

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 1GB. 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 15-20 min. on the newer computers. 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: Create and Setup the Installation Directory

- If you already have a full version of the OrCAD software installed on your machine, I would recommend that you uninstall it before installing this version. Then reboot your computer after the un-installation.
- If you already have a C:\Cadence directory from a previous version of these instructions, you can skip the rest of this step.
- Login to your Administrator (localmgr) account
- Create the following directory: C:\OrCAD
- You must install AND RUN this software from your Administrator (localmgr) account

Step 2: Copy the files from DFS

- Open up a windows file browser.
- Type \\Rose-hulman.edu\dfs\AcademicAffairs\ECE\CadenceSoftware\ at the top and hit return
- Copy the folder OrCAD16.2 to your laptop

Setp 3: 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**
- Go into the OrCAD16.2 folder that you saved on your laptop and click on **SETUP.EXE**
- 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
Cadence Product Components	PRODUCT/DEMO INSTALLATION
License Agreement	I ACCEPT then NEXT
Turn off virus protection	Should already be done from before, click OK
Control File	Leve blank and click NEXT
Choose Desitnation	Use the default and click NEXT
Select Products	Select all of the primary boxes
Working Directory	Use the default and click NEXT
Footprint Viewer Options	PCB EDITOR FOOTPRINT VIEWER

Table 1: Windows Requiring a Response that Appear During Installation

Window Title	Actions
Layout Footprint Library	Use the default. NEXT
Select Program Folder	NEXT
Installation Summary	NEXT
License Path	Enter <i>5280@rhit-lmgrd1.rose-hulman.edu</i> in the blank
Configuration Status	Click on FINISH
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 Extensions	YES
Text Editor File Extensions	YES
The installation software will tell you that it's setting up the environment. This can take a couple of minutes.	
Setup Complete	Click on FINISH
Restart Windows	This is not an install option. You should do this before updating the software with the hotfix.

Step 4: Installing the Updates

- Turn off your virus protection software
- Go into the OrCAD16.2 folder that you saved on your laptop and double click on *Hotfix_ORCAD16.20.002_wint_1of1.exe*
- If you followed the instructions above, then all the defaults in the pop-up windows should be correct, simply click on **OK** or **NEXT** for all of them

Setp 5: Install the RHIT Libraries

- You must install the RHIT libraries to obtain model files for the parts we will be using in the lab this quarter.
- Go to the following URL and download the libraries for the professional version of OrCad.
http://www.rose-hulman.edu/~herniter/Rose_Classes/classes2.htm
 - Click on the link that says...“Click here to install the Rose-Hulman class libraries for SPB_15_2(600 KB).”
 - Save the file to your desktop and then run it.
 - In the “Choose Destination Location” window, browse and then select the Cadence installation directory from the previous steps, *C:\OrCAD\OrCAD_16.2*, and otherwise, follow the installation defaults

Step 6: Running the Tools for the First Time

- Login to the normal user account on your computer.
- You can now run the software from the desktop Start Menu:
START -> ALL PROGRAMS -> ORCAD 16.2 -> ORCAD CAPTURE CIS
- Every time you start the software, the Studio Suite Selection window will appear. Click on **ALLEGRO PCB DESIGN CIS XL** and then click on **OK**. This will start the OrCad Capture CIS package.
- Once you are sure that you can run the tools, you can delete the folder on your desktop that you downloaded from DFS for installation.