Creating and Analyzing Circuits

Figure 1-1 illustrates the process of creating a circuit, simulating the circuit, and displaying the simulation results. This process consists of the following:

1 The circuit schematic is created and prepared for simulation using MicroSim Schematics.

The term "simulator" refers to either PSpice A/D or PLogic. PSpice A/D simulates analog, digital, or mixed mode (analog and digital) circuits, and PLogic simulates digital circuits only.
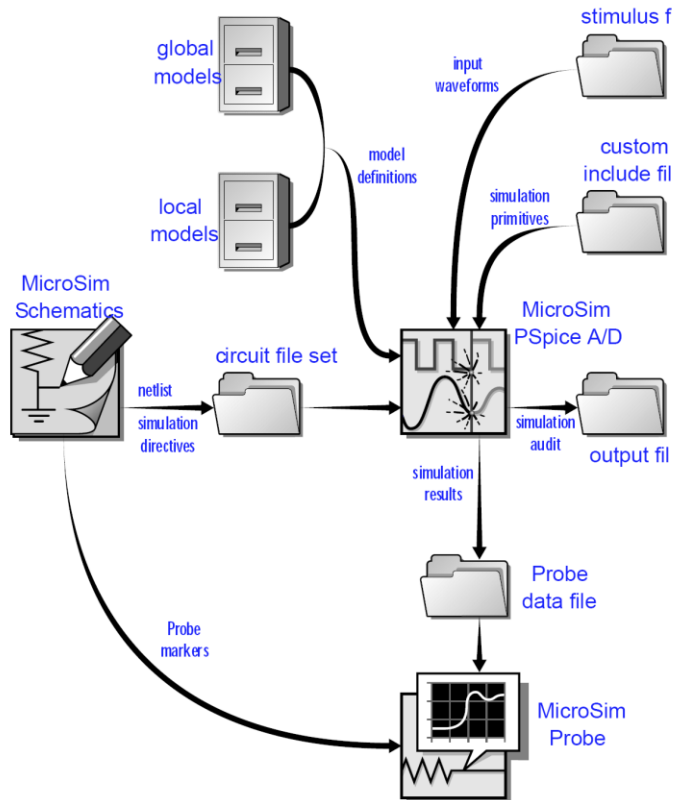
2 Circuit simulation and analysis are performed by either PSpice A/D or PLogic.

