from 2 to 5 time steps. The default value is 3. This means that if the model is running at 50 µs, there will be a time shift of 150 µs between the two signals.

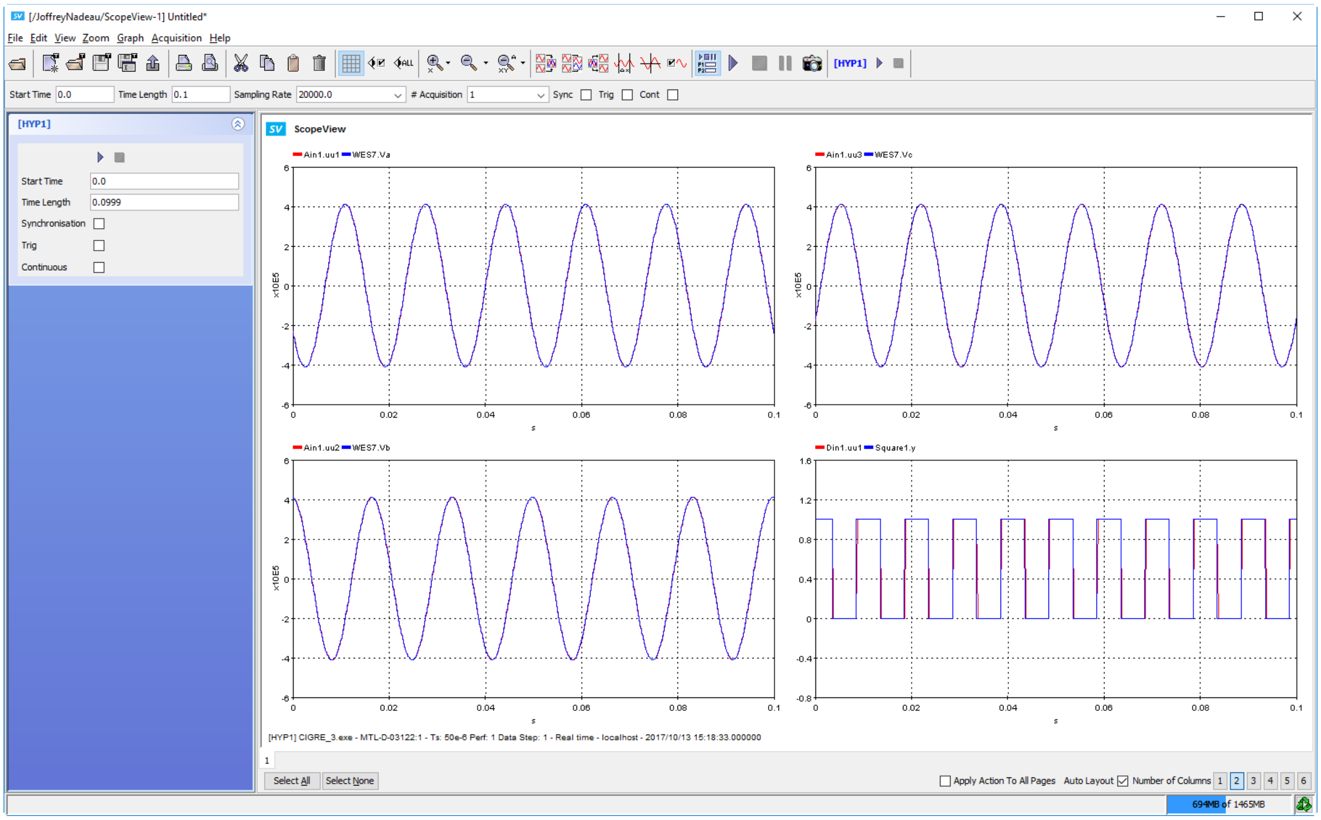


Figure 55: Viewing all signals in one ScopeView window

That's it for this tutorial. There are a lot of other features for you to discover in ScopeView. **Here are a few suggestions for you to try:**

- Change the acquisition parameters to better understand their impact.
- Undo the overlay of one graph and redo it with the double abscissa option (this is the button right next to the **Superimpose** button.
- Use the cursor to measure the slope and time difference between two data points.
- Use the right-click menu to change the axis names and scaling.
- Use the right-click menu to change the page layout, title, legend, etc.
- Overlay the three bus voltages and use the right-click menu to show phasor data.
- Use the various mathematical functions.

