

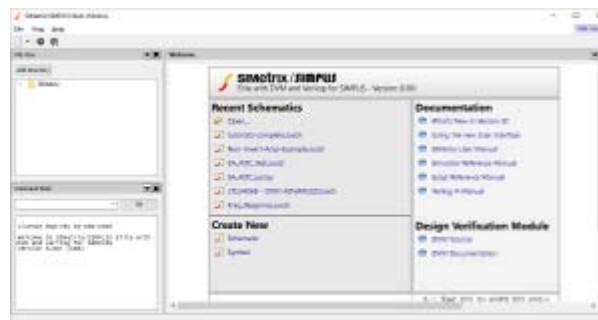
Tutorials

Introductory Tutorials

These tutorials are designed to give new users a basic understanding of how to use SIMetrix and SIMetrix/SIMPLIS.

Tutorial 1: Getting Started

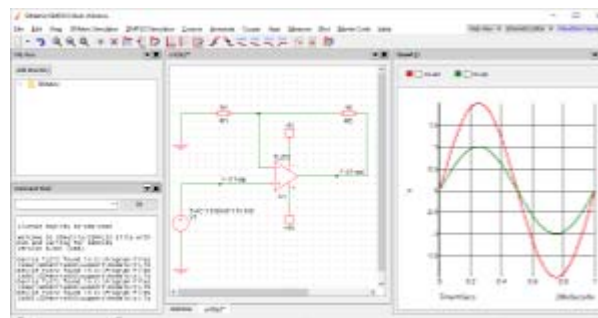
Guides you through getting started with the application and learning about the how the fundamentals of the user interface operates.



[Read Tutorial](#)

Tutorial 2: Creating a Schematic and Running a Simulation

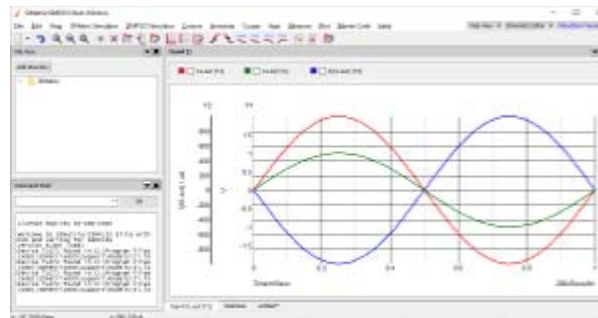
Leads you through the creation and simulation of a simple analog circuit.



[Read Tutorial](#)

Tutorial 3: Using the Waveform Viewer

Introduces the basics of using the waveform viewer.



[Read Tutorial](#)

SIMPLIS Tutorials

SIMPLIS provide a range of tutorials and webinars on using SIMetrix/SIMPLIS.

[View SIMetrix/SIMPLIS training at SIMPLIS.com](#) 



Tutorial 1: Getting Started

This tutorial will help familiarise yourself with the SIMetrix front-end.

To begin you will need to instal SIMetrix, SIMetrix/SIMPLIS or our free demonstration program SIMetrix/SIMPLIS Elements. This tutorial is designed for version 8 and beyond of SIMetrix and SIMetrix/SIMPLIS, and all versions of SIMetrix/SIMPLIS Elements.

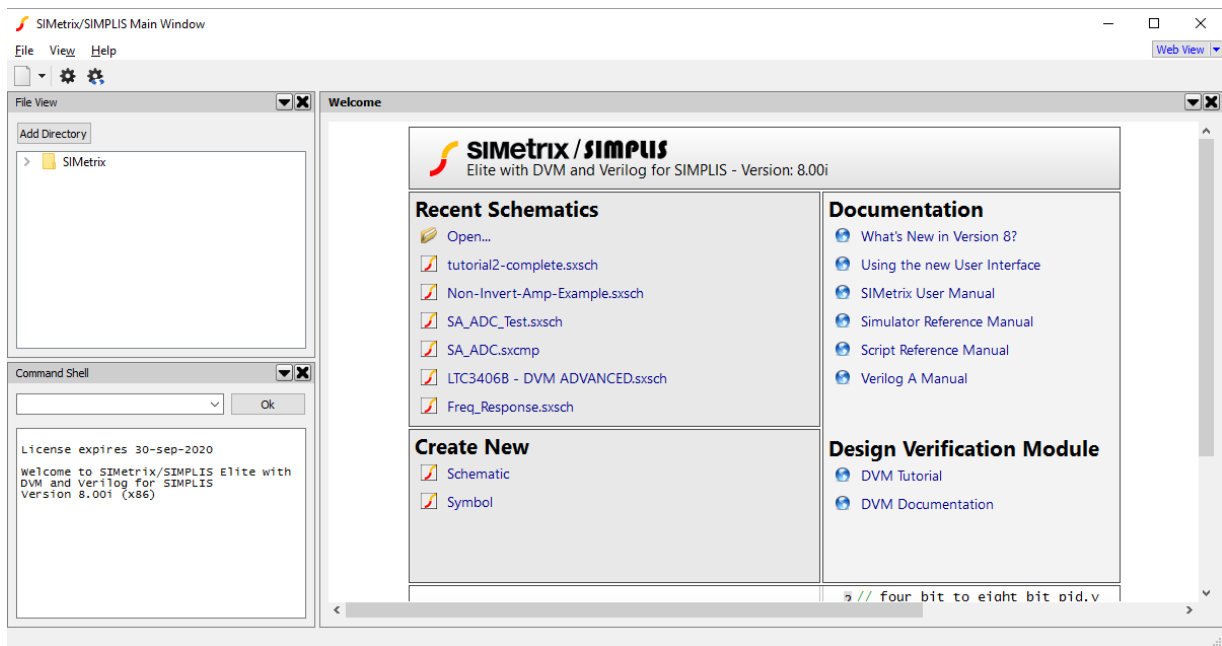
[Download SIMetrix and SIMetrix/SIMPLIS Full Version](#)

[Download SIMetrix/SIMPLIS Elements](#)

Learning the Interface

When you start the program, a single window will appear you will be presented with the welcome page, similar to the one below. Depending on your product and version this welcome page will appear differently. From this page you can perform common first actions, such as opening recent files, viewing the documentation and creating new schematics.

The default settings will also provide panels showing the File Viewer and Command Shell (the Command Shell is not available in SIMetrix/SIMPLIS Elements). The File Viewer displays the content of the SIMetrix directory in your Documents folder and can be used to quickly access all associated files such as schematics and scripts. The Command Shell allows for advanced interaction with the application, giving access to the scripting system.



View Panels

All panels within SIMetrix and SIMetrix/SIMPLIS can be relocated, resized, moved to another window, or closed completely. Note, there are some restrictions to these actions in SIMetrix/SIMPLIS Elements.

To move panels around, simply click and drag the darker grey title bar at the top of each panel. A blue outline will show you where the panels can be placed. At the moment the placement options may be fairly limited as we have only a small number of panels visible. When working with a larger number of schematics or waveforms, this behaviour becomes more useful, so you can view content side by side, or in any layout you require.

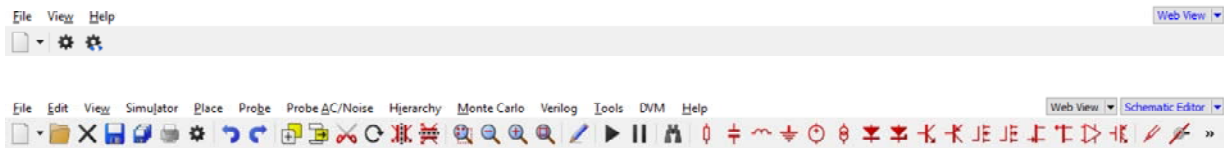
Each panel has its own context menu that provides options to move or close the panel. This context menu can be opened by pressing the down arrow button on the menu, as shown below, or by right clicking on the menu.



Panels are split into two categories: *System Panels* and *Workspace Panels*. System Panels provide methods to operate the program, such as the file viewer and command shell. Workspace Panels provide methods of using the program to create content and review designs, for instance with the schematic editor and waveform viewer. In using the panels, the only differences that occur are that System Panels will always appear on the outside edge of a window, whilst Workspace Panels will appear in the center or main part of the window. Additionally, Workspace Panels have a menu bar and toolbar associated to them.

Menu Bar and Toolbar

The menu bar and toolbar that appears at the top of the window is dependent on the Workspace Panel that is currently selected. Below are examples of the menu bar and toolbars that appear for the welcome page and a schematic.



You can tell what Workspace Panel is selected, by looking at the title bars of the panels in the window. Each title bar contains the name of the panel, in the selected Workspace Panel, this title will be in bold. You can swap between panels by either clicking within them or on the title bar. You will notice that the System Panels never have bold titles, as they cannot be selected for displaying a menu bar or toolbar.

A second way of determining the type of Workspace Panel the toolbar is being displayed for is to look at the buttons on the top right of the menu bar and toolbar block. As you can see in the picture below, one of the them is highlighted blue, this is the type of workspace panel that is selected. These buttons can also be used to quickly swap between workspace panels. Clicking on the button will display the last viewed panel of that type in this window. Pressing the arrow will bring up a menu allowing you to quickly move all panels of this type to a new window, or close all panels of this type.



Next Steps

In the following tutorial, we will create our first schematic and perform a simulation.

[Tutorial 2: Creating a Schematic and Running a Simulation](#)

© 2018 SIMetrix Technologies Ltd. All rights reserved.



Tutorial 2: Creating a Schematic and Running a Simulation

This tutorial will show you how to create a simple schematic and perform a transient analysis upon it. It is intended for beginner users.

