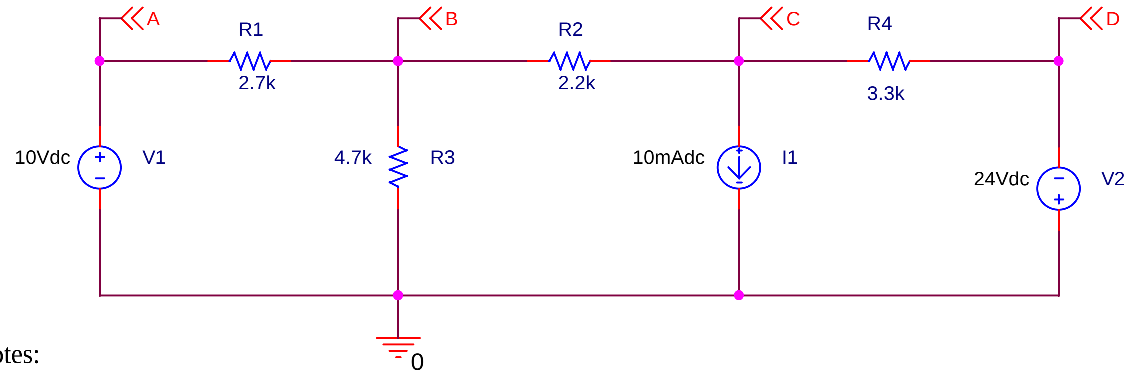
EGR 260
Circuit Analysis
File: DC Circuit.opj

DC Circuit - Determining Node Voltages

Purpose: Determine the node voltages in the circuit shown below.

Analysis: The simplest type of analysis in PSPICE is a bias point analysis. This type of analysis determines node voltages. Currents through voltage sources and total circuit power dissipation are also automatically determined.



Notes:

1) All circuits must include a ground symbol. Use the GND symbol from the toolbar (pick that ground symbol labeled 0).

2) The voltages sources are part VSRC from the SOURCE library.

3) Parts may be rotated using Ctrl + R. Alternately, right-click on the part and pick ROTATE or MIRROR.

4) Note that the current source has the units mAdc. PSPICE ignores all letters after the first letter m, which is a valid SI prefix. So the current could have just as easily have been labeled 10milliamperes. Be sure not to include any spaces.

5) PSPICE uses its own node numbering technique and it is often not apparent what node numbers will be used. Specific node names can be assigned using OFFPAGE symbols (>>C on the toolbar). Note that nodes A - D have been labeled above and will be referred to in the OUT file.

3 Basic Steps in analyzing a circuit using ORCAD Capture.

1) Draw the schematic
   A) Start ORCAD Capture. Pick FILE - NEW - PROJECT from the main menu.
   B) Enter a project name. Pick Analog or Mixed-Signal Circuit Wizard under "Create a new project using." See attached page as an example.
   C) Add any necessary libraries. The default libraries are often sufficient, but the EVAL library is sometimes needed for digital parts, switches, op amps, and more.
   D) A blank schematic screen will appear. Draw the schematic.

2) Create the Simulation Profile.
   A) Pick PSPICE - NEW SIMULATION PROFILE from the main menu.
   B) Choose a name for the Simulation Profile. For example, the name BIAS was used for a bias point analysis (see attached page).
   C) Select the type of analysis (see attached page as an example).

3) Analyze the circuit.
   A) Pick PSPICE - RUN from the main menu.
   B) Once the analysis screen appears and the analysis is completed, select VIEW - OUTPUT FILE to see the results. See the attached OUTPUT FILE.

