SPICE models and PSpice

Using SPICE models and PSpice is also covered in, at least, two other resources, Appendix C of the PSpice User's Guide, pspug.pdf in the doc\pspug of the Cadence, OrCAD product installation, and sloa070, an Application Note at www.ti.com that discusses using Texas Instruments SPICE models with PSpice.

Fundamentals

To simulate a circuit from Capture using PSpice, two library elements are required. The graphical representation of the device, from a Capture OLB file that defines the pins for connecting and contains associated properties to invoke the model when the simulator runs, and the model text itself, from a LIB file containing one, or more, simulation models. The SPICE model element in this is the model text; this text needs to be associated with a graphical symbol to define the connections and netlist the model correctly for the simulator to use.

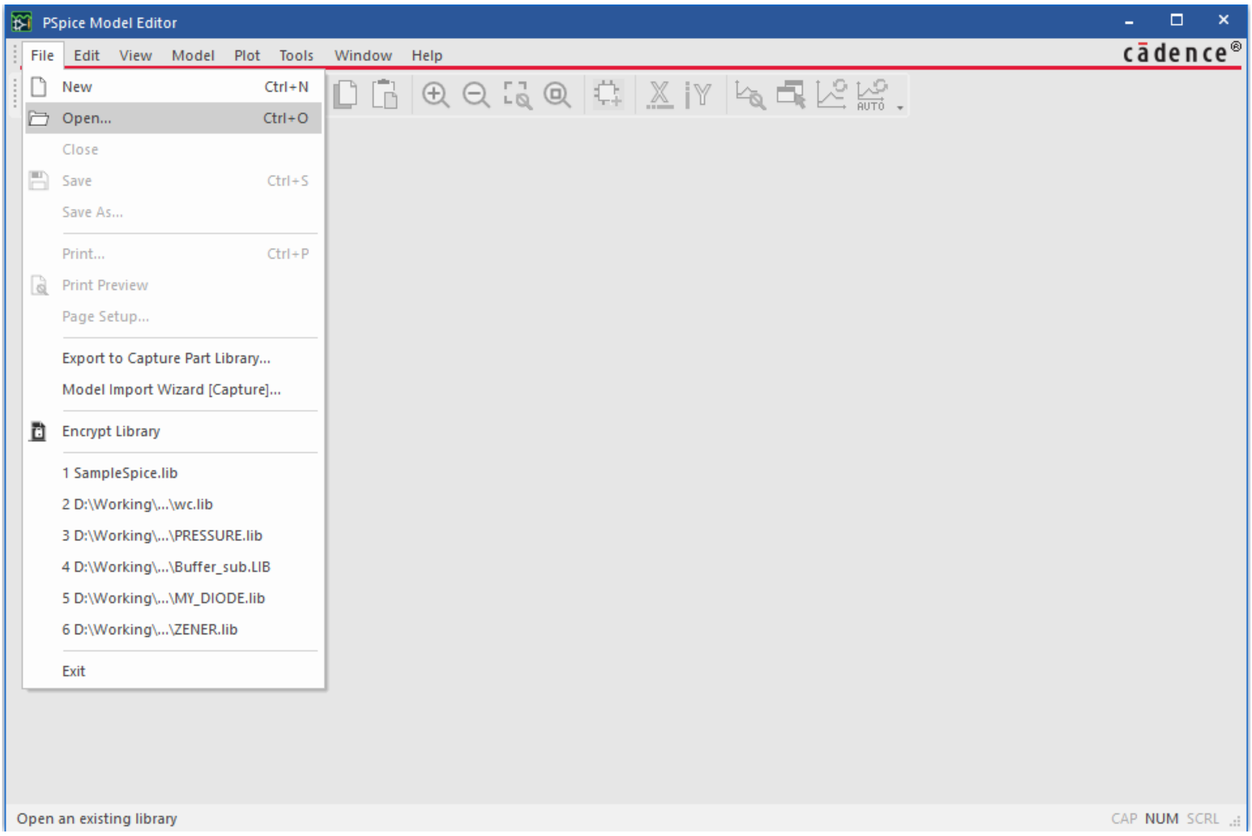
Getting Started

First, locate and download the required model from the vendor site. The exact details vary a bit from vendor site to vendor site but, whatever the exact process, the result will usually be a text file to download, or a page of model text to copy and paste to a text editor. Save the model text and name the file to represent the component that is modelled and change the file extension to .LIB - this will be more convenient for use with the PSpice Model Editor. Some vendors may provide an archive of all, or many, or their models in one large text file, the Model Editor will be able to handle this as well, check that the archive extracts correctly and change the name of the resulting text file to reflect the vendor, or model family, as required, and change the file extension of the extracted text file to .LIB

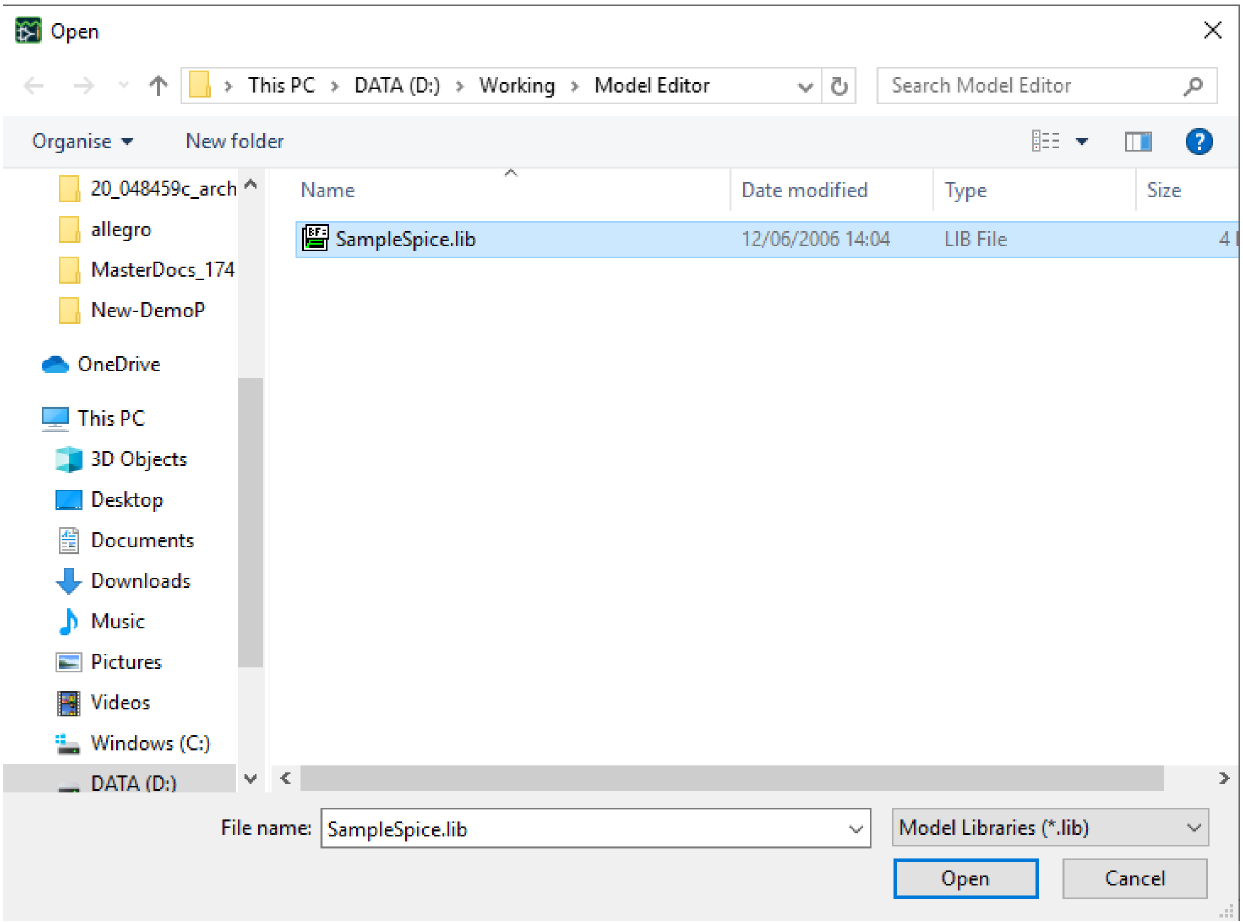
The model text will usually contain a number of comment lines, they begin with *, that describe the model and maybe something about the usage and what behaviour is modelled for more complex devices. The actual model implementation text will start at a line that begins either .MODEL or .SUBCKT The components implemented with the .MODEL type contain a list of parameters for the intrinsic models within PSpice, the components implemented with the .SUBCKT type contain a "netlist" of intrinsic devices that implement the component model, in many cases there may also be some .MODEL entries that are specific to the .SUBCKT model.

