

A Concise Introduction to TINA™

Adapted from

Computer Programming with Python™, Multisim™ & TINA™ by James M. Fiore

This work is copyrighted under the terms of a Creative Commons license:



This work is freely redistributable for non-commercial use, share-alike with attribution.

Published by James M. Fiore via dissidents

For more information or feedback, contact:

James Fiore, Professor
Department of Physical Science, Engineering and Applied Technology
Mohawk Valley Community College
1101 Sherman Drive
Utica, NY 13501
jfiore@mvcc.edu

or via www.dissidents.com

TINA™ is a trademark of DesignSoft. Neither the author, nor any software programs or other goods or services offered by the author, are affiliated with, endorsed by, or sponsored by DesignSoft.

“Without deviation, progress is not possible”

- Frank Zappa

1 Introduction to TINA

Objective

The objective of this exercise is to become familiar with the TINA™ electrical circuit simulation package in order to create simple schematics and perform basic simulations. TINA stands for Toolkit for Interactive Network Analysis and is produced by DesignSoft. The differences between US (ANSI) and Euro (DIN) schematic symbols, and 3D components will be examined along with the use of virtual instruments to make simulated measurements. Please note that the precise look of the windows, menus and dialog boxes may be slightly different from that pictured depending on the version of TINA that is being used. Regardless of appearance, the functionality remains. A free version of TINA, known as TINA-TI, is available from the Texas Instruments web site at <https://www.ti.com/tool/TINA-TI>

Procedure

After logging into the computer, open TINA. If a desktop shortcut is not available TINA may be accessed via the Programs menu under the TINA menu item. You will be greeted with something similar to the screen shown in Figure 1-1. Depending on the version, the precise look of the program may be a little different from that shown (TINA-TI). There are two toolbar strips immediately below the menus. The topmost toolbar contains the usual File Open, Save, and similar commands, along with buttons for text insertion, zoom level and the like. At the extreme right end of this strip is search box for electronic components. The lower strip is made up of a set of tabbed toolbars for electronic components, measurement instruments and other devices. Selecting a given tab (e.g., *Basic*, *Meters*, *Semiconductors*, etc.) shows the toolbar for that set of items.

TINA's schematic capture facility is object based, that is, you “draw” a circuit by selecting predefined objects such as resistors and transistors, and drag them onto the workspace. They are then wired together using the mouse. You can zoom into or out of the workspace using the mouse or the zoom toolbar button.

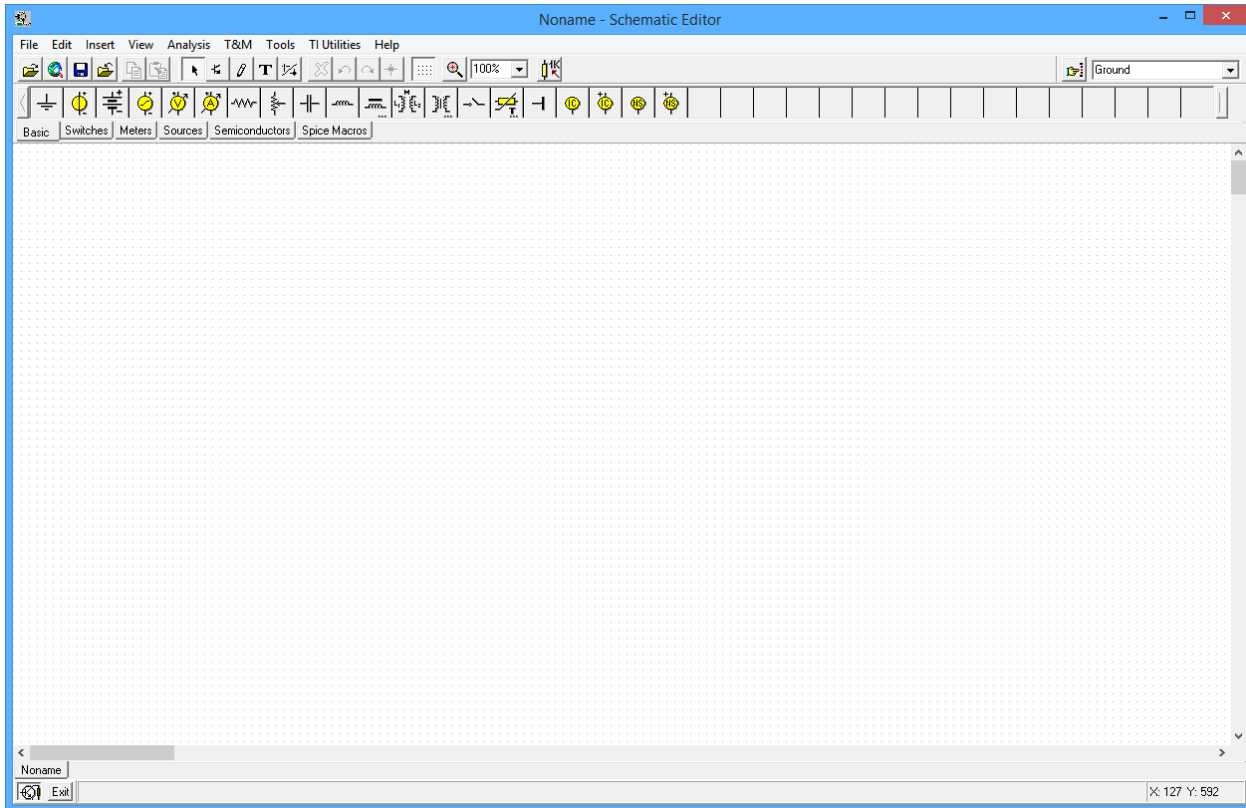


Figure 1-1

TINA will draw schematics using one of three formats. The first style uses the ANSI (North American) standard for schematic symbols. The second style uses the DIN (Euro) standard. The final style is called "3D" where components look more like photographs of the actual component. Examples are shown in Figure 1-2 using ANSI, DIN and 3D. You can set the preferred format via the View->Options menu item. More on this in a moment.

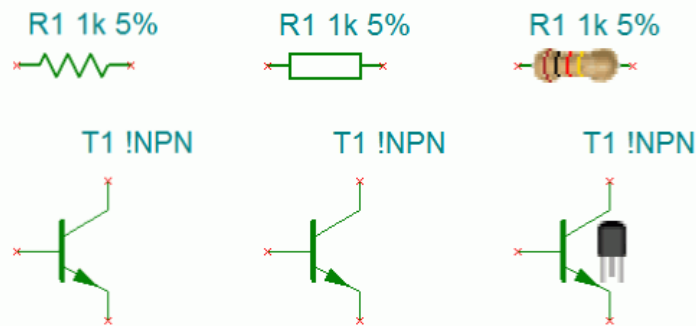


Figure 1-2

Generally, we do not use the 3D style for most work. It is, however, useful for newcomers as a way of

reinforcing and practicing use of the resistor color code, and for general package identification (such as the TO-92 transistor case pictured in Figure 1-2). TINA will show the proper color sequence for resistors using either two or three significant digits along with a proper tolerance band (i.e., four or five bands). A comparison is shown in Figure 1-3.

