Running Cadence/PSpice from EECS Machines

From the Windows desktop,

1) click on the “Stop Button” at the bottom far-left side of the screen.

2) Scroll down and you’ll see one or more folders with Cadence in the name. Click on the one named: Cadence Release 17.2 – 2016.

3) Click on “Capture CIS.” This will launch what we typically call PSpice.

4) A window called “Cadence Product Choices.” There will probably be only two choices, but regardless of how many, choose “Allegro PCB Design CIS XL.” Then, click OK.

5) From the startup page, choose “New Project,” then select a name and a folder to place the project in. Since PSpice will eventually create a number of files for this project, I find it best to create a new folder for each project.

   Note that there are 4 options in this New Project window. It is VERY important to choose the right one: **Analog or Mixed A/D** . (Note that this correct option may not be the default option, so watch out!)

6) From this on, you can follow the PSpice tutorial from the 211 website entitled “PSpice with Cadence”