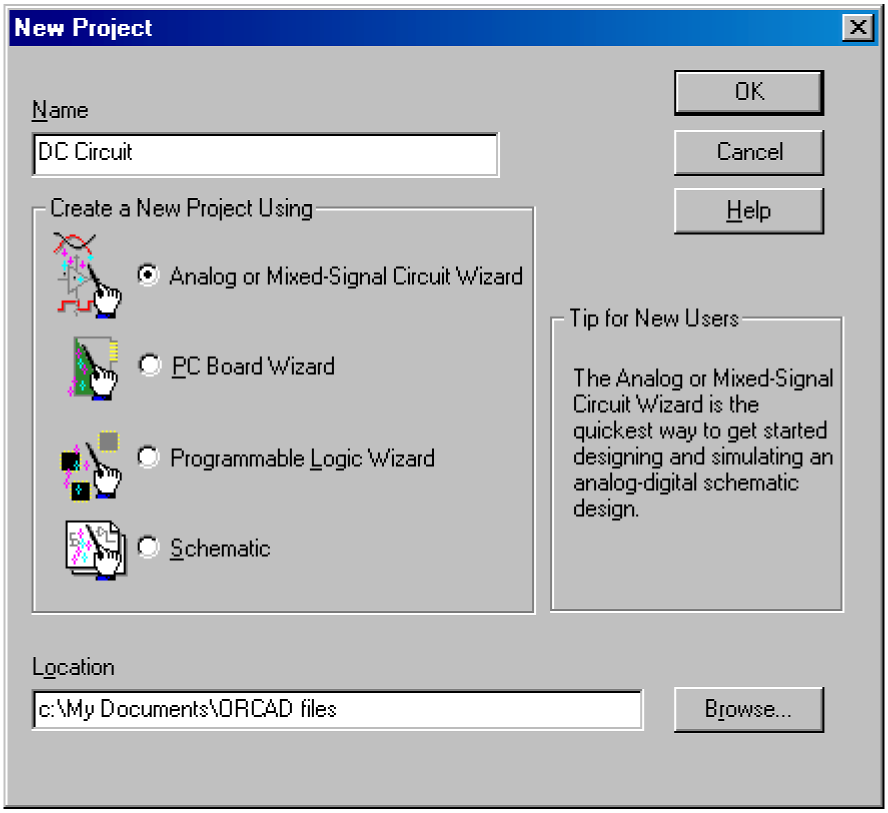
Title <Title>		
Size A	Document Number <Doc>	Rev <RevCode>
Date: Saturday, February 05	Sheet 1 of 1	

3 Basic Steps in analyzing a circuit using ORCAD Capture.

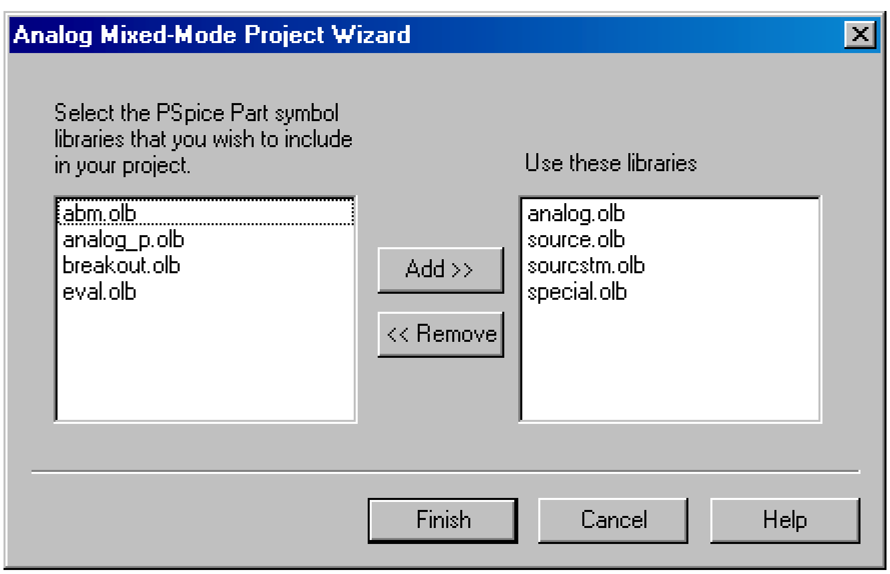
1) Draw the schematic

   A) Start ORCAD Capture. Pick FILE – NEW – PROJECT from the main menu.

   B) Enter a project name. Pick Analog or Mixed-Signal Circuit Wizard under Create a New Project Using.

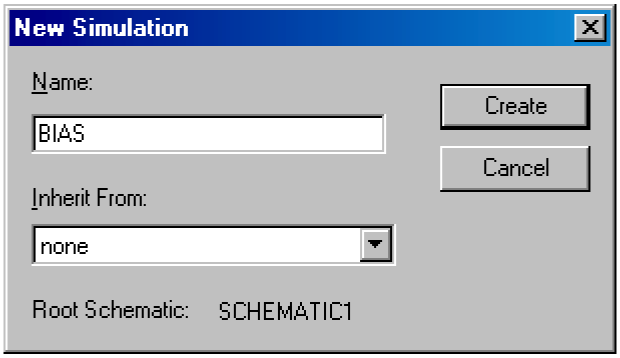


   C) Add any necessary libraries. The default libraries are often sufficient, but the EVAL library is sometimes needed for digital parts, switches, op amps, and more.