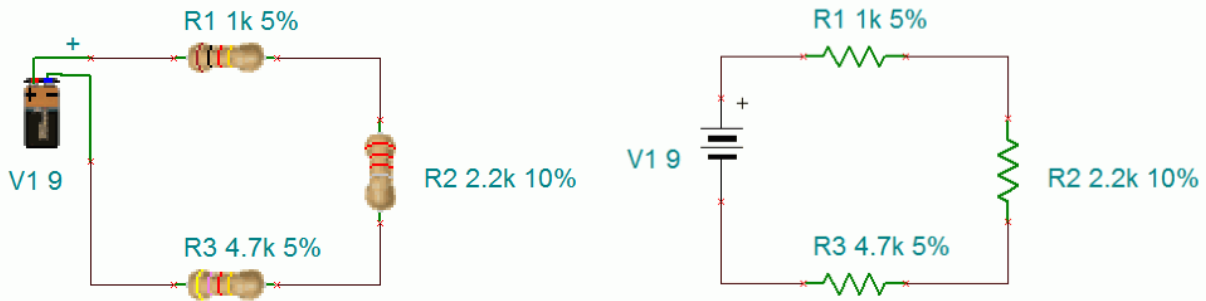


Figure 1-3

Along with the appropriate electronic parameters of a device, physical data regarding shape and pinout are required if a printed circuit board (PCB) will be created. For example, a particular device such as an operational amplifier may be available in several different package styles including through-hole or surface mount variants. Our goal here is to become familiar with capturing the schematic and performing simulations, such as verifying a laboratory exercise. Consequently, we shall not concern ourselves with the needs of PCB layout at this point.

To illustrate the use and editing of components, select the *Basic* tab and drag a DC voltage source (third item) onto the workspace. It will show up with a default voltage value and label. Double click the symbol. A dialog box will pop up next to it as shown in Figure 1-4.

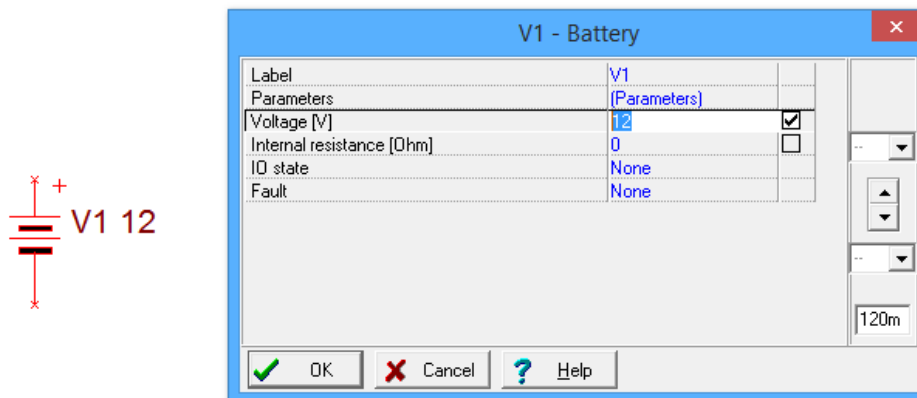


Figure 1-4

From this dialog you can change a variety of attributes, the most important of which are the voltage value and the label. Double clicking any component will bring up a settings dialog box although the precise items contained within it will vary from component to component. For example, for a resistor there will be a resistance setting instead of a voltage setting. If you need to remove a component, simply select it and hit the Delete key.

Electronic components can be placed into two broad categories; generic parts and more accurate "real world" models typically generated by the manufacturer of that part. This is particularly true with complex components or devices. For example, Figure 1-5 shows a pair of NPN transistors. The one labeled "!NPN" is a simple generic model. The lower, highlighted transistor references a specific part number, in this case the 2N3904, a popular general purpose device.

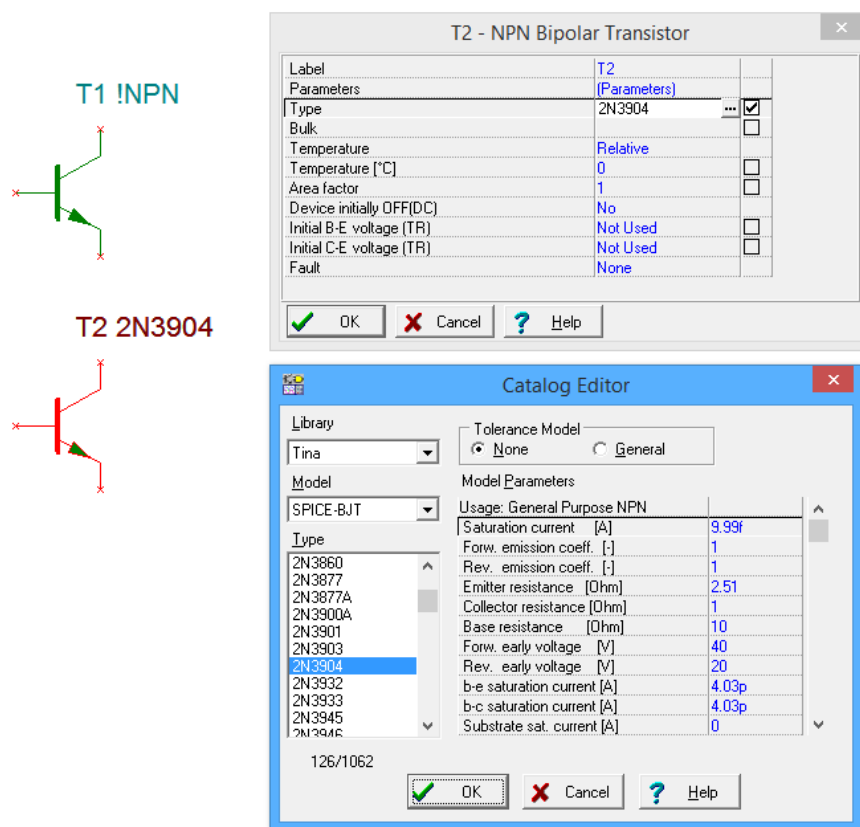


Figure 1-5

Double clicking the generic transistor opens up its settings dialog box. Initially, under Type, it will say "NPN", referring to the generic. Next to this is a small box labeled with an ellipsis (...). Clicking on the ellipsis will open a second settings box, the Catalog Editor. From here, a specific manufacturer's part can be selected from the scrolling list on the left. If need be, specific parameters can be altered in the list on the right.

Remember, whenever you see the small ellipsis box, clicking on it will open another settings box so that you can specify further details.

Editing the position and orientation of a component is straightforward. Once the item is selected (highlighted in red by default) it may be moved using either the mouse or the cursor keys. If you need to move a group of components, the mouse may be used to select several items by clicking and then dragging the mouse over them. Every component within the selection box will be highlighted and will move as a group. Components may also be rotated and flipped. These commands can be accessed from the main menu, however, it is handy to remember certain keyboard shortcuts (such as Ctrl-R for rotate right, or clockwise). Also note that it is possible to select the text labels of components and move or rotate them independently of the component. This can be handy if a label becomes obscured by a wire. To do so, make sure the component is not selected (unhighlighted) and then click directly on the label. It can now be moved or rotated.

Along with editing the components, you may also customize the look of the work space. From the file menu, select Page Setup. This opens the dialog box shown in Figure 1-6 and allows you to specify the sheet size for the schematic and related properties.

