Chapter 8 Conclusions

A system modeling and simulation procedure is proposed in this chapter. Implementing system simulation on a digital PCB assembly before the actual hardware is built enables the designer to pinpoint potential problems and optimizes system performance on the CAD/CAE system. This would cut down development cost and time. A four step closed-loop digital hardware design sequence is suggested in Figure 8.1. Common models such as PCB traces and discontinuities can be derived on a one-time basis for various PCB configurations and stored as SPICE sub-circuits netlists. A CAD software can be used to convert the schematics into PCB layout in typical formats such as Gerber, DXF, HPGL or other graphical standards. By linking the geometrical information of PCB layout, electrical specifications and physical specifications of the components, a logical connection of the system can be constructed including the components, interconnections and power distribution system. Each of these elements are then assigned suitable SPICE sub-circuits from the database. If the need arises, derivation of models not included in the database can be performed using the methods proposed in Chapter 2 to Chapter 6. Finally a complete SPICE netlist can be declared for the digital PCB system by linking all these sub-circuit netlists in a main netlist declaration. Time and frequency domain simulation are performed for the system to validate the system against parameters such as bandwidth, noise margins or noise immunity, idle and full load power consumption. Signal integrity and stability of the system can also be carried out if required.
Nevertheless modeling a large digital PCB system is not as straightforward as it seems. Summarized below are the obstacles which the writer feels will severely impede the application of system modeling using the methods compiled in this thesis:

- Insufficient workstation and personal computer power. Due to constraint on memory size, memory access time and the central processing unit computing power, simulations are being limited to simple systems for the moment. In order to design a practical digital PCB system with hundreds of components, the system has to be divided into smaller modules to which modeling and simulation are performed. In this respect these smaller modules are assumed to be decoupled from other and they only affect each other through the boundary of the modules comprising of connectors and traces. Even then the task of assigning the appropriate stimulus due to the other modules at the boundary concerned could be very daunting.

- Differential current distribution on power planes due to electrical signal on PCB traces is not incorporated.

- Common-mode current distribution on power planes is ignored. Unintentional radiation is largely due to common-mode current (Daijavad and Rubin 1992, Paul 1989). Full-wave formulation (Naishadham et al 1993) would have to be employed instead of planar circuit approach in order to incorporate common mode current on the power planes.
• Automated model extraction and creation system from CAD layout is required as manual model extraction of model is extremely tedious and time consuming.

• Frequency range of system only valid from d.c. to 5GHz or as long as quasi-TEM approximation remains valid.

• Extremely Large and complex schematics usually encounter convergence and stability problems during numerical solution in SPICE. This is due to the fact that SPICE utilizes multi-dimensional Newton-Raphson method to determine the transient solutions of nodal voltages for the non-linear network during intermediate steps. Risk of non-convergence of solutions increases when the number of nodes is large and with the presence of signals with rapid transition rate.

• Susceptibility analysis is not included, therefore the effect of the system when under external radiation is not investigated.

It is therefore suggested that in future undertaking, efforts should be concentrated on incorporating full wave solution to extract equivalent circuit for systems such as discontinuities and power planes. Common mode current will be taken into account and the models extracted will be valid to a higher frequency.