DOWNLOAD PDF PSPICE 9.1 MANUAL

Chapter 1 : Page 5 â€“ Electronics-Lab

2 4. After clicking OK, the Create PSPICE Project dialog box will pop up. It will ask you to choose which type of project you want to create. 5. Once you have clicked OK in the Create PSPICE Project dialog box, the schematic window.

This performs some basic checks on your circuit listing the errors if there are any. Enter zero if no offset required. There are other fields in this dialogue box but they can be left empty as default. Set up your input sinewave as shown above. Figure 3 You now need to enable and setup the analysis conditions for your circuit. A screen appears as shown in Figure 4. To enable a particular analysis, click in the box and a tick appears. More than one kind of analysis can be enabled at once. The types you are most likely to use are Transient: A time controlled analysis AC Sweep: A frequency response analysis DC Sweep: Produces the voltage transfer characteristic plotting Vx against Vy. More than one simulation type can be enabled at one time and the different plots viewed in the Probe display. Figure 4 The final time you choose will of course depend on the frequency of the AC input signal. If you have a sine wave input frequency 1khz, then choosing a final time of us shown below will simulate for 2 entire cycles of the input signal. The probe output screen starts automatically by default. Set up the Transient analysis as shown above in Figure 5. The current marker has to positioned on a device pin, an error message will display if you try to place the current marker on the wire. Multiple markers can be used to display values from different parts of the circuit. There is a problem around scale. Typically the currents will be of orders less than the voltages. As they are displayed on the same graph, this would normally show the current as a straight line along the origin. To view the currents, delete the voltage traces off the plot by selecting the particular voltage on the legend and hit the delete key. Figure 7 Figure 7 shows the trace from the circuit you have constructed if the markers were placed as shown in Figure 6. If your trace does not look like this, check your circuit. Make sure your input sinwave voltage is set up correctly. A box is present which tells you the waveform values at that position of the cursor. You could do this for example, to measure the current value without deleting the voltage trace. To change the voltage or current the cursor is tracing, click on the symbol by the side of the plot name eg Ic Q2 Common simulation errors: Figure 8 shows the box that appears using the following command. Assuming the simulation is error free, this normally happens when in setup you only have "bias point detail" enabled. Enable a more useful simulation. This pages are just a starter guide to Pspice. There is an extensive Help File for both the Schematics and the Probe sections. Make sure you are looking at the correct Help files. There are also Pspice simulation manuals held by the technicians on the LG floor. For the purpose of the tutorial questions the parametes of the FET to be entered are After starting Schematics, using Draw-get new part, get part Mbreakn3 from the breakout library. This is a generic model for an NMOS transistor with the Bulk connected to the Source, Select the NMOS transistor, it turns red, then using edit-model-edit instance model to bring up the parameter box below. This is because these capacitances are a feature relating to the width. There is no explicit parameter for cds. To set the width and length parameters, dble-click on the device and the parameter box below appears. Enter the required width and length as shown above. All the other parameters can be left blank as they will assume a default value. Plotting Small Signal Characteristics in Spice:

Chapter 2 : Page 5 â€“ Electronics-Lab

Manual Pspice 1. SimulaciÃ³n AnalÃ³gica PSPICE Dpto. Sistemas ElectrÃ³nicos y de Control Apuntes Pspice VersiÃ³n Dpto.

Chapter 3 : PSpice Starter Manual

Home Downloads Circuit Design - Emulation PSPICE student version PSPICE student version blog.quintoapp.com - A/D Reference Manual blog.quintoapp.com - Optimizer User's Guide.

DOWNLOAD PDF PSPICE 9.1 MANUAL

Chapter 4 : PSPICE For Beginner - PDF Free Download

1 PSPICE Schematic Student Tutorial --X. Xiong This tutorial will guide you through the creation and analysis of a simple MOSFET circuit in.

Chapter 5 : PSPICE For Beginner - PDF Free Download

Pspice Student Version Manual So the lab report is the chance students have to show that they can successfully build upon instructor's feedback, they have one week to resubmit their revised version.

Chapter 6 : PSPICE student version

analyses in PSpice. Included in this manual are detailed command descriptions, start-up option definitions, and a â€¢ PSpice your Microsoft Windows User's Guide.

Chapter 7 : PSPICE Student Version - S4Student Tech

PSpice Schematics is a schematic capture front-end you learn and use PSpice Schematics efficiently, this manual is separated into the following sections.

Chapter 8 : PSPICE student version

Version , which is free from ORCAD via either CD or download. (Go to ORCAD's PSpice and the part's pin numbers. When a part is placed, it is automatically.

Chapter 9 : PSpice Starter Manual

PSPICE LAB MANUAL ECE-BEC 3 PROGRAM: 1 AIM: To verify the characteristics of Low pass and High pass filter CIRCUIT Here's a simple circuit for you to dive into running SPICE simulations and plotting results.