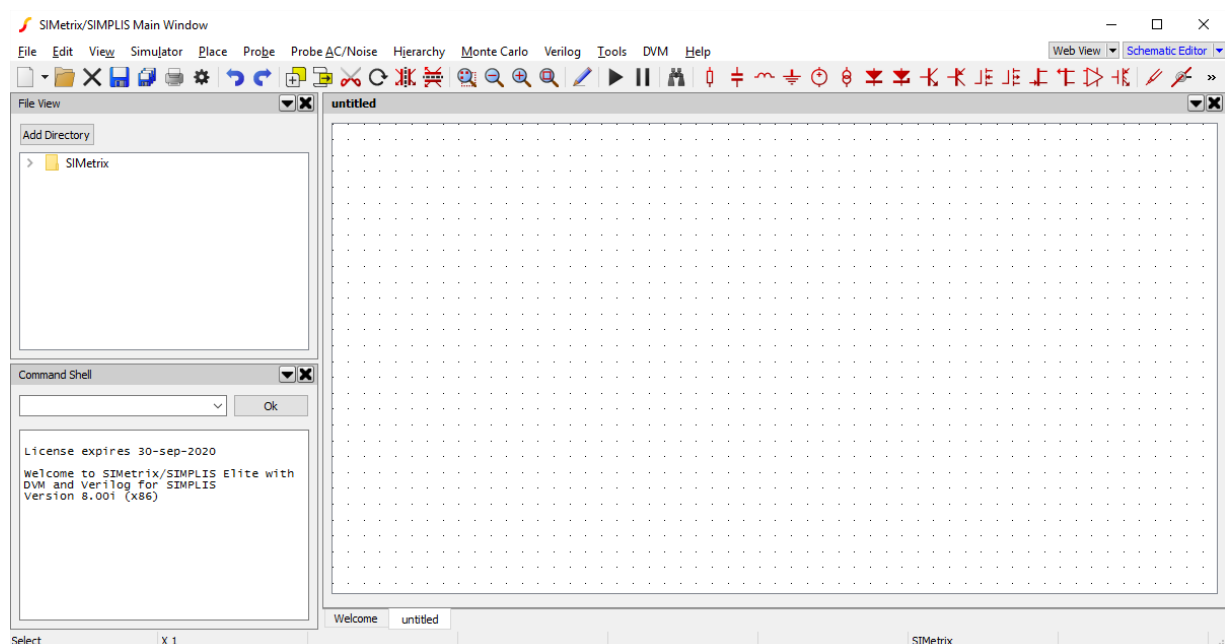
This assumes you have SIMetrix, SIMetrix/SIMPLIS or our free demonstration program SIMetrix/SIMPLIS Elements installed. This tutorial is designed for version 8 and beyond of SIMetrix and SIMetrix/SIMPLIS, and all versions of SIMetrix/SIMPLIS Elements.

[Download SIMetrix and SIMetrix/SIMPLIS Full Version](#)

[Download SIMetrix/SIMPLIS Elements](#)


Creating a Schematic

From any window, go to the menu at the top of the window and select: **File > New > SIMetrix Schematic**. The window will display a new schematic panel, similar to that below:





Placing Components



For this tutorial, we will create a simple non-inverting amplifier. To start, we'll place the components that we require: a voltage source as input, two resistors, an op-amp and ground.

To place the first component, press the *Place Voltage Source* button  on the toolbar. Now when you move the cursor back onto the schematic, the outline of the voltage source component will appear. Left click on the schematic to place the component at that position.

If you move the mouse, you will notice that the outline of the voltage source is still attached to the cursor. This would allow you to place another instance of that component. In this tutorial we do not want a second voltage source, so right click on the schematic to end component placement. The cursor will now return to normal.

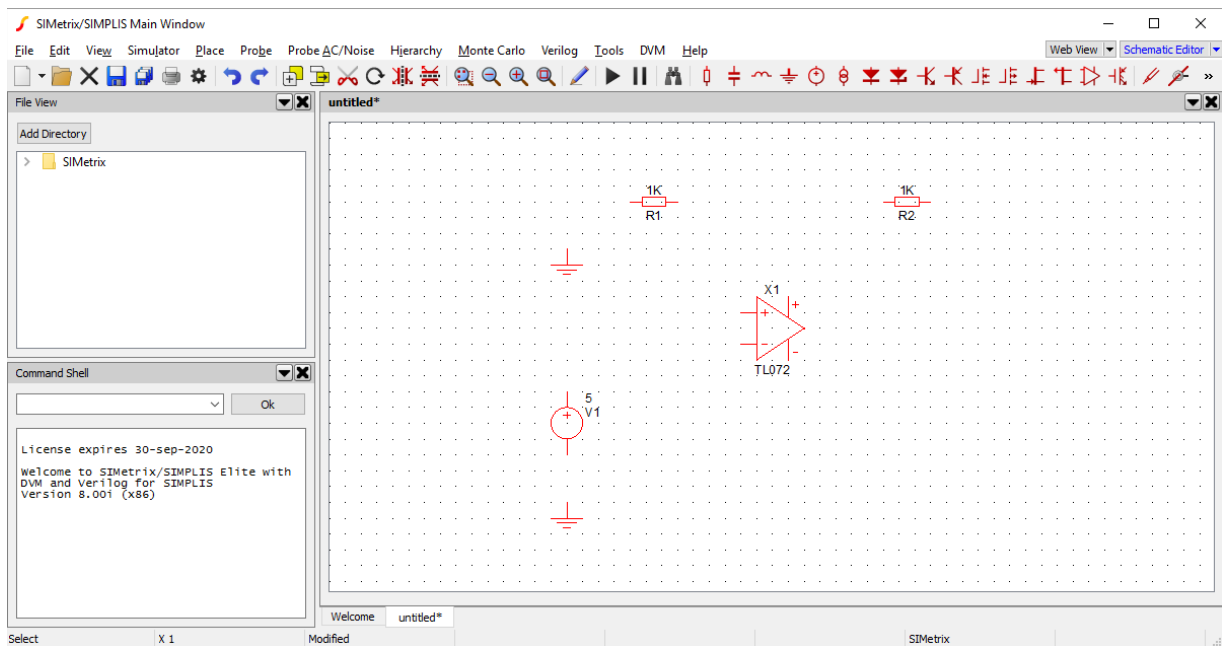
Next we will place two resistors. Like last time, press the *Resistor* button , but unlike last time, before placing the element we are going to rotate it 90 degrees. To rotate the element, move the cursor back onto the schematic so the resistor outline appears, then press F5. The resistor should rotate 90 degrees clockwise. Now left click and place a resistor, then move the mouse to another part of the schematic and left click to place a second resistor. When both resistors are placed, right click to end component placement.

Tip: All schematic elements can be rotated, either by pressing the default rotate shortcut of F5, or by pressing the rotate button  on the toolbar.

Finally add two ground components  and an op-amp component  using the same method as we have for the voltage source and resistors. At the moment we are not worried about their position and the components can be placed anywhere on the schematic.

After components have been placed on the schematic they can be moved by left clicking on the component, then dragging it to the new location. Multiple components can be selected by left clicking on the first component, then press and hold the *control* key and click on the other components to move together. When all components have been selected, release the *control* key and click-on then drag any of the selected components. Selected components by default have a blue colour.

Arrange the components on the schematic so that they are laid out roughly in the same way as shown below.



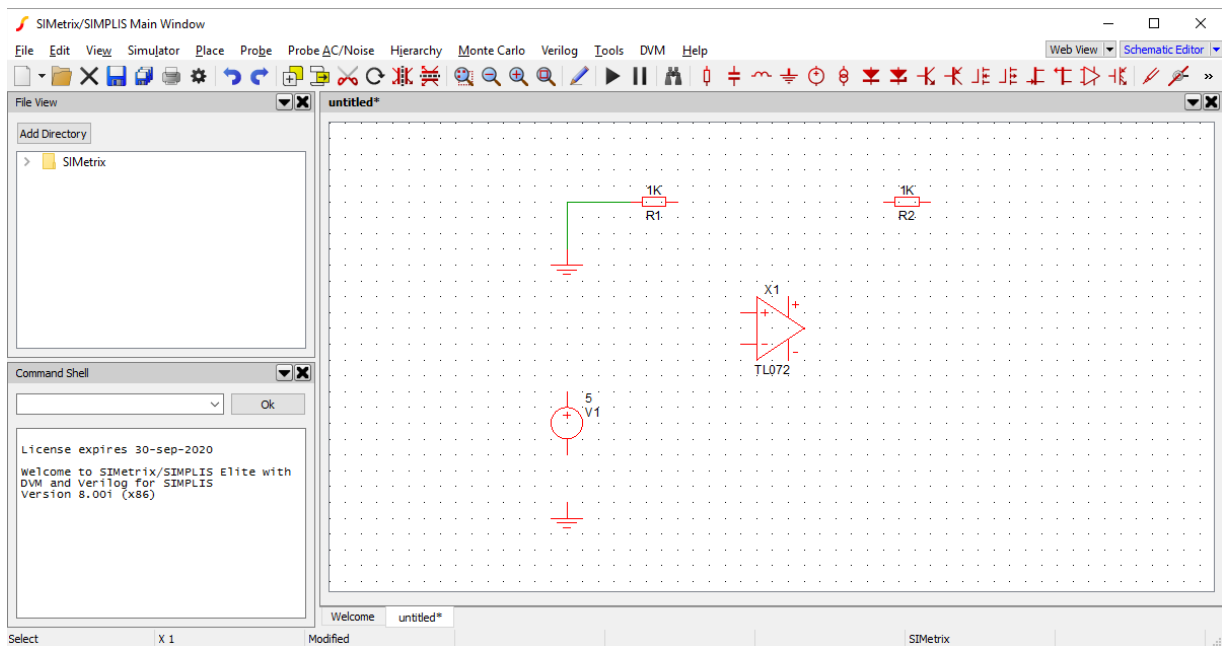
Placing Wires

Next we will add wires to connect the components. SIMetrix provides a Smart Wiring facility that makes placing wires faster and easier, by simply specifying the points that the wire should attach to.


To begin, move the cursor to the left-hand pin on the left most resistor (R1 in our example picture above). The cursor will change from a cross-hair to a pencil. Left-click on the pin and will notice when you move the cursor that a straight line will be drawn from the pin to the cursor.

Now move the cursor to the pin on the nearest ground component and click on the pin. A wire will be added that automatically connects the two pins. Your schematic should now look like the one below.

Tip: The Smart Wiring capability can handle complex layouts. The wire that is automatically generated should always avoid going through placed components and generally attempt to provide a sensible route.