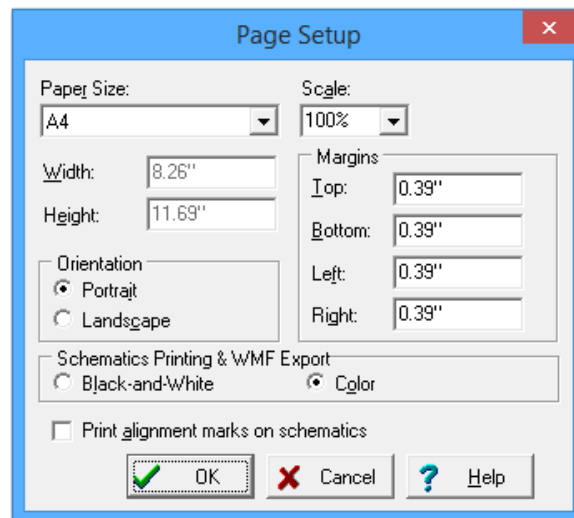


Figure 1-6

Other attributes, such as whether or not to show the grid, component values, labels, and the like can be selected from the View menu. This is shown in Figure 1-7. Of particular importance is the final item in this list, Options, which was mentioned earlier. Selecting Options brings up the Editor settings box shown in Figure 1-8.

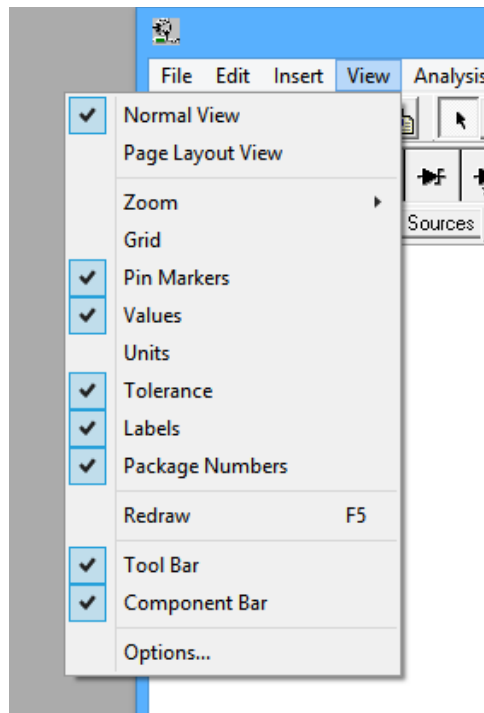


Figure 1-7

The Editor options include the aforementioned ANSI, DIN and 3D styles. It also includes the base function for AC (it is recommended this be set to sine). Further, you can adjust the color scheme for your workspace and a few other global parameters.

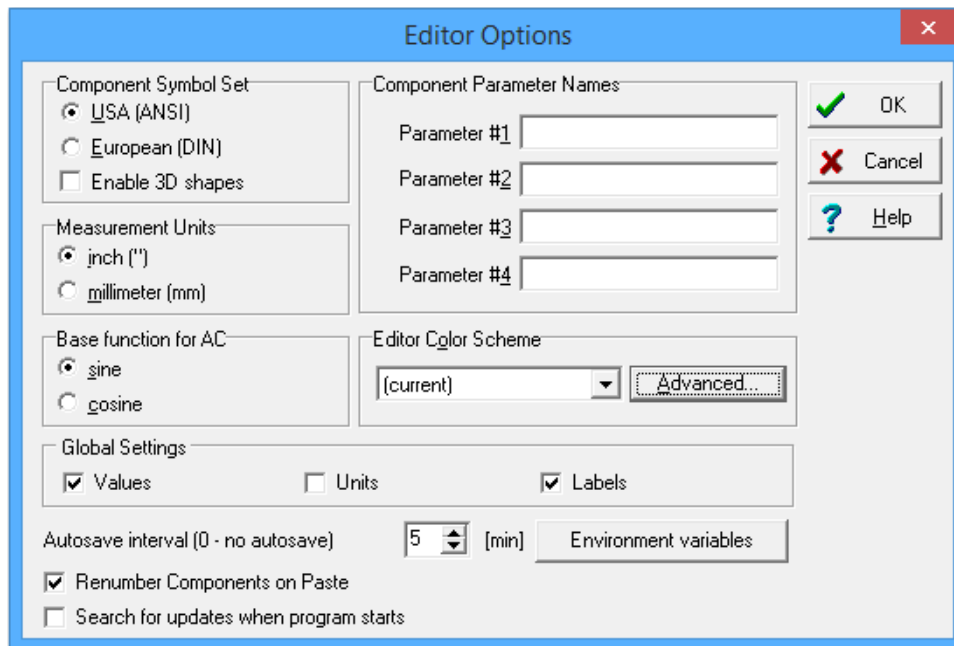


Figure 1-8

OK, let's create a circuit and perform a simulation. If you have been experimenting, remove any components on your workspace (select with the mouse and then press the Delete key). From the *Basic* components tab, select a DC voltage source, two resistors and an earth ground symbol. We shall make a series loop of the three elements with the negative end of the power supply at ground. One resistor will need to be rotated 90 degrees (one horizontal and one vertical). In order to wire the items together, move the mouse to the end of one component you wish to wire. The connection ends are denoted with a small red x. When the mouse cursor is close, it will turn into a pen shape. Simply click on the lead of the component and move the mouse to the desired lead of another component. While moving, TINA will draw a red line. Clicking on the second component will create a proper wire (by default, colored black). Wires are always drawn along the horizontal and vertical with 90 degree bends, not directly from point to point. This is the proper way to draw a schematic in the vast majority of cases.

To delete a wire, click on it to select it. You will see a set of small "handles" on it. To remove the wire, simply hit the Delete key. Note that you can also move the wire with those handles if desired.

Note that if you move a component, TINA will automatically move the wires along with it. You do not have to rewire it. If a component is later deleted, this will result in "dangling wires" which also should be deleted.

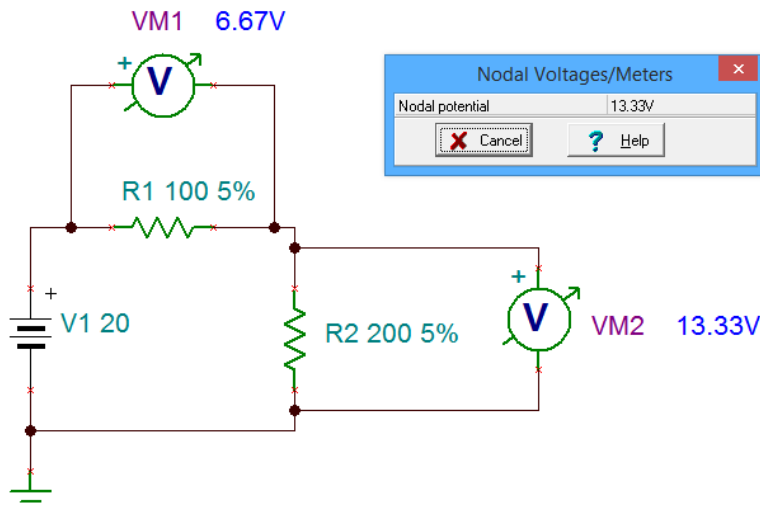


Figure 1-9

Once the components are in place and wired, double-click each component to set their values as shown in Figure 1-9. We shall then add two the two voltmeters (fifth item in the *Basic* tab). Please note that it is perfectly acceptable to change the component values immediately after dragging them onto the work space; you don't have to wait until they are wired together. It is suggested, though, that you decide upon a particular workflow and stick to it otherwise the chances of forgetting to change components from their default values increases.

