

## ADS Overview-Part1

1. The installed version of Advanced Design System (ADS) runs on Windows only and is only installed on the Windows boxes located in Eaton Hall.
2. Create a directory named ADS on your account (H: Drive)
3. Open ADS by running the following menu command: **Start->Programs->Advanced Design System 1.5->Advanced Design System** (this runs the following command: C:\AdvDesSys1.5\bin\hpads.exe). The main ADS window is shown in Figure 1 below.

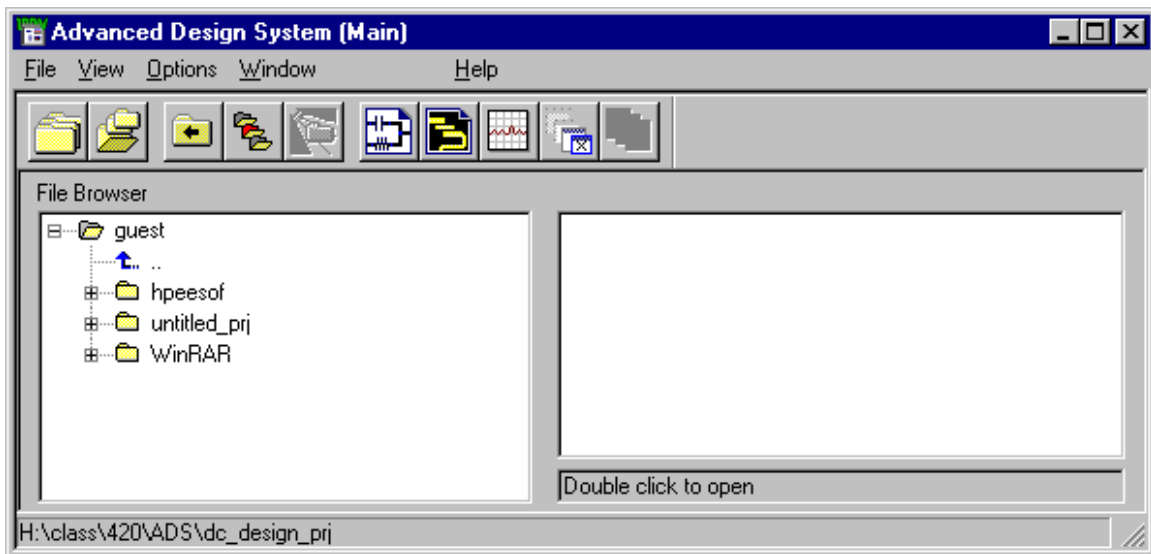


Figure 1: Main ADS Window

4. The “startup directory” is c:\guest\. Create a new project by choosing **File->New Project**. Either type in the path yourself as shown below or use the browse feature to find the ADS directory you created on your user account. Name the new project “design1” as shown in Figure 2 below. Click “OK” when you are done.

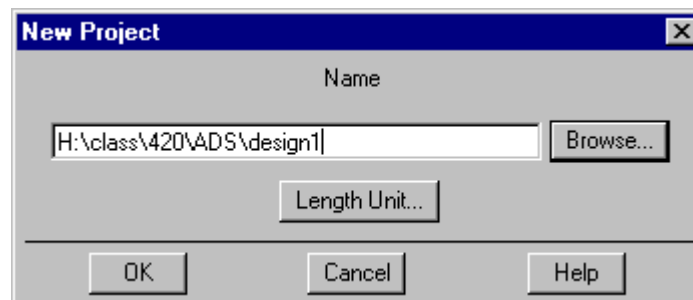


Figure 2: New project dialogue box

- When you create a new project, the schematic window will automatically open. Close all secondary windows by choosing **File->Close All**. When a new project is created, a directory named “design1\_prj” is created with several configuration files and sub-directories. In general, you should modify files within this directory through ADS only. These directories should be showing in your display as shown in Figure 3 below. Browse through the subdirectories and turn on/off the **View->Show All Files** preference. Your schematics will be stored in the networks subdirectory.