Congratulations, you've made it through the Quick Start Guide!

4. CONTACT AND SUPPORT

If you have any questions, please refer to our online Knowledge Base:

http://www.opal-rt.com/support-knowledge-base/

…or contact the Support team using our web form:

http://www.opal-rt.com/contact-technical-support/

You can also check out the **Troubleshooting** section in this document, as it is a good reference for a first experience. If you don't find answers for the issues you are having, do not hesitate to contact us, and we'll make sure to help you through your first steps and answer all your questions.

Hardware Information:

http://www.opal-rt.com/hardware-overview/

http://www.opal-rt.com/hardware-signal-conditioning-external-modules-accessories/

Return Merchandise Authorization (RMA):

http://www.opal-rt.com/return-marchandise-authorization/

Download latest software versions:

http://www.opal-rt.com/download-center/

5. TROUBLESHOOTING

5.1. BLOCKS CANNOT BE MOVED, ADDED OR EDITED

Make sure that your design is not in read-only mode: right-click the design (without selecting a component), choose **Properties…** and uncheck **Read Only**.

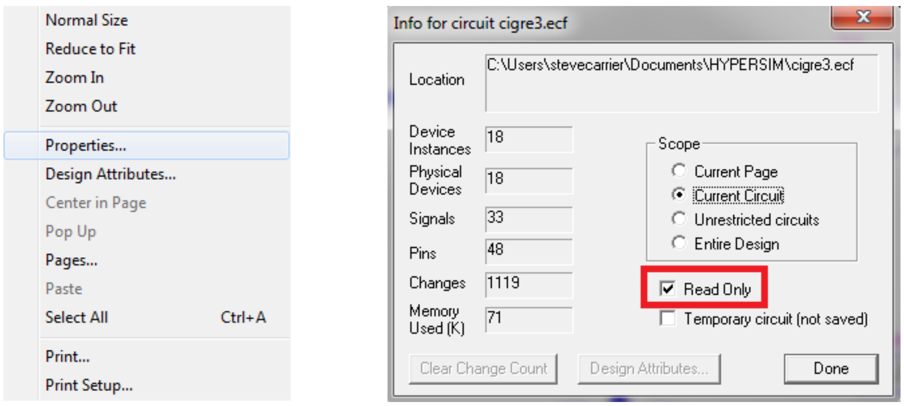


Figure 56: Adding or removing Read Only mode

5.2. MAP TASKS DO NOT WORK WHEN USING I/O

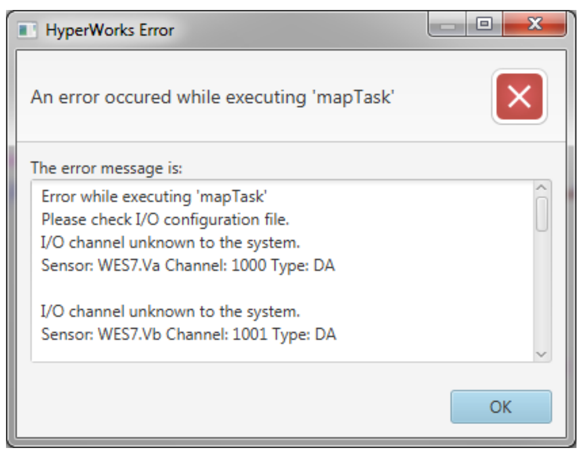


Figure 57: Map Tasks error message

Possible causes:

1. Simulation preference may be set to **Offline** instead of **Real-Time**.
2. Another simulation using I/O is already running. HyperView Real-Time Monitoring can be used to verify whether this is the case.
3. I/O-type and/or I/O-Number do not correspond to the values in the .io I/O configuration file.
4. I/O configuration files are set incorrectly (also in the .io file).

5.3. NO ANALOG OR DIGITAL INPUT SIGNAL IS RECEIVED IN SCOPEVIEW

1. Verify that sensors have been loaded correctly. The **Map Tasks** output in the terminal window should list all programmed I/O sensors.
2. Verify that the loopback cables have been connected correctly to the back of the simulator.
3. Verify the integrity of the output signals. If the model is stable, they should not be zero.