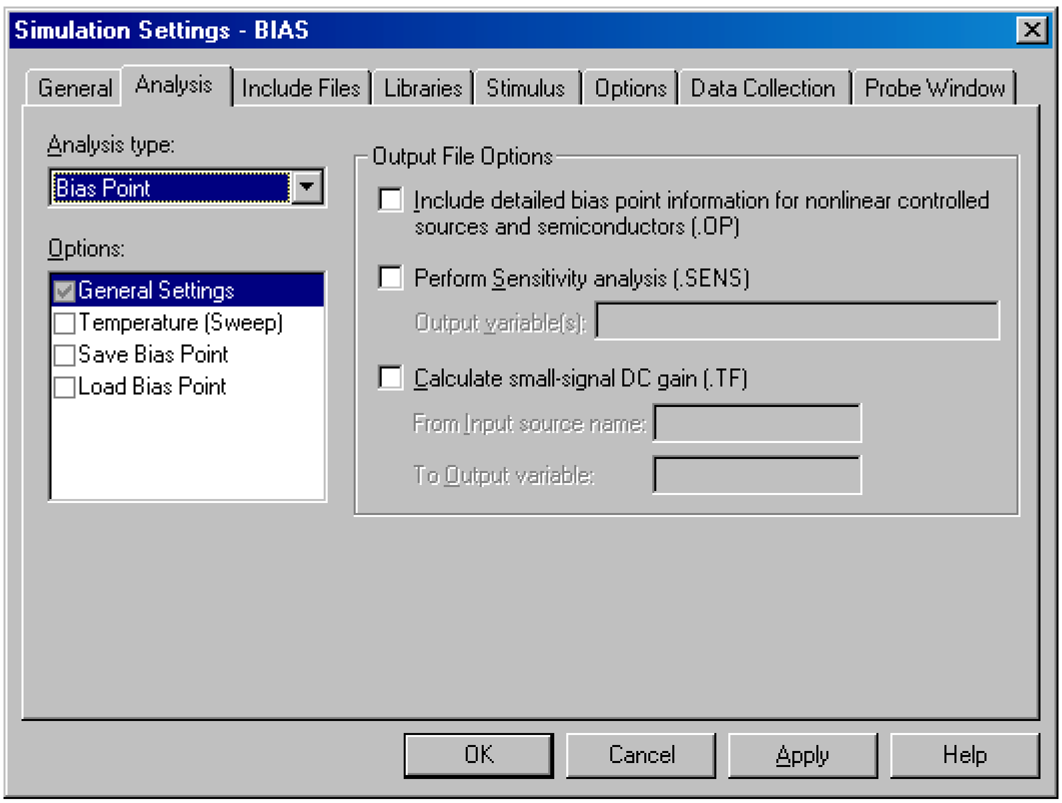


   D) A blank schematic page will appear. Draw the schematic.

2) Create the simulation Profile.
   A) Pick PSPICE – NEW SIMULATION PROFILE from the main menu.
   B) Choose a name for the Simulation Profile. The name BIAS was chosen below since a Bias-point analysis will be performed.



   C) Select the type of analysis.



3) Analyze the circuit.
   A) Pick PSPICE – RUN from the main menu.
   B) Once the analysis screen appears and the analysis is complete, view the OUTPUT file to see the results. See the attached OUTPUT file.

```
**** 02/05/ 22:50:44 *********** Evaluation PSpice  **************

 ** circuit file for profile: BIAS

 ****     CIRCUIT DESCRIPTION

******************************************************************************

** WARNING: THIS AUTOMATICALLY GENERATED FILE MAY BE OVERWRITTEN BY SUBSEQUENT
PROFILES

*Libraries:
* Local Libraries :
* From [PSPICE NETLIST] section of pspiceev.ini file:
.lib nom.lib

*Analysis directives:
.PROBE
.INC "dc circuit-SCHEMATIC1.net"

**** INCLUDING "dc circuit-SCHEMATIC1.net" ****
* source DC CIRCUIT
V_V2         0 D DC 24Vdc AC 1Vac
V_V1         A 0 DC 10Vdc AC 1Vac
R_R1         A B  2.7k
R_R2         B C  2.2k
R_R3         0 B  4.7k
R_R4         C D  3.3k
I_I1         C 0 DC 10mAdc AC 1Aac

.END

**** 02/05/ 22:50:44 *********** Evaluation PSpice  **************

 ** circuit file for profile: BIAS

 ****     SMALL SIGNAL BIAS SOLUTION       TEMPERATURE =   27.000 DEG C

******************************************************************************

 NODE   VOLTAGE     NODE   VOLTAGE     NODE   VOLTAGE     NODE   VOLTAGE

(    A)   10.0000  (    B)   -8.7063  (    C)  -28.0240  (    D)  -24.0000
Note: Node voltages are shown using the letters assigned using OFFSET symbols

    VOLTAGE SOURCE CURRENTS
    NAME         CURRENT

    V_V2         1.219E-03   Note: The current through voltage sources is shown
    V_V1        -6.928E-03

    TOTAL POWER DISSIPATION   4.00E-02  WATTS
                              Note: Total power dissipation is shown
          JOB CONCLUDED

          TOTAL JOB TIME             .27
```