Installing OrCAD and PSpice onto Your Laptop

These instructions describe how to install and setup the OrCAD suite of tools. This suite of tools will allow you to use the Capture schematic entry tool and the PSpice A/D simulation tool. You will need to be connected to the Rose-Hulman network during the installation. It's best if you are physically on campus using a wired network connection to take advantage of the fast network because the total data transfer is over 600MB. The installation can be done off campus using the Rose VPN software, but it will take much longer. Please **do not** use the wireless network to do the installation, not only will it take forever, if you loose the connection halfway through, you will have to restart the process. You can expect the installation process to take approximately 1 hr. In the instructions below, text that appears in bold small-caps refers to **MOUSE CLICKS** and text that appears in bold-italics refers to ***typed text*** that you must enter into a blank space.

Step 1: Accessing the files on Tibia

- If using WinXP login to your LocalMgr account, If using Win98 just login to your computer with your proper network username and password
- Go to **START -> RUN** from your windows desktop.
- Type ***tibia.rose-hulman.edu\public\apps\cadence\psd15.0\cdrom1*** click **OK**

Setp 2: Installation

- Turn off your virus protection software.
  - To do this with the latest version of McAfee Anti-Virus software: in the lower-right corner of your screen, right-click on the blue shield, then click on **DISABLE ON-ACCESS SCAN**
- Double-click on **SETUP.EXE** in the directory that you just opened on tibia
- The following table lists the windows that appear and the appropriate response for each

Table 1: Windows Requiring a Response that Appear During Installation

Window Title	Actions
Software Licensing Agreement	**YES**
Install Options	**INSTALL PRODUCTS**
License Manager Communication	Use bottom fields only (leave top field blank) Port: ***5280*** Host Name: ***rigel.rose-hulman.edu***
User Information	**NEXT** (Check to see if information is correct)
User Information Confirmation	**NEXT** (Check to see if information is correct)
Control File	***//tibia.rose-hulman.edu/public/apps/cadence/controlfile.txt***
Select Product Group	**SELECT FROM BOTH**
Select Products	The contolfile.txt automatically selected those products that are usable with our current license. However some of these tools are redundant. Look through the list of tools and **unselect** the following two tools: **PCB DESIGN STUDIO W/ CONCEPT HDL** and **PCB DESIGN EXPERT W/ CONCEPT HDL**
Working Directory	Must enter a path without spaces. The default works fine. Click on **NEXT.**
Intellicad popup	**NO**
Footprint Viewer Options	**ORCAD**
Layout Footprint Library	Use the default. **NEXT**

Table 1: Windows Requiring a Response that Appear During Installation

Window Title	Actions
Select Program Folder	This is the name that appears in the Start Menu on your windows desktop. You can choose any name, but these instructions will assume that you choose the default. **NEXT**
Installation Summary	**NEXT**
At this point files will begin copying. This will take several minutes and then you will be prompted with the next two questions.	
Product File Extenstions	**YES**
Text Editor File Extensions	**YES**
The installation software will tell you that it's setting up the environment. This can take over 10 minutes.	
Setup Complete	**FINISH.** It will then take a couple of minutes for the installation program to disappear.
At this point it is best to restart your computer before using the software, even though you are not prompted to do so.	

Step 3: Running the Tools for the First Time

- Login to the normal user account on your computer.
- Before running your tools for the first time, from your desktop go to
  START -> ALL PROGRAMS -> CADENCE PSD 15.0 -> SET CONFIGURATION
- The Product Configuration window will appear, which is divided into three regions each of wich has several possible selections. The defaults should be correct, but just to make sure, the three regions and the correct selection for each is given in the table below:

Table 2: Three Regions of the Product Configuration Window

Region Title	Select
Capture	**CAPTURE CIS**
PSpice	**PSPICE STUDIO**
PSpice Advanced Analysis	**PSPICE STUDIO**

- You can now run the software from the desktop Start Menu:
  START -> ALL PROGRAMS -> CADENCE PSD 15.0 -> CAPTURE CIS
- Every time you start the software, the Studio Suite Selection window will appear. Click on the downward pointing triangle and then scroll down to the selection **PCB DESIGN EXPERT WITH CAPTURE CIS** and then click on **OK**. This will start the OrCad Capture CIS package.