3 Probe is used to display simulation results.

Figure 1-1 *Circuit Analysis File Interactions*

2.3 PSpice File Interactions

Figure 1 illustrates the relationship of PSpice to the circuit file and other files of importance. The directives in the circuit file instruct PSpice to read the appropriate library files in order to gather device information for the components used in the circuit netlist.
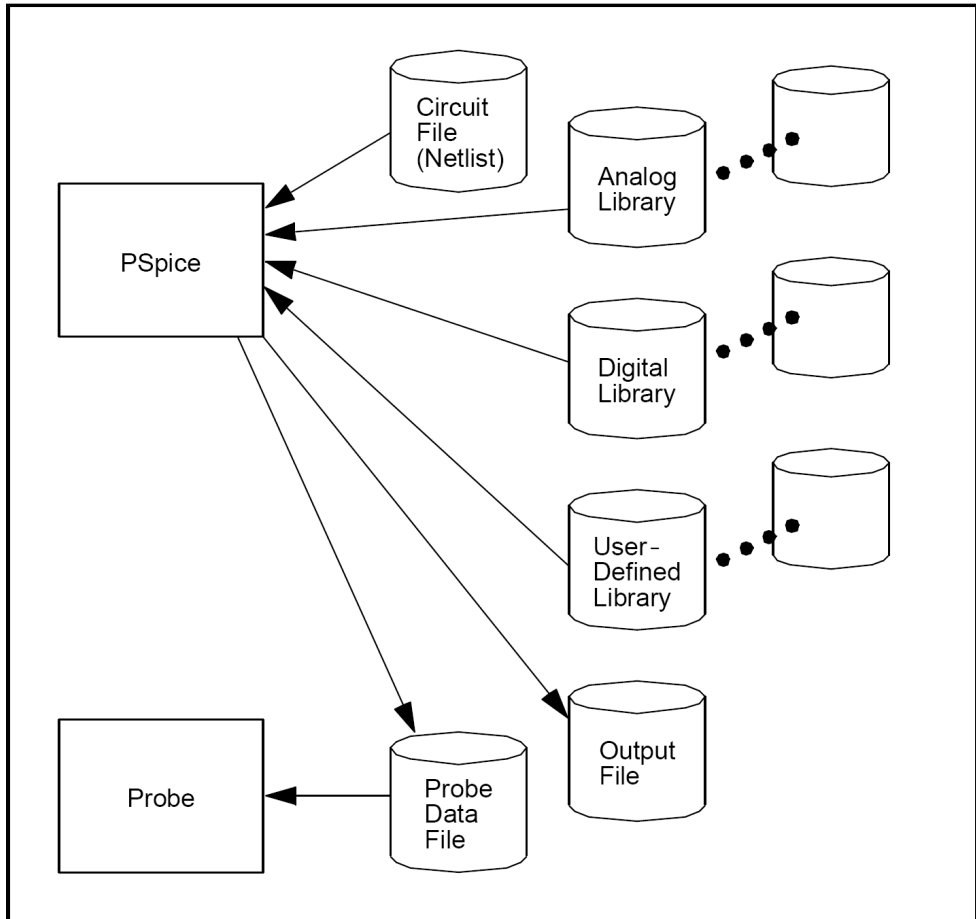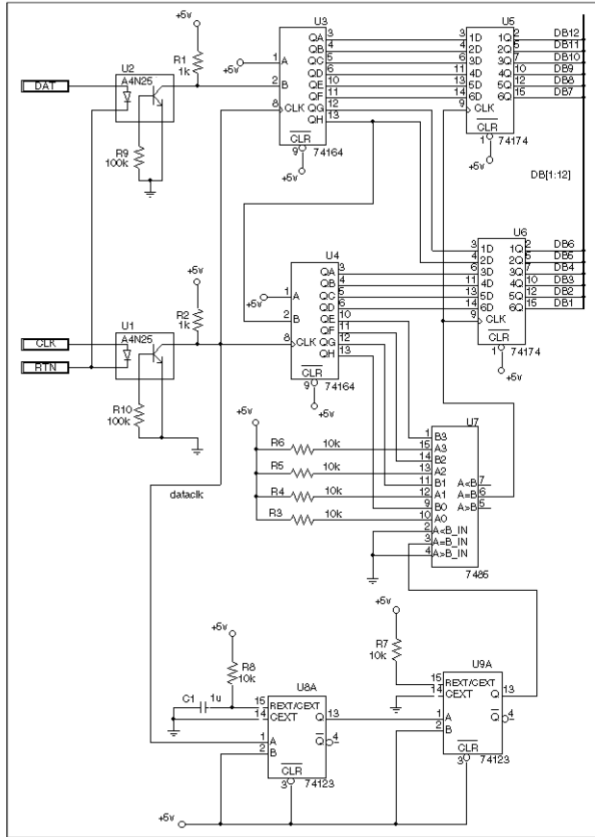