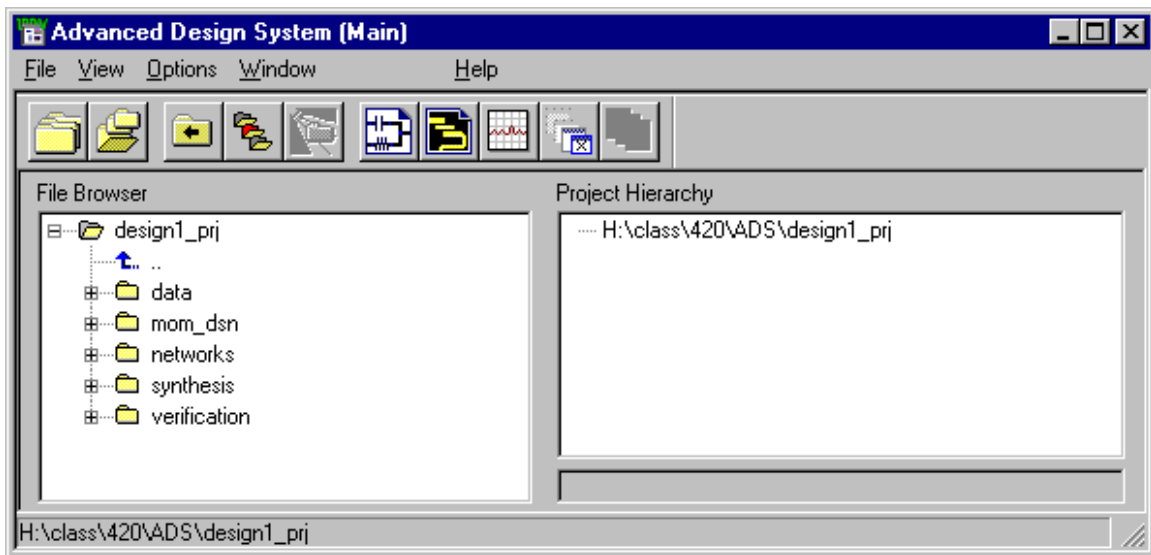


Figure 3: Main ADS window with design1\_prj opened

- To create a new design/schematic within your project, press **File->New Design**. Name your new design DC\_Simulation as shown in Figure 4 below. Make sure all of the other settings are the same as those in the figure. Schematics show the electrical connections, but they don't show the physical dimensions. Layouts show the physical and electrical connections. When you actually fabricate your design, you must create a layout from your schematic. Click “OK” when you are done.

