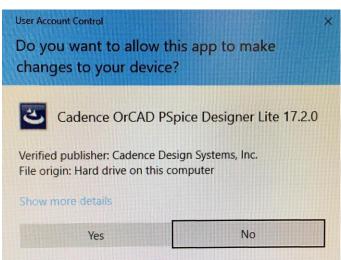
## **Installing PSpice 17.2**

**NOTE:** You cannot have different versions of OrCAD PSpice installed on one computer (to my knowledge). Before attempting this install, remove any previous versions of PSpice.

## **MS Windows 10 computers**

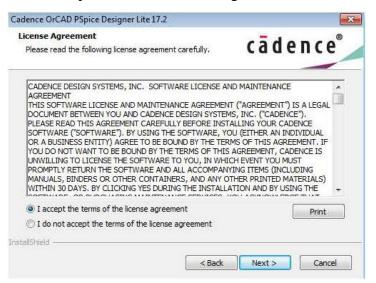
- Download '17.2\_OrCAD\_Lite\_Capture\_PSpice.zip', a zip file with the student/lite version of OrCAD PSpice 17.2 and supporting files/programs, from the EE 220 course webpage under the <u>Labs</u> link or click <u>17.2\_OrCAD\_Lite\_Capture\_PSpice.zip</u>.
- 2) Place the downloaded '17.2\_OrCAD\_Lite\_Capture\_PSpice.zip' in a directory/folder selected by you (e.g., use File Explorer).
- 3) Using File Explorer go to appropriate directory/folder (if necessary) and right click on '17.2\_OrCAD\_Lite\_Capture\_PSpice.zip' ⇒ select Extract All ... A subfolder or directory named 17.2\_OrCAD\_Lite\_Capture\_PSpice should be created.
- 4) Go into subfolder or directory named 17.2\_OrCAD\_Lite\_Capture\_PSpice. Now, double click 'setup.exe' to begin installation.
- 5) You should see the window below. Click 'Yes'.



6) At window, click 'Next >'.



7) At window, click button to accept terms of the license agreement. Then, click 'Next >'.



8) At window, select 'Anyone who uses this computer (all users)' button. Click 'Next >'.

Cadence OrCAD PSpice Designer Lite 17.2	
Setup Type Select the setup type to install.	cādence°
Install this application for:	
Only for me (Recommended)	
Anyone who uses this computer (all users)	
InstallShield	
	< Back Next > Cancel

9) At window, click 'Next >'. [Note: It wouldn't work when I used 'Browse...' to customize.]

Cadence OrCAD PSpice Designer Lite 17.2	
Installation Settings	cādence®
Installation Directory	
C:\Cadence\SPB_17.2	Browse
nstallShield	
nstalisinelu	< Back Next > Cancel

10) At window, click 'Install'.

itart Copying Files Review settings before copying files.	cāden	Ce
Setup has enough information to start copying the p change any settings, click Back. If you are satisfie	program files. If you want to review	y or
copying files. Current Settings:		
- Products to install: PSpiceAD Lite		* III
Product destination path: C:\Cadence\SPB_17.2		
<ul> <li>Working directory: C:\Users\FlowCAD\AppData\Roaming\SPB_I</li> </ul>	Data	-
4		,
allShield		
	< Back Install	Cancel

11) The installation will take a couple minutes. At window, click 'Finish'.

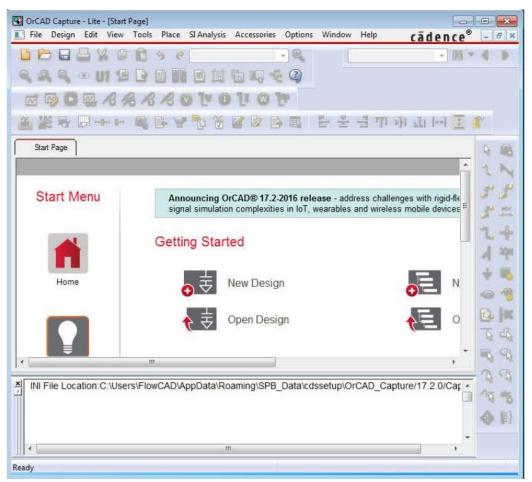
adence OrCAD PSpice Designer Lite 17.2	
Setup Complete Click Finish to complete installation.	cādence <sup>®</sup>
You can select any of the following options before yo	ou proceed:
Dpen Cadence web page	
🕅 View Product Notes [What's New in Release]	
Remove Cadence paths from PATH variable. Do	not select if you have older releases installed
Generate doc index to enable search in Cadence	Help [Must for server installation]
staliShield	
	< Back Finish

12) Click windows icon at lower left corner of screen, scroll down to **Cadence Release 17.2-2016**, click down arrow to display software options, and select/run 'Capture CIS Lite'. You are ready to roll!

