

Accelerator Power Engineering – USPAS June 12 – 16, 2017

The contents of this folder are as follows:

1. The file “Readme First.docx”
2. The File “2017 USPAS Accelerator Power Systems 17-05-10.pptx” - this is the PowerPoint file that we will use and discuss during the weeklong class. You are welcome to open it and browse through it prior to the class for familiarity with the material. As you look through the file, the material might provoke some questions that we can discuss and perhaps answer in class.
3. The LTspiceXVII.exe executable file.
4. The file “LTspiceXVII* Installation Instructions” that holds the LTspiceXVII circuit analysis tools that we will use in the computer laboratory to simulate some of the circuits we will discuss in the class. We will have the circuit analysis program installed on the laboratory computers prior to the start of the laboratory session. You do not need to install the program now or even look at the material. However, if you are curious and/or ambitious, click (or double-click) on the “LTspiceXVII.exe” file and install in the directory given in the Installation directions.
5. The file “LTspiceXVII Laboratory Exercise.docx” that contains instructions for the exercise that we will implement in the computer laboratory. Then if you are curious and/or ambitious, run the simulations and good luck. If you have trouble, do not panic. We will discuss the setup and circuit simulations in the laboratory in June.
6. The file irgp4063dpbf.asy. This file will be copied into the C:/Files/LTspiceFiles/Sym folder that we will create as part of the LTspiceXVII installation.
7. The file irgp4063dpbf.spi. This file will be copied into the C:/Files/LTspiceFiles/Lib folder that we will create as part of the LTspiceXVII installation.

*SPICE is a software simulator, developed at U.C. Berkeley, to analyze non-linear analog circuits. LTspiceXVII is a free version of SPICE developed and supported by Linear Technology as an aid to its customers in their design of circuits using Linear Technology integrated circuits. <http://www.linear.com/designtools/software/>

Paul Bellomo and James Sebek May 2017