# Lab 11-3 Importing Third party models from Internet and using it with PSpice

**Objective: Learn how you can quickly download a model from Internet & use it in PSpice simulation**

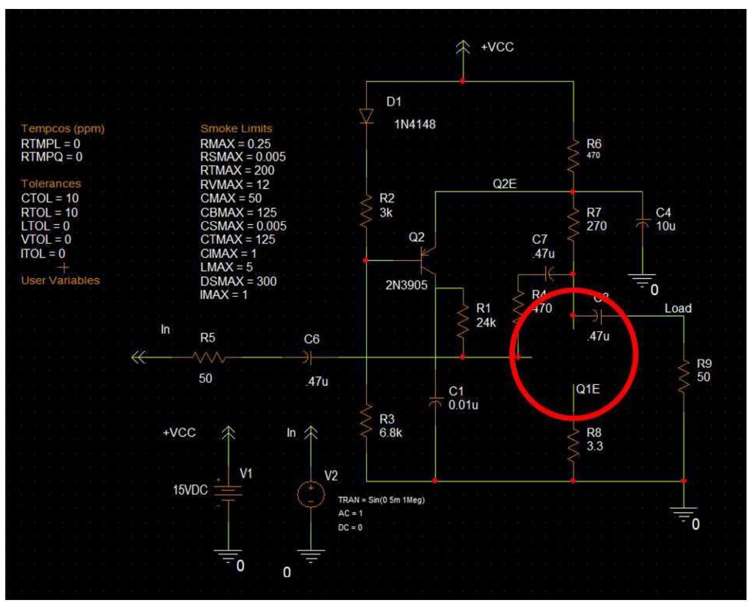
---

[PSpice is the only Spice simulator that allows you to download any component right from the web into your design environment from any third-party model vendor and use it in a design to simulate]

### Simple plug & play steps to use any model from Internet in PSpice simulation

1. Open **rf_amp.opj**

2. Review the circuit schematic of the project,




3. Notice that we have left a space open for a component to be connected. We will be connecting an NPN high voltage power transistor here.

4. Visit this webpage - https://www.st.com/en/power-transistors/stn0214.html#resource. Scroll down and download the PSpice model for STN0214

HW Model, CAD Libraries & SVD

HW MODEL, CAD LIBRARIES & SVD

| | Description | Version | Size | Action |
|---|---|---|---|---|
| ☐ | STN0214 PSpice model | 1.0 | 850 Byte(s) | ZIP |

5. You will notice that a .zip file has been downloaded. Unzip the contents of the zip file into a desired folder.

6. The content of the folder contains a .lib file of the component which has the necessary PSpice netlist information required for it to work with PSpice.

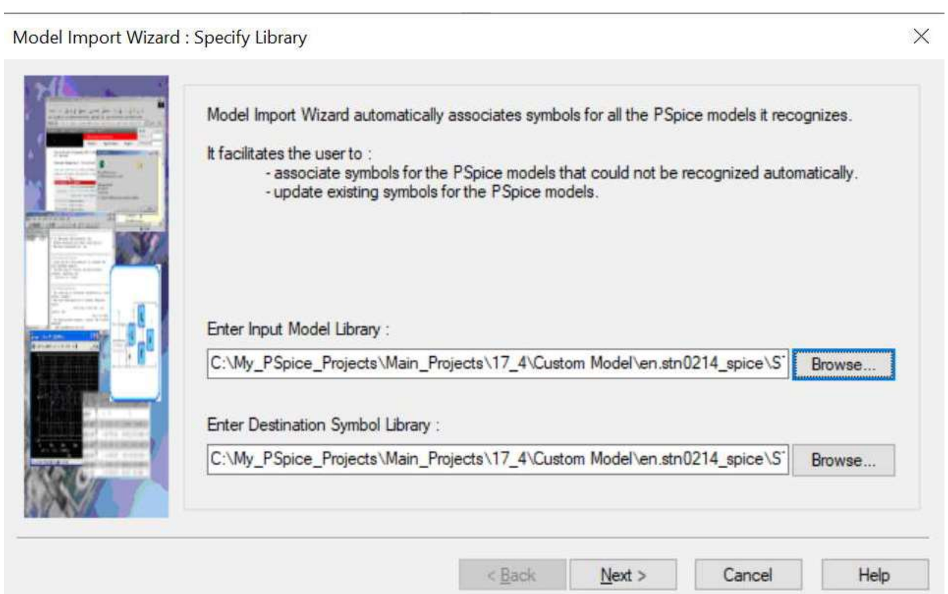
| Name | Type | Compressed size |
|---|---|---|
| STN0214.lib | LIB File | 1 KB |

But, along with a .lib file – we need a schematic symbol that represents this component on the schematic

7. You can quickly create a schematic symbol for a component and associate the .lib file to it using the PSpice Model Editor.

