Installing the NGspice Simulation Environment for ECE 214

February 4, 2020

Step 1. Download and install Matlab® version 2019b from https://umaine.edu/it/software/matlab. Click the Matlab icon and make sure Matlab® opens without any errors. 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 NGspice, the Spice simulation engine; Sue2, the schematic capture program; and CppSimView, the results browser, on your computer. 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 subdirectories including: SpiceModels, Import_Export, Sue2, SueLib, SimRuns, CppSimShared, Netlist, ...

   1. Locate the file named `ngsim.m` in the `CppSim/CppSimShared/HspcToolbox` directory. Open the file using Matlab® or a text editor. On line 133, replace `win32` with `win64`.

   2. 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.

   3. 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 214. 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 214 simulations. Click "Create" next to Library Operations. A "Create New Library" window will appear. Enter ECE214 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. 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. Move this file to the "SpiceModels" directory within the "CppSim" directory. Unzip this file. You should now have a number of files in the "Spice-Models" directory including a file called "ECE214_models.mod". If this file does not exist in the "SpiceModels" directory, check to ensure that you unzipped the the ECE214_models.zip file 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 "ECE214 Library" in the Library Selection Box in the upper right hand corner of the schematic window. Click on the module called: "first_module_for_ECE214." Click: File → New Schematic. Enter "Lab1" as the schematic name and make sure the destination library is ECE214. A blank schematic window should now appear. You can now begin to generate the schematic to simulate the circuit for Lab 1.

You are now ready to design and simulate!!!