

Using Mentor Graphics for Analog Simulation

Introduction

This project is designed to give the first-time **Mentor Graphics** user an introduction to the tools needed to perform schematic capture (design entry) and simulation of analog circuits. Before working this project, you should have worked the **Design Manager** and **Bold Browser** tutorial. If you have not, stop and work that tutorial now. By the end of this project, you should be able to use **Design Architect** to enter and verify schematics, and to use **AccuSim** to simulate analog designs.

Design Architect

To capture designs for simulation, **Design Architect** is used. **Design Architect** is a drawing program that knows about electrical components, wires, busses and electrical rules. You can use **Design Architect** to draw schematics for simulation, or just to get a neatly formatted schematic printout.

Start up **Design Manager** by clicking on `Design Tools⇒Mentor Graphics`, just like in the tutorial. Once **Design Manager** has loaded, maximize its window. Use the pop-up menu (right mouse button) command to update the Tools window. The icon for **Design Architect** is labeled `design_arch`. Double click on the **Design Architect** icon; you should see a message in the lower left letting you know **Design Architect** is starting up. You can minimize **Design Manager** now. Remember, it may take some time, but you will soon see a **Design Architect** window.

Once loaded, (after the window maximizes itself), you will see a palette of icons on the right side of the window. Most every Mentor tool has a palette of this type. You can press these icons (with one mouse click) instead of going through the menus to perform certain functions. These icons will change depending on what you are doing, as you will see.

The design that you will enter is a simple RC circuit, shown in Figure 1. Use the following steps to enter the circuit:

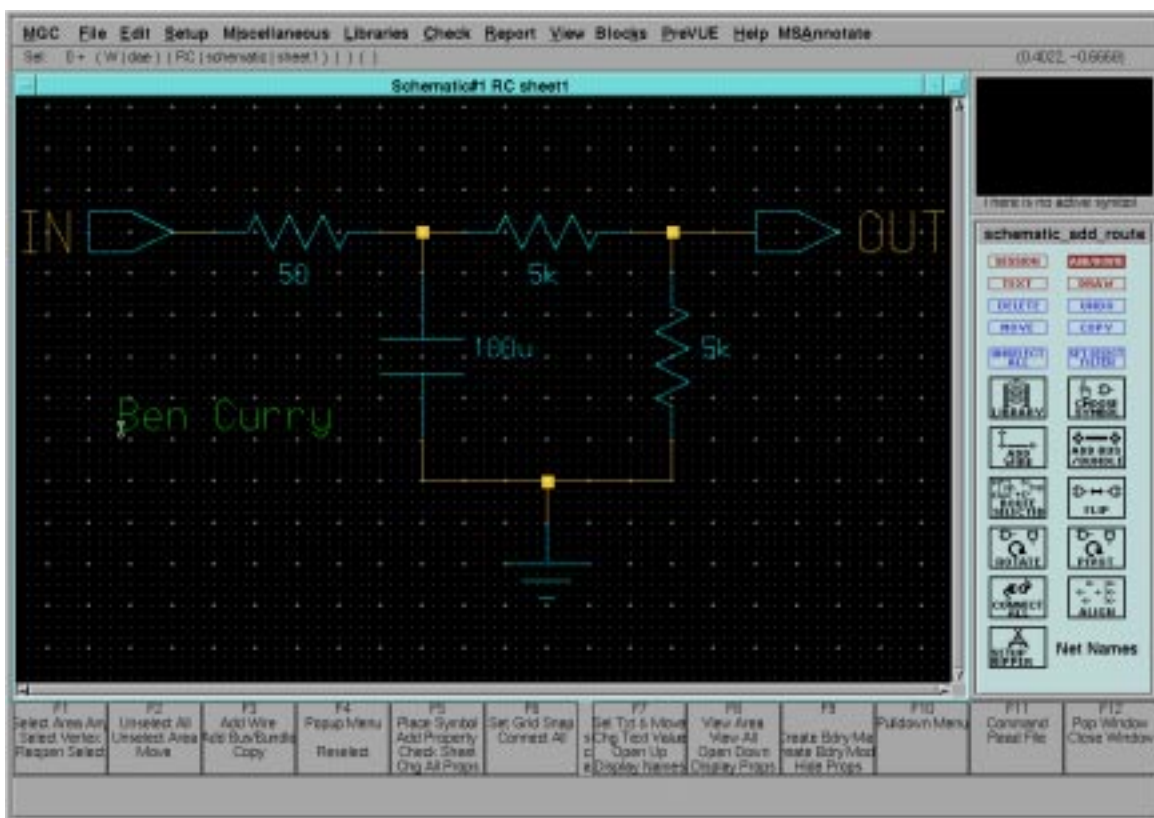


FIGURE 1. RC circuit

1. Click on the OPEN SHEET palette icon or select File⇒Open⇒Sheet from the menu bar. Mentor refers to schematics as “sheets” because the schematic for one circuit could span multiple “sheets” of paper. Mentor also refers to circuits as “components” because any circuit could be used as a single component in a higher-level design.
2. A form will appear. Note the path in the Component Name box. This is the path to *your* mgc directory; you should write this down for later use. Click in the Component Name box and append /RC to the path in the box. This will create a component (circuit) called “RC”. When you have typed this in, OK the form.
3. An empty sheet will appear in the **Design Architect** window. This sheet is where you will put the parts for your circuit and wire them together. To begin, you need to select a library of components to use. Do this by selecting Libraries⇒MGC Analog Libraries from the menu bar.
4. Note how the palette changes to show a list of analog parts libraries. You will be using the Generic Parts library for your circuit. Open this library by clicking on Generic Parts in the palette area.
5. Again, the palette changes, displaying the various parts you can choose from. First, add a resistor. Select a horizontal resistor (RESIST-H) by clicking on its icon.

6. Now, move your mouse pointer onto the schematic sheet. A ghost image of the resistor will appear; you can move it to the position you want to place it. To place the resistor, simply click once. If you accidentally select a part you don't want, click on `Cancel` in the gray prompt box (at the lower left of the window), or use the `Esc` key to cancel the command.
7. To place the other horizontal resistor, you can click in the `Active Symbol` window (in the far upper right corner - it shows a picture of the resistor) to get another copy of the part, which can then be placed. You could also copy the part you already placed on the sheet, or click on `RESIST-H` in the palette again.
8. Next, place the capacitor in the same way: click on the `CAP` icon, move the ghost image to where you want it, and click to place it on the schematic. Be sure to leave space between all of the components, especially their terminals. Do not overlap any components, or you will not be able to simulate your design later.
9. You may soon find that your sheet is rather crowded. If you need to, you can zoom out to get more area to work with. Choose `View⇒Zoom Out` from the menu bar to do this. Using `View⇒All` will zoom the view properly to display everything on your sheet.
10. Place the ground connection (`GROUND`) and vertical resistor (`RESIST`) using the same process you used for the other parts.
11. Place the input port, labeled `IN` in Figure 1, by selecting the `PORTIN` icon. This is where you will input signals to the circuit. Also, place the output port, `OUT`, the same way. It is called `PRTOUT`. They will automatically be labeled `NET`; this label will be changed later.
12. Next, add wires to connect the components. If you like to use function keys, you can start the wiring tool by pressing `F3`. Notice the function key definitions at the bottom of the window. You could also click `ADD WIRE` from the `ADD/ROUTE` palette. To show the palette icons while the libraries palette is visible, use the pop-up menu (right mouse button) while the mouse is over the palette, then select `Display Schematic Palette`.
13. Wire the circuit as shown in Figure 1. To start a wire, click once. To place a bend in the wire, click once again. To end a wire, double click. For a wire to be electrically connected to a component, it must start or end on the purple diamond on the terminal of the component. You can also start or end a wire on any part of an existing wire (e.g. between the horizontal resistors). To undo your last click, you can use the `Backspace` key. When you have finished wiring the circuit, click on `Cancel` in the prompt at the lower left, or use the `Esc` key, to quit the wiring tool.
14. Notice how Figure 1 has different values for the capacitance, resistances and the input and output port names. All of these names/values are referred to as "text properties" in Mentor. Let's change the values of those properties now. First, unselect everything by choosing `Edit⇒Unselect⇒All` from the menu bar. You could also use the `F2` key.
15. Next, if you haven't already done so, get the schematic palette icons back. use the pop-up menu in the palette to select `Display Schematic Palette`.
16. Go to the `TEXT` palette by clicking on its button in the second row of the palette. These brown buttons are the palette selectors. Clicking on each one brings up the corresponding palette.