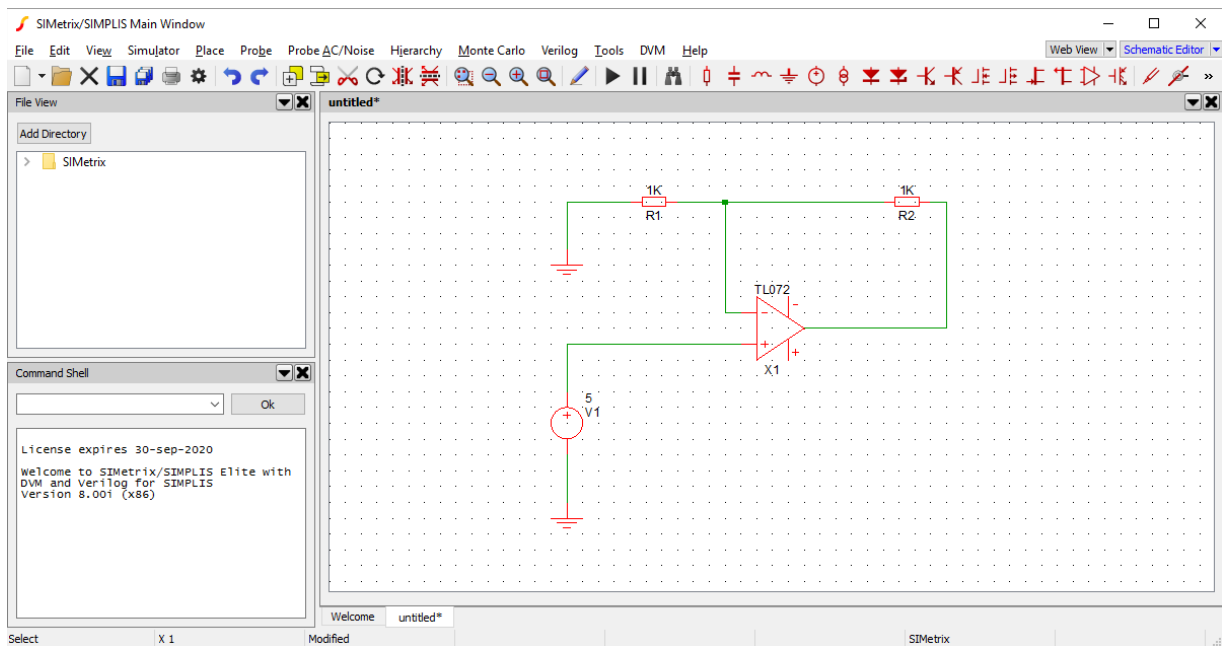


Next we will connect the positive output of the voltage source to the positive input of the op-amp. We could do this as before, however we would end up with wires crossing near the op-amp when we later connect the left resistor to the negative input. Ideally we then want the op-amp to be flipped horizontally so that the input pins are the opposite way around.

To flip the component, left-click on the op-amp (ensuring it is the only component selected) and press the *Flip* button  on the toolbar. The component should now flip over with the positive and negative sides reversed. The component will also appear a little higher than it was before the flip, so move it back down and then connect the voltage source positive output and op-amp positive input.

Tip: The default shortcut key for flip is Shift-F6. You can also mirror an element by pressing the mirror button or shortcut F6. All schematic elements can be freely rotated, flipped and mirrored, in any combination.

Continue to wire all connections, so that the non-inverting amplifier looks like the image below. If you need to move wires, you can click and drag them as you do for components. Attached wires will attempt to resize with the movement.

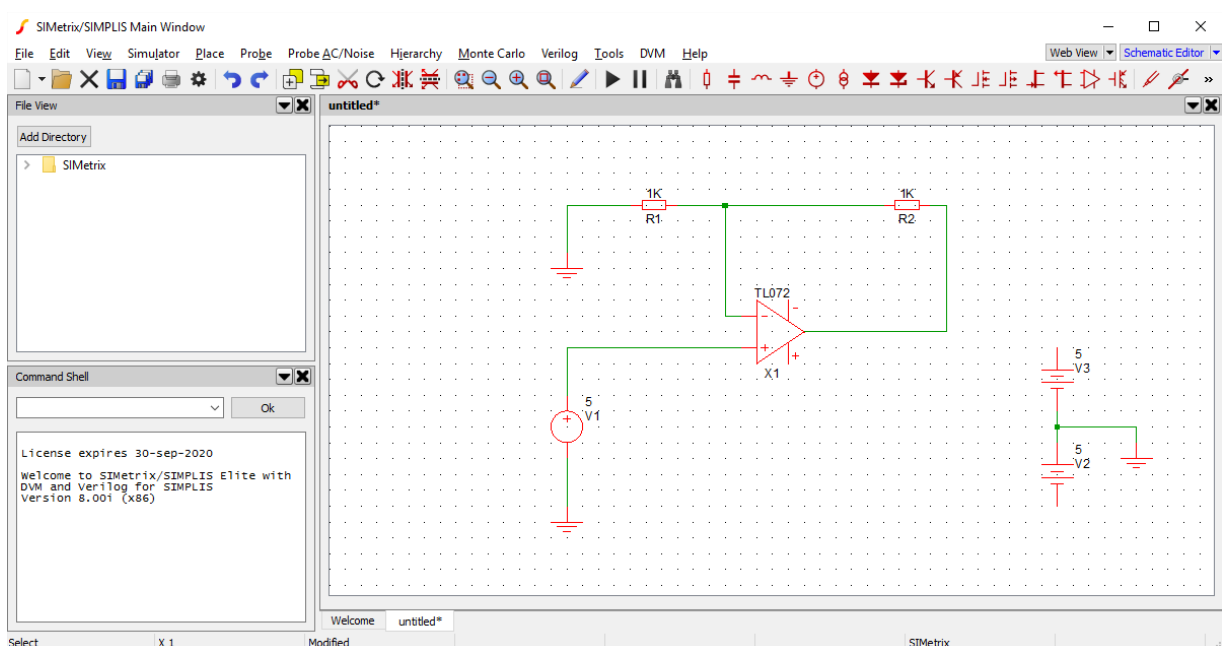


Connectors

Next we will add our supply rail to the op-amp. To keep our design clean, instead of routing wires from the power supply directly to the op-amp, we will use connectors.

We will start by building a power supply that runs between + and - 5V. To create the power supply, use the menu **Place > Voltage Sources > Power Supply** then place two power supply components vertically next to each other.

Next add a ground component and connect all three together as shown in the picture below.



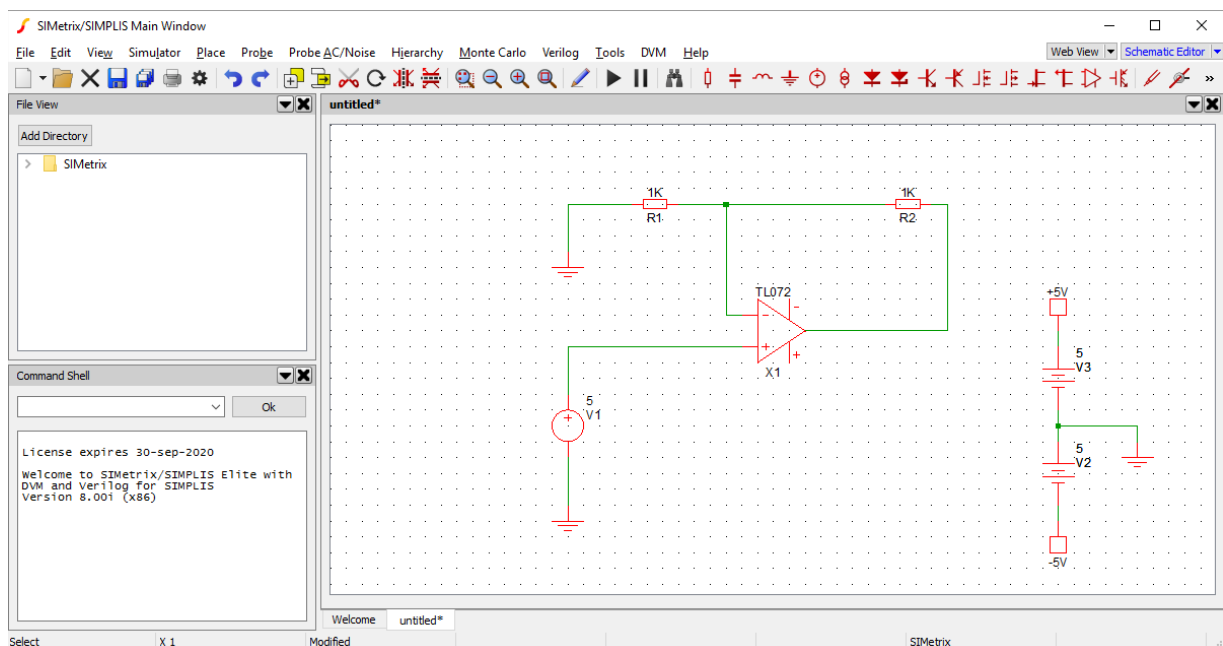
By default, the power supplies generate 5V, so we now have our +5V and -5V outputs.


Next we will add two connectors, one of the +5V source and the other for -5V. Connectors with the same name are all directly attached, as such the ground component is a commonly used case of the connector component, which by default has it's own symbol.

To add a connector, go to the menu **Place > Connectors > Terminal**, then place connectors above and below the power supply, rotating the connectors as required.

By default all connectors connect to VOUT. We will need to change the name of the connectors so we have our +5V and -5V connections.

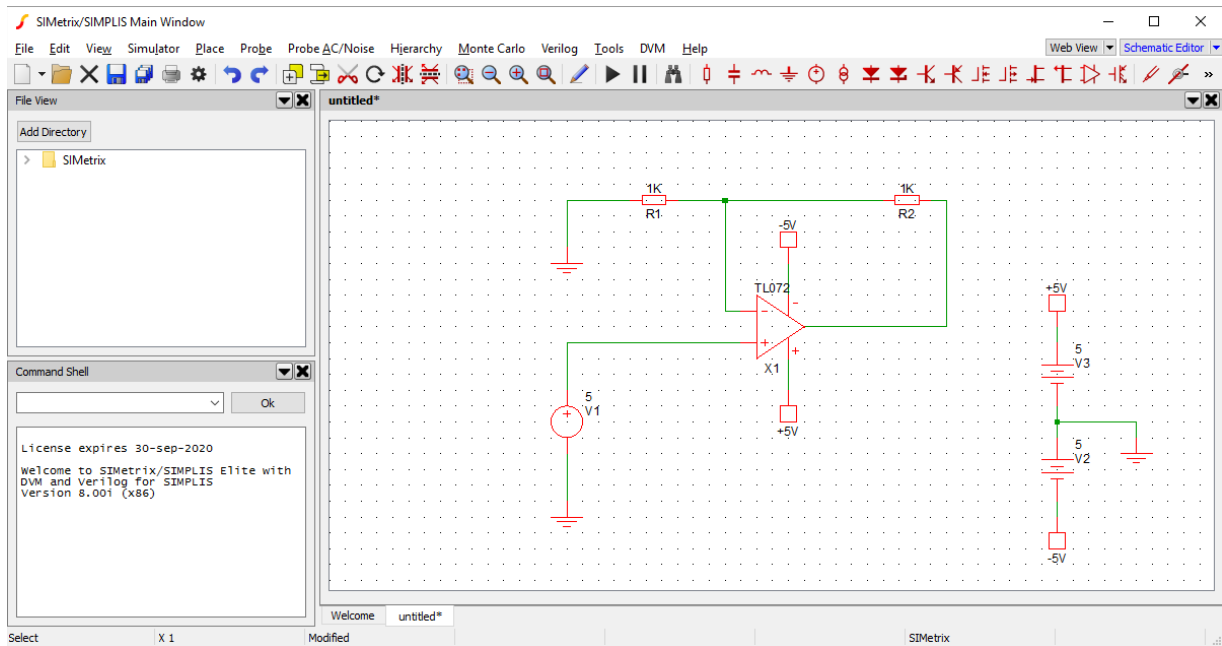
To change the name of the connector, double click on one of the connector elements. A pop-up box will appear asking you to enter text, with a default value of VOUT. Change the text to +5V or -5V, depending on which connector you have double clicked, then press ok. You will notice the label displayed above or below the connector will have changed. Repeat this renaming step for the other connector. Your schematic should then look like the one below.



