Cadence Virtuoso With MMSIM

Course for low power analog and digital circuit Design

S No.	Module	Name of the Topic	Prerequirement/Others
1	Module 1	Introduction to circuit simulator	**Mosfet Basics**
2		Open source circuit Simulator LT Spice	
3		LT spice Schematic, Library	
4		Different type of circuit analysis	
5		PWL waveform creation	**Strong knowledge of analog and digital circuit**
6		Analog and Digital Circuit with simulator	**Shortcuts,Library browsing etc**
7		Limitation of LT spice	
8		Waveform Tools	
9		Full operation with netlist concept	
10		Model file & Analysis with Model file	
11	Module 2	Matlab Basics	**Matlab required to provide strong calculation functions**
12		Plot command in Matlab	
13		Lt spice Interfacing with Matlab	
		Assignment	
14	Module 3	Linux Platform basics	**How to use the terminal to perform non gui operation?**
		Assignment and Tutorials	
15	Module 4	Cadence virtuoso Basics,	**Support spice, hspice, spectre, verilog, vhdl ,Matlab etc**
16		Schematic operation	**Shortcuts,Library browsing etc**
17		ADE windows utility	**Ac, transient,DC, noise etc**
18		Different type of Basic analysis	
19		Parametric analysis	
20		Corner analysis	
21		Montecarlo analysis	
22		Basic use of calculator	
23		Matlab interface for plots	
24		Script based operation	
25		CDB to OA Conversion	**Require for newer version of MMSIM**
Assignment (Analog and Digital circuit based)			
Books		**Rabey**	**Digital integrated circuit**
		Jacob Baker	**Cmos IC and layout design**

Design by Rajeev Pandey