

Instruction to Install the NGspice Simulation Environment for ECE 342

August 25, 2017

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.

Note If you installed NGspice and Sue2 on your computer in ECE 214, start Sue2 Schematic Capture program and skip to step 4.3 below.

Step 2. Download and install CppSim from <http://www.cppsim.com/download>. 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://web.eece.maine.edu/kotecki/ECE342/docs/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://web.eece.maine.edu/kotecki/ECE342/docs/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 342. 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 342 simulations. Click “Create” next to Library Operations. A “Create New Library” window will appear. Enter ECE342 as the library name, then click OK.
4. Close Sue2.

Step 5. Download the “ECE342_devices library” from http://web.eece.maine.edu/kotecki/ECE342/docs/ece342_devices_A.tar.gz. Move this file to the “Import_Export” directory within CppSim directory. Do not unzip this file.

Step 6. Download the “ECE342 Spice Models” from http://web.eece.maine.edu/kotecki/ECE342/docs/ece342_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 “ECE342_models.mod”. If this file does not exist in the “SpiceModels” directory, check to ensure that you unzipped the the ECE342_models.zip file in the correct directory.

Step 7. Start Sue2. Click: Tools → Library Manager. Click “Import Library Tool,” select the ECE342_devices_A.tar.gz file and click Import. Close and restart Sue2.

Step 8. Select the “ECE342 Library” in the Library Selection Box in the upper right hand corner of the schematic window. Click on the module called: “first_module_for_ECE342.” Click: File → New Schematic. Enter “Lab1” as the schematic name and make sure the destination library is ECE342. 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!!!