	с					
-		Cadence Release 17.2-2016 ^ New	AB	P]	×Ш	0 🗹
	?	Cadence Help New	N	PB	<u>≜</u> ≞	
7	9	Capture CIS Lite			Excel	Outlook
-	<b>9</b>	Magnetic Parts Editor	w			
		Model Editor		2		
		PSpice AD Lite New	Word			
		PSpice Advanced Analysis Lite New	Utilities			
	hnbl	1988 18823- 2886				
8	X	Simulation Manager				100
8		Simulation Manager Stimulus Editor	Acrobat	Reader	Tz 7-Zip File	<b>in</b>
® D	Property lies		Acrobat DC	Reader		File Explorer
		Stimulus Editor		Reader	7-Zip File	
		- Stimulus Editor Calculator	DC		7-Zip File Manager	
S S S S		Stimulus Editor Calculator Calendar	DC		7-Zip File	
		Stimulus Editor Calculator Calendar Camera	DC		7-Zip File Manager	

Bonus material (I'll add as/if I get time):

13) This program window should open. Click File  $\Rightarrow$  New  $\Rightarrow$  Project.



14) When the New Project window opens, enter a project Name (no blank spaces), select the 'PSpice Analog or Mixed A/D' option, click 'Browse...' and select a folder in which to store the project, and click 'OK'.

lame	ОК
EE220L_lab4	Cancel
Create a New Project Using	Help
Spice Analog or Mixed A/D	Tip for New Users
PC Board Wizard	Create a new Analog or Mixed A/D project. The new project may be blank or copied from an existing template.
Rogrammable Logic Wizard 💿	
Schematic	Learn With PSpice - Examples And AppNotes
Location	
c:\Temp	Browse

15) A 'Create PSpice Project' window should appear. Select 'Create a blank project' and click 'OK'.

reate PSpice Project	
Create based upon an existing project	ОК
demo_all_libs.opj	 Browse
Oreate a blank project	Cancel
	Help

16) The new project (name.opj) should now be created and you should be at the 'PAGE1' tab displaying a portion of a simulated blueprint page on which you can create your circuit schematic. Click on the 'Place part (P)' icon to bring up the 'Place Part' window.

lace Part	¢ *	
Part		
		~
Part List:	<b>T</b>	55
		J ab
		1.4
		1 7
Libraries:		· 주 + · ·
	COX	<ul> <li>▲ <sup>2</sup></li> <li>↓ ■</li> <li>↓ ■</li> <li>↓ ■</li> </ul>
Libraries: Design Cache	C 🗆 🗙	
	C 🗆 X	
	C X Packaging Parts per Pkg: 1	
	Packaging	
Libraries: Design Cache	Packaging Parts per Pkg: 1	

17) In the 'Place Part' window, click on the 'Add Library' icon.

Design Cache	G	1	×	
erosigi redullo.	/	<u> </u>		
	- Packag	6800)		

- 18) A 'Browse File' window will open. Use the mouse to highlight/select all the \*.olb files and click 'Open'. This will give you access to all the includes parts in these parts libraries.
- 19) Use the 'Place Part' window to create and wire your circuit (instructor will demonstrate).

- Do NOT forget to create a reference node, i.e., need a ground.
- To rotate a part, press the letter 'R' key.
- To mirror a part, press the letter 'V' or letter 'H' keys to mirror vertically or horizontally
- To place a wire, press the 'W' key or the 'Place wire (W)' icon

Place Part		ф ~ X		-
Part			3	10
OPAMP		-	1	N
Part List:	Y	-	100	F
OPA660X1/BB/BURR_BRN	^			

20) After the circuit is created, go to the top menus and click PSpice  $\Rightarrow$  New Simulation Profile. In the New Simulation window enter a Name and click 'Create'.

lame:	S.
Example	Create
nherit From:	Cancel
none 🔹 🗸	)

- 21) In the 'Simulation settings- Example' window, click on 'Configuration Files' (on left)  $\Rightarrow$  'Library'  $\Rightarrow$  'OK'.
- 22) Save your project and click the 'Run' icon. This cause the PSpice A/D Lite program to open and simulate your circuit. You can now display voltages, currents, and powers using the appropriate icons (instructor will demonstrate. You can change the displayed precision of these items by clicking PSpice ⇒ Bias Points ⇒ Preferences ⇒ Bias Points. Then, in the 'Bias Point Preferences' window, enter your desired 'Displayed Precision' and click 'OK'.
- 23) To copy displays from PSpice to MS-Word, use the mouse to left click and drag to highlight what you would like to copy, CTL-C, and go to MS-Word and hit CTL-V.
- 24) I like to draw arrows for my currents. To copy bitmap images into Corel PHOTO-PAINT, I had to use the 'Snipping Tool' as CTL-V and Copy from Clipboard gave me gibberish.