

Capture and PSpice

An Introduction to Electronic CAE at MSU

© 2004, 2006 John Getty

Documenting a Project

- Start a documentation folder
- Summarize design criteria
- Brainstorm designs, evaluate and select
- Enter the chosen design in Capture
- Perform a SPICE analysis
- Use Layout to create the PCB artwork
- Import the data base to ISOPRO and mill PCB
- Build and test the PCA
- Organize and then archive the documentation

© 2004, 2006 John Getty

Blizzard of Names, a Short History

SPICE, Simulation Program with IC Emphasis - 1972

↓

Microcap, HSPICE, MicroSim – 1980's

↓

OrCAD, complete PC based design environment - 1989

↓

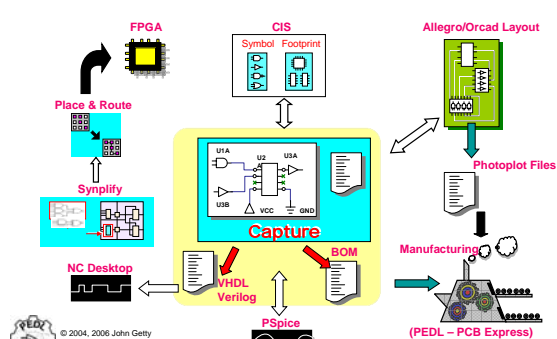
Cadence purchases OrCAD
Uses PSD distribution scheme, early 21st Century

↓

OrCAD distribution assumed by reseller (EMA)

© 2004, 2006 John Getty

OrCAD Design Flow



The diagram illustrates the OrCAD design flow. It starts with 'Capture' at the center, which is connected to 'FPGA', 'CIS Symbol Footprint', 'Synplify', 'Place & Route', 'NC Desktop', 'VHDL Verilog', 'PSpice', and 'BOM'. From 'Place & Route', the flow goes to 'Allegro/OrCAD Layout', then to 'Photoplot Files', and finally to 'Manufacturing (PEDL - PCB Express)'.

© 2004, 2006 John Getty

Capture Design Flow

1. Create a new project
2. Place and connect parts
3. Assign part references
4. Check design rules
5. Add inter-sheet references
6. Cross reference report
7. Edit part and net properties
8. Netlist for PCB design
9. Generate Bill of Materials
10. Backannotate from PCB design

© 2004, 2006 John Getty

Capture Learning Goals

- Create a new Capture project, save to a new directory
- Identify the Project Manager
- Open and hide schematic pages
- Add and remove libraries from the project
- Search for parts, note difference between searches for '74', '74*', '*74'
- Add *components* to a schematic, determine presence of PSpice *model*
- Navigate schematic with zoom in and out ("i" and "o") and refresh (F5)
- Open, pivot and close part's properties, change property values
- Wire the schematic
- Run annotation and design rules check
- Create a Bill of Materials (BOM)

© 2004, 2006 John Getty

Map \\gilbert\micromill

- Login using your MSU account
- Map the OrCAD data drive
 - Right click on My Computer>Map Network Drive
 - In the Folder box type \\gilbert\micromill
 - Click “different user name”
 - Enter PHYSICS\yourusername (or MSU) and the password for your Physics domain login



Create a directory structure

- Under \\gilbert\micromill, create a directory using your last name (plus an initial if needed.)
- Under this new directory, create another called “volt_div”, then another under “volt_div” called “ckt”

The directory structure should look like,:
[\\gilbert\micromill\yourname\volt_div\ckt](#)

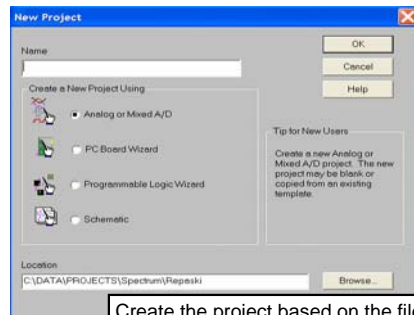


Create Documentation

- Open MS Word, create a new document called volt_div_doc.doc
- Put in here the date, your name and brief description of the circuit.
- Save the file but don't close the Word application



Start a New Project



Create the project based on the file:
[\\gilbert\micromill\MSU Templates\msu.opj](#)

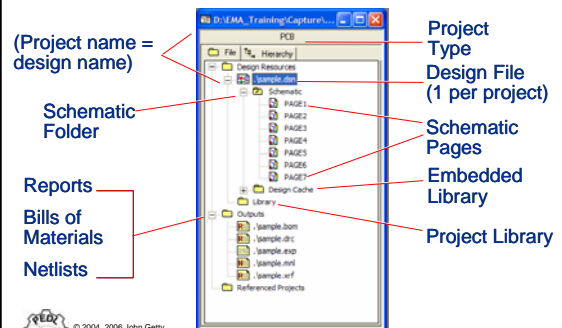


Common File Extensions

.OPJ	Capture project file
.DSN	Capture design file
.OLB	Capture part library file
.UPD	Property update file
.DRC	Design rules check file
.BOM	Bill of Materials file
.EXP	Export properties file
.MNL	Layout netlist file
.SWP	Layout backannotation file
.XRF	Cross-reference report
.NET or .ASC	Other netlist files
.VHD or .VHO	VHDL source file
.EDF	EDIF 2.0 netlist



Project Manager Window



Capture Tool Bars

Main Tool Bar

- File operations
- Editing
- Auto Tiling
- Parts History List
- Viewing
- Design Processing
- Snap to Grid Setting
- Online Help
- View Project Manager

Schematic Tool Bar

- Electrical Objects
- Selection Mode
- Non-Electrical Objects

Add the "Analog" Library

- Double Click on Page1 in the Project Manger
- Add the PSpice library "analog.olb"