Now we need to add matching connectors to the supply rail of the op-amp. First select the +5V connector by clicking on it, then press the duplicate button  (or the default shortcut Control-D). The cursor will show the outline of a connector, move and rotate it as required to place it next to the +ve supply rail of the op-amp. Repeat this for the -ve supply and wire the new connectors to the supply rail pins on the op-amp.

Tip: The duplicate key is a quick method of performing copy and paste, both of which are supported within SIMetrix. After selecting any component element and performing Copy (Control-C), you can then paste the selected elements anywhere within the same schematic, a different schematic, or even as a picture to other programs.

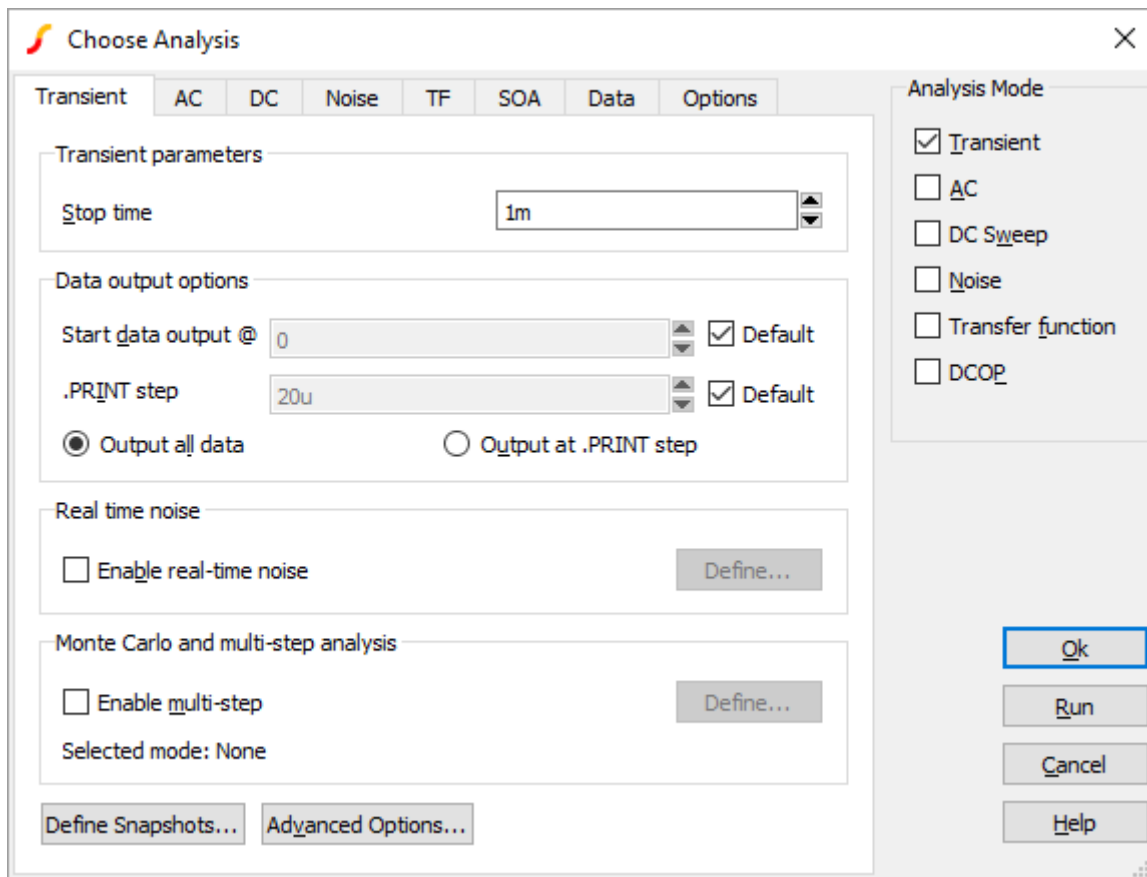
Your completed schematic should look like this.



Running a Simulation

With the schematic laid out, we can now run our first simulation. To simulate our schematic, first we must choose the type of analysis that we want to perform.

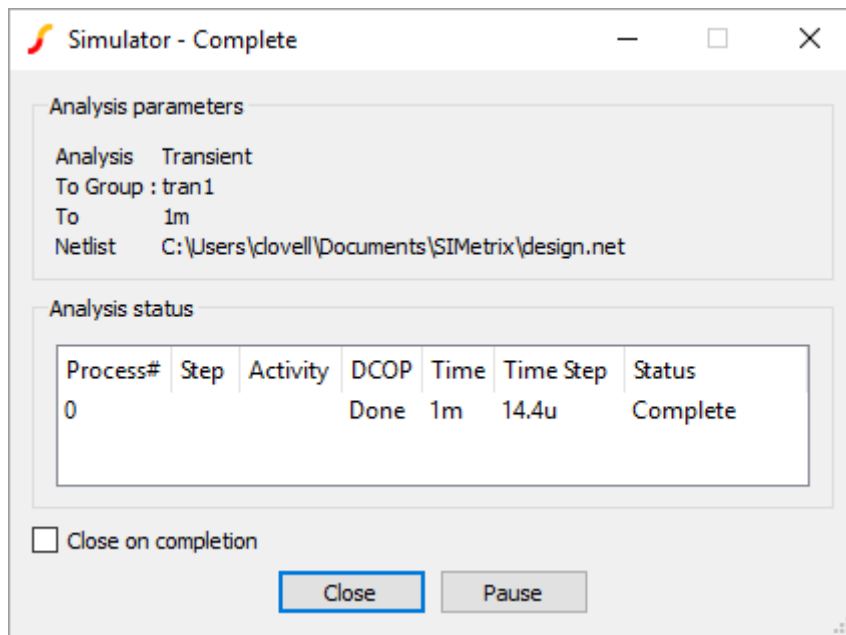
Go to the menu *Simulator > Choose Analysis....* The window below will pop-up.



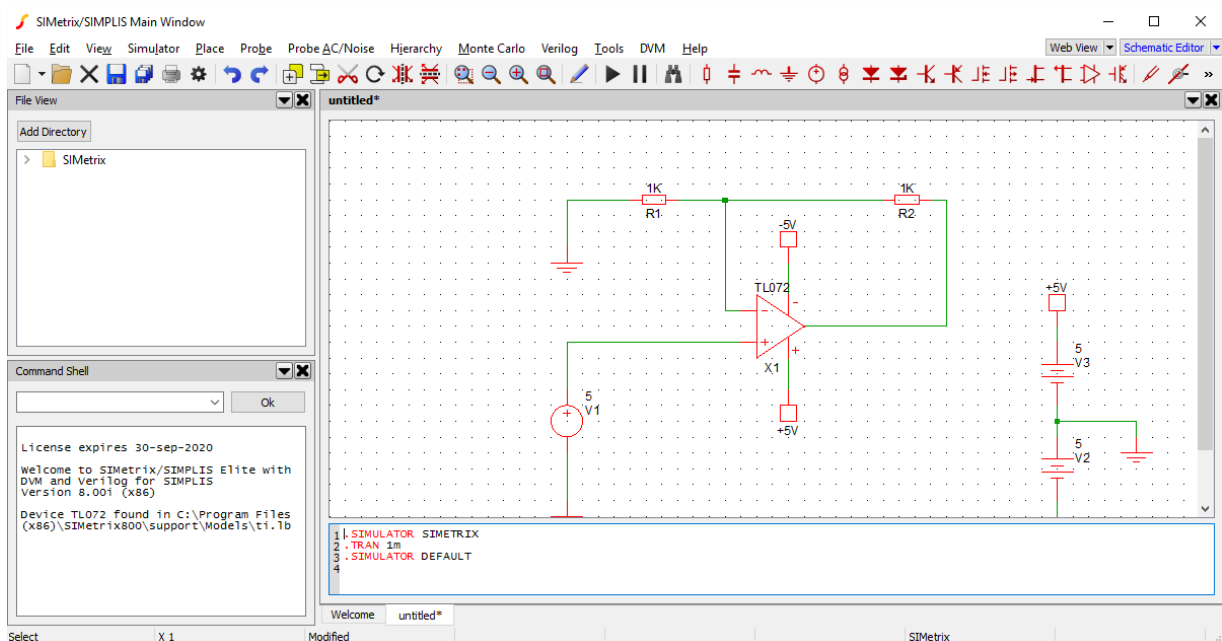
If the window you see looks different to the one displayed here and the title of the window has the word SIMPLIS in it, then you are using the wrong simulator. In this case, simply close the analysis box and select menu **Simulator > Switch to SIMetrix Mode**. Then re-open the Choose Analysis window and the one shown above should appear.

We want to run a transient simulation, so check the *Transient* box on the right hand side of the window, under *Analysis Mode*. When this is done, press the *Run* button in the bottom right of the window.