17. To change the properties, you will first need to select them. There are several ways to do this; we'll use the simplest. Place the mouse over the text you want to select and, while holding down the `shift` key, click. While the shift key is held down you can only select text properties, and they can only be selected as long as you hold down the shift key. Use this method to select the properties you wish to modify. In this case, you should select both NET properties on the ports, the 10P on the capacitor and all the 10K's on the resistors.
18. As you click on them, notice how they turn white, denoting they are selected. If anything else is selected, you should unselect it by clicking on it, or unselecting all and selecting only the properties mentioned above. Unlike other windowed applications, once something is selected in Mentor, it stays selected. Once they're all selected, `Se1 : 6+` should be displayed in the upper-left corner. If the number is different, too few or too many items are selected.
19. Now, click `CHANGE VALUE` in the `TEXT` palette. A prompt box will appear at the bottom of the screen. Notice how the property value it is displaying is also the only highlighted (white) object in the circuit. This is so you know which property you are changing. Simply type the correct new value in the `New Value` box. In the case of the left NET, you should change it to `IN`. In the case of the right NET, you should change it to `OUT`. Press the return key after you have typed in the new name and the value changes. When the last value is changed, the prompt disappears. The new resistor values are 50, 5k and 5k, as in Figure 1. The new capacitor value will be 100u.
20. Now, add your name to the circuit. In the `TEXT` palette, click on the `ADD COMMENT` button. Just type your name in the prompt in the lower left corner and press return. Then, move the cursor onto the sheet and click where you want to place your name. If you unselect your name, you will see your name is in green text. This means the property is a comment (i.e. it has no electrical meaning).
21. Finally, you must check and save your work. You must check your sheet before you save it to be sure you have not made any basic mistakes. If your sheet does not pass the check, the simulator will not simulate the circuit. To check your work, choose `Check⇒Sheet` from the menu bar. A report will appear. If you have any errors or warnings, you will need to fix them and check your sheet again. If you have an error or warning, the report will tell you it is at something like `N$483` or `I$372`. This is the handle tag Mentor uses to keep track of things. To figure out what it refers to, highlight the handle in the report, then look at your schematic. The corresponding part will be highlighted in your circuit as well (this is called *cross-selection*). Fix any problems you may have and check your circuit again.
22. When your circuit checks with no errors or warnings, close the report window. Save your design by choosing `File⇒Save Sheet` from the menu bar.
23. Don't forget to get a printout of your new circuit. You will first need to setup which printer you will use by choosing `MGC⇒Setup⇒Printer . . .`. In the form that appears, click on the desired printer name, click the `Select Printer` button and OK the form. Then, from the menu bar, choose `File⇒Print Sheet :`, set any options desired (the defaults should be suitable), OK the prompt bar and go collect your printout.

At this point, you are done with **Design Architect**. Exit it by closing its main window. You have now completed the schematic capture of the RC circuit. In the next section, you will do a transient analysis of the circuit using **AccuSim**, the analog simulator.

