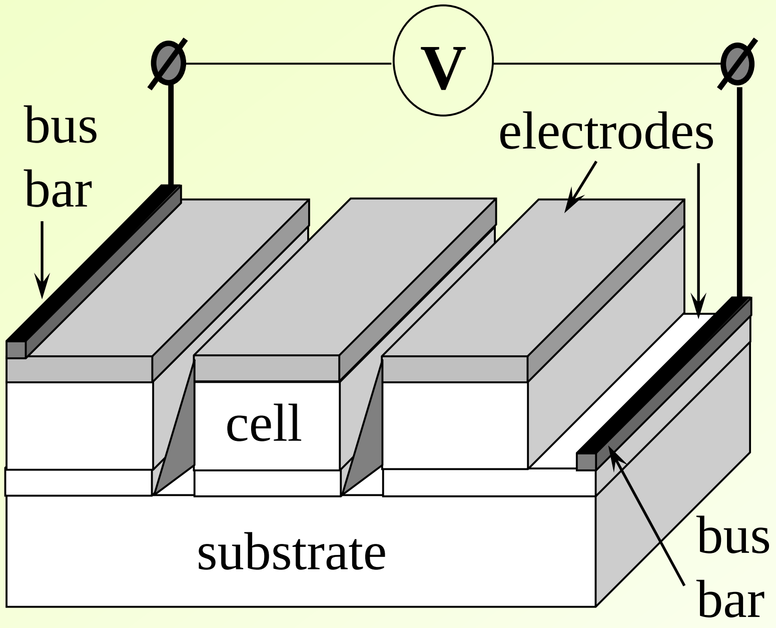
PSPICE: device non-uniformity modeling and other examples

Lecture 10 hands-on companion

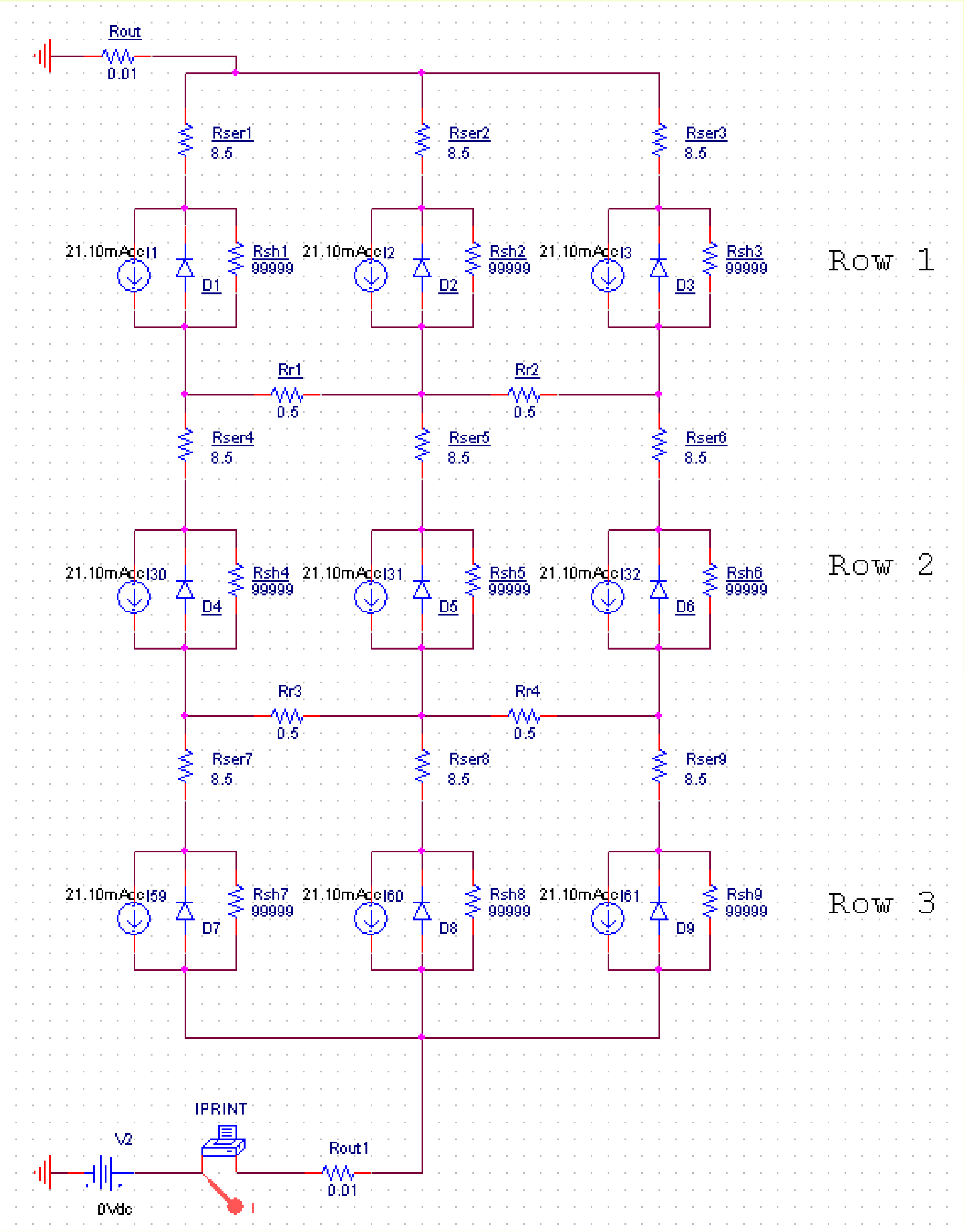
Special Topics:

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 PSPICE 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:

Simulation Parameters

Property Name	Description	Value	Default	Unit	Distribution	Postol	Negtol	Editable
IS	Saturation current	7.6E-11	10f	A				☑
N	Emission coefficient	1.5	1					☑
CJO	Junction capacitance	5E-9	0	F				☐
VJ	Junction potential	0.7	1	V				☐
RS	Ohmic resistance	0	0	Ohm				☐
TT	Transit time	0	0	sec				☐
M	Grading coefficient	0.5	0.5					☐
EG	Activation energy	1.11	1.11	eV				☐
XTI	Isat temperature exp	3	3					☐
KF	Flicker noise coef.	0	0					☐
AF	Flicker noise exp.	1	1					☐
FC	Depletion cap. coef.	0.5	0.5					☐
BV	Rev breakdown volt	100	100	V				☐
IBV	I at V-breakdown	.001	.001	A				☐

PV diode model

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

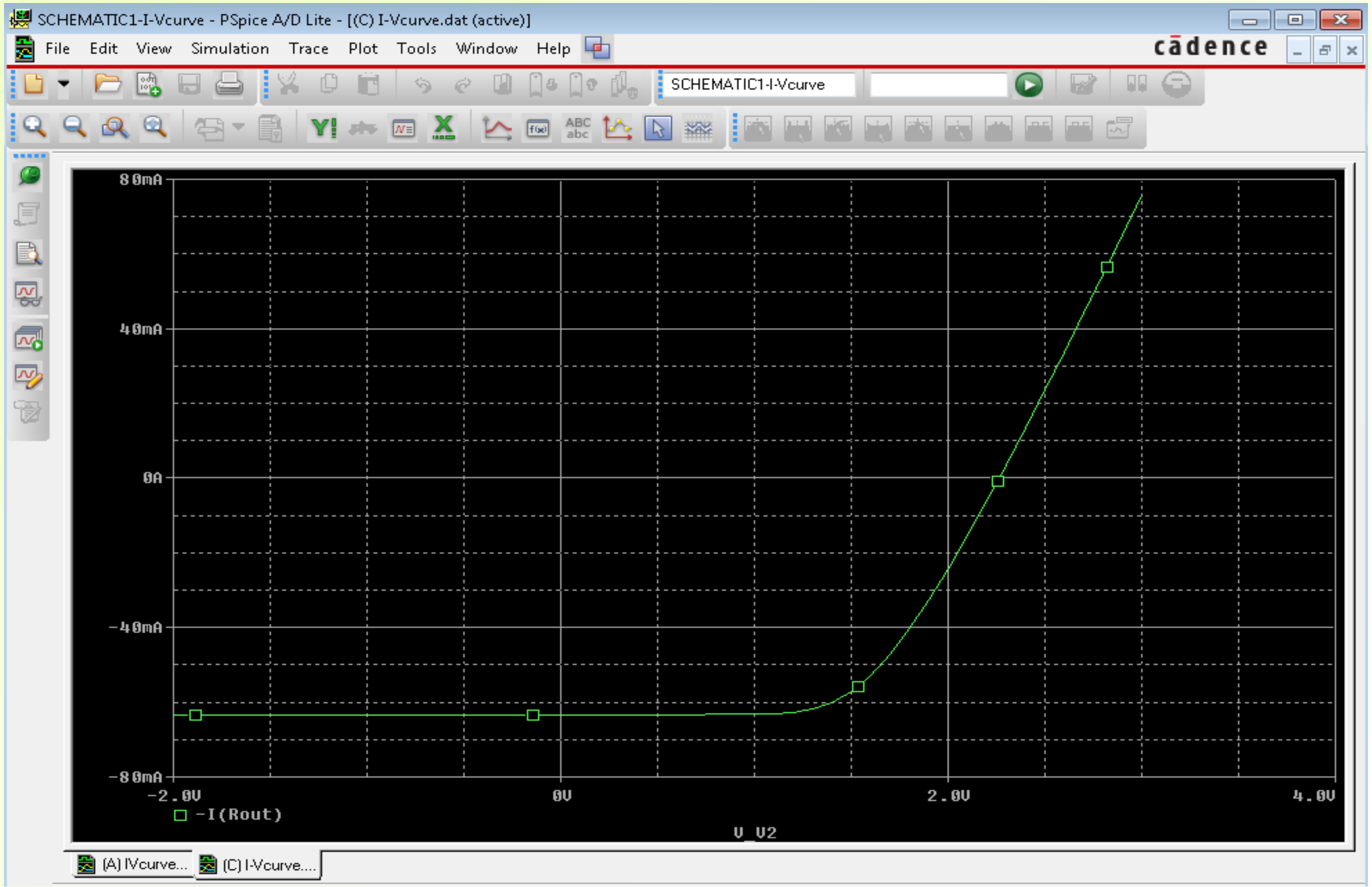
$$V_{OC} = \frac{NkT}{q} \ln\left(\frac{IL}{IS} + 1\right)$$

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

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 ( ) - you should get IV curve shown above
- V_{OC}=3 x 754mV, J_{SC}=3 x 21.1mA (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 Rsh2 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
- Diana Shvydka and V. G. Karpov, Power generation in random diode arrays, Phys. Rev. B 71, 2005, pp. 115314-1-5.