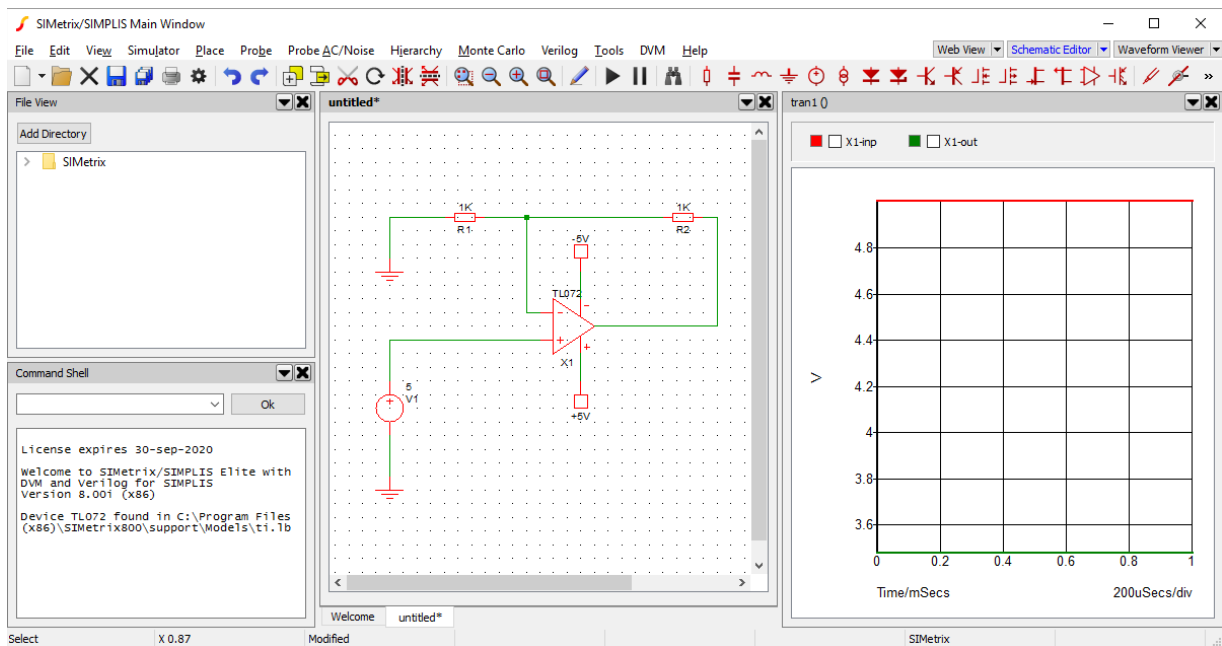
The Choose Analysis window will disappear and a Simulator window will open. If everything worked as expected, it should give a status of complete and look like the one below.



Visibly this dialog is the only immediate output of the simulation, based on how we have set up our circuit. A netlist has been generated, which we can view by clicking back on the schematic and then pressing F11. A text box will appear below the schematic, like the one shown below. Pressing F11 again will make text box disappear.

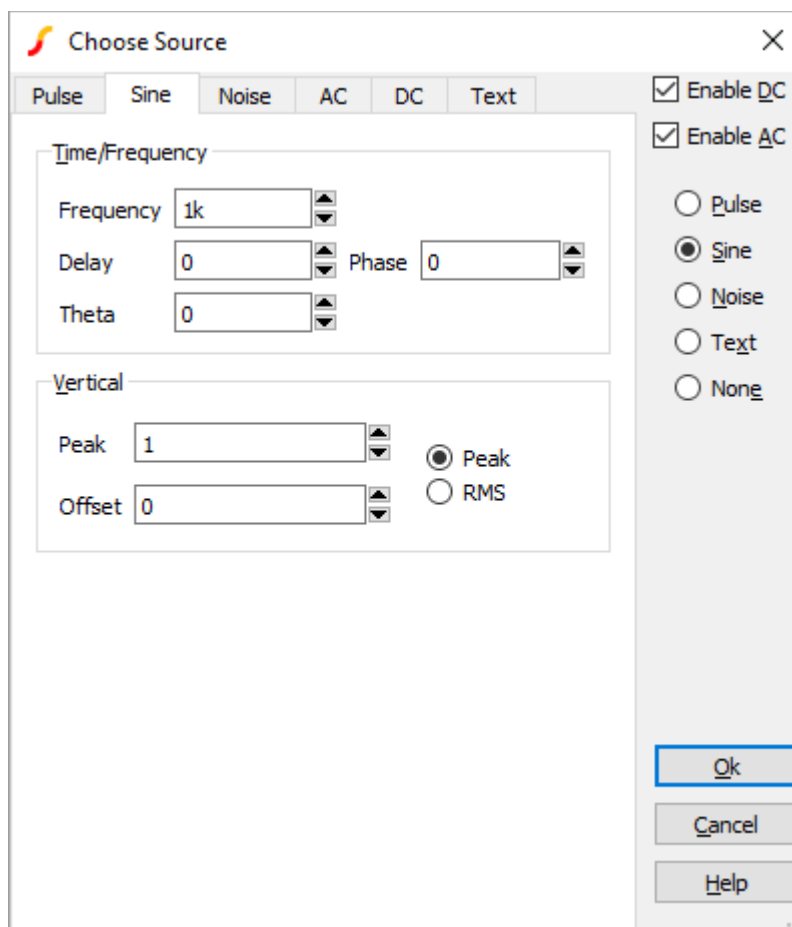


We can probe the voltage within the circuit by using the menu **Probe > Voltage....** The cursor will change to an icon of a voltage probe. You can then click on a wire to inspect the voltage, the waveform viewer will automatically appear in a new panel next to the schematic. In the following picture, the wire connecting the output of the voltage source to the op-amp and the wire connected to the output of the op-amp have been probed.



This is clearly a fairly boring simulation that has been run, as the input voltage has remained constant throughout. So we will need to adjust the parameters on the voltage source.

To adjust the properties, double click the voltage source component, the following window will pop-up (note the Pulse tab will be selected by default).




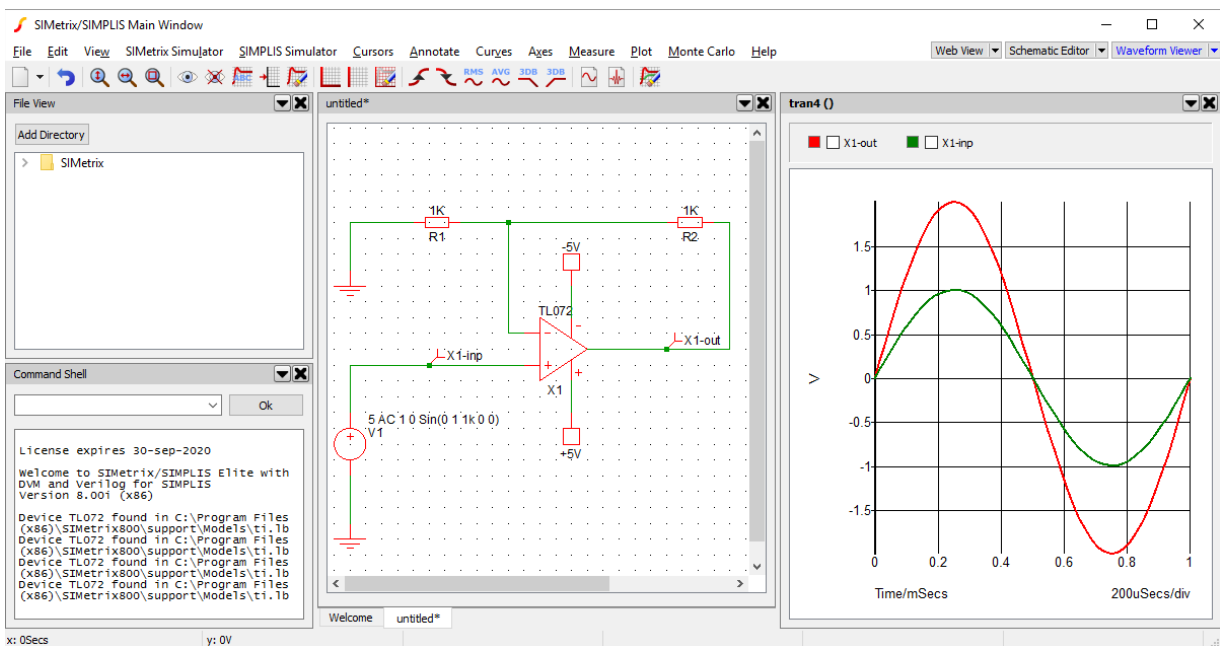
Tick the *Enable AC* check box and select the *Sine* option on the right hand side of the dialog, then press Ok. The label next to the voltage source will change to reflect the adjustments we have made to the component.

Before we re-run the simulation, we will make the process of checking the voltages a little quicker, by placing fixed probes to the schematic, instead of having to perform a manual probe after each simulation.

To place the fixed probes, go to menu **Place > Probe > Voltage Probe** (or use keyboard shortcut B), and place the probes like we did for the previous components, one on the wire connecting the voltage source and the op-amp and the other on the wire connected to the op-amp output.

Close the waveform viewer panel, so we get rid of our previous result, by pressing the cross in the title bar for the waveform panel.

Finally, re-run the simulation by pressing the run toolbar button  or by clicking on the schematic and pressing F9. The final output should look similar to that shown below.



Next Steps

Now that we have successfully run a simulation, we can advance on to learn more about the SIMetrix and SIMetrix/SIMPLIS program.

[Tutorial 3: Using the Waveform Viewer](#)

© 2018 SIMetrix Technologies Ltd. All rights reserved.

Tutorial 3: Using the Waveform Viewer

In this tutorial you will learn more about using the SIMetrix waveform viewer.

