

Download Allegro Pcb Si User Guide

Allegro PCB SI User Guide The High-Speed Design Flows dialog box displays a list of all models in the design and shows related source and status information for each model. For further details, see Chapter 3, Managing Models Resident in a Design, October 2012. 76. Product Version 16.6 Allegro PCB SI User Guide Integrated with Cadence ® Allegro ® PCB and IC package design, editing, and routing technologies, Allegro Sigrity™ SI provides advanced SI analysis both pre- and post-layout. Operating early in the design cycle allows for “what if” scenario exploration, sets more accurate design constraints, and reduces design iterations. Using EMA Timing Designer and Cadence Allegro PCB SI Interface 7 Figure 7: Allegro PCB SI Signal Analysis Dialog B. Run Simulation For this example, you are looking at a single byte lane of one of the on-board memory devices. Cadence Allegro PCB Designer Interface User Guide, V2.23. Allegro PCB Designer Interface. Installing the Allegro Interface. Installing the Allegro Interface. Follow this procedure to install the interface: Procedure. 1. Identify the location where the interface should be installed. The recommended installation directory is the .allegro_site ...