Draw the Circuit

- Add parts
 - R – analog library
 - Vdc – source library
 - Off-Page Connector
 - "0" ground ("0" reqd!)
- Connect the parts (wire)

Automatic Reference Designator Assignment

Design Rules Check

Create BOM to List Footprints

Highlight *.DSN>Reports>CIS...>BOM Add Footprints to export and select export to Excel

Qty	Value	Pit Ref	PCB Footprint
1	D1N4468/-55C/RAD	D1	DO-41
2	D1N4468/-55C/RAD	D2	DO-41
3	CON9	J2	
4	100k	R1	AX/RC05
5	100k	R2	AX/RC05
6	100k	R3	AX/RC05
7	100k	R4	AX/RC05
8	10k	R5	VRES16
9	LM741	U1	DIP-1008/W.300L.450

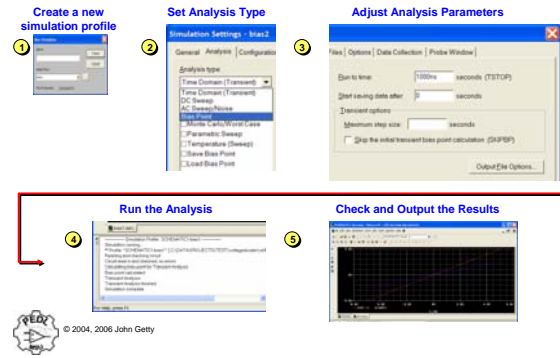
Fini (with schematic capture)

- Now would be a good time to update the documentation
- Printing can be done to virtually any size paper
- Next is PSpice circuit analysis

5 minute break



PSpice Analysis Flow

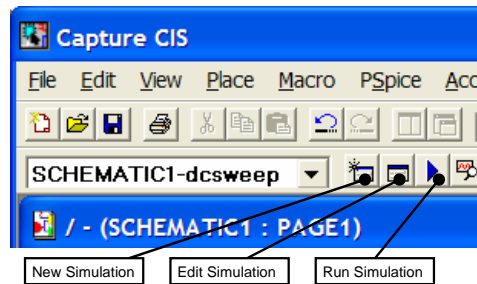


PSpice Learning Goals

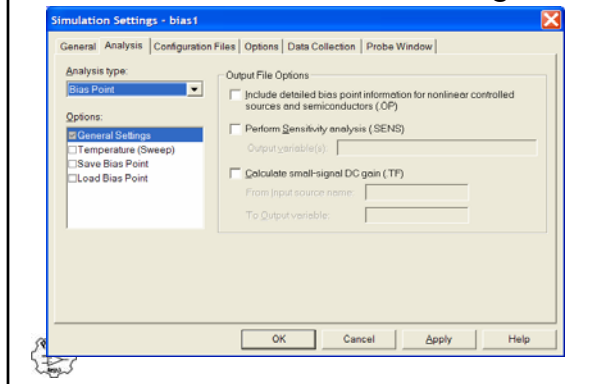
- Create a Simulation Profile, understand some of the parameters
- Generate and examine a PSpice netlist, examine the syntax
- Run a bias analysis on a voltage divider
- On your voltage divider circuit, run a DC Sweep analysis
- Gain familiarity with the PROBE application
- Plot the results of the PSpice DC Sweep analysis



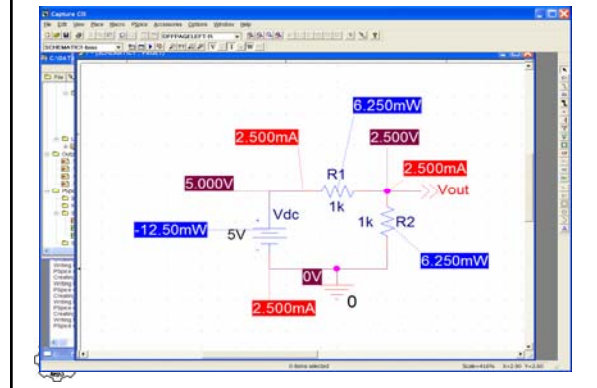
New Simulation Profile

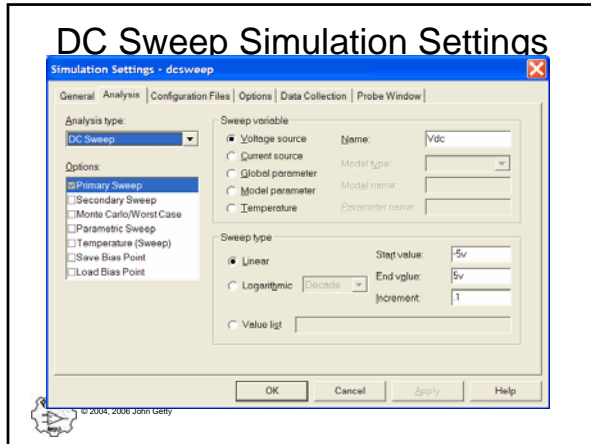


DC Bias Simulation Settings



Bias Point Display





Complete documentation

- Use alt-PrtScr to capture Bias Point and DC Sweep display
- Paste these into your Documentation
- Cut 'n paste the BOM into the Documentation
- Save this Word file to the project directory
- Archive the directory to CD
- Print copies of documentation & schematic

Finì, Finì

Now go forth, draw schematics and prosper.

References

1. A Brief History of SPICE, <http://www.ecircuitcenter.com/SpiceTopics/History.htm>, accessed 3/28/2004
2. Paula M. Clark, Sr. Technical Instructor/AE, EMA Design Automation, 64 West Street, Milford, NH 03055, (603)673-9183