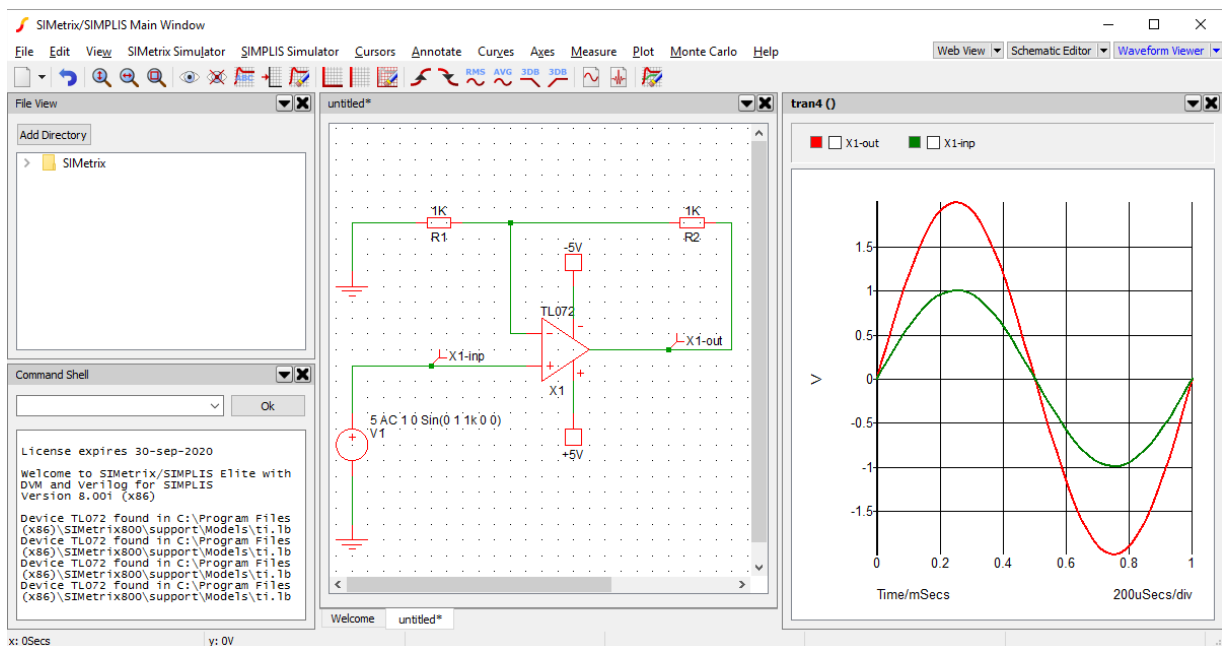
This tutorial is designed for version 8 and beyond of SIMetrix and SIMetrix/SIMPLIS, and all versions of SIMetrix/SIMPLIS Elements.

We continue on from the end of [Tutorial 2: Creating a Schematic and Running a Simulation](#), which you should complete first if you are unfamiliar with running simulations. The schematic as completed from the end of tutorial 2 is provided.

[Download Tutorial 3 Schematic](#)

Getting Started

To begin, we will need the waveform output from the non-inverting amplifier that we developed in the previous tutorial. If you are continuing on from the previous tutorial, then you should already have the output and your SIMetrix workspace should look similar to the one below. If you are starting from this tutorial, download the schematic from above and run a simulation.

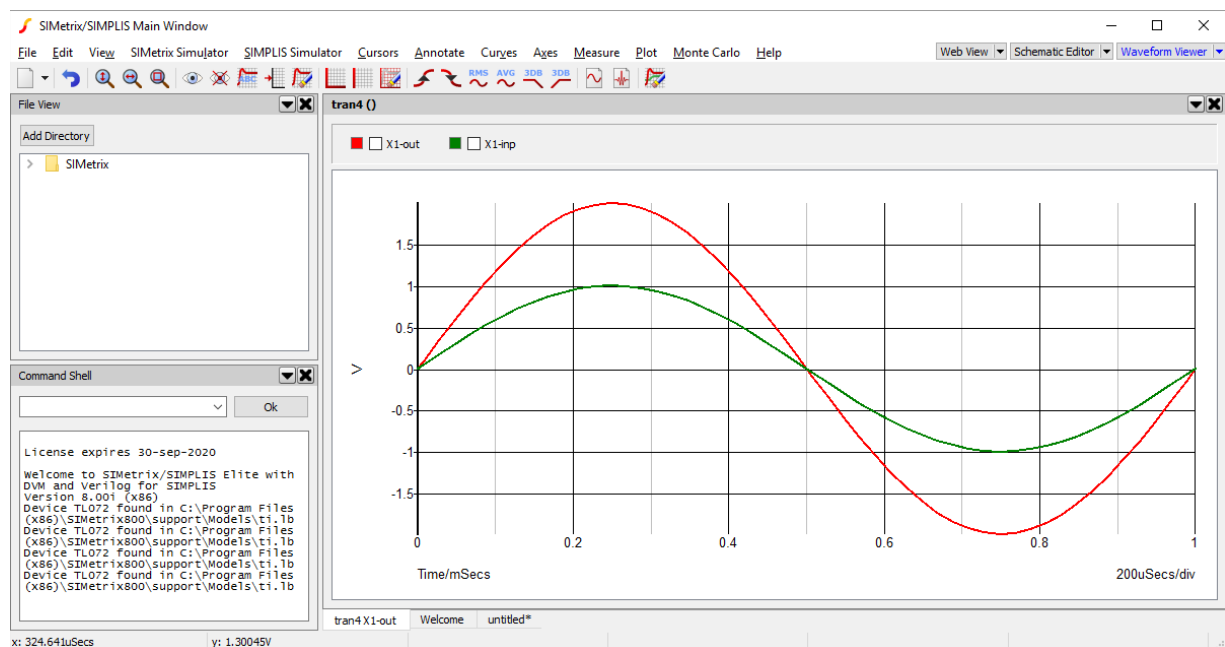


So that we can focus solely on the waveform, we will move the waveform panel so that it takes up the full workspace panel area. To do this, click and drag on the titlebar for the waveform (in our example pictures this has the title tran4()), but

your title may appear with a different number and include a file path if you have saved your schematic previously). When you drag the panel a blue highlighted region will appear over the main workspace panel area, release the mouse in the blue highlighted region and the window will move there.


If when you release the waveform panel it opens up in a new window, open up the menu context panel by clicking on the down arrow in the panel title bar and choose the menu option **Dock to... > SIMetrix Main Window** (note the window name may appear differently depending on whether you are using SIMetrix, SIMetrix/SIMPLIS or SIMetrix/SIMPLIS Elements, but the name Main Window will remain). The waveform should now return to the main window and be docked in the position that we want it.

Your workspace should now look like the following:




Moving and Zooming

We will begin with a common requirement on viewing a waveform, adjusting what is being displayed on the screen by moving the view region and zooming in and out. Throughout we will use the default shortcut keys, but all operations can also be performed by using the appropriate menu option in the View menu.

Tip: It is easy when starting out to end up in a situation where the waveform disappears from the screen. The *Fit Window* button  (shortcut key Home) will return the display so that the plotted waveforms all fit fully within the visible region.

To begin, we can move the areas we are viewing in the waveforms up, down, left and right, by using the keyboard cursor keys. We can zoom out by pressing F12 and zoom back in again by pressing Shift-F12. All of these movements are made by a fixed constant amount.

We can view a specific area of the waveform by clicking and dragging around the area that we want to view. When we click and drag on the waveform, a box will appear showing the region that will be displayed after releasing. This will adjust the movement and zoom automatically to show just that region. Remember to go back to the full waveform view, use the shortcut key Home.

To go back to the previous zoom or movement setting, you can use the *Undo Zoom* button  (shortcut Control-Z).

Measurements

The waveform viewer can provide certain measurements for plotted curves, such as RMS, frequency and minimum/maximum values, as appropriate.

First we need to select the curves that we want to receive the measurements for. Curves can be selected via the legend box that is above the waveform plot. The legend box, as shown below, displays all plotted curves, showing their colour, name and any measurements the have been made.

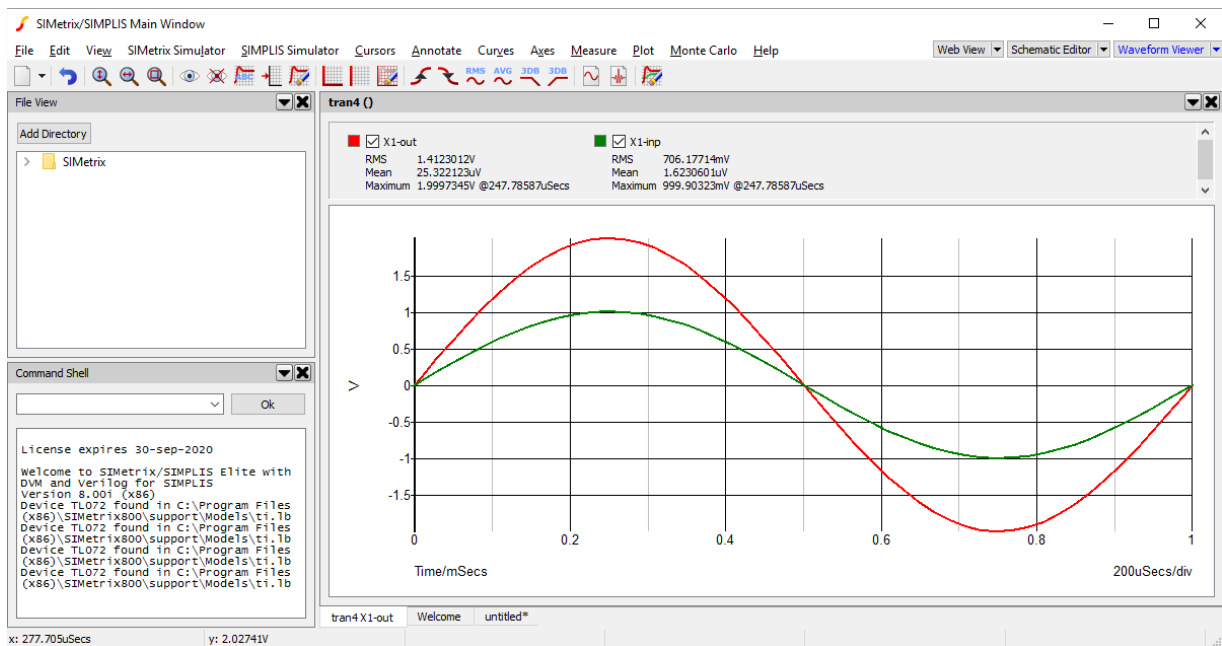


We are going to select both the X1-out and X1-inp curves. To do this, go to the legend box and click on the check boxes next to the names X1-out and X1-inp.

Next we will request the measurements for RMS, Mean and Maximum for these two curves. To do this, go to the **Measure** menu and press the menu items for each of these measurements. The measurements will appear in the legend box, underneath the check box and name of the selected curves.

Tip: If you choose a measurement that is not applicable for the curve type selected, an error message will display in the Command Shell telling you why the measurement could not be provided.

Your workspace should now look similar to the one below, with the measurements being displayed for the X1-out and X1-inp curves.