AccuSim

You are now ready to simulate your design. Your circuit should appear in the Navigator window of **Design Manager**. If it does not, select the Navigator window and use the pop-up menu command `Update Window`. Now your design, RC, will appear. To invoke **AccuSim** on your design, click on the RC icon to select it. Then use the pop-up menu in the Navigator to select `Open⇒AccuSim`. After a short time, a new window will appear.

Once **AccuSim** has loaded, it will maximize its window, and you will see your schematic in a smaller window inside the **AccuSim** window. The white window in the upper left will let you watch the status of your simulation. The following steps will guide you through a transient analysis of your circuit.

1. First, you need to specify an input signal. Mentor calls this a “force.” You need to specify which signal is to be forced, by selecting the signal. Simply click the IN port in your schematic. You should see the wire turn white, denoting that the input is selected. **NOTE:** The default size for the schematic is sometimes too small to be readable. If this is the case, you can enlarge the window and/or make the schematic fill the window. To make the schematic fill the window, type `Shift-F8` with your mouse pointer in the schematic window. (You can see from the function key definitions at the bottom of the window that this is the “view all” key). Optionally, you can select `View⇒All` from the pop-up menu or the menu bar. All of these actions result in the schematic filling the window.
2. With the input selected, click the `ADD FORCE` button on the palette. In the form that appears, note how the selected signal, IN, is the signal name in the form. We’ll use a sine wave for the input. It will be specified in the time domain, so click on the `Time` button then click the `SIN` button. Now, the magnitude and frequency of the sine wave must be defined. Enter 5V for the magnitude, 2Hz for the frequency and `OK` the form.
3. Now you need to tell Mentor what type of analysis you wish to perform. To do this, click `SETUP ANALYSIS` from the palette. A form comes up and you want to do a `Transient` analysis, so click that button now.
4. The form changes, and now you can enter the information needed for a transient analysis (as in SPICE). You’ll use a three second analysis with ten millisecond steps, so enter a time step of 10ms and a stop time of 3s. Also, be sure to turn the `Use Initial Conditions` box on, by clicking on it, to be sure the capacitor is properly initialized. `OK` the form.
5. You can now run the simulation. Simply click the blue `RUN` button on the palette. Select `OK` on the `Keeps` form that appears. This tells the simulator what signals to keep track of. In this case, the simulator will keep track of everything in the schematic.
6. You can watch the `Status` box to see the simulation run. When the simulation is done, you have several options for gathering and examining output results. One option is to plot the results graphically. To do this, select the signal(s) in the schematic window that you wish to plot. In this case, we’ll use both the input and the capacitor voltage. Click on the IN port to select the signal. Since the capacitor is connected to ground, you can look at that node voltage above the capacitor to see the capacitor voltage. Click on the wire connected to the top of the capacitor to select it. You should now have both signals (IN and the capacitor) selected. If you accidentally

choose a terminal and not a wire, you will see the current and not the voltage. (A terminal is highlighted with a small white cross at the terminal.) **NOTE:** In **AccuSim**, when you want to view a voltage, you select the wire. When you want to view the current into a component's terminal, you select that terminal.

7. If you selected an object you didn't want to select, simply unselect it by clicking on it again. Once the desired signals are selected, click TRACE on the palette. Both signals will now be plotted.
8. To add signals to the Trace window, simply select them and click TRACE. Let's add the current into the capacitor now. First, unselect everything, then select a *terminal* on the capacitor by clicking on the topmost point of the capacitor. You should see a white cross, but the wire should not be highlighted.
9. Click the TRACE button again and the trace is added to the window. You do not have to rerun the simulation to get a window update. This is because you chose to have all signals kept in the Keeps form.