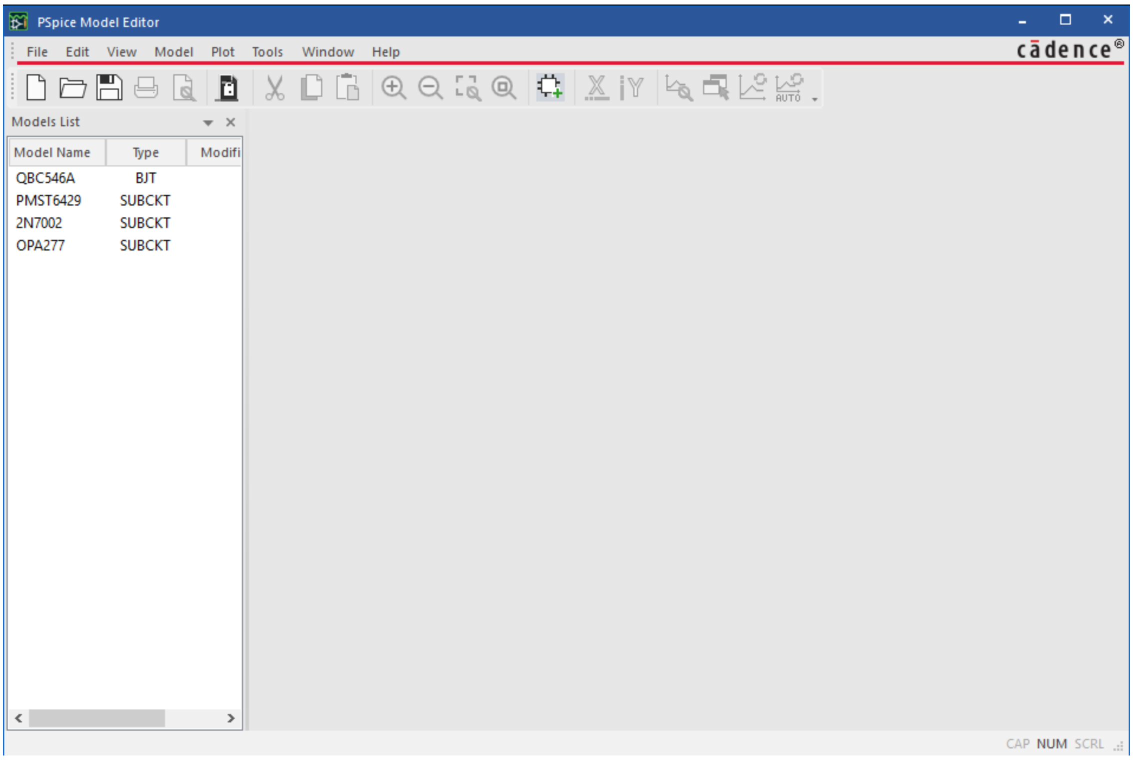
Running the Model Editor

Start the Model Editor from Windows icon>Cadence PCB Utilities 17.4-2019>Model Editor 17.4. Then choose either Design Entry HDL or Capture as the default design tool. For this example, we will use Capture. Now select File>Open and open the LIB file. For the purposes of having "something" to refer to as a LIB file, this note uses a file that contains four components, a transistor model, a transistor sub-circuit, a power MOSFET sub-circuit and an Op-Amp sub-circuit, this file is called "SampleSpice.lib" See the Sequence in the following screenshots:

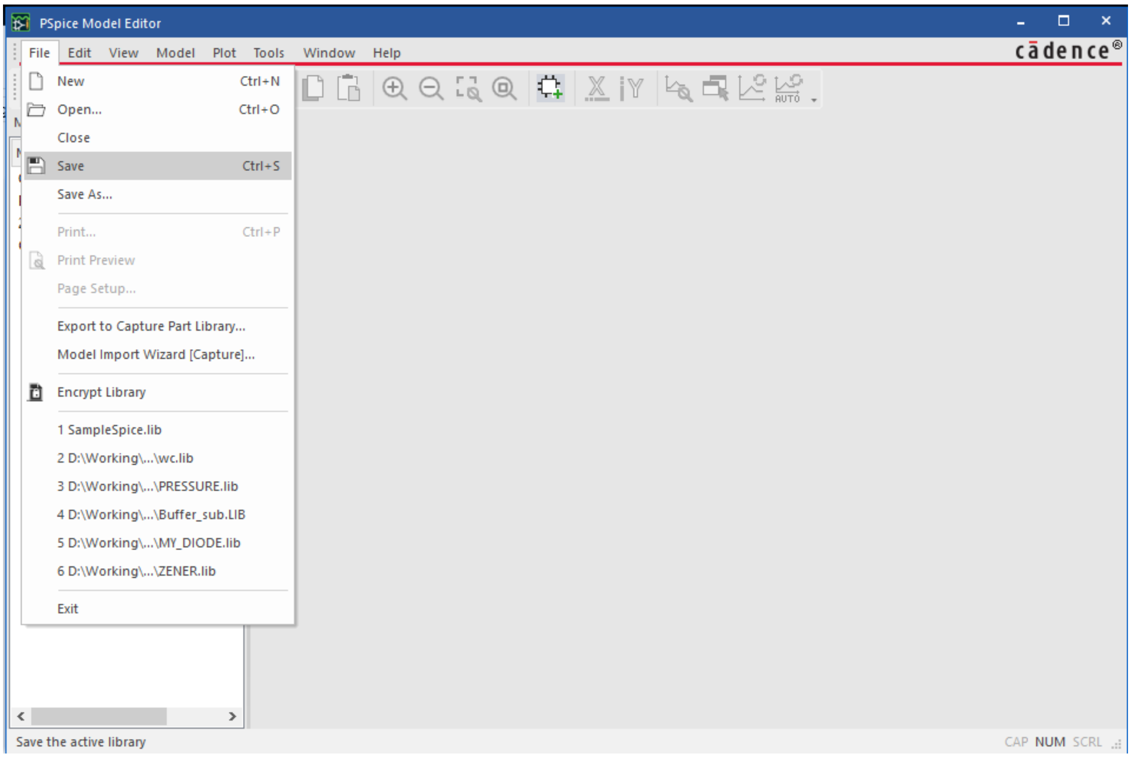


File>Open and select the file.





With the LIB file open, the model names and types are listed.



Save the LIB file from the Model Editor to add the “PSpice indexing marks” to the LIB file.

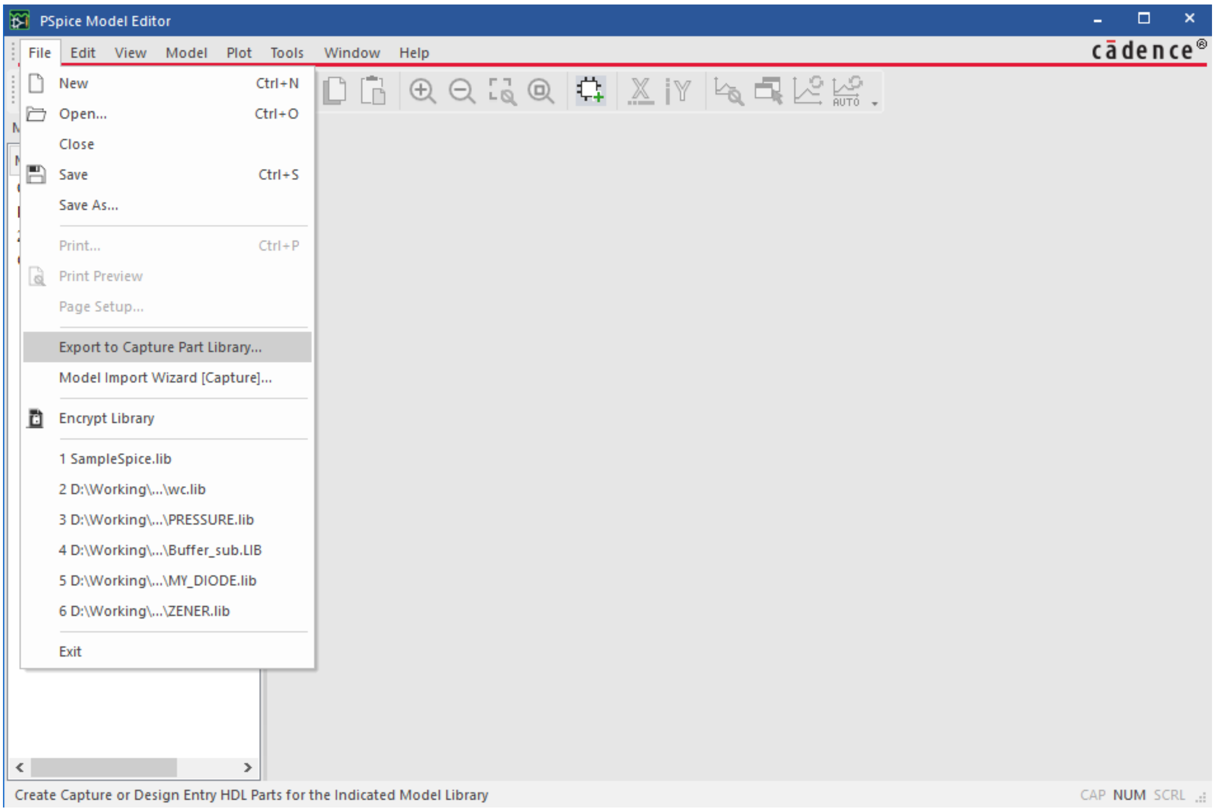
Aside

All of the models supplied with the product installation and models available from vendors are text, or encrypted text. Searching large amounts of text to find models within the configured simulation libraries is not very efficient so PSpice uses index, IND, files and “indexing marks” to make searching for models significantly more efficient. The libraries supplied with the product are indexed during installation, any additional libraries are indexed when the library is first used with a simulation, the index and LIB file timestamps are also compared at the start of every simulation and, if required, the indexes are rebuilt.