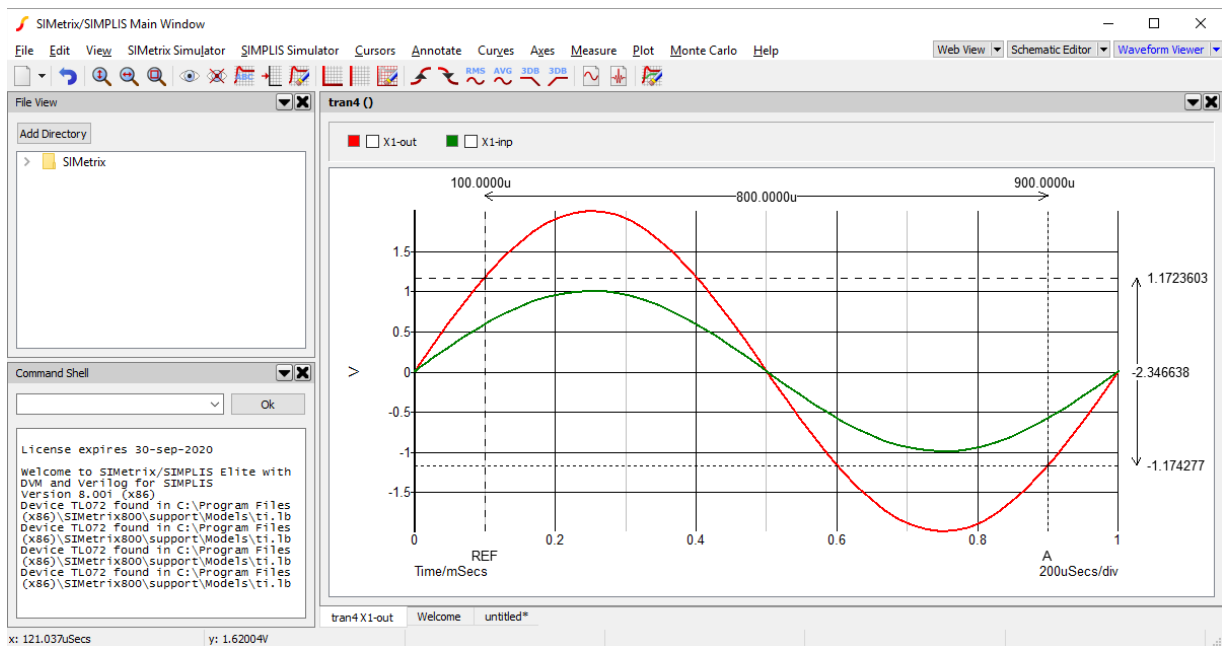


To remove the measurements, use menu **Measure > Delete All Measurements**. Alternately to remove specific measurements, make sure only a single curve is selected then use menu **Measure > Delete Measurements...** and choose the measurements to remove.

Cursors

Cursors provide a method of measuring the interval or difference between two points within a waveform window. They follow specific curves and allow markers to be positioned at specific points along a curve, with the difference between two markers being displayed.

To turn on the cursors, use the menu **Cursors > Toggle On/Off**. A pop-up box may appear, giving you brief instructions on cursor usage, which can be closed. Your workspace should now look similar to the one below.



Initially we are provided with two cursors, labelled *REF* and *A*, these labels can be found on the x-axis. The cursors are positioned at two points on a curve, the points are by default placed at the intersection of the vertical line going up from the label name and the horizontal line with the same line-style. The cursors are visually shown with a horizontal-vertical cross at this intersection.

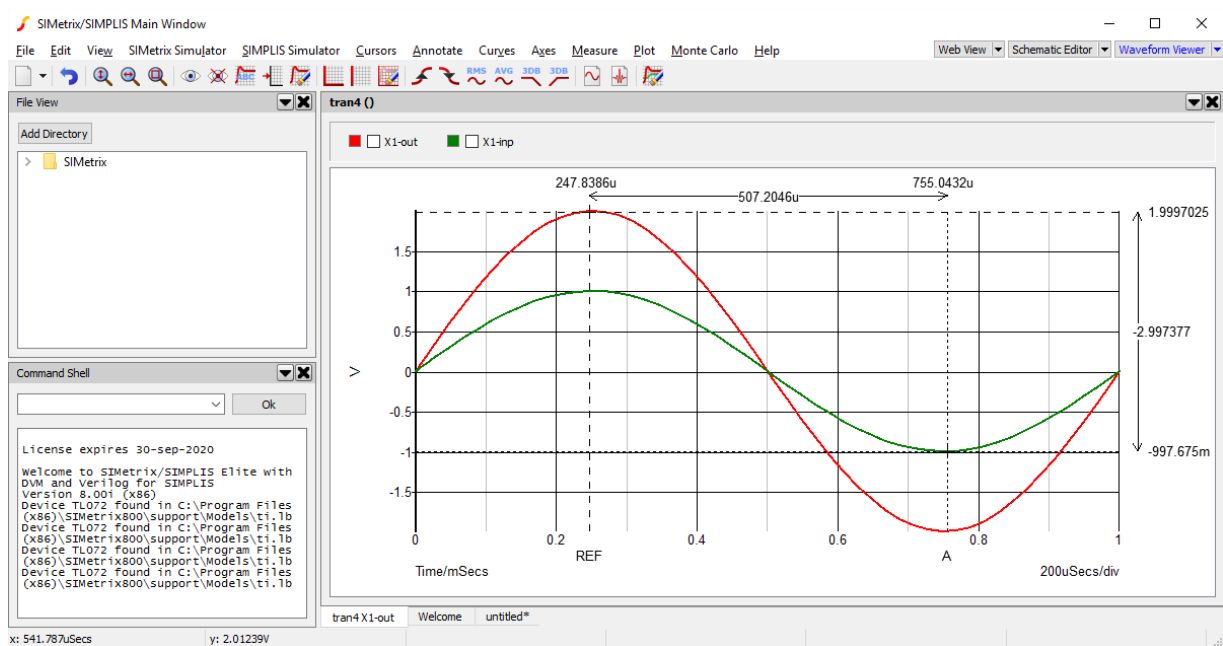
The x and y values of the cursor positions are shown above the waveform for the x values and to the right of the waveform for the y values. The difference between the x and y cursor values are also shown in the same regions.

We can move the cursors along the x-axis, by moving the mouse pointer to one of the horizontal lines, where the mouse pointer will change to a horizontal resize icon, then left clicking and dragging the cursor, moving it left or right. The cursor y value will follow the curve it is attached to. Similarly we can move along the y-axis by dragging one of the horizontal cursor marker lines. This movement will be locked to the minimum and maximum values of the curve in the direction the cursor is being moved, or the maximum of the visible axis region, whichever is less. Dragging the cursor fully up, down, left or right, will move the cursor to the maximum extent on the curve, which may mean that the cursor appears to jump to another area of the curve if that area is the maximum or minimum as required.

Currently the cursors are both attached to the same curve. Now lets say we want to measure the voltage difference between the maximum of X1-out and the minimum of X1-inp. To do this we will need to move one of the cursors to the other curve. For this example we are assuming both cursors are currently attached to X1-out and we will move the A cursor to X1-inp. If your cursors are currently attached to a different curve, use the instructions below but ensure that REF is attached to X1-out and A is attached to X1-inp.

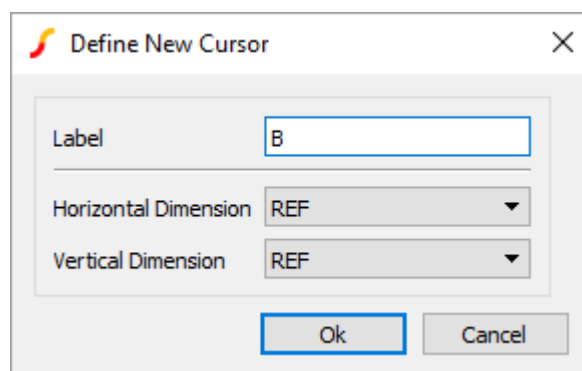
Move cursor A by placing the mouse pointer over the intersection point for cursor A. The mouse pointer should change to an icon showing arrows going up, down, left and right. Click and drag with the left mouse button when this icon is shown and move the mouse pointer to a point along the X1-inp curve and release the mouse button. The cursor should now snap to the X1-inp curve.

Next, to select the maximum of curve X1-out, move the REF cursor fully upwards by dragging the horizontal cursor marker. Depending on product version, you may need to make minor adjustments to the x-position to get a maximum value. Similarly, move the A cursor fully down to get the minimum of the X1-inp curve. Your workspace should look similar to the following, showing a voltage difference of 3V.



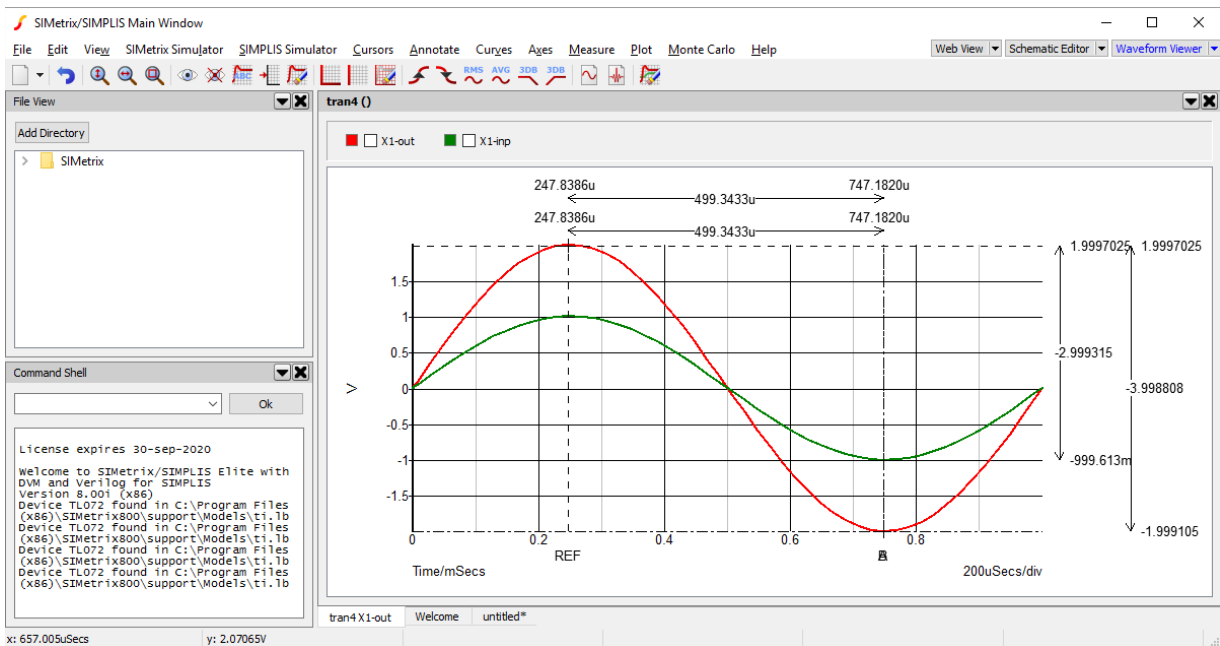
Next we'll also measure the minimum of X1-out and compare this to the maximum of X1-out. Instead of moving our cursor A, we will add a new cursor, so then we can see the two difference measurements at the same time.