The circuit is now ready to perform a simulation.

From the Analysis menu, select DC Analysis->Calculate nodal voltages. In a moment, the values of the voltages will be printed next to the voltmeters. Also, a small window will pop up. With this window, you can investigate particular node voltages or component voltages and currents. You will notice that the mouse cursor will have changed shape into a measurement probe. You can click on a node to see a voltage with respect to ground (a node is denoted with a small, solid black circle on the wire where items connect, such as where a meter connects to a wire). Further, if you click on a component, the voltage across it and the current through it will be displayed in the little pop up window. This is a nice interactive feature.

TINA also has a wide variety of virtual instruments. These can be accessed via the T&M (Test and Measurement) menu. For example, Figure 1-10 illustrates a virtual oscilloscope. This has the advantage of immediacy (assuming you've used an oscilloscope before), however, it is not the ultimate way to perform a simulation. We will look at even more powerful and flexible ways of creating simulations in the next exercise.

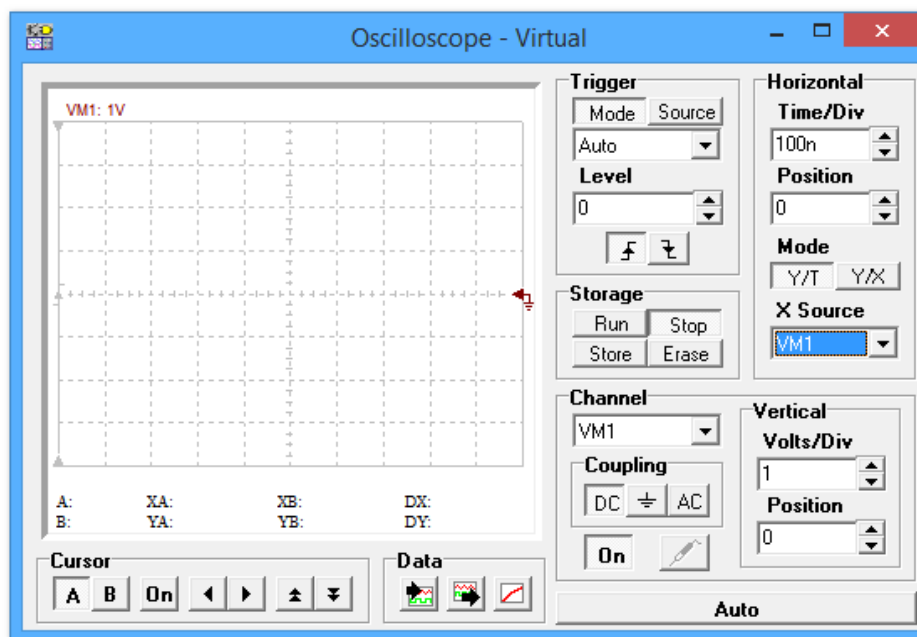


Figure 1-10

For now, you may wish to delete this new instrument from the work space and then save the existing schematic. Remember to always save to either your student account or to a USB drive. Never save directly to the hard drive in lab.

There are three very good “every day” uses for TINA during your studies: First, it is a very handy tool for verifying lab results. That is, you can recreate a lab circuit, simulate it, and compare the simulation to both your theoretical calculations and lab measurements. Second, it is a handy tool for checking

homework if you get stuck on a problem. Third, it is convenient for the creation of schematics, for example, for a lab report. An easy way to do this is to draw the circuit, capture the screen image (Windows key+Print Screen key copies it into the clipboard) and then paste the image into your favorite image manipulation program. From there you can crop it, change contrast, etc. as needed and then paste the modified image into your lab report.

At this point you may wish to experiment a bit by rewiring the circuit or to try building new circuits. As with any tool, continued practice will hone your skill. Once you are done and have saved the file (the extension should be .TSC), close TINA and then shut down the computer.

2 TINA Extensions

Objective

The objective of this exercise is to become more familiar with the TINA electrical circuit simulation package in order to use more generalized simulations via the Output Windows.

Procedure

In previous work we have examined the basic functionality of TINA, namely basic schematic capture functions such as component selection, placement, and parameter editing, along with simple simulations using voltmeters and the nodal voltages probe to measure DC voltage. While this method is relatively quick and easy to use, it is limited in other aspects. Some of the issues include:

1. Limited measurements per unit, for example a single measurement for a voltmeter, requiring multiple units for multiple measurements.
2. The need to rewire the instruments (and hence the schematic) in order to take different readings.
3. Excessive amount of work space area obscured by the instrument(s) with accompanying clutter.
4. No convenient way of storing and recalling prior simulations.
5. No convenient means of exporting simulated measurement data to other programs.

To address these issues TINA allows for an all-in-one analysis using a Table of Results and also a graphing window. Large amounts of data may be displayed simultaneously. These data may also be saved as a text file for later examination or for input into other programs.

Nothing “extra” needs to be added to the circuit. Instead of choosing the DC Analysis->Calculate nodal voltages menu item, instead, select the DC Analysis->Table of DC results menu item. To illustrate, begin by building the circuit shown in Figure 2-1. Drag a DC voltage source, an earth ground and two resistors onto the work space. Connect them in a series loop and edit the component values to 20 volts for the source, and 4000 and 6000 ohms for the two resistors.

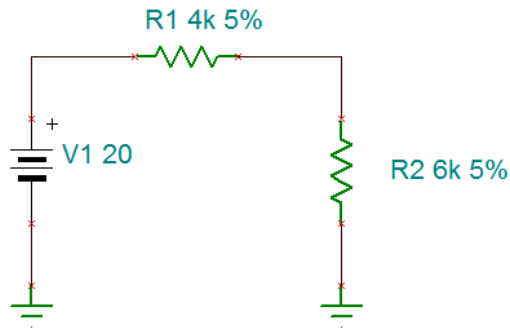


Figure 2-1

Notice that no voltmeters are required. From the Analysis menu, select DC Analysis->Table of DC results, as shown in Figure 2-2.

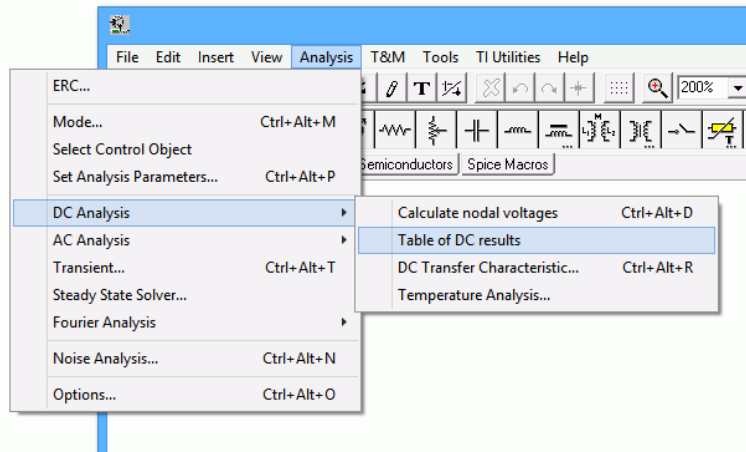


Figure 2-2

In a moment, the Table of Results window will open as shown in Figure 2-3. You may need to enlarge the window in order to see all of the values.