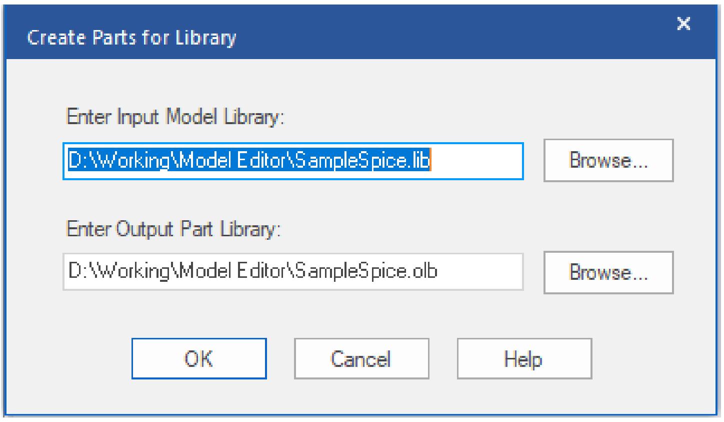
Creating the Graphical Symbols for use in a Schematic

There are two options for generating graphical symbols for use in a schematic, in both cases, models of the .MODEL type will be associated with the appropriate intrinsic PSpice device graphical symbol. In the "default" method, which will be described first, the .SUBCKT type will have a rectangular shape with pins created as the graphics.

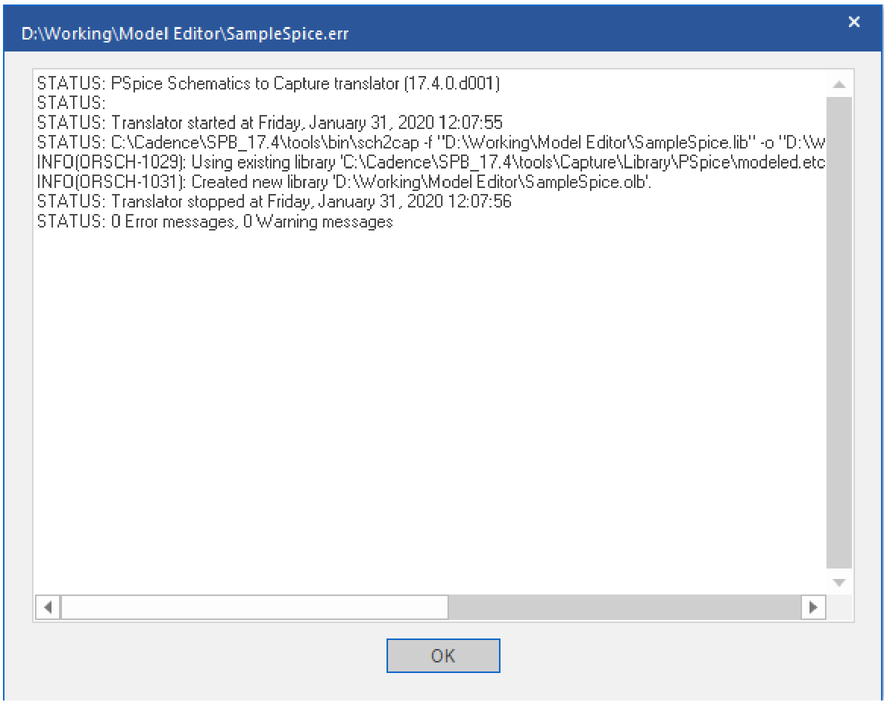
Use File>Export to Capture Part Library:



The library selected for input will automatically be the library that is currently open:

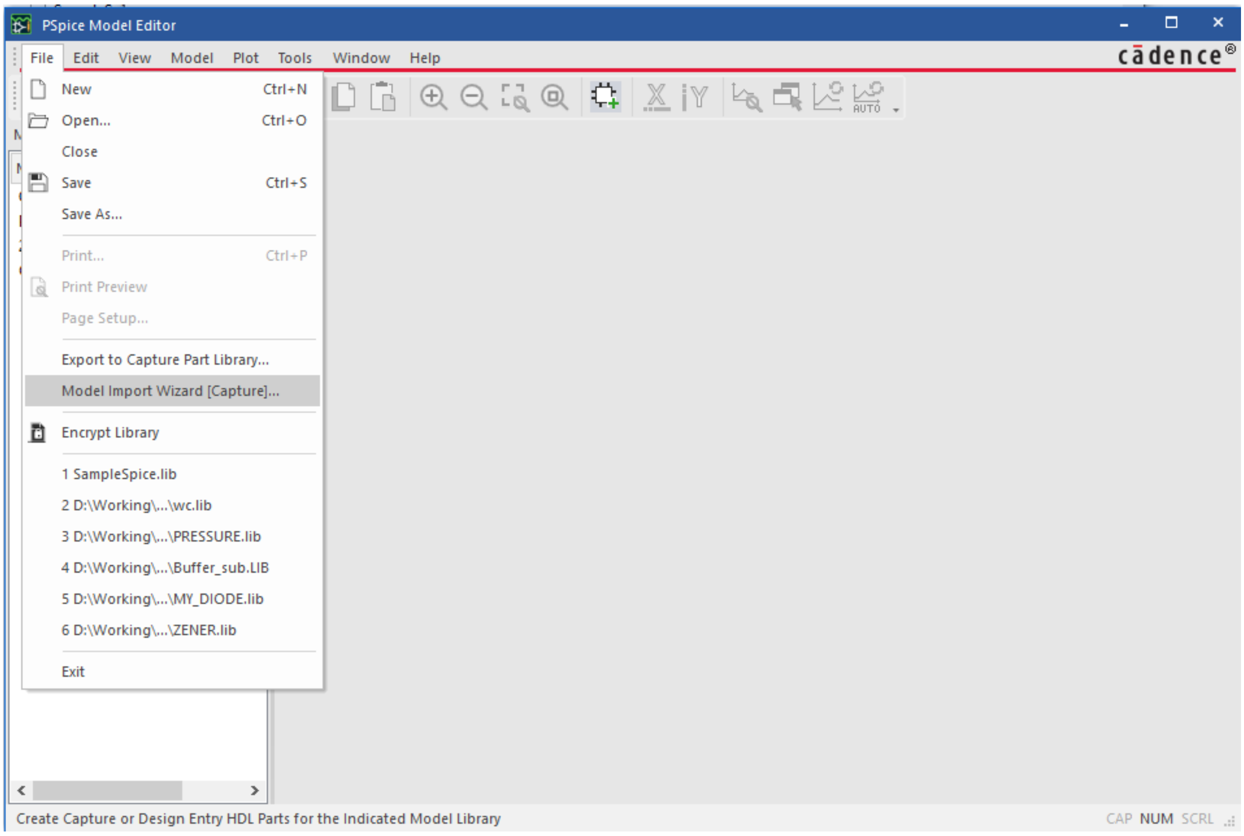


Select OK to complete the library creation and the resulting status will be displayed:

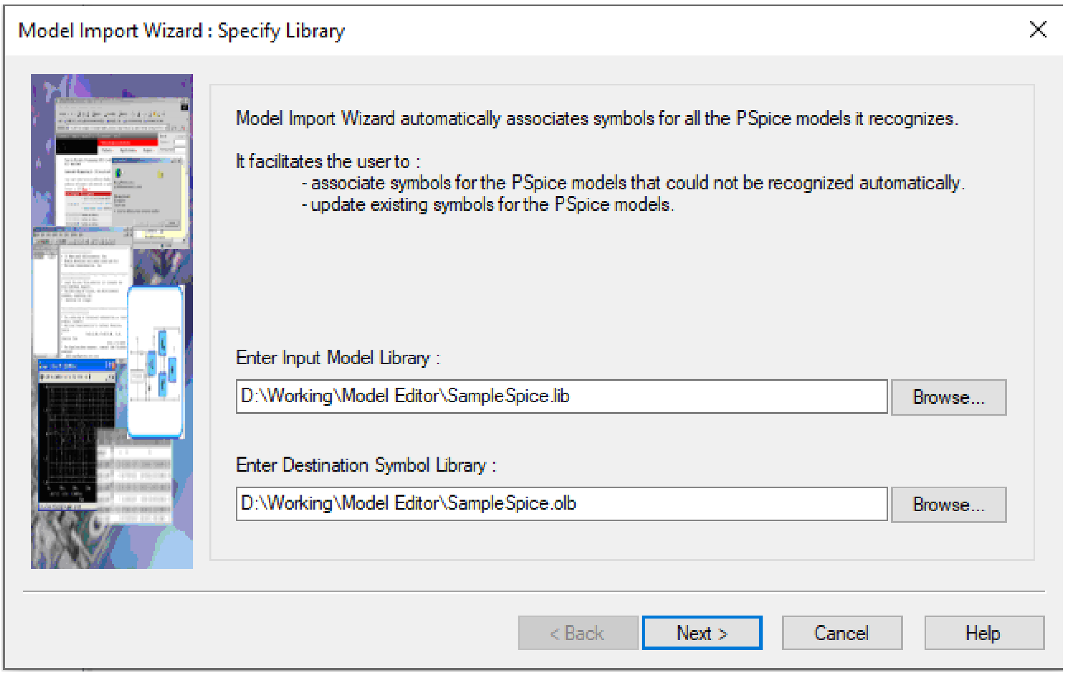


That completes the "default" method, the resulting graphical symbols will be rectangles with pins for the .SUBCKT model type.