Figure 1: PSpice file interactions

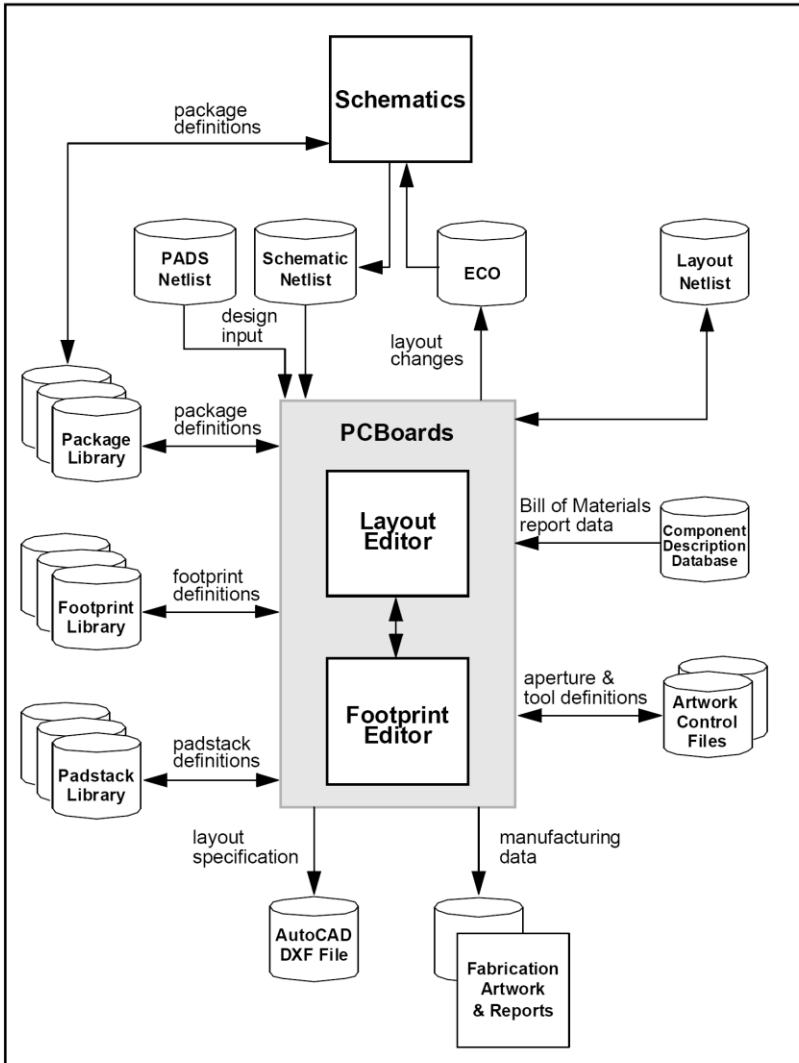**Figure 1-2** *Opto-isolated, Addressable Serial-to-parallel Converter Circuit*
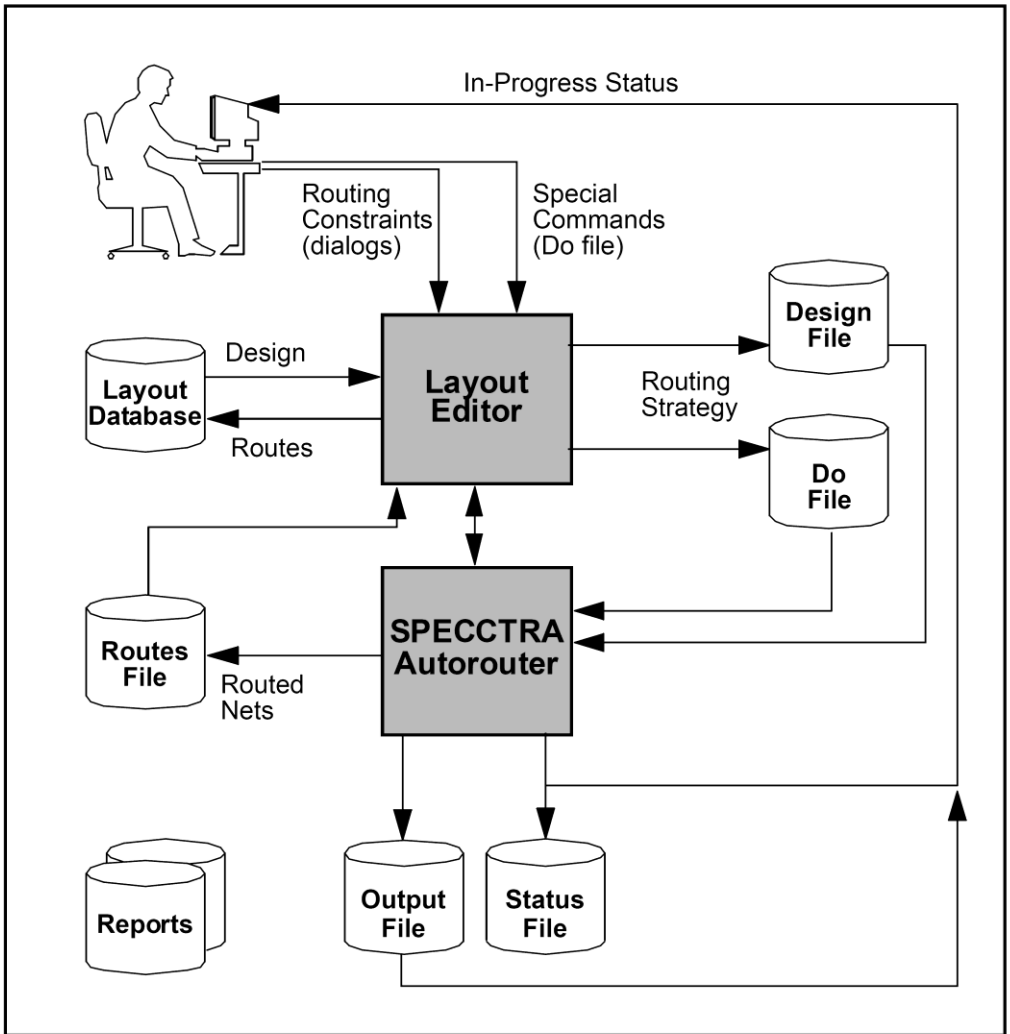
Figure 1: PCBoards file interaction

Figure 1: PCBoards autorouter file interactions