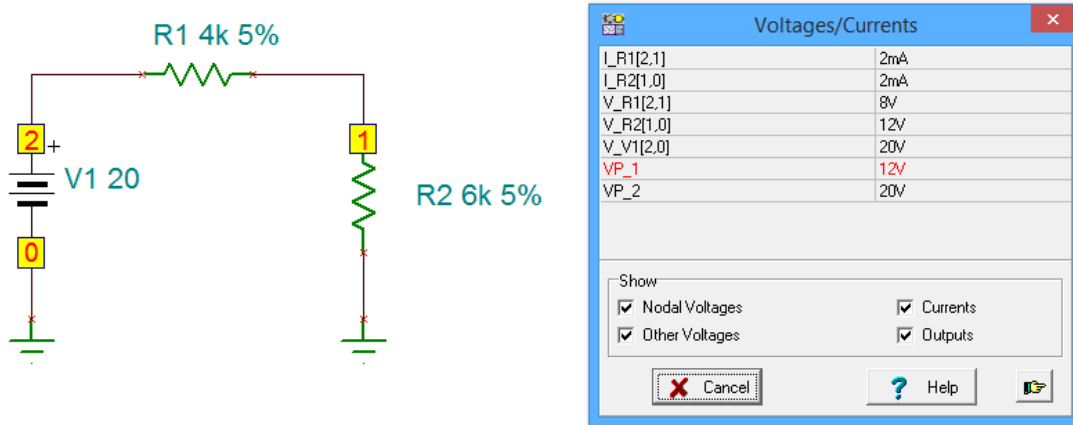


Figure 2-3

This window lists all of the component voltages and currents in the circuit. Essentially, it's like using the probe on everything and then collecting all of the data in one place. This list can be saved as a text file by selecting the small “finger point” button in the lower right. All values are identified by the component's name. You will also notice that every connection point in the circuit has been given a number. Ground is always 0 and the remaining points are sequentially numbered as the circuit is wired together. Note that voltage points 1 and 2 (VP_1 and VP_2) are included in the list and that each component's current or voltage is also identified by these numbers. For example, the voltage across resistor R1 is identified as V_R1[2,1]. The [2,1] indicates that the component is connected from node 2 to node 1. The first number is the connection for the positive lead of the meter while the second is the connection for the negative lead. Thus, this indicates the polarity and direction of the voltage and current.

It is worth noting that although TINA also has an ammeter similar to its voltmeter, this method tends to be much more efficient for current measurement because ammeters must be inserted in series with the component of interest. That is, the circuit must be rewired to accommodate the ammeter, unlike a voltmeter which is simply placed across the component(s) of interest.

At this point, you might be wondering why you would ever *not* use this method, abandoning the original method and voltmeters entirely. There are two cases where using the original method and/or

voltmeters may be preferred. First, in very large circuits the table of values can be overwhelming. Second, as is, there is no obvious way of determining a voltage that appears across several components, short of adding up the individual voltages manually. In this case, a voltmeter can still be used, its value will show up in the table alongside the other component voltages.

Finally, you may prefer to give the connection points names, such as *Vin* or *node A*, instead of the default numbering scheme. This can be achieved by adding *Voltage Pins* to the schematic. This is the first item in the *Meters* tab and looks like the letter C with a tail. These items can be placed anywhere on a schematic, and once added, the names can be changed. This is illustrated in Figure 2-4. Two voltage pins have been added to the locations formerly known as nodes 1 and 2. The names have been changed to “b” and “a”, respectively. Note that the table of voltages and currents now uses these letters instead of the numbers.

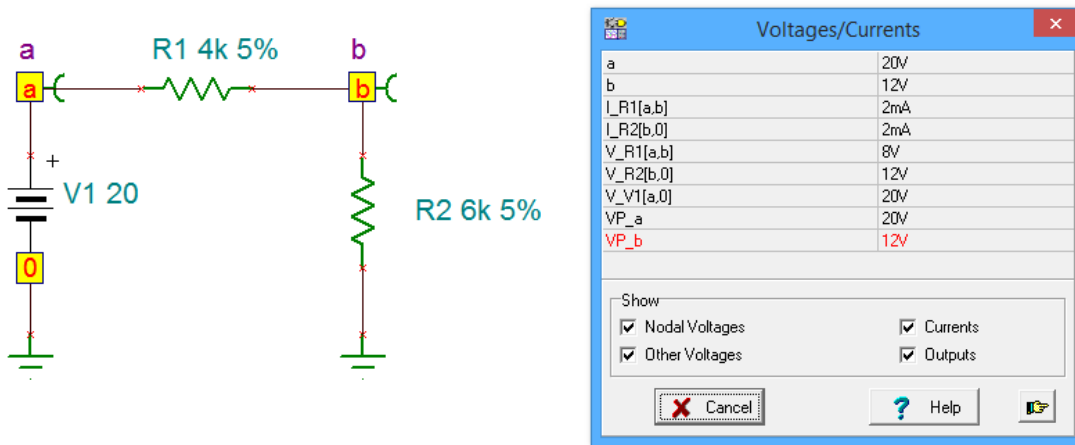


Figure 2-4

Finally, the table still makes use of the mouse probe. If you click on a Voltage Pin, its entry will be highlighted in the table. If you click on a component, the entries for both its current and voltage will be highlighted.

There are times when graphical data is preferred over simple numeric data. To illustrate this, we will shift our attention to time domain waveform analysis using an AC source. To do this, first delete the DC source in your existing circuit. Replace it with an AC voltage generator. This is the fourth item in the *Sources* tab. Your circuit should look similar to the one shown in Figure 2-5.