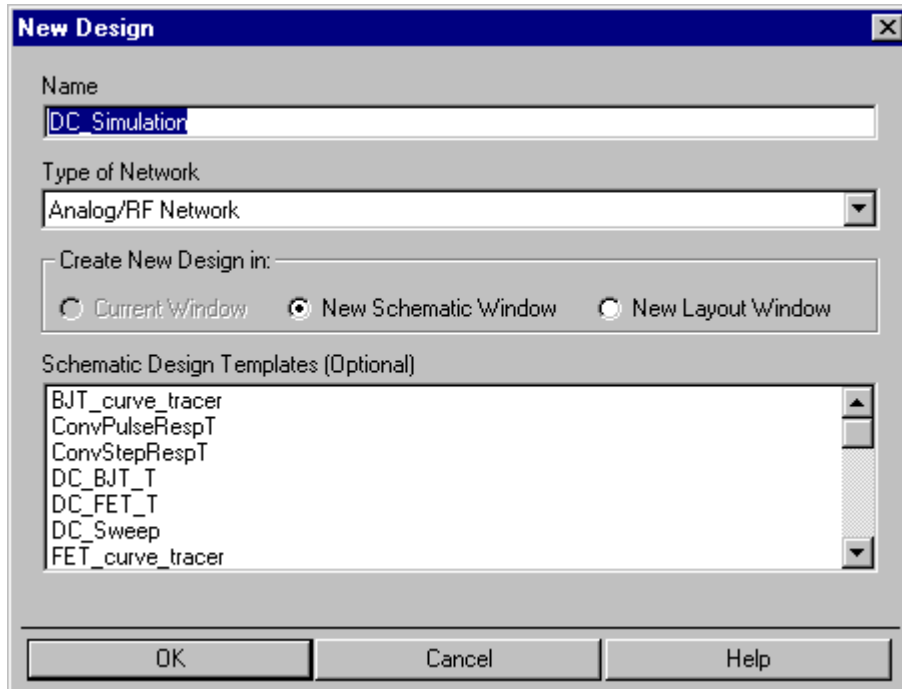


Figure 4: New design dialogue box

7. When you click “OK”, the schematic window in Figure 5 below should open. Note the title bar that tells us that we are editing a schematic called “DC\_Simulation”. You can save the schematic by pressing Ctrl-S or **File->Save Design**. The box circled in red allows us to choose which component tool bar (library) will be displayed. Currently the “Lumped Components” library is selected and you should see a vertical toolbar with various lumped components to choose from. Place a resistor on your schematic by clicking on the resistor icon (w/o a box around it). When you move the cursor over the drawing part of the schematic window, the ghost of the resistor will be shown. Place the part on your schematic with a left mouse click. Note that you are in “place component” mode and you can place as many resistors as you need. Right now, just place one more resistor. Get out of “place component” mode by pressing the “Esc” key or by right clicking with the mouse and choosing **End Command**.

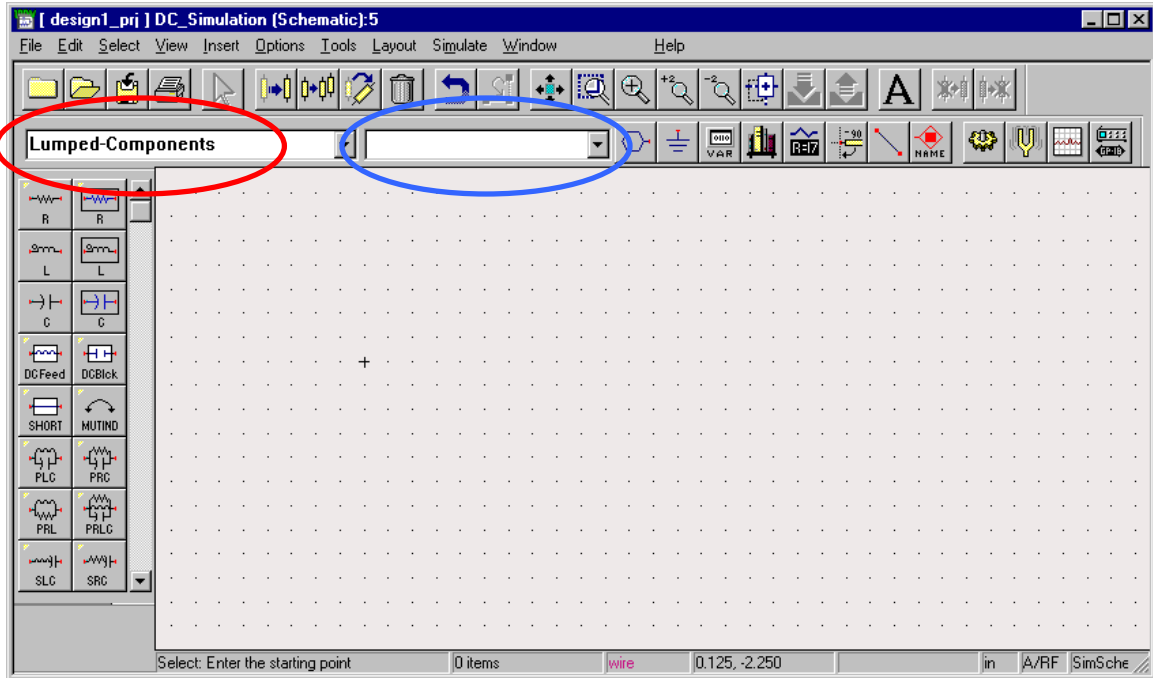


Figure 5: Schematic window with the “DC\_Simulation” schematic opened


8. Note that if you know the part name (R for resistor), you can type it in directly in the dialogue box circled in blue. Place the following components on your simulation:
  - a. Syntax of the following list is “**Library: Part Type (part name)**”
  - b. Need two “Lumped-Components: Resistor (R)” – should have already placed these in the previous step.
  - c. Need one “Sources-Freq Domain: DC voltage source (V\_DC)”
  - d. To place a ground, click on this icon: 

Figure 6 below shows the completed circuit. Once the components are placed, arrange the components as shown in the figure and then connect the components with wires as shown by pressing Ctrl-W or **Insert->Wire**. Note that each resistor that you place shows its part name (R), instance name (R1 or R2), and its nominal value (50 Ohm). Similarly, the DC voltage source shows its part name (V\_DC), instance name (SRC1), and its nominal value (1.0 V).