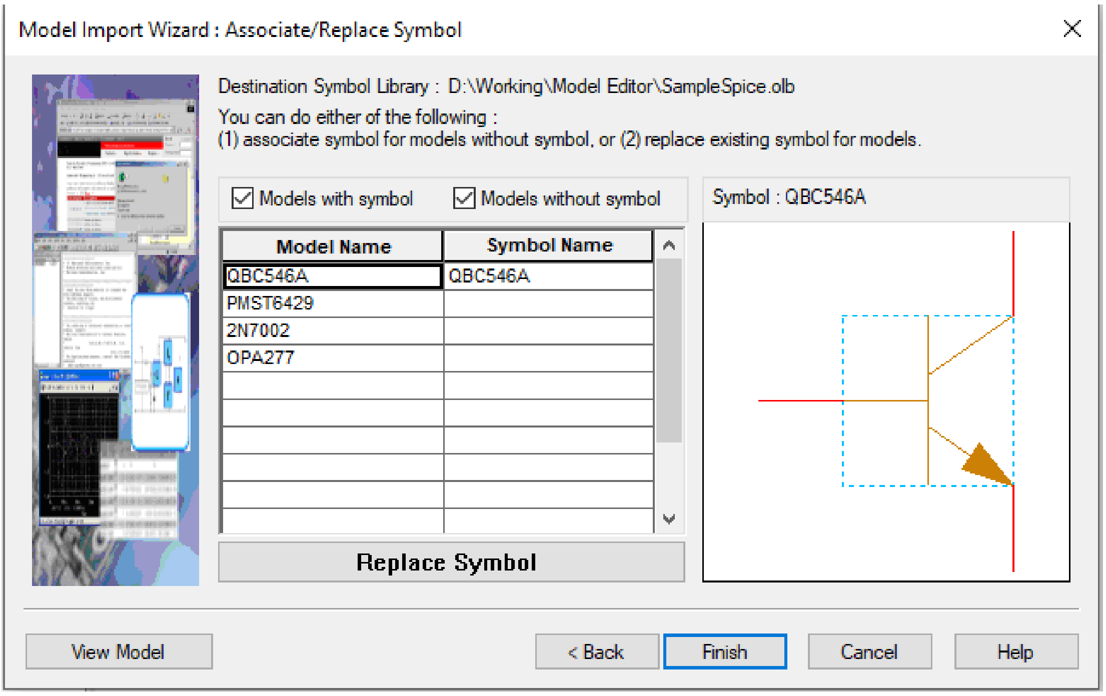
The second method uses the Model Import Wizard, in this the .SUBCKT model types can be associated with existing graphical symbols with a matching number of pins, either from the existing graphical libraries, or a user library generated for the purpose that holds "generic" symbols for common models that are likely to be imported – this just results in a much shorter list of symbols to search. In this case the user library is called "zzGenerics.olb". Start the Model Import Wizard:



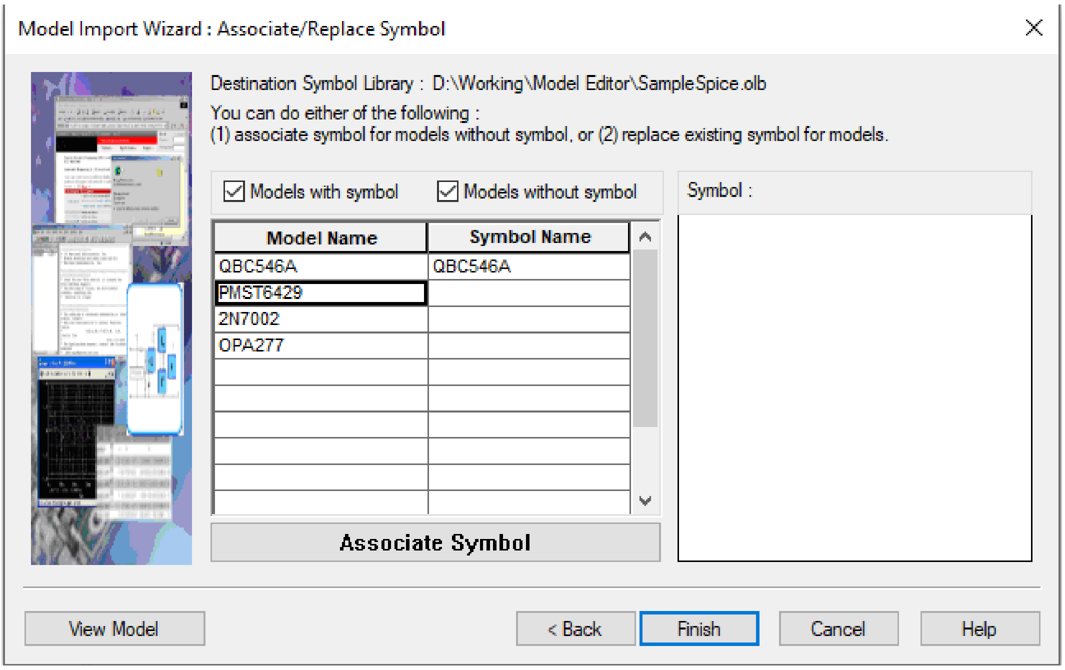
Current library is again selected:



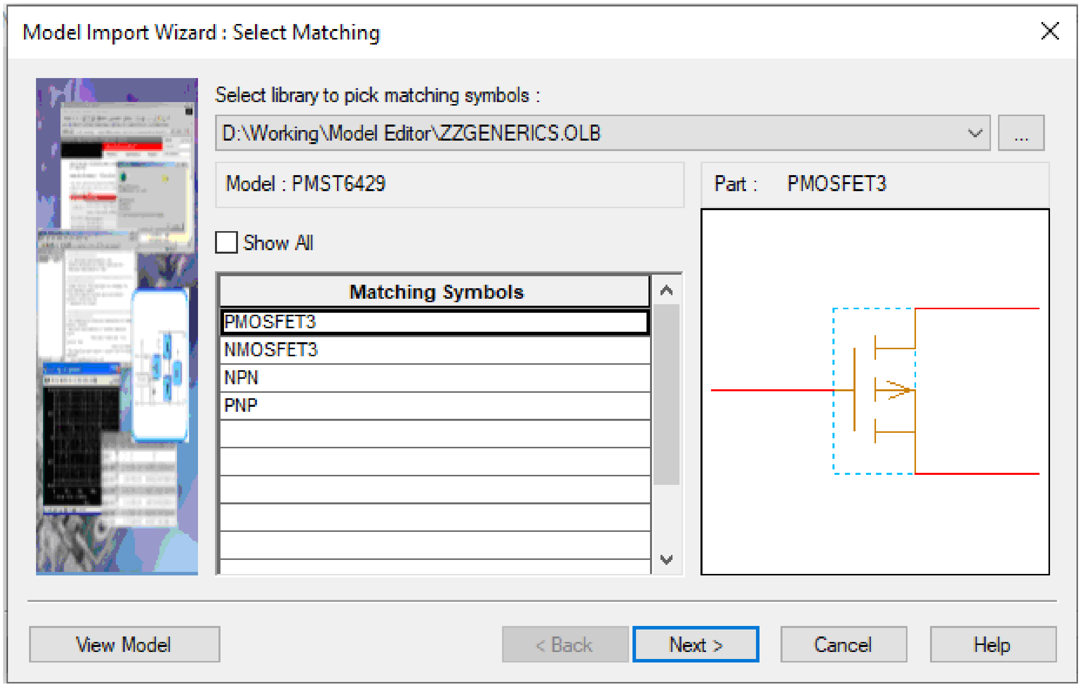
Next will list the parts in the LIB file, the intrinsic part will have an associated symbol by default:



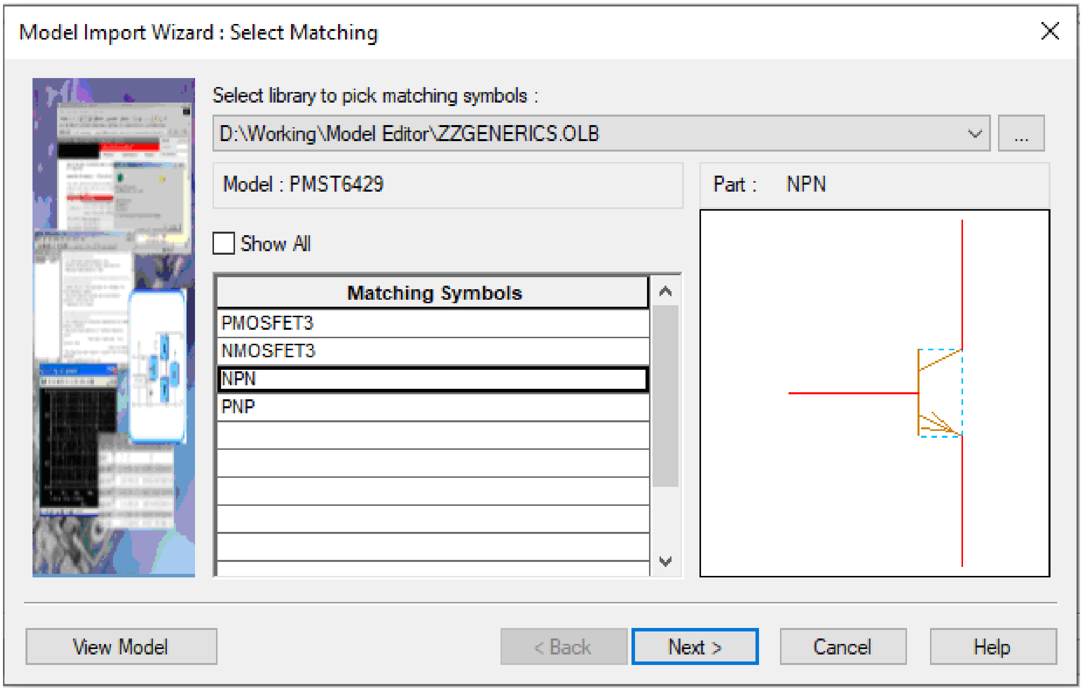
The Symbol can be replaced if desired. The next part is selected:



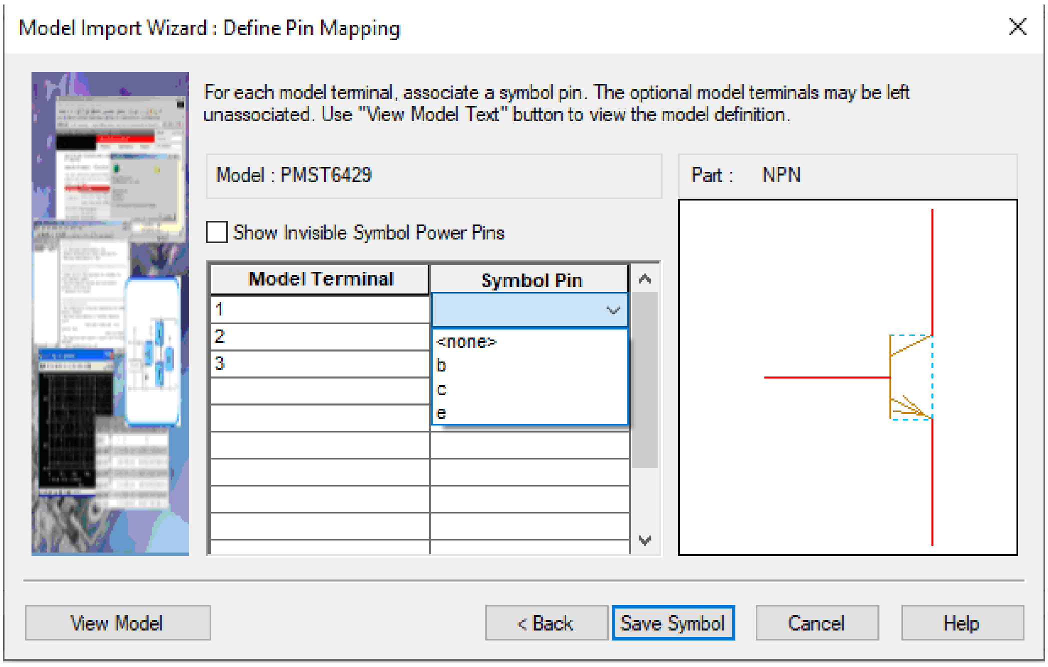
This is not an intrinsic part so a graphical symbol is required, use the Associate Symbol:



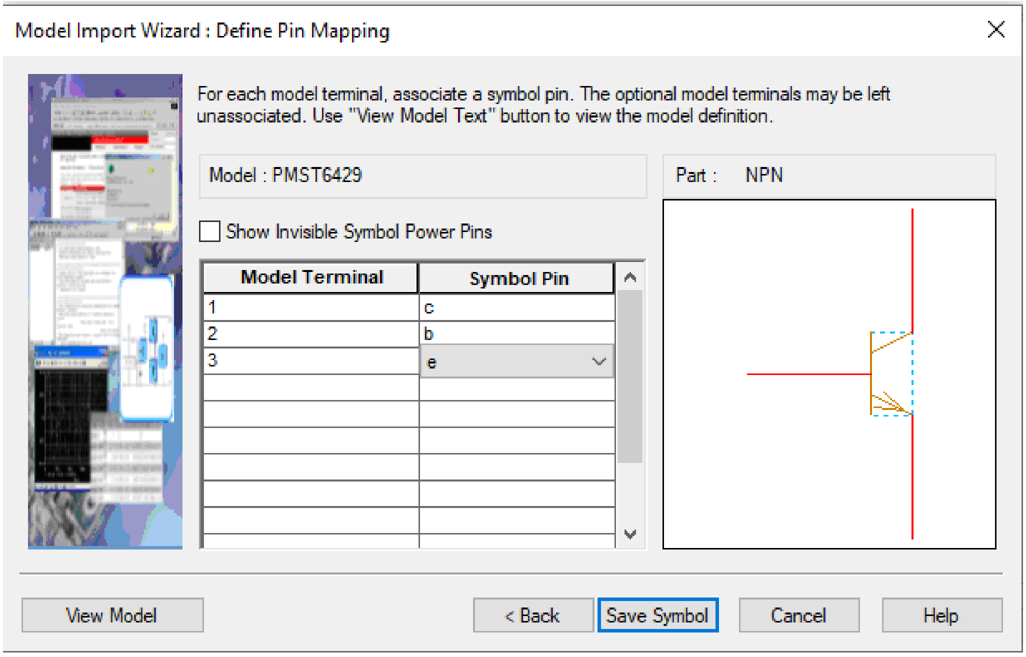
The zzGenerics library has been chosen as the graphical symbol source in this case, the model has 3 pins so all the symbols in the graphical library are listed as candidates, and the NPN symbol is the required one:



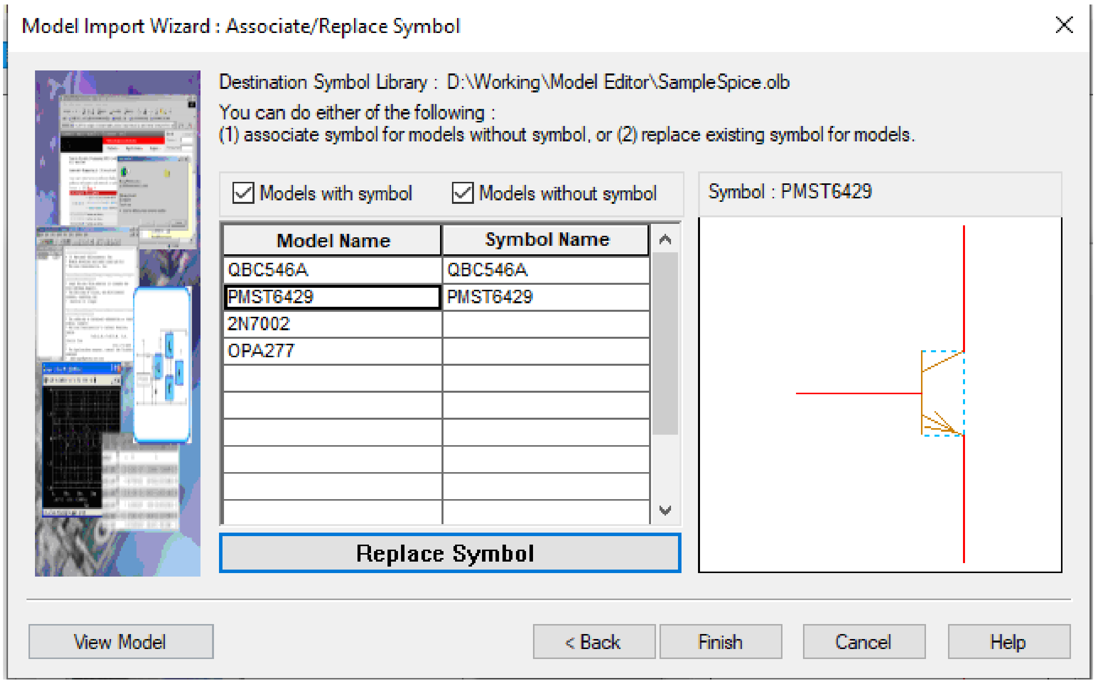
Select Next to get to the Pin Association stage:



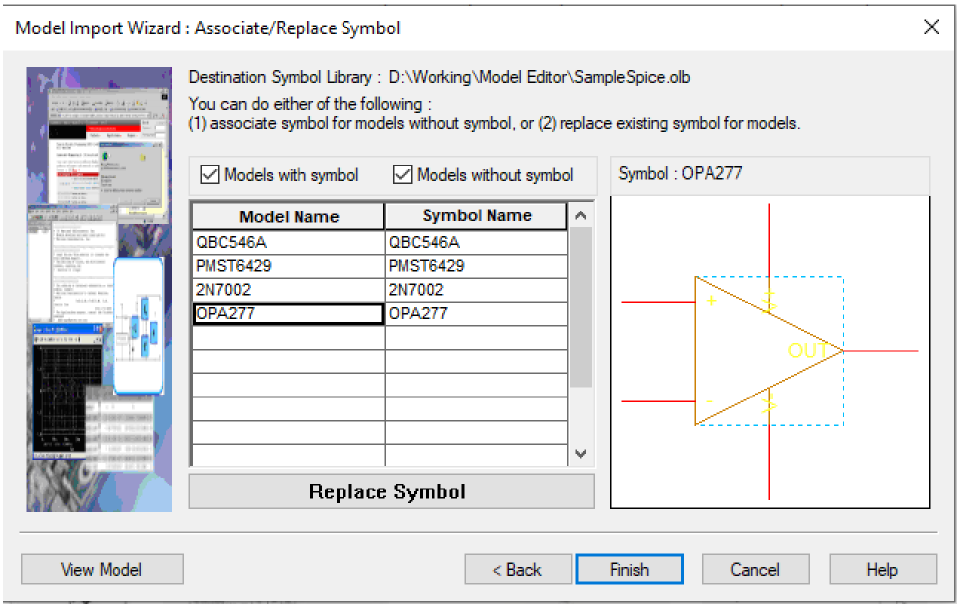
The model pins are 1, 2, 3 for an NPN transistor the Pin Order c, b, e. The pins are selected from the drop-down list:



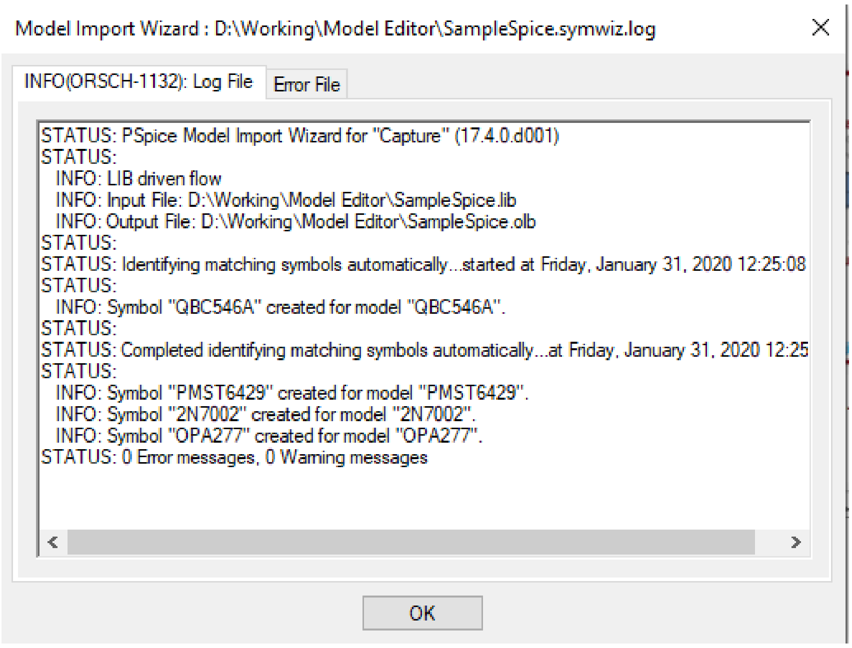
Save Symbol will save the Symbol and Pin Association and name the created symbol after the model name.



Repeat the process for the other models to complete the association, any .SUBCKT types not mapped will have default, rectangular graphics with pins created.



Click on Finish when the symbol association is complete.



Again the library creation status is reported and the OLB file created.

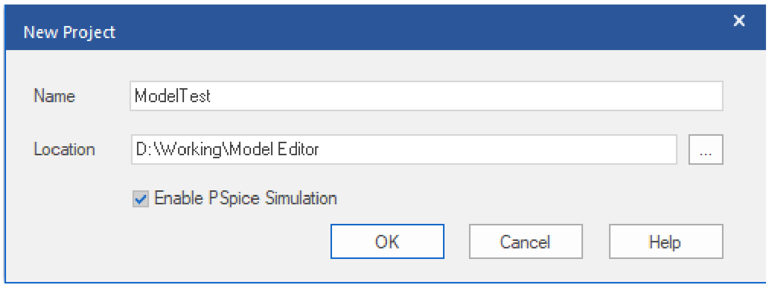
The Model Editor can be closed at this point.

Using the created parts in Simulations

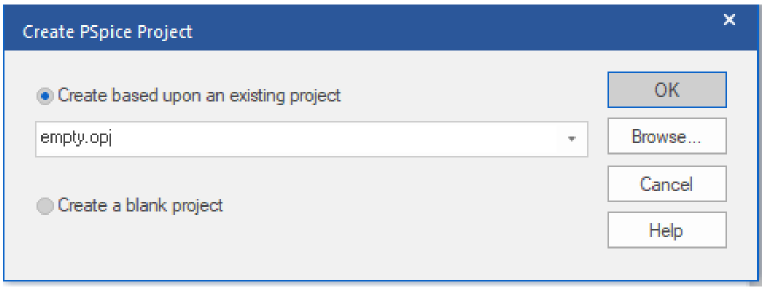
Now that the graphical symbol, OLB, files and simulation model, LIB, files have been created, the libraries can be added to OrCAD Capture, a schematic created and the simulation run using the new models. Start OrCAD Capture, and a new project, File>New>Project. Make sure you check "enable PSpice Simulation"

Default symbols:

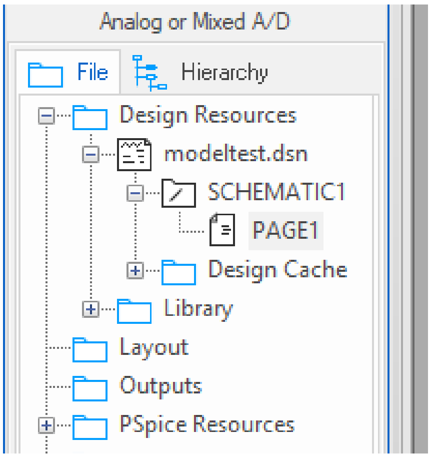
To test this, a simple design with two Sallen-Key filters and the default rectangular sub-circuit symbol will be used. Whilst probably not aesthetically pleasing, the default rectangular symbol is perfectly adequate for "quick test" simulation purposes.



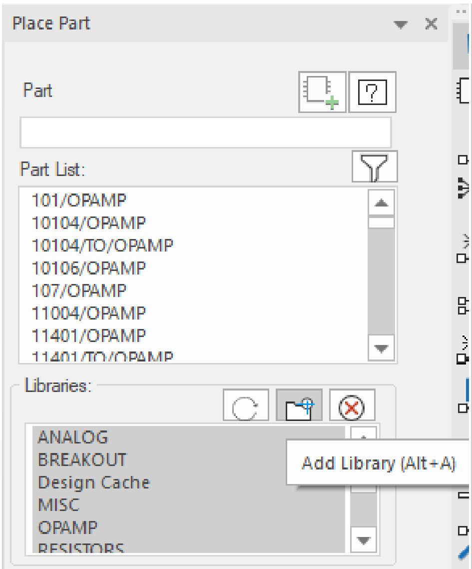
Specify the project name and directory, base this project on an "empty" project, or create a blank project.



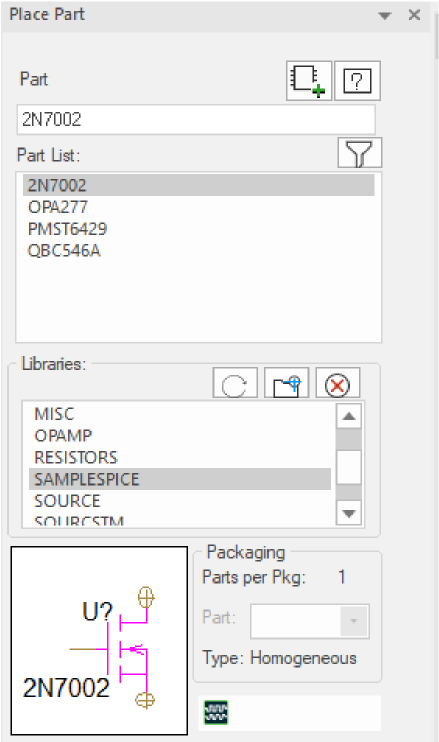
Open PAGE1 of the project and draw the schematic.