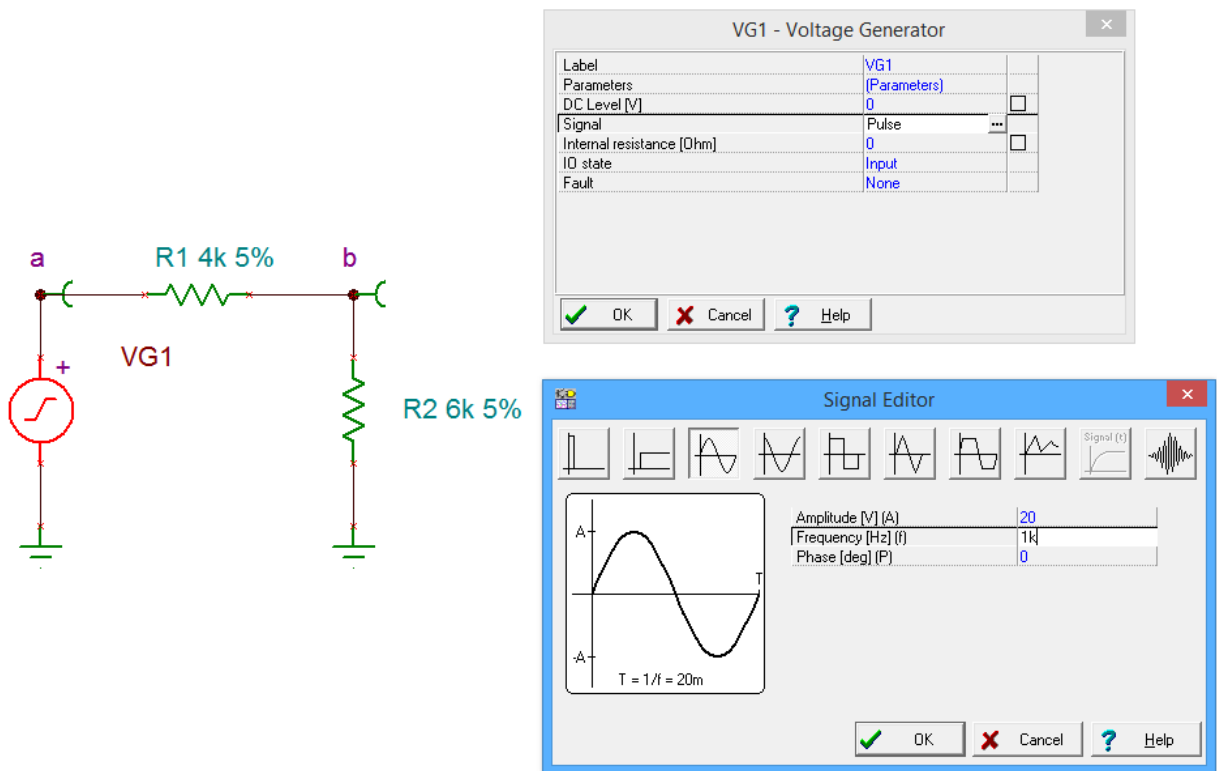


Figure 2-5

The AC generator is very flexible. We would like to create a 20 volt peak sine wave at a frequency of 1 kHz. To do this, double click on the generator to open its settings dialog box. Click on the Signal row to reveal a small ellipsis button (...). Clicking on the ellipsis brings up the Signal Editor. Select the sine shape (third item) and set the Amplitude to 20 and the frequency to 1000 Hz (1k, for short). Select OK for both of these windows.

From the Analysis menu, select Transient. The Transient Analysis dialog opens, as shown in Figure 2-6.

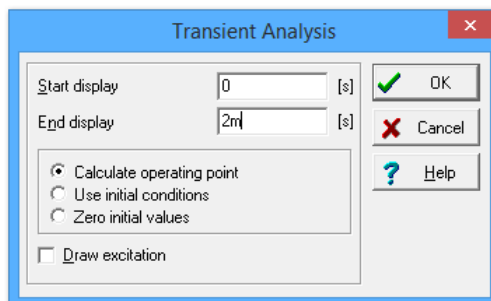


Figure 2-6

We would like to see the first two milliseconds of our waveform. At 1 kHz, we should see two full cycles. Enter a start time of 0 and an end time of 2m (0.002 seconds). After selecting OK, a graphing window opens as shown in Figure 2B-7. Notice that the time scale runs from 0 to 2 milliseconds and the amplitude spans from -20 to +20 volts. Two waveforms are drawn, node voltages *a* and *b*, as requested by our Voltage Pins.

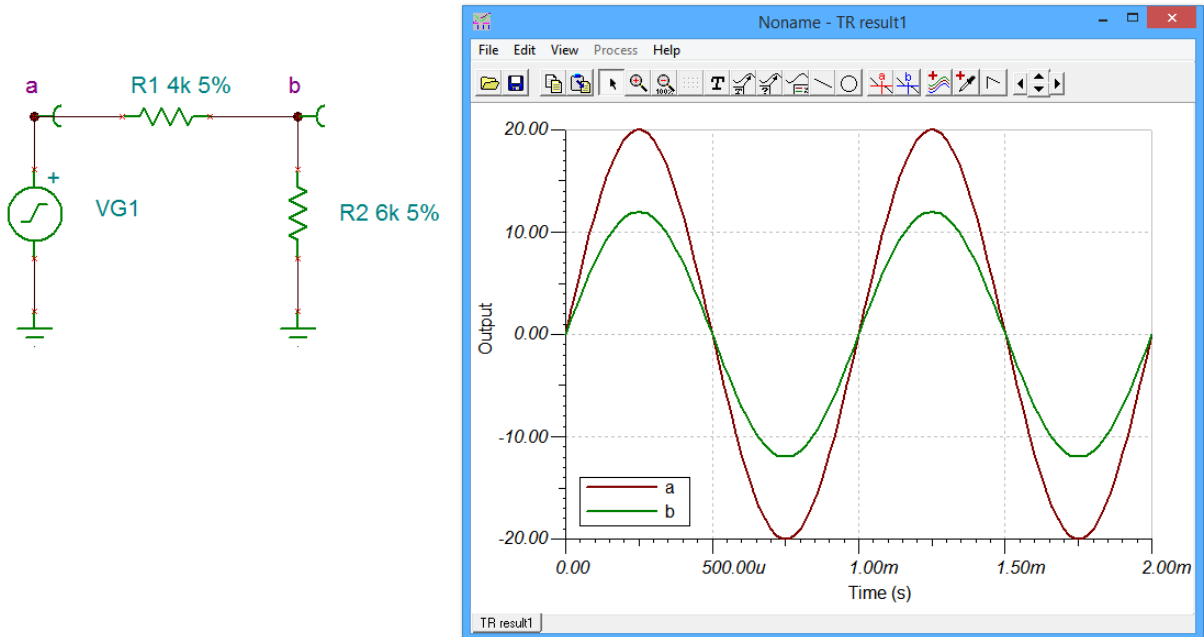


Figure 2-7

There are several useful tools and variations you can utilize with this graphing window. They are accessed either through the menus or the toolbar. For example, you can zoom into any part of the wave for closer inspection by using the zoom tool (magnifying glass). For precise measurements of the waveforms, you can make use of the cursors. There are two cursors labeled *a* and *b* (red and blue crosses in the toolbar). To use a cursor, select the cursor button and then hover the mouse over the waveform of interest. The mouse pointer will change shape into a cross. Click on the waveform. This will produce a pair of horizontal and vertical lines that track the waveform. You can move the cursor by grabbing the associated letter tab at the top of the graph with the mouse. The values at the intersection are displayed in a small output window. If both cursors are active, the output window will also include the differences between the two sets of cursor values. This is illustrated in Figure 2-8 where the bottom line displays the difference in the horizontal X values (here, that's time), and the vertical Y values (in this case, voltage).

