User Manual

(For LTspice Models)

Import LTspice Model:

The provided zip file contains an LTspice model library file (.lib) and a symbol file (.asy).

There are two ways the models can be imported into the LTspice.

Method-1:

- Identify the LTspice default library path/location where all the components symbol/library are placed.
- In general, the default location for model file is as follows:

   **.lib file path:**   C:\Users\......\Documents\LTspiceXVII\lib\sub\

    .asy file path: C:\Users\......\Documents\LTspiceXVII\lib\sym\

- After identifying the library path, place (copy & paste) the "**.lib file and .asy file"** at their respective locations.
- Restart the LTspice and now one should be able to see the newly added component in the component library.

> **Note:** In the newer versions of the LTspice, the default library location may differ. One need to identify the default library location for this method.
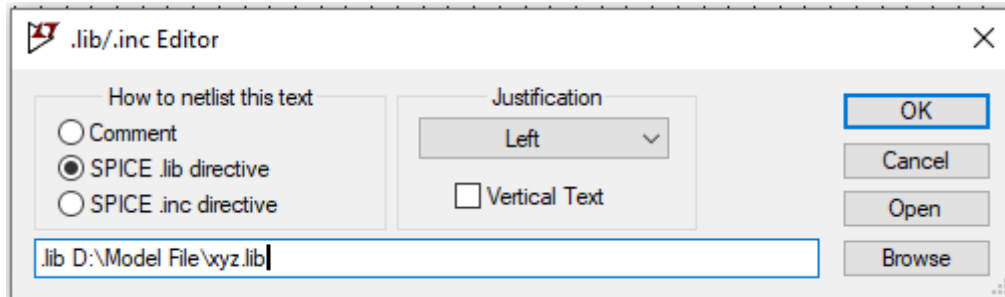
Method-2:

- In this method, there is no need to copy the model files (symbol file & library file) to the LTspice default library location.
- **To import the symbol file (.asy)**, the path for the .asy file needs to be written as follows:
    - Open LTspice > Tools > Sym. & Lib. Search Paths
    - Provide the path in the "Symbol Search Path":
        - E.g.: D:\Model Files\; the symbol file needs to be at this location.
