Instruction to Install the NGspice Simulation Environment for ECE 445

January 21, 2018

Step 1. Download and install Matlab® version 2017b from https://umaine.edu/it/software/matlab. Click the Matlab icon and make sure Matlab® opens without error. If you experience errors, you may need to update your operating system or install additional libraries.

Step 2. Download and install CppSim from http://www.cppsim.com/download.html. This will install the Spice simulator: NGspice; a schematic capture program: Sue2; and a simple results browser: CppSimView on your computer.

   1. Click the CppSimView icon. Two windows should appear on your screen. If you experience errors, check the documentation at: http://www.cppsim.com/Manuals/cppsim_vppsim_primer5.pdf for possible sources of errors. Close CppSimView.

   2. Click the Sue2 icon. A schematic capture window should appear. Again, if you experience any errors, check the documentation at: http://www.cppsim.com/Manuals/cppsim_vppsim_primer5.pdf for possible sources of errors and try and correct any installation errors.

Step 3. Locate the CppSim directory on your computer. The CppSim directory should contain sub-directories including: SpiceModels, Import_Export, Sue2, SueLib, SimRuns, Netlist, ...

   1. If using MS Windows 10, download the `spinit` file from http://davidkotecki.com/ECE214/docs/ngspice/spinit and place it in a folder called: `C:\Spice\share\ngspice\scripts`. You will need to create this directory structure. Important: the file must be called `spinit` and not `spinit.txt`. Turn on "show file extensions" and verify that `spinit` is not actually named `spinit.txt`. If it is, rename the file `spinit` without any file extension.

   2. If using OSX or Linux, download the `spiceinit` file from http://davidkotecki.com/ECE214/docs/ngspice/spiceinit. Edit the file so that the lines beginning with `codemodel` all have a valid directory structure and point to valid `.cm` files. Rename this file `.spiceinit` and place in your home directory.

Step 4. Configure Sue2:

   1. In the Sue2 schematic capture window, click: Tools → Library Manager. The Library Manager window should open.

   2. A Library is the name of a folder where you store schematics. Each schematic is referred to as a module. Sue2 comes with a lot of libraries which are not used in ECE 445. From the Library text box, select each library except for "devices" and "spice," and click "Remove Library." This will not delete the library from your computer, but will remove it from your current design environment.

3. Create a new library to store your ECE 445 simulations. Click "Create" next to Library Operations. A "Create New Library" window will appear. Enter ECE445 as the library name, then click OK.

4. Close Sue2.

Step 5. Download the "ECE214 devices library" from http://davidkotecki.com/ECE214/docs/ngspice/ece214_devices.tar.gz and the "ECE445 devices library" from http://davidkotecki.com/ECE445/docs/ece445_devices.tar.gz. Move this file to the "Import Export" directory within CppSim directory. Do not unzip this file.

Step 6. Download the "ECE214 Spice Models" from http://davidkotecki.com/ECE214/docs/ngspice/ece214_spicemodels.zip and the 'ECE445 Spice Models" from http://davidkotecki.com/ECE445/docs/ece445_spicemodels.zip. Move this file to the "SpiceModels" directory within the "CppSim" directory. Unzip this file. You should now have a number of files in the "SpiceModels" directory including a file called "ECE214 models.mod" and "ECE445 models.mod". If these files do not exist in the "SpiceModels" directory, check to ensure that you unzipped the the ECE214 models.zip and ECE445 models.zip files in the correct directory.

Step 7. Start Sue2. Click: Tools → Library Manager. Click "Import Library Tool," select the ECE214 devices.tar.gz file and click Import. Close and restart Sue2.

Step 8. Select the "ECE445 Library" in the Library Selection Box in the upper right hand corner of the schematic window. Click on the module called: "first module for ECE445." Click: File → New Schematic. Enter "NAND2" as the schematic name and make sure the destination library is ECE445. A blank schematic window should now appear. You can now begin to generate the schematic for the ECE445 design project.

You are now ready to design and simulate!!!