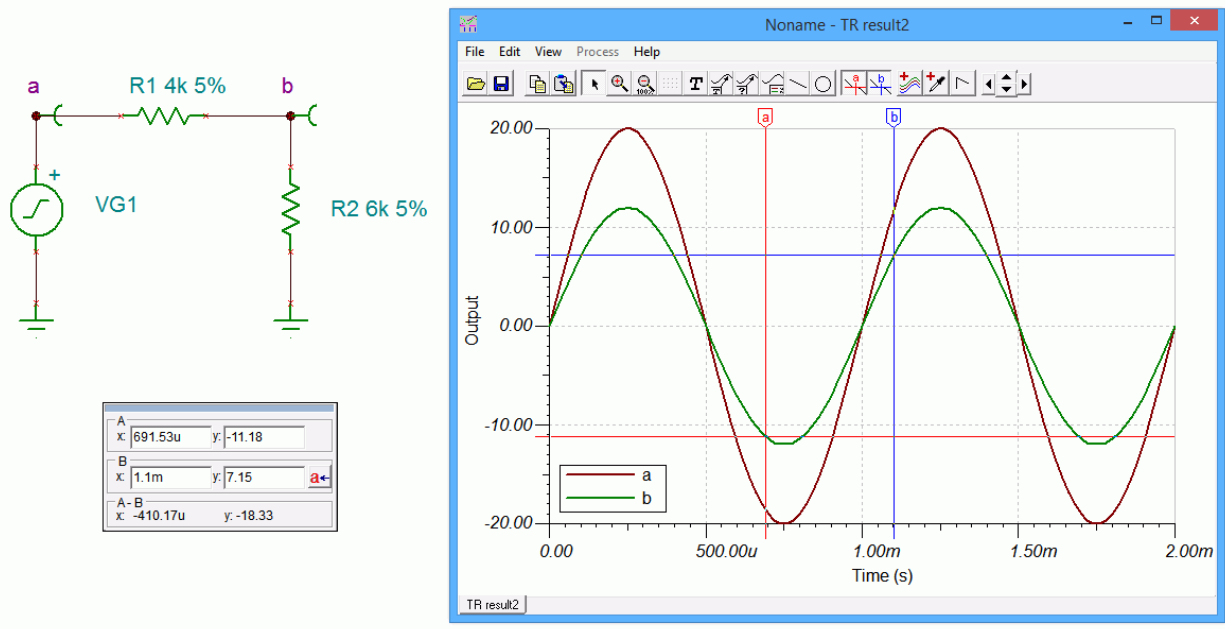


Figure 2-8

You may wish to separate the curves instead of displaying them together. This can be achieved through the View menu options Separate/Collect curves, as shown in Figure 2-9.

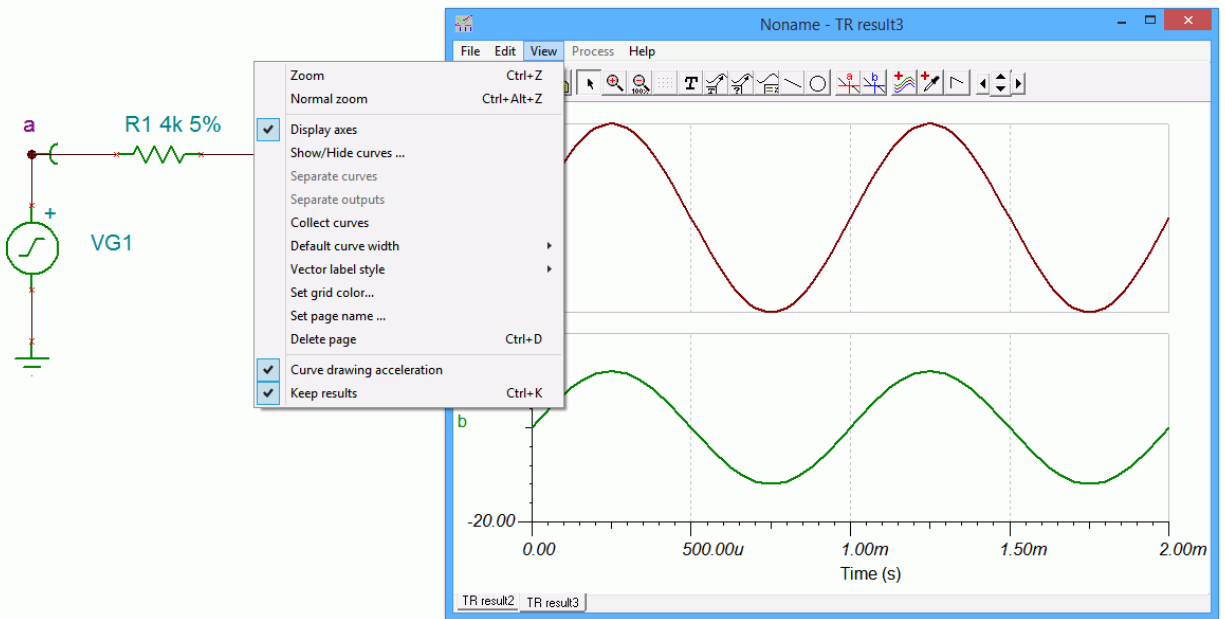


Figure 2-9

Some other graph attributes can be modified through the View menu such as the color of the grid and the visibility of the axes. Double clicking on a curve will allow you to set the color of the curve, its thickness and other characteristics. Unique labels can be applied using the Text tool (capital T on the toolbar and essentially the same as the one found in the main program toolbar). A pointer can be added to the text with the use of the Text Pointer button (a small T with an arrow). The horizontal and vertical axes can be formatted by double clicking on them. This will bring up the Set Axis dialog box. Here you can change the label, set the numeric format and precision, and also set the scale range and type. This is illustrated in Figure 2-10.

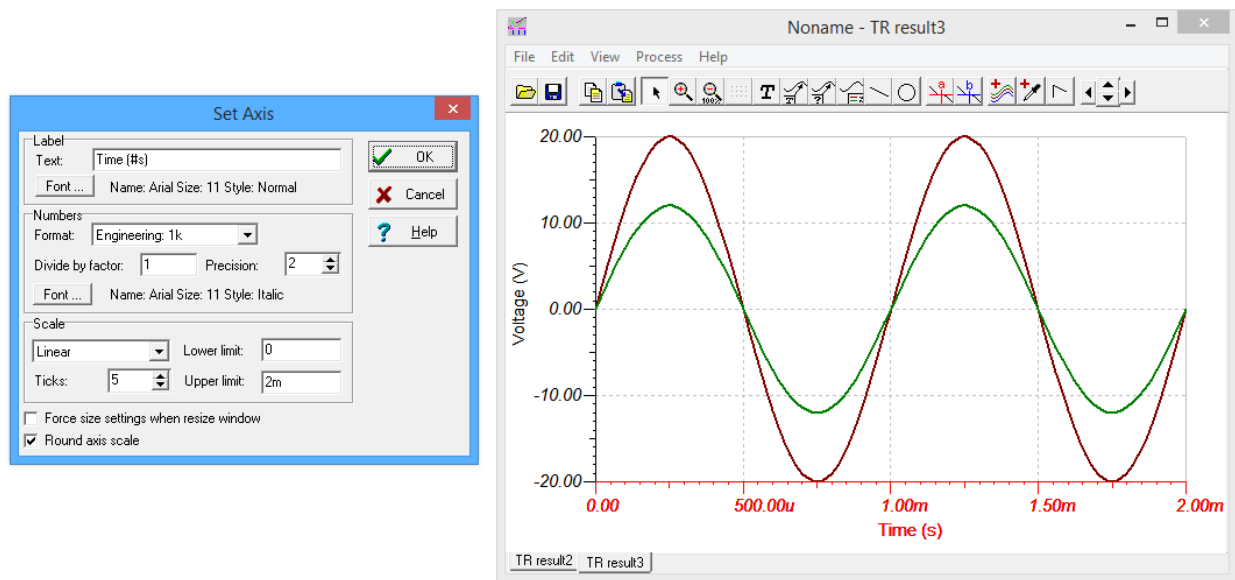


Figure 2-10

One particularly interesting toolbar button is the one immediately to the right of the cursors. This is the Post Processor. Selecting this will allow you to create complex expressions that can then be plotted. For example, you might wish to find the difference between two voltages or the current through a section of a circuit by dividing its voltage curve by the associated resistance.

The process is fairly straightforward. The Post Processor dialog is illustrated in Figure 2-11. On the left is a list of available signals. In this example we have the voltages from node a to ground and from node b to ground. Suppose we would like to plot the voltage seen across R1. This is voltage *a* minus voltage *b*. Select *a* from the list and copy it to the Line Editor by clicking on the large downward arrow. Now, select the minus sign from the drop down list of built-in functions and copy this down to the Line Editor. Finally, select *b* from the curves list and copy it down. We now have an expression to plot. It

can be previewed with the Preview button before selecting OK. This curve is shown in bright blue in the graphing window.