Sometimes, it may be difficult to determine what a signal in the plot corresponds to in the schematic. It is obvious that one of the signals plotted is the input since it has the label "IN". But what about the current? Here, the names are system assigned handles and not obvious to us what they correspond to. Using cross-selection, as you did in **Design Architect**, is an easy way to see what the correspondence is. Make the Trace window active and follow these steps:

1. Unselect everything, so no signals are selected in the legend of the Trace window.
2. Now, select the current signal that has the handle that looks something like /I\$2/POS:i. Be sure to select the text on the line that shows the trace color in the legend. If you select the line above it, you will not see the proper results.
3. When you select the line in the legend, the corresponding signal is selected in your schematic too. Now you can see the correspondence between the trace and the circuit signal.
4. Cross-selection works the other way too. If you select the IN signal in the schematic window, its plot will be selected in the Trace window as well.

Another way of gathering output data is to examine the *Outfile*. The VIEW OUTFILE icon is located on the RESULT palette. The *Outfile* is much like the output file created by a **SPICE** simulator. You can print the data from both the *Outfile* and the Trace window. However, the Trace window is usually more informative and compact.

To print the Trace window, setup the printer just as you did in **Design Architect**. (MGC⇒Setup⇒Printer...) Then with the Trace window active, choose File⇒Print⇒Active Window... from the menu bar. OK the prompt bar and go get your printout. To print the *Outfile*, you could just print the entire file using the method above with the *Outfile* window active. However, the file is often as long as ten pages, and most of the data may not be needed (remember your print quota of 500 sheets per quarter?) A better approach is to highlight only the information you want to print. Then choose File⇒Print⇒Document...

from the menu bar. In the prompt, change the *portion* box to *selected*. and click OK. This will cause Mentor to only print the portion you have selected in the `Outfile`.

Now you have gathered the data you need from the simulation. But what if you made a mistake, and the capacitance should really be 50 UF? Do you have to go back to **Design Architect**? The answer is no. Let's see how you can continue your simulation and yet change the component value.

1. First, close any report windows you may have open (like the `Outfile`). You can leave the Trace window open if you like.
2. To change the capacitance from 100 UF to 50 UF, select the capacitor (*not* its value) in the schematic window.
3. Next, select `Edit`⇒`Property`⇒`Change` from the pop-up menu in the schematic window or from the menu bar.
4. In the form, one property is named `instpar` - it's the value of the capacitance. Select this property line and click OK.
5. Another form appears, and you can type in the new value. Do not change any other items. Press return or click OK when you are done.
6. Notice how the value turned red on the schematic. This means that you have edited the original schematic. If you wish, you could propagate that change back into your original schematic later. (Mentor calls this a "back annotation.") For now, though, you have only changed the value that **AccuSim** sees.
7. Everything is now ready, so click `RUN` to rerun the simulation.
8. The new results you see in the Trace window will be from the circuit with a 50UF capacitor.

You can continue this process as many times as you need to. Be sure to get a printout of this new data for your report. To help determine what circuit this data came from, you may want to put a label on the Trace before printing it. With your mouse in the Trace window, choose `Chart`⇒`Add Text`: from the pop-up menu. In the prompt box, enter the text or label you want and click OK. Now move the cursor back to the Trace window, where you will get a set of cross hairs. Place the cross hairs where you want the origin (lower-left corner) of your label to be and click to place it. When you have printed out the data you need, you will be done with **AccuSim**. Exit the program by closing its main window. You will get a form, and you could click OK to save your simulation results and setup. If you have made printouts, though, you probably will not need to save your results, so select `Without Saving` and OK the form.

You can now exit **Design Manager** by closing its window. You now know how to enter a design into the system and simulate it using the analog simulator. In the next project, we'll perform other types of analysis on a simple transistor circuit.

Report

The report should be typed and follow the report format discussed or handed out in class. Be sure to include:

- The printout of your circuit from **Design Architect**.
- The Trace printouts from the 100 UF capacitor and 50 UF capacitor simulations.
- Any portions of the Outfile you feel are needed for your report (probably none).