Use menu **Cursors > Add Additional Cursor....** A pop-up will display, shown below, asking for the definition of the new cursor.



The cursor is defined with a label and cursor it is attached to when displaying the measurements on the horizontal and vertical dimension. We want to use the default settings, which will give us a new cursor named B that will measure the difference between itself and the REF cursor. Press Ok to add the cursor.

The new cursor will now appear on the waveform, you will most likely find it at the far left of the visible waveform area. As we did earlier, move this new cursor so that is at the lowest point of X1-out, you may need to change the curve that the cursor is attached to. Your workspace should now look like the one below.



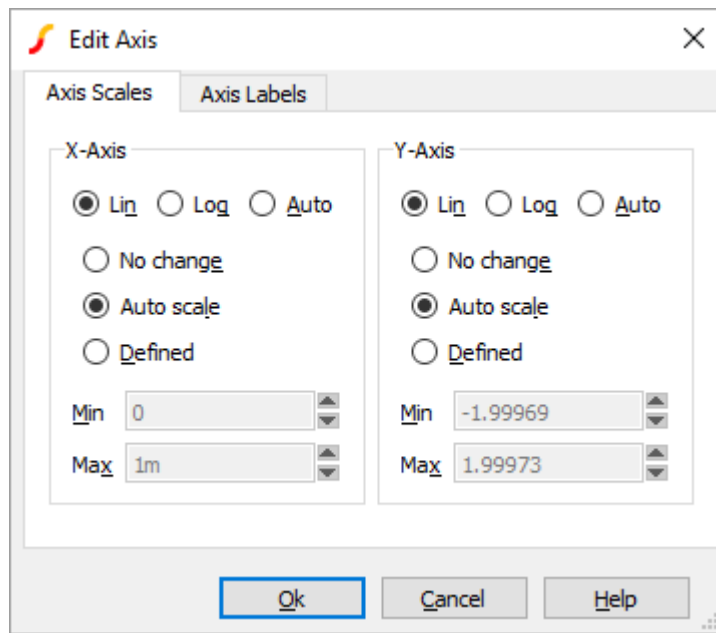
When multiple cursors are placed on the waveform, multiple measurement indicators are shown alongside the waveform. In this case it allows us to read-off the voltage differences between the two curves as 3V and 4V respectively.

To turn cursors off you can use menu **Cursors > Toggle On/Off**, alternately to remove cursors that were added in addition to the two first cursors you can use the menu **Cursors > Remove Additional Cursors...** and choose the additional cursor to remove.

Axes

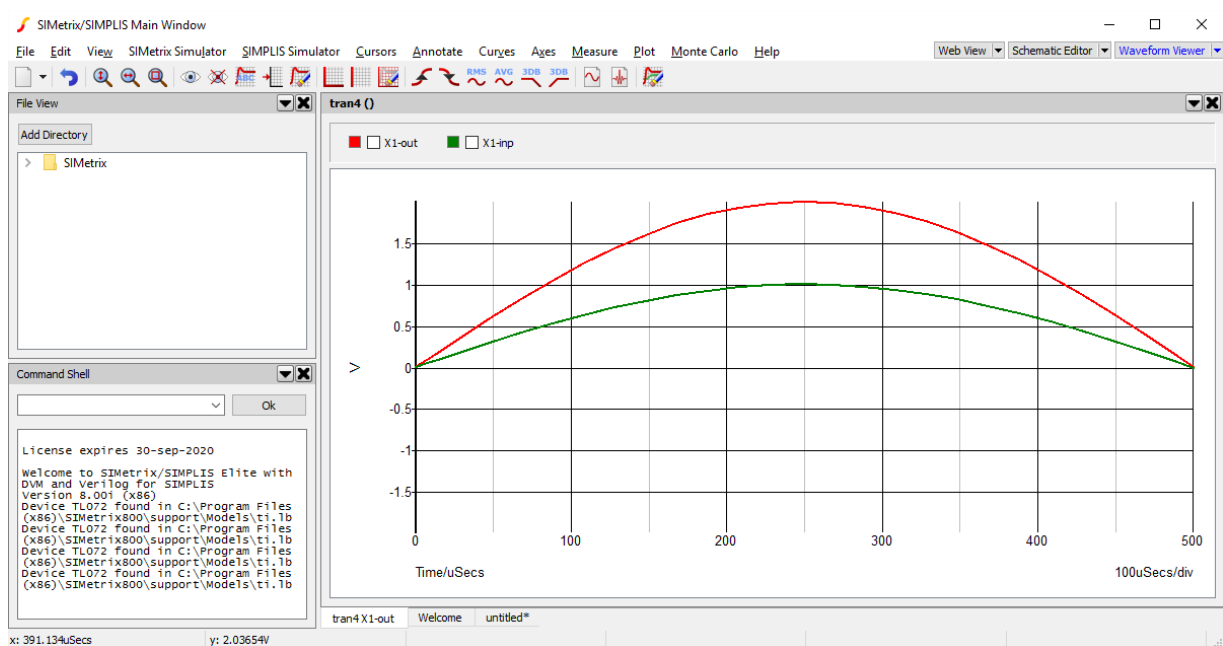
This section will cover some of basic operations that can be performed on axes. This will include adjusting the visible region of axes, handling additional y-axes and splitting curves up so each curve has it's own axis.

To begin we will adjust the visible limits on an axis. Use menu **Axes > Edit Axes...**, a window will pop-up similar to the one below.



From here we adjust the limits used on the x and y axes, change the scale type and adjust the labels used. In this tutorial we are only going to adjust the axis limits. By default the axis limits are automatically derived from the data within the curves, as can be seen by the option *Auto scale* being selected for both the x-axis and the y-axis.

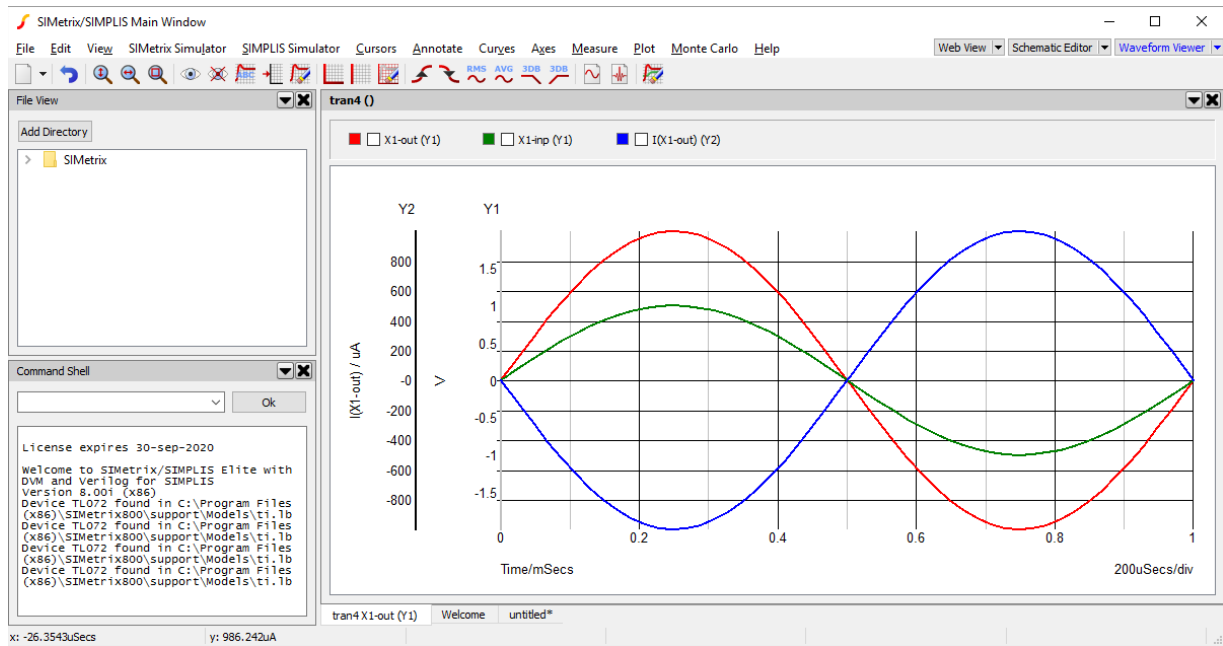
On the x-axis box, change the selection from *Auto scale* to *Defined*. The *Min* and *Max* fields will now become active for the x-axis. Adjust the *Max* value to 0.5m and press Ok. The waveform will now adjust so that the x-axis is between 0 and 500 μ s. If wanted, we can use the right cursor key to move along the x-axis and display more of the curve keeping the same level of zoom.



Reset the view using the shortcut key Home.

Next we will plot a curve that uses a different measurement unit. Using the tabs at the bottom of the waveform panel, switch back to the non-inverting amplifier schematic we used to generate this waveform from.

We are going to plot the current at the output of the op-amp. To do this, with the schematic visible and selected, use menu **Probe > Current in Device Pin...**, the cursor will change to an icon of a probe. Click on the output pin of the op-amp. The view will automatically change back to the waveform viewer, where it will now have a third curve showing the current we selected, which should appear similar to the one below.



We can now see that a second y-axis has been added. Additionally, the y-axis being used by each curve is now also displayed in the legend box in brackets.

Previously, when we used the edit axis pop-up we were given options to adjust the x-axis and y-axis, which leaves some ambiguity over which y-axis would be adjusted now that we have more than one y-axis. We resolve this issue with the concept of a *selected axis*. The waveform viewer shows which y-axis is selected by highlighting one of the stacked y-axes. In the picture above Y2 is selected as it appears with bold highlighting. To change the selected axis, you just click on the y-axis that you want to select. Now when you use the edit axis menu, any changes requested on the y-axis will be made to the selected y-axis.

Next Steps

This tutorial has provided the basics of using the waveform viewer. Nearly all aspects of the waveform viewer have more advanced functionality, which can be learnt from the documentation.

[View the SIMetrix and SIMetrix/SIMPLIS Documentation](#)