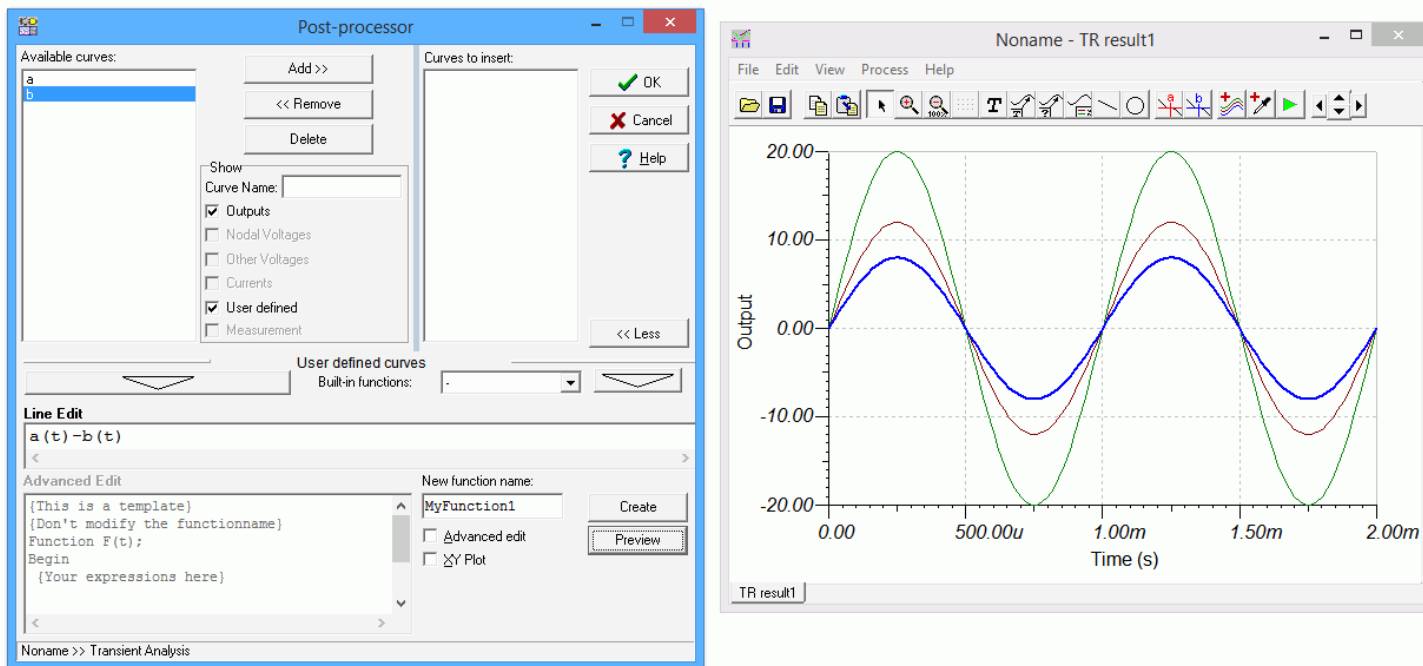


Figure 2-11

Assignment

Recreate the transistor amplifier schematic shown in Figure 2-12. Try to make it as close to this drawing as possible. All parameters must be the same. Device designators must also be the same (after all, these will be cross referenced to the PCB layout and bill of materials). The only items to change are the date (use today's date) and the designer's name entry (insert your name).

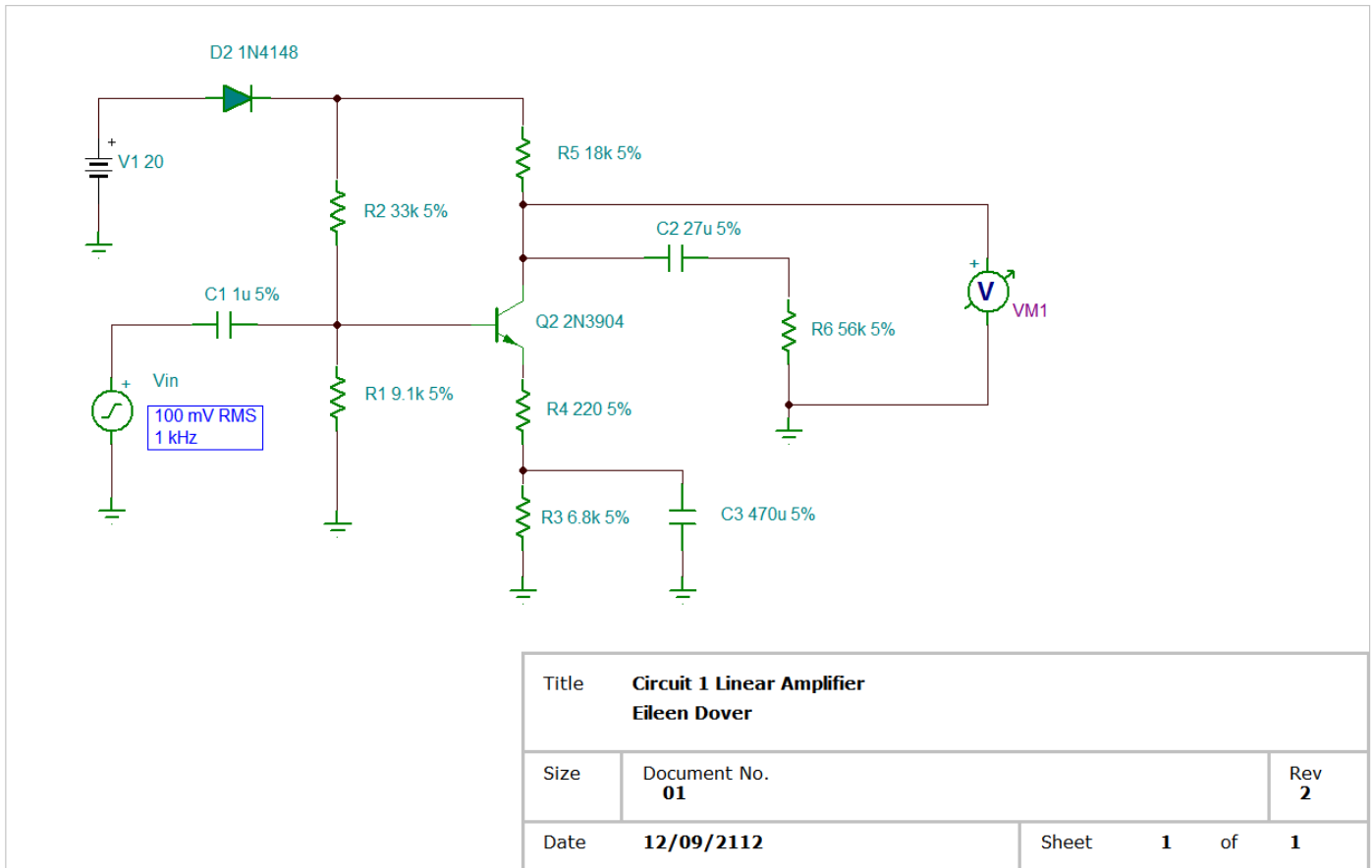


Figure 2-12