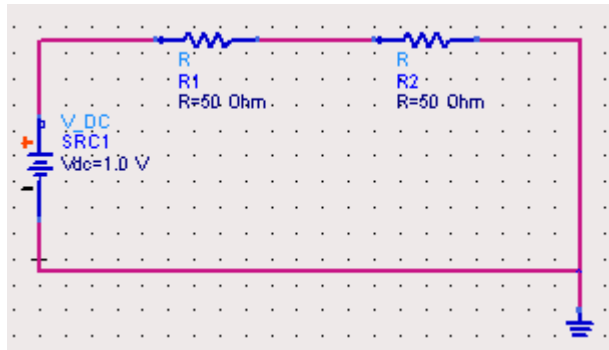



Figure 6: Completed circuit for DC simulation

9. Save your design by pressing Ctrl-S or **File->Save Design**.
10. Now that we have created our circuit we are ready to simulate. For this circuit we will be doing a DC simulation. Choose the “Simulation-DC“ library. Place a DC simulation controller on your schematic by clicking on this icon  and then clicking in your schematic window. When we run the simulate command, all the simulation controllers are run. In this case we only have one simulation controller: DC simulation. Before we simulate, run simulation setup by choosing **Simulate->Simulation Setup**. The dialogue box in Figure 7 below should appear. Set the settings as shown. Make sure the “Open Data Display when simulation completes” checkbox is disabled. We will be displaying our DC simulation results directly on the schematic rather than in a separate “Data Display” window. Press the “Apply” button to save the settings and then the “Cancel” button to close the dialogue box.

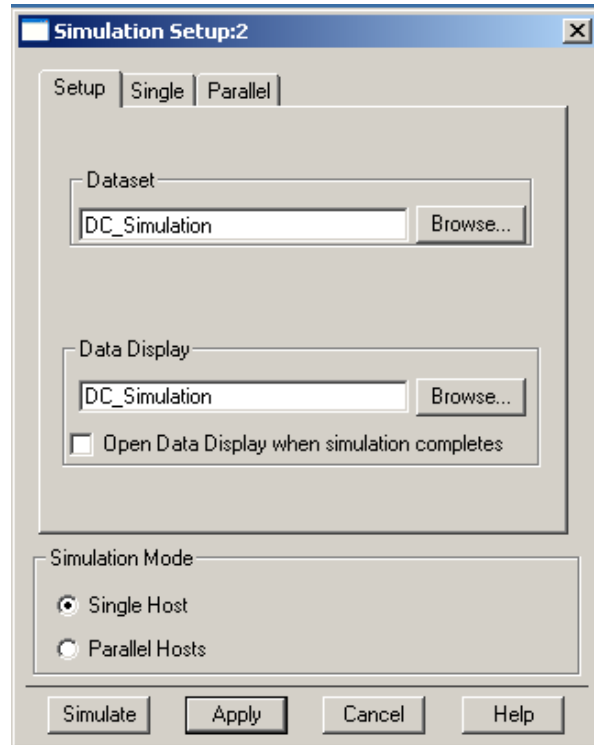


Figure 7: Simulation setup

**11.** Run the simulation by pressing F7 or choosing **Simulate->Simulate**. A simulation window should pop up and the following information should appear in the text box:

```
HPEESOFSIM (ver. "150" rev. "200" -- 12/13/00 23:36:43)
Copyright Agilent Technologies, 1989-2000.
```

```
DC DC1[1] <(GEMX netlist)>
..
```

```
Convergence achieved in 3 iterations.
```

```
-----
Simulation finished: dataset `DC_Simulation' written in:
`H:\class\420\ADS\design1_prj\data'.
-----
```

```
Resource usage:
Total stopwatch time: 1.07 seconds.
```

If an error occurs, double-check your schematic and each of the above steps.

**12.** Print the DC simulation results on the schematic by choosing **Simulate->Annotate DC Solution**. Your schematic should look like Figure 8 below.

