Orcad Tutorial

ajay_bhargav, Sat Apr 14 2007, 09:33 am

An Orcad tutorial for making Schematics and PCB using Orcad Capture and Orcad Layout.
A complete tutorial of 55 pages, containing all the information which you need to know for making a PCB design.

Inside you will come to know.. how to:
1. Create project
2. Create new part symbol or schematic
3. Create new PCB footprint
4. Make PCB design
5. Route, explaining all good and bad aspects
6. Take Output as Gerber Files or Print manager

So its a complete handy tutorial. You must download
Have a nice time making PCBs

Visit Download section for more information.