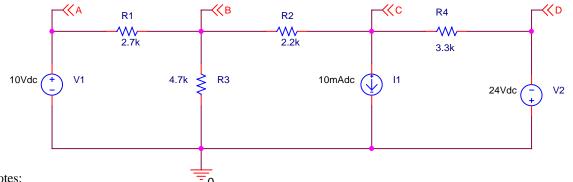
#### EGR 260 Circuit Analysis File: DC Circuit.opj

# 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.

4

- 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)	Analy	vze	the	circuit.	
-,		4-1	DO	DICE	T

5

- 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	<b>-</b>							
	<title>&lt;/td&gt;&lt;td&gt;&lt;/td&gt;&lt;td&gt;&lt;/td&gt;&lt;td&gt;&lt;/td&gt;&lt;td&gt;&lt;/td&gt;&lt;td&gt;&lt;/td&gt;&lt;td&gt;&lt;/td&gt;&lt;td&gt;&lt;/td&gt;&lt;/tr&gt;&lt;tr&gt;&lt;td&gt;Size&lt;/td&gt;&lt;td&gt;Document Number&lt;/td&gt;&lt;td&gt;&lt;/td&gt;&lt;td&gt;&lt;/td&gt;&lt;td&gt;&lt;/td&gt;&lt;td&gt;&lt;/td&gt;&lt;td&gt;Rev&lt;/td&gt;&lt;td&gt;&lt;/td&gt;&lt;td&gt;&lt;/td&gt;&lt;/tr&gt;&lt;tr&gt;&lt;td&gt;A&lt;/td&gt;&lt;td&gt;&lt;Doc&gt;&lt;/td&gt;&lt;td&gt;&lt;/td&gt;&lt;td&gt;&lt;/td&gt;&lt;td&gt;&lt;/td&gt;&lt;td&gt;&lt;/td&gt;&lt;td&gt;&lt;Rev&lt;/td&gt;&lt;td&gt;v¢o&lt;/td&gt;&lt;td&gt;de&lt;/td&gt;&lt;/tr&gt;&lt;tr&gt;&lt;td&gt;Date:&lt;/td&gt;&lt;td&gt;Saturday, February 05&lt;/td&gt;&lt;td&gt;Sheet&lt;/td&gt;&lt;td&gt;1&lt;/td&gt;&lt;td&gt;of&lt;/td&gt;&lt;td&gt;1&lt;/td&gt;&lt;td&gt;&lt;/td&gt;&lt;td&gt;&lt;/td&gt;&lt;td&gt;&lt;/td&gt;&lt;/tr&gt;&lt;tr&gt;&lt;td&gt;&lt;/td&gt;&lt;td&gt;2&lt;/td&gt;&lt;td&gt;&lt;/td&gt;&lt;td&gt;1&lt;/td&gt;&lt;td&gt;1&lt;/td&gt;&lt;td&gt;&lt;/td&gt;&lt;td&gt;&lt;/td&gt;&lt;td&gt;&lt;/td&gt;&lt;td&gt;&lt;/td&gt;&lt;/tr&gt;&lt;tr&gt;&lt;td&gt;&lt;/td&gt;&lt;td&gt;&lt;/td&gt;&lt;td&gt;&lt;/td&gt;&lt;td&gt;&lt;/td&gt;&lt;td&gt;&lt;/td&gt;&lt;td&gt;&lt;/td&gt;&lt;td&gt;&lt;/td&gt;&lt;td&gt;&lt;/td&gt;&lt;td&gt;&lt;/td&gt;&lt;/tr&gt;&lt;/tbody&gt;&lt;/table&gt;</title>							

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.

New Project	×
Name DC Circuit Create a New Project Using	OK Cancel <u>H</u> elp
<ul> <li>Analog or Mixed-Signal Circuit Wizard</li> <li>PC Board Wizard</li> <li>Programmable Logic Wizard</li> <li>Schematic</li> </ul>	Tip for New Users The Analog or Mixed-Signal Circuit Wizard is the quickest way to get started designing and simulating an analog-digital schematic design.
L <u>o</u> cation c:\My Documents\ORCAD files	B <u>r</u> owse

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.

Analog Mixed-Mode Project Wiz	zard		×
Select the PSpice Part symbol libraries that you wish to include in your project. analog_p.olb breakout.olb eval.olb	Add >> << Remove	Use these libraries analog.olb source.olb sourcstm.olb special.olb	
	Finish	Cancel	Help

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.

New Simulation		×
<u>N</u> ame:		Create
BIAS		Cancel
Inherit From:	<b>_</b>	
Root Schematic:	SCHEMATIC1	

C) Select the type of analysis.

Simulation Settings - BIAS	×
General Analysis Include File:	s   Libraries   Stimulus   Options   Data Collection   Probe Window
Analysis type: Bias Point  Dptions:  General Settings  Temperature (Sweep) Save Bias Point Load Bias Point	Output File Options         Include detailed bias point information for nonlinear controlled sources and semiconductors (.OP)         Perform Sensitivity analysis (.SENS)         Output variable(s):         Calculate small-signal DC gain (.TF)         From Input source name:         To Output variable:
	OK Cancel <u>A</u> pply Help

### 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.

\*\* 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 0 D DC 24Vdc AC 1Vac V\_V2 A 0 DC 10Vdc AC 1Vac V\_V1 R\_R1 AB 2.7k BC 2.2k r r2 0 в 4.7k r r3 C D 3.3k r r4 I Il C 0 DC 10mAdc AC 1Aac .END \*\* circuit file for profile: BIAS \* \* \* \* TEMPERATURE = 27.000 DEG C SMALL SIGNAL BIAS SOLUTION 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 -6.928E-03 V V1 TOTAL POWER DISSIPATION 4.00E-02 WATTS Note: Total power dissipation is shown JOB CONCLUDED TOTAL JOB TIME .27