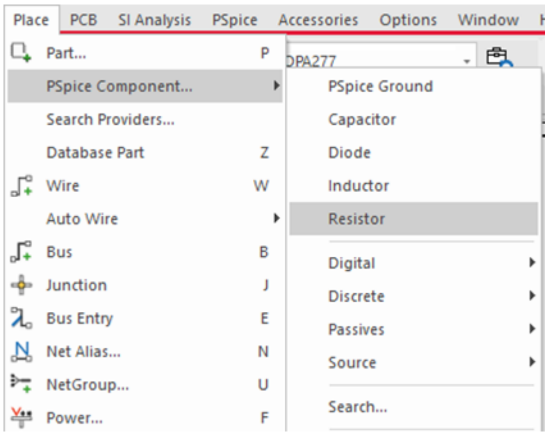


Use Place>Part and the rectangular Add Library icon button to add the created OLB file.

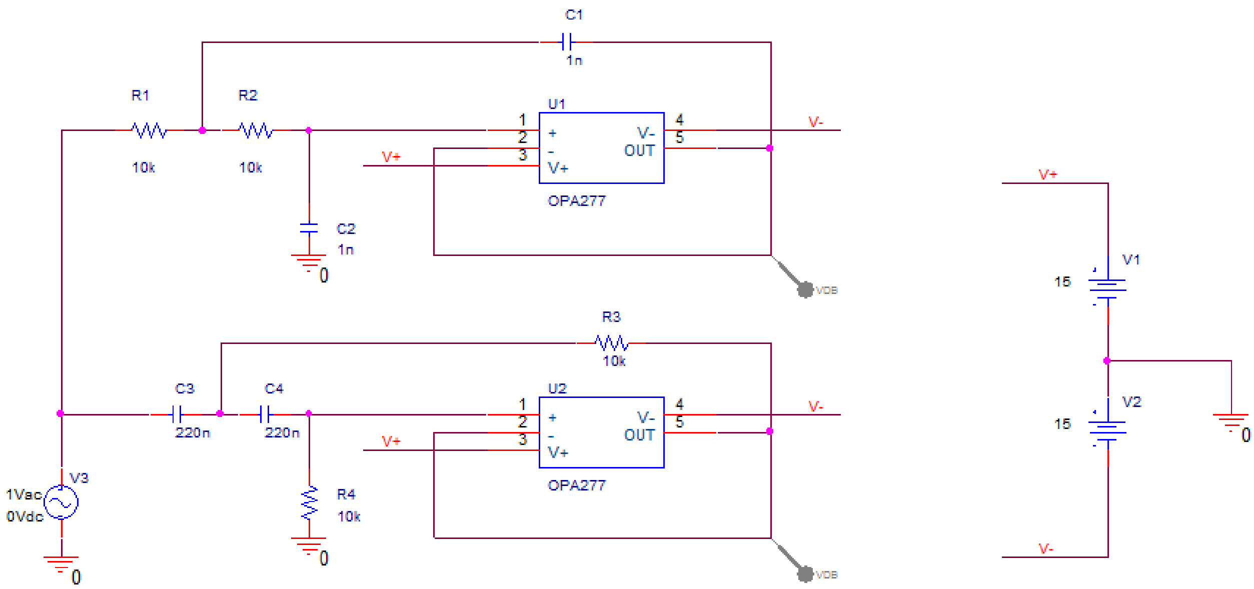


This will select the created OLB file in the library list.

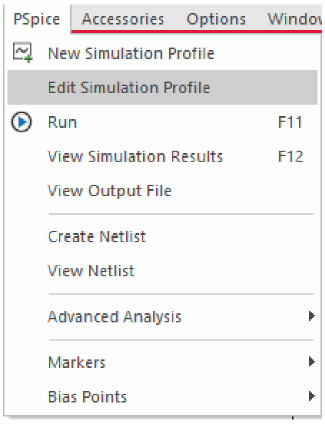




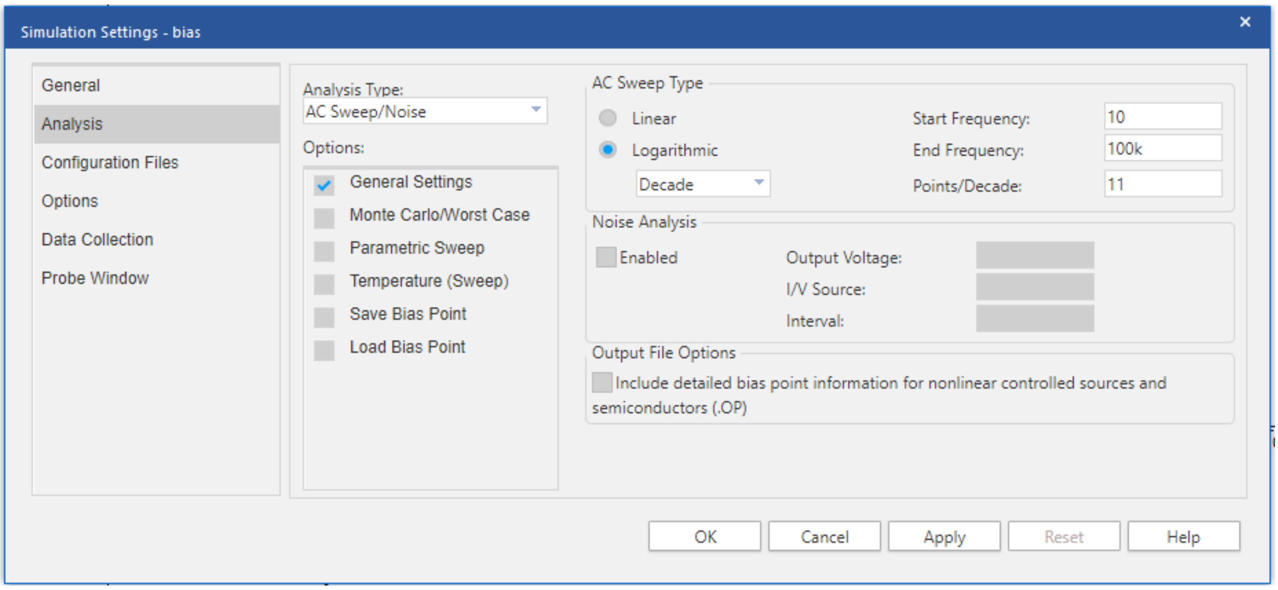
Select the OPA227, place two of them and complete the circuit. Ensure you place parts from the PSpice enabled libraries.



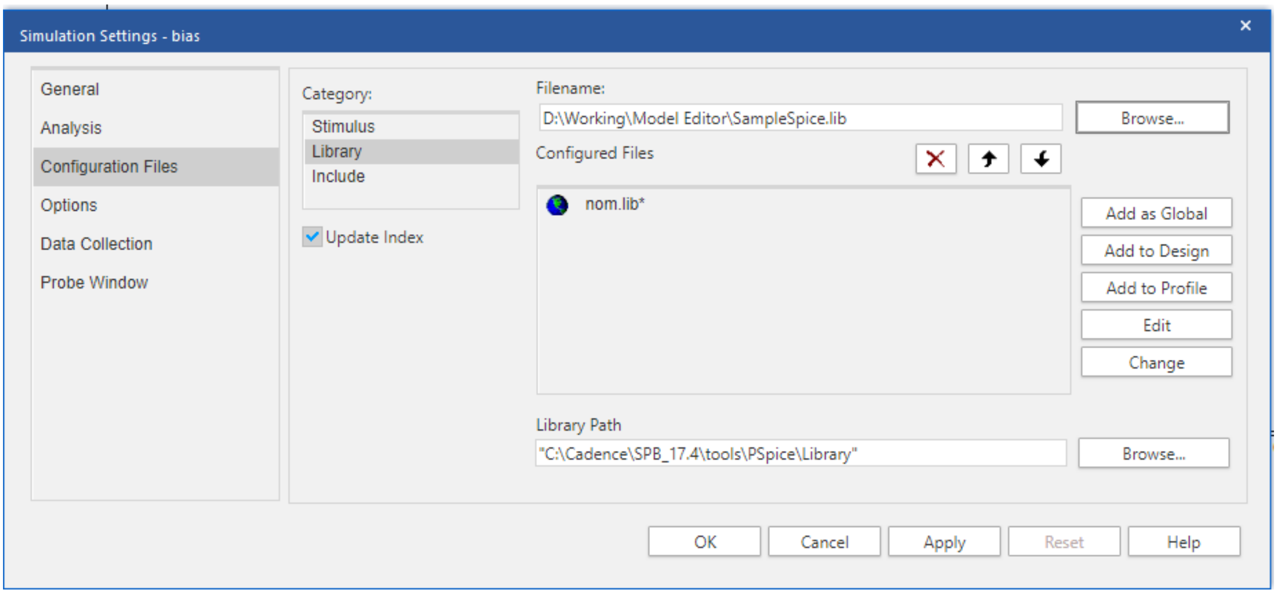
Use PSpice>Edit Simulation Profile, or create a New Simulation Profile.