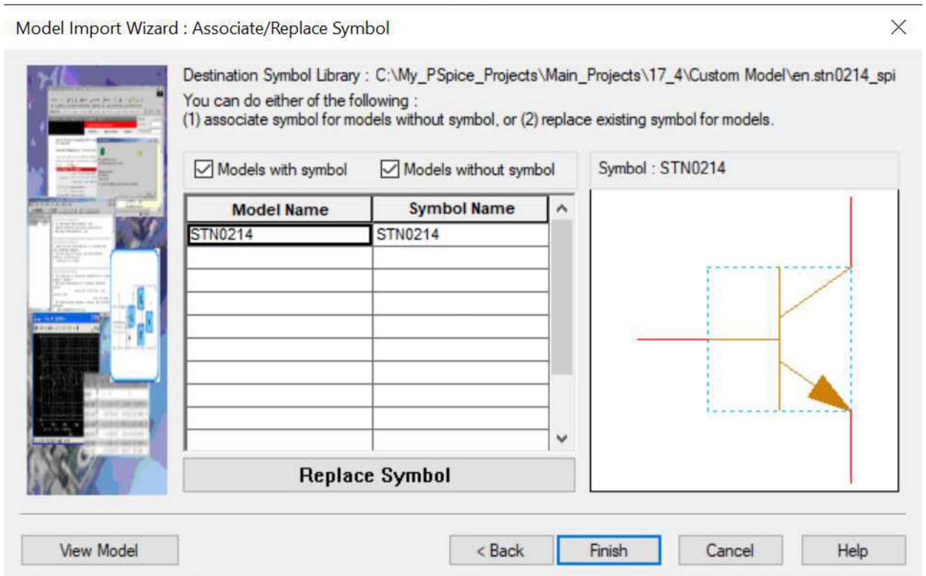
To open the PSpice Model Editor, choose **Start – Cadence PCB Utilities 17.4-2019 – PSpice Model Editor 17.4**

8. Go to **File > Model Import Wizard [Capture]**

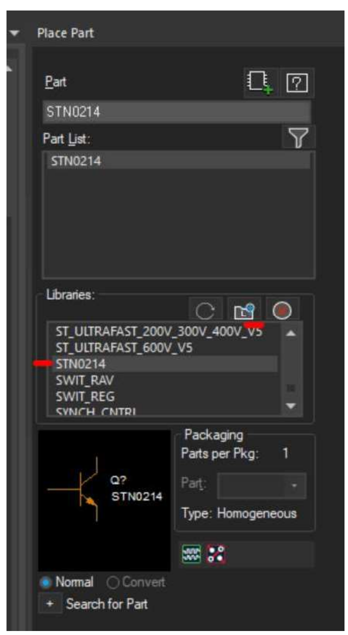



9. Browse to the location of the **Input model library,** where you saved the .lib file**.** The destination **schematic symbol library path** will be configured automatically to the same location. Click **Next.**

10. The schematic symbol will be pre-displayed as PSpice reads through the .lib file and suggests a symbol of an NPN transistor. Click **Finish** to complete schematic symbol creation for this model

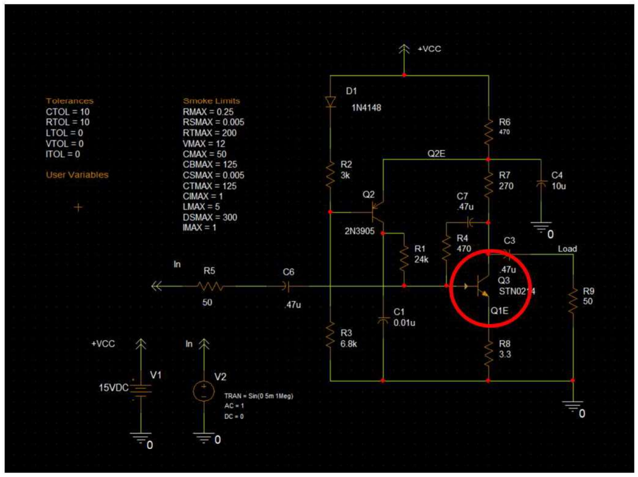



11. Go back to Capture window. Select the add library icon and browse to the schematic symbol .olb file we just created.



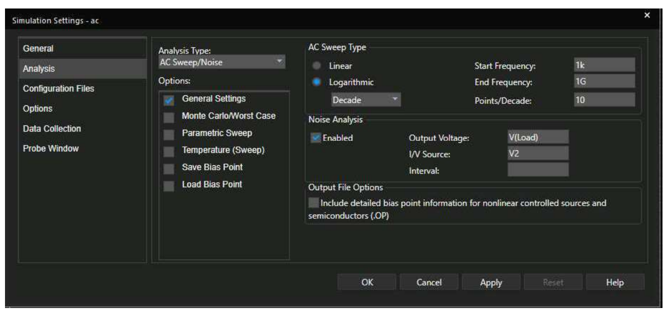

12. Once you load up the .olb file, the part **STN0214** shows up in the list

13. You can quickly double click on it and place it on the schematic.

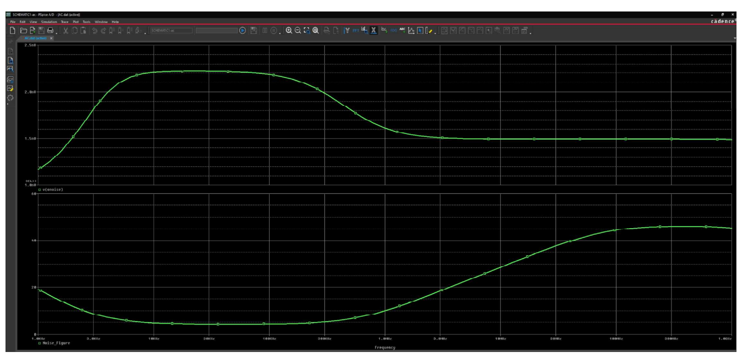



You have now successfully placed the downloaded part on the schematic for PSpice simulation.

14. The AC simulation profile is already set up, so you can go ahead and run the simulation. Review the simulation settings below.




The results will be as shown below,



## Lab Summary

In this lab, you

- Learnt the process to bring in third party vendor models to use in PSpice simulation

End of Lab