

How to Install the NGspice Simulation Environment for ECE 214

February 19, 2017

Links to the files you will need to download are located on the ECE 214 course web site (<http://web.eece.maine.edu/kotecki/ECE214/>) or at: "<http://web.eece.maine.edu/kotecki/ECE214/html/Components>"

Step #1 Download and install 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. Try to correct any installation errors before Lab #2.

Step #2 Download and install CppSim (<http://www.cppsim.com/download>). This will install four programs on your computer. We will only be using two of these: Sue2 (schematic capture) and CppSimView (results browser).

1. First, 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. Try to correct any installation errors before Lab #2. Close CppSimView.
2. Next, 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 before Lab #2.
3. Perform the following steps to configure Sue2:
 - (a) In the Sue2 schematic capture window, click: Tools → Library Manager. The Library Manager window should open.
 - (b) 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 needed for ECE 214. From the Library text box, select each library except for “devices” and click “Remove Library.” This will not delete the library from your computer, but will remove it from your current design environment.
 - (c) 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.
 - (d) Close Sue2.

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

Step #4 Download the “ECE214_devices library” (http://web.eece.maine.edu/kotecki/ECE214/docs/ngspice/ece214_devices.tar.gz). Move this file to the “Import_Export” directory within CppSim directory. Do not unzip this file.

Step #5 Download the “ECE214 Spice Models” (http://web.eece.maine.edu/kotecki/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 “SpiceModels” 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 #6

1. If using MS Windows 10, download the `spinit` file (<http://web.eece.maine.edu/kotecki/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 (<http://web.eece.maine.edu/kotecki/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 #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 “Lab2” as the schematic name and make sure the destination library is ECE214. A blank schematic window should now appear. You can not begin to generate your Lab #2 schematic.

At this point NGSpice should contain three libraries: ECE214 (used to store all schematics you will generate in ECE214), ECE214_devices (contains all of the devices – resistors, capacitors, inductors, voltage sources, diodes, switches... – used in ECE 214), and devices (contains pins, labels, gnd, and globals). Schematics contain symbols. The symbols you need for your schematics are contained in the ECE214_devices and the devices libraries.

You are now ready to design and simulate!!!