PSPICE: device non-uniformity modeling and other examples

Lecture 10 hands-on companion

Special Topics:
Device Modeling
Hands-on session model

- PV mini-module with 3 cell connected in series
- Each cell’s area is 1x3cm
- Each cell is presented as 3 sub-cells connected in parallel
PSpice model

- PSpice schematics for 3x3 mini-module
- Diode will be modeled with Model Editor
- Resistors, and other parts are from the standard P Spice component library
**PSpice: Model editor**

- Model Editor allows creating new device models, editing parameters for many standard devices types, and creating subcircuit models for more complex device types (e.g., operational amplifiers).
- Simpler devices, such as resistors, may only need the resistance value to have a complete model.
- In the Lite version of Pspice only Diode model can be modified.
Model editor: CdTe diode model

- Open Model Editor
- Select Model-> Copy From; Under Brows locate library:
  - “C:\OrCAD\OrCAD_16.6_Lite\tools\pspice\library\evalp.lib”
  - Select pdiode model; type new model name “DiodePV”
- Set the following parameters:

<table>
<thead>
<tr>
<th>Property Name</th>
<th>Description</th>
<th>Value</th>
<th>Default</th>
<th>Unit</th>
<th>Distribution</th>
<th>Postol</th>
<th>Negtol</th>
<th>Editable</th>
</tr>
</thead>
<tbody>
<tr>
<td>IS</td>
<td>Saturation current</td>
<td>7.6E-11</td>
<td>10f</td>
<td>A</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>N</td>
<td>Emission coefficient</td>
<td>1.5</td>
<td>1</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>CJ0</td>
<td>Junction capacitance</td>
<td>5E-9</td>
<td>0</td>
<td>F</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>VJ</td>
<td>Junction potential</td>
<td>0.7</td>
<td>1</td>
<td>V</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>RS</td>
<td>Ohmic resistance</td>
<td>0</td>
<td>0</td>
<td>Ohm</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>TT</td>
<td>Transit time</td>
<td>0</td>
<td>0</td>
<td>sec</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>M</td>
<td>Grading coefficient</td>
<td>0.5</td>
<td>0.5</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>EG</td>
<td>Activation energy</td>
<td>1.11</td>
<td>1.11</td>
<td>eV</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>XTI</td>
<td>Isat temperature exp</td>
<td>3</td>
<td>3</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>KF</td>
<td>Flicker noise coef.</td>
<td>0</td>
<td>0</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>AF</td>
<td>Flicker noise exp.</td>
<td>1</td>
<td>1</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>FC</td>
<td>Depletion cap. coeff.</td>
<td>0.5</td>
<td>0.5</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>BV</td>
<td>Rev breakdown volt</td>
<td>100</td>
<td>100</td>
<td>V</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>IBV</td>
<td>I at V-breakdown</td>
<td>0.001</td>
<td>0.001</td>
<td>A</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
PV diode model

- Saturation current $I_S$, emission coefficient (ideality factor) $N$, light generated current $I_L$, and temperature $T$ define $V_{OC}$:

$$V_{oc} = \frac{NkT}{q} \ln \left( \frac{I_L}{I_S} + 1 \right)$$

- $I_L$ is modeled with DC current source
- $k$ – Boltzmann's constant, $q$ – electron charge, $T=300\text{K}$, $N=1.5$
- For $I_L=21.1\text{mA}$, resultant $V_{OC}=754\text{mV}$
Editing circuit model

- In Capture open project module3x3-> Module3by3.opj
- Under Schematic1 node double click Module3x3, this will open schematic page
- Select any Diode part and right-click->Associate Pspice model and associate with the model file you created
- Under Pspice ->Edit Simulation profile-> Configuration files->Libraries check for diodepv.lib; if it is not present, click Browse, find your model file, and Add to Design
- Under Analysis check setting to DC sweep, V2 sweep parameters: -2 to 3V, increment 0.1
Running the model

- Run Pspice (ampton) - you should get IV curve shown above
- $V_{OC}=3 \times 754\text{mV}$, $J_{SC}=3 \times 21.1\text{mA}$ (3x3 mini-module)
Running the model

- Toggle cursor (Trace->Cursor->Display) and find the value of $V_{OC}$ at $I=0$
- Switch back to the Capture schematic page, change the resistance of $R_{sh2}$ to 0.001, and re-run the model
- Again toggle cursor (Trace->Cursor->Display) and find the value of $V_{OC}$ at $I=0$
- The difference represents loss due to a dead shunt
References

- OrCAD Capture user manual
- OrCAD PSpice user manual