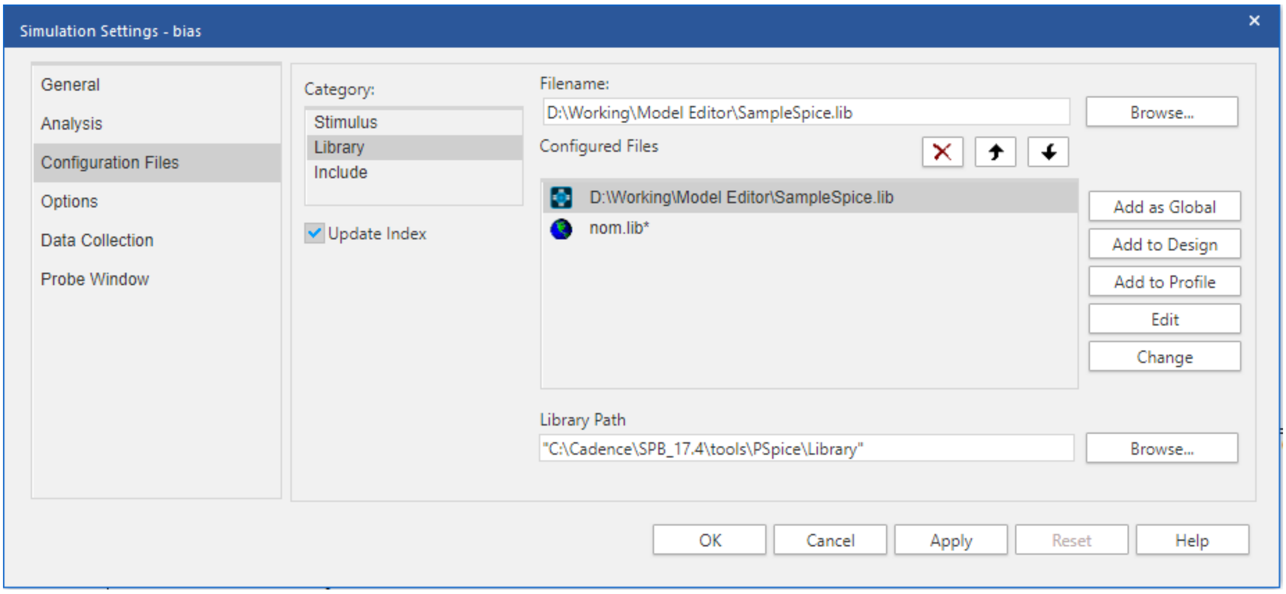
On the Analysis tab, for this case, set the profile for an AC Sweep from 10Hz to 100kHz.



Also, while still editing the profile, go to the Configuration Files, select the Library entry on the left and use the upper Browse button to browse for the LIB file that defines the simulation models.

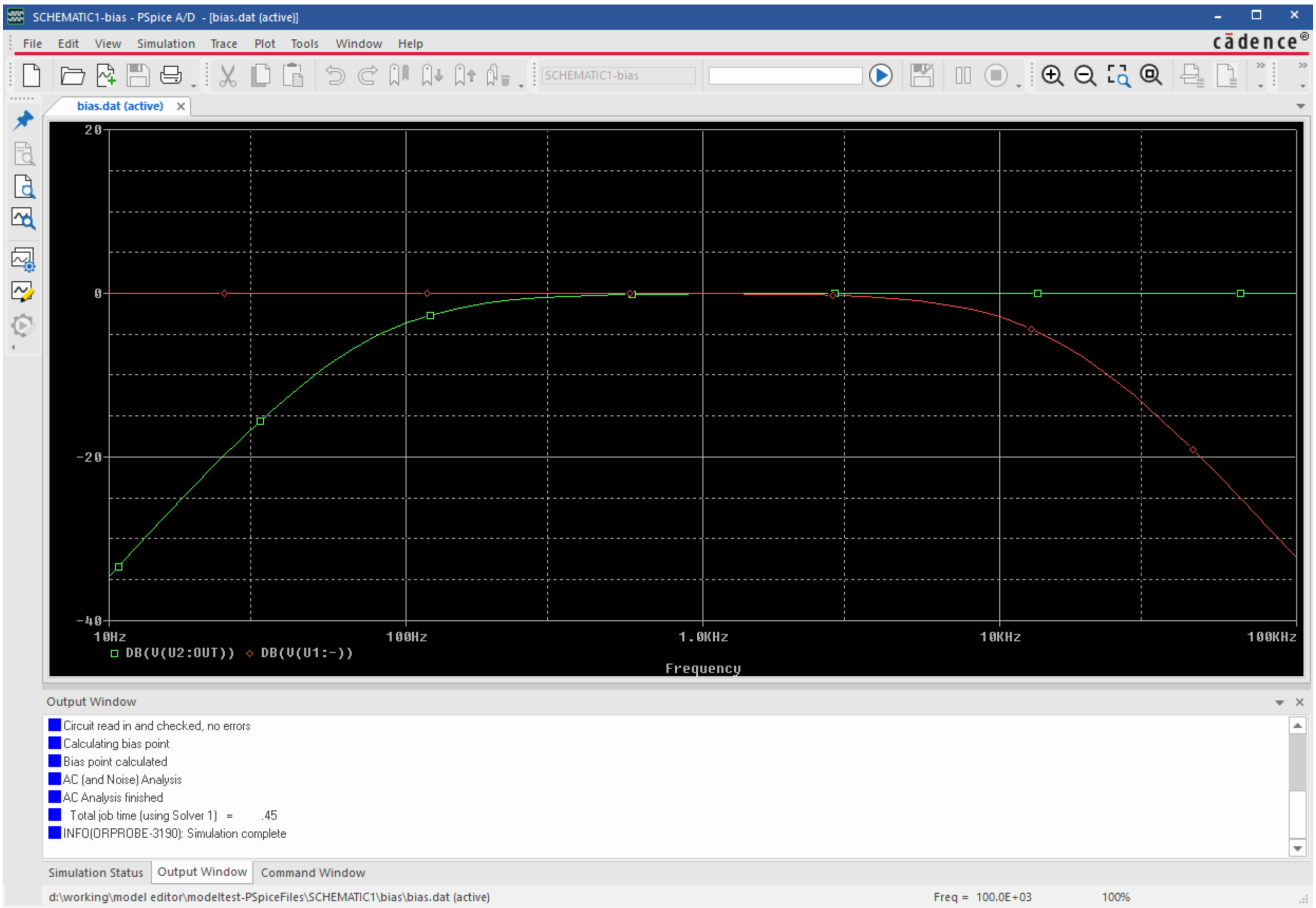


Use the Add to Design button to add the simulation library.



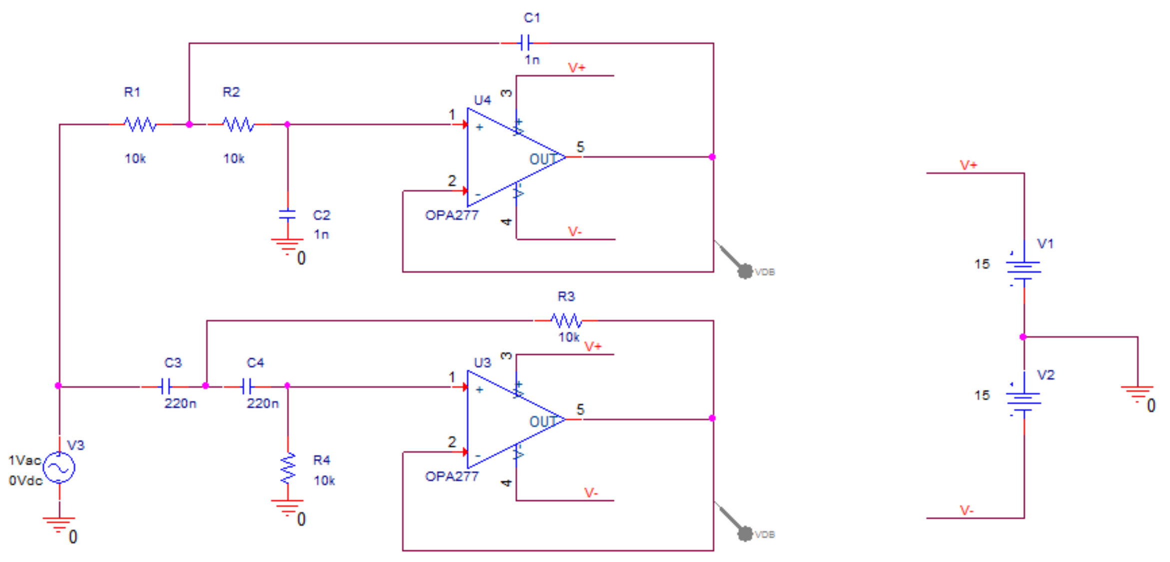
OK the Simulation Settings to close the simulation Profile.

From the PSpice menu use Run to run the simulation. Once the simulation has completed the Probe window will open with the simulation results and the traces for the active probes displayed.



Model Import Wizard Symbols

The circuit is redrawn with the Symbols created by the Model Import Wizard process and the simulation is run again. The modified circuit is shown below:



When the simulation is run again, the same results are displayed, as before: