Tutorial of HP-ADS2002

for EE710

Patrick Roblin and Xian Cui

October, 2002
1. Tutorial for ADS

1.1 First go through the Quick Start available at:


Read the following sections:

- Projects in ADS
- Designs in ADS
- Simulations in ADS
- Results in ADS

1.2 Running OSU demo files

1 Log in a HP workstation (DL 517, DL 557)

2 Create a general directory where you will store all EE710 ADS projects
   - open a Unix window
   - mkdir EE710

3 Start ADS from a Unix window (the ADS main window opens)
   - cd EE710
   - source /apps/adst2002/Startup.EE
   - hpads

4 Create a new project (the schematic window opens):
   - in ADS (Main) windown select File-->New Project ..
   - for the name select: 710demo
   - keep the default length unit of mil

5 In the Unix window copy class demo files from archive:
   - cd
   - cd EE710/710demo_prj/networks
   - cp ~/roblin/EE723/demo/* .

6 Demo of design using ideal transmission lines
Figure 1.1: Schematic of the circuit in demo.dsn

- select File->Open Design ... in the ADS schematic window
- select demo.dsn: the opened dsn file is shown in Figure 1.1
- save that file in case ADS has changed: click on floppy disk icon
- run ADS by clicking on Simulate (gear icon)
- the status summary window will be displayed and a data display window will appear when the simulation has completed.
- in the display window click on the Rectangular Plot icon and position the graph window in the white field of the display window
- the Plot Traces and Attributes window appears: select S(1,1) and click on "Add", select Magnitude and then click on OK and OK again.
- the graph obtained is shown in Figure 1.2
Figure 1.2: Data display—magnitude plot of reflection coefficient (S11) for the circuit in demo.dsn

7 Demo of optimization for ideal transmission lines

- open the demo1.dsn design file (see above), shown in Figure 1.3
- run ADS (see above)
- in this demo, if the optimization process stops very quickly, you can change the
Figure 1.3: Schematic of the circuit in demo1.dsn

TL2.E from 3.231161e+01 to 3.341161e+01 so that you can observe more iterations in the status window.

- to add marker to the curve in the data display window, select Marker->New... and then click on the objective curve
Figure 1.4: The window for changing Goal expression

- if you want to change the default optimization settings, just double click on the rectangular OPTIM block in the schematic window, and then you can change settings in the Nominal Optimization Window, shown in Figure 1.5. Normally, the default settings are OK.
- if you want to change the Goal expression, just double click on the Goal block in the schematic window and adjust settings in a window shown in Figure 1.4
- display data (see above)

8 Demo of microstrip implementation

- open the demo2.dsn design file, shown in Figure 1.6
Figure 1.5: The Nominal Optimization window

- to adjust the settings of microstrip substrate, double click the MSub block in the Schematic window. The Microstrip Substrate window will appear, as shown in Figure 1.7
- run ADS (see above)
- display data (see above)

9 **Demo of optimization for microstrip implementation**

- open the demo3.dsn design file, shown in Figure 1.8
- if you want to adjust optimization settings, see above
- run ADS (see above)
- display data (see above)
1.3 Some Hints for Creating New Project

1 Component Palette List

- in the Schematic window, you can select the desired components from the list is on the upper-left corner of the schematic window
- in this list, TLines-Ideal, TLines-Microstrip, Simulation-S_Param and Optim/Stat/Get/Yield/Doe are very useful for our simulations

2 LineCalc
Figure 1.7: The window for changing parameters of microstrip substrate

- in the Schematic window, select Tool→LineCalc→Start LineCalc to start using LineCalc
- the interface is shown in Figure 1.9

3 Editing Component’s Parameters

- to change a component’s parameters, just double click that component symbol in the Schematic window, and then you can make changes to parameters.
- for example, by double clicking Term in the Schematic window, you can see a window for editing the characteristic impedance of the termination, as shown in Figure 1.10
- A shortcut for getting the online help is to click the ”Help” button, which is also shown in the lower-right corner of Figure 1.10
Figure 1.8: Schematic of the circuit in demo3.dsn
Figure 1.9: The LineCalc window
Figure 1.10: The window for editing characteristic impedance of Term