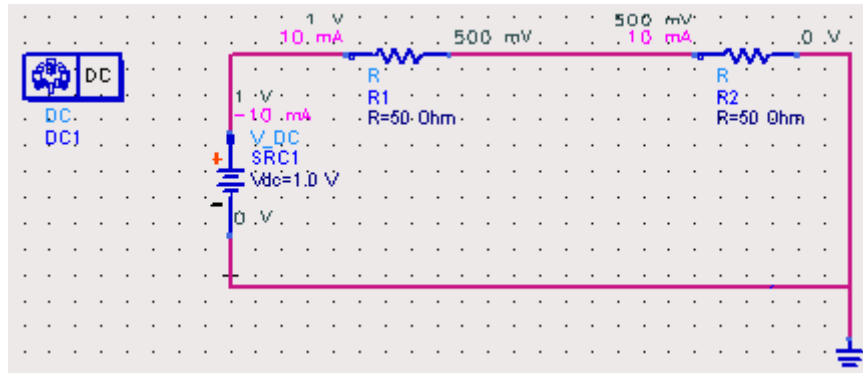


Figure 8: DC simulation results

13. Edit the component parameters for R1 (your R1 resistor may be named differently than the one in Figure 8, therefore, to clarify, the R1 resistor is the resistor directly following the positive terminal of the DC voltage supply). You can do this in four different ways:
- Double click on the R1 resistor.
  - Right click on the R1 resistor and choose “Edit Component Parameters”
  - Select the R1 resistor and choose **Edit->Component->Edit Component Parameters**.
  - Left clicking on the component parameters on the display and changing them directly.

The first three methods will open the parameter as shown in Figure 9. Change the instance name to R\_Generator.

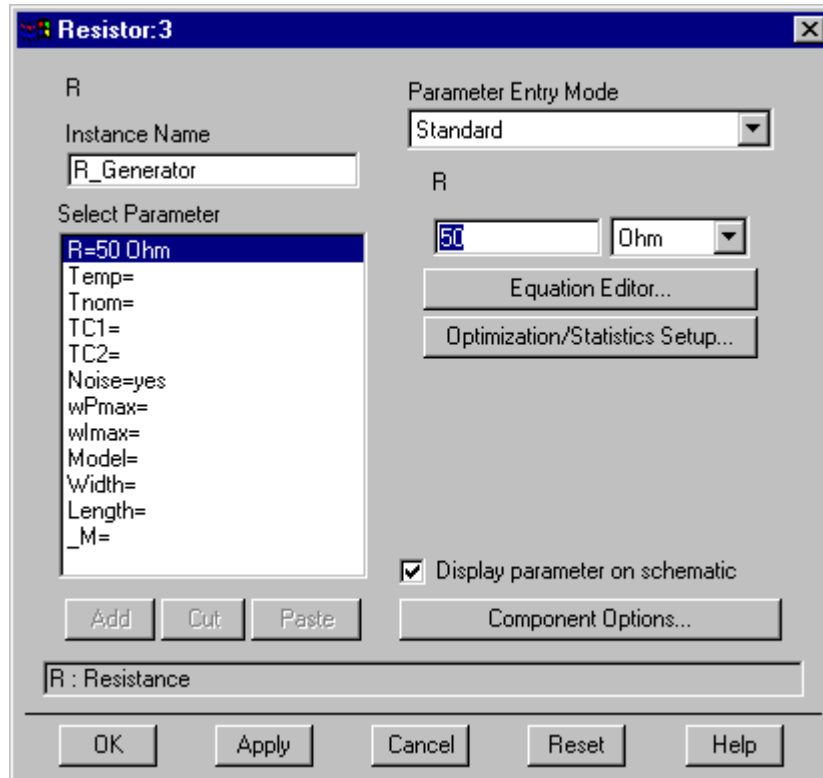


Figure 9: Component parameter box for R1

Change the instance name of the other resistor to R\_Load and the nominal value to  $R = 2 \text{ k}\Omega$  :

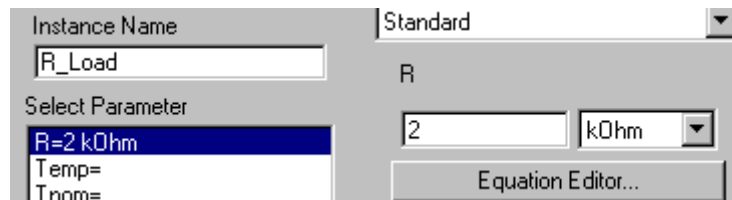


Figure 10: Component parameters for R2

**14. Simulate the design (press F7) and place the DC results on the schematic window (choose Simulate->Annotate DC Solution).**