6. ANNEX A: COMMON KEYBOARD SHORTCUTS

The following table lists key combinations used in the schematic drawing and symbol editor windows.

COMMAND	SCHEMATIC	SYMBOL EDITOR
AddPins	-	Ctrl+E
Auto Create Symbol	-	Ctrl+J
Bring to Front	-	Ctrl+B
Close	Ctrl+W	Ctrl+W
Copy	Ctrl+C	Ctrl+C
Create Subcircuit Block	Ctrl+Shift+Q	
Cut	Ctrl+X	Ctrl+X
Design Attributes	Ctrl+Shift+D	
Design Preferences	Ctrl+J	-
Draw Bundle	Ctrl+Shift+B	-
Draw Signal	Ctrl+Shift+S	-
Duplicate	Ctrl+D	Ctrl+D
Enlarge	Ctrl+Shift+E	Ctrl+Shift+E
Export	F2	-
Find	Ctrl+F	-
Go to selected	Ctrl+G	-
Properties	Ctrl+I	-
Group	-	Ctrl+G
Magnify	Ctrl+Shift+M	-
Move to Grid	-	Ctrl+Shift+G
New	Ctrl+N	Ctrl+N
New Breakout	Ctrl+B	-
Normal Size	Ctrl+Shift+N	Ctrl+Shift+N
Normal Size	Home	-
Open	Ctrl+O	Ctrl+O
Open/Close Breaker	Alt-Click	-
Orientation (for device)	Ctrl+Q	-
Part Attributes	-	Ctrl+I
Paste	Ctrl+V	Ctrl+V
Paste Special	Ctrl+Shift+V	-
Pop Up (to parent circuit)	Ctrl+Shift+U	-
Print	Ctrl+P	-

COMMAND	SCHEMATIC	SYMBOL EDITOR
Push Into (subcircuit)	Ctrl+Shift+I	-
Redo	Ctrl+Shift+Z	-
Redraw	End	-
Reduce	Ctrl+Shift+R	Ctrl+Shift+R
Reduce to Fit	Ctrl+Shift+F	Ctrl+Shift+F
Reduce to Fit	Ctrl+Home	-
Rotate Left	Ctrl+R	
Rotate Right	Ctrl+Alt+R	
Save	Ctrl+S	Ctrl+S
Save As	Ctrl+Shift+A	-
Select All	Ctrl+A	Ctrl+A
Send to Back	-	Ctrl+Shift+B
Subcircuit and Part Type	-	Ctrl+Q
Undo	Ctrl+Z	Ctrl+Z
Text Font	-	Ctrl+T
Ungroup	-	Ctrl+U
Zap	Ctrl+H	-

Other Key Usage

KEY	FUNCTION
Delete	Deletes selected items
Arrow Keys	Flip and rotate items while placing a device or pasting
Arrow Keys	Move selected items in small steps
Ctrl or Shift or Alt	Path controls over signal lines while drawing signals. Press these keys after starting to draw with the mouse pointer.
Ctrl+Shift	Control device alignment while placing devices. Press these keys while a device is stuck to the mouse pointer.
Ctrl+Tab	Moves between design windows in the main HYPERSIM® window.
Page Up (PgUP)	Scroll Up
Ctrl+PgUp	Scroll Left
Page Down (PgDn)	Scroll Down
Ctrl+PgDn	Scroll Right
Spacebar or Esc	Exits and moves back to the normal pointer when a signal pointer is active. Also stops the sticky action of placed or duplicated objects. If you are working with the text tool pointer, then you must first exit the text and then press the Spacebar. To exit the text, click on an empty space in the design.

Holding down the CTRL button while holding moving the mouse pointer is a quick way to navigate around a drawing.

7. ANNEX B: OVERVIEW OF THE HYPERVIEW TOOLS

This is the communication server between HYPERSIM®, ScopeView and TestView. It also offers advanced features as listed below.

1. **Real-time performance**

This tab enables monitoring simulation performance in real-time. Are there overruns or stretched steps? How big is the real execution time? How much idle time is available? All these questions and more can be answered with this tool.

2. **Line, transformer and synchronous machine programming tools**

These tools are meant to:

1. Compute the parameters of the DPL from the geometric configuration.
2. Compute the inductance matrix values of a transformer according to the BCTRAN model.
3. Compute the synchronous machine with internal fault parameters from the standard parameters.

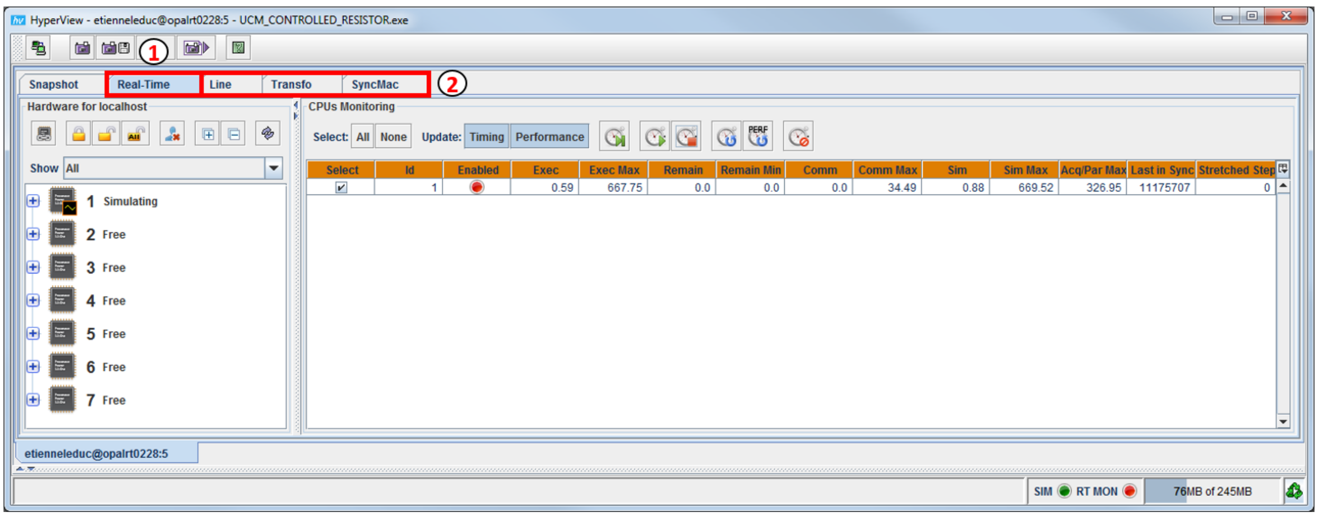
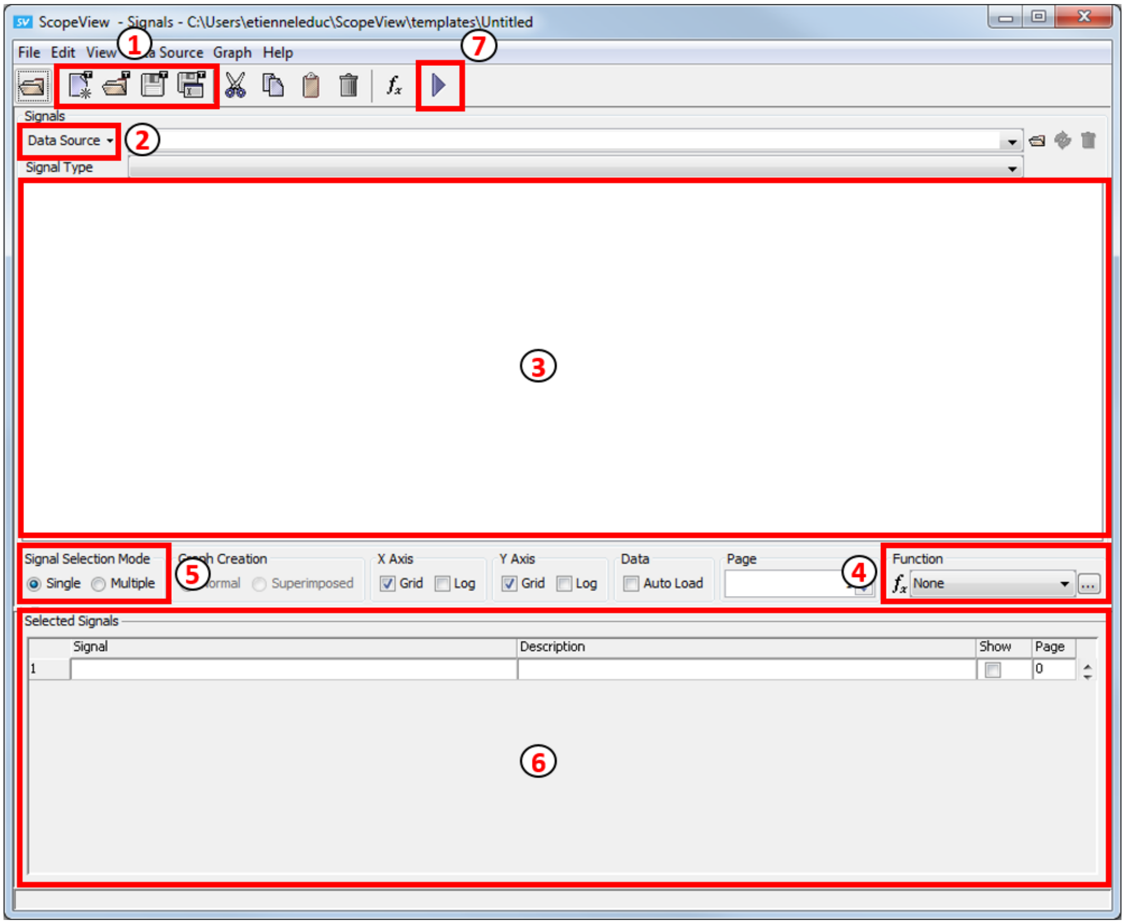


Figure 58: HyperView tools

8. ANNEX C: OVERVIEW OF SCOPEVIEW

This is the visualization software of the suite and offers various result analysis options. There are two main windows in ScopeView:

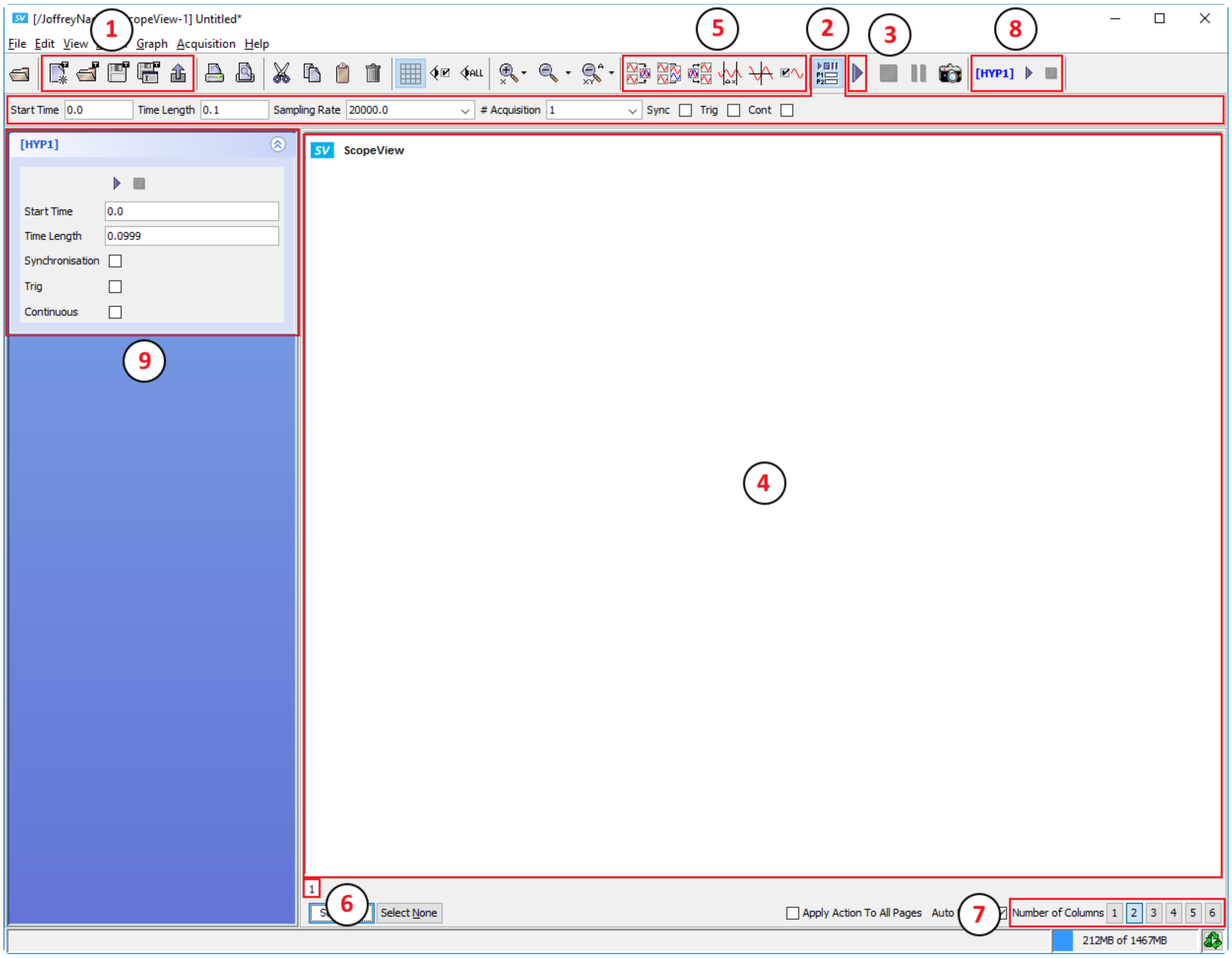
SIGNAL SELECTION WINDOW



1. **New template / Open template / Save template / Save template as**
2. **Data Source:** Where signal data is located; this could be a running simulation or previously exported file.
3. **Signal list:** Displays the list of all signals. Once a source has been selected, all signals appear here, grouped by category (e.g. Breaker, Line). Categories can be selected right below the Signal Source; it is the Signal Type option.
4. **Functions:** There are 96 different mathematical functions available to analyze data ranging from trigonometrical functions to harmonic and spectral analysis as well as statistics.
5. **Signal Selection Mo..:** By selecting either **Single** or **Multiple**, signals can be selected either one by one or in groups by using the CTRL or Shift keyboard shortcuts.
6. **Selected signals:** All functions and signals chosen appear in this list. The signal name and the page on which it is to be displayed can be modified.
7. **Start acquisition.**

SIGNAL DISPLAY WINDOW

This window is the scope where mathematical analysis and signals are monitored.



1. **New template / Open template / Save template / Save template as / Export data.**
2. **Acquisition settings:** Clicking the button opens a menu. Those settings are described in detail in step 2.4.3.
3. **Start acquisition:** Clicking this button starts an acquisition.
4. **ScopeView:** Where the graphical representation is displayed.
5. **Display tools:** The user can zoom in or out, select one or multiple signals to display, overlay the signals, use a cursor to follow the acquired values, calculate the slope and much more.
6. **Pages**: Display various graphs on multiple pages.
7. **Columns**: Specifies the number of columns to display. For a given page, signals can be displayed on up to 6 columns. Depending on the type of signal and screen resolution, typically no more than 3 columns is suggested.
8. **Start source acquisition:** Clicking this button starts an acquisition for the source written on the left side. Each new source is added to the right side.
9. **Data source options:** Various options depending on the data source type. These settings are described in details in step 2.4.3.

CONTACT

OPAL-RT Corporate Headquarters

1751 Richardson, Suite 2525

Montréal, Québec, Canada

H3K 1G6

Tel.: 514-935-2323

Toll free: 1-877-935-2323

Technical Services

www.opal-rt.com/support

NOTE:

While every effort has been made to ensure accuracy in this publication, no responsibility can be accepted for errors or omissions. Data may change, as well as legislation, and you are strongly advised to obtain copies of the most recently issued regulations, standards, and guidelines.

This publication is not intended to form the basis of a contract.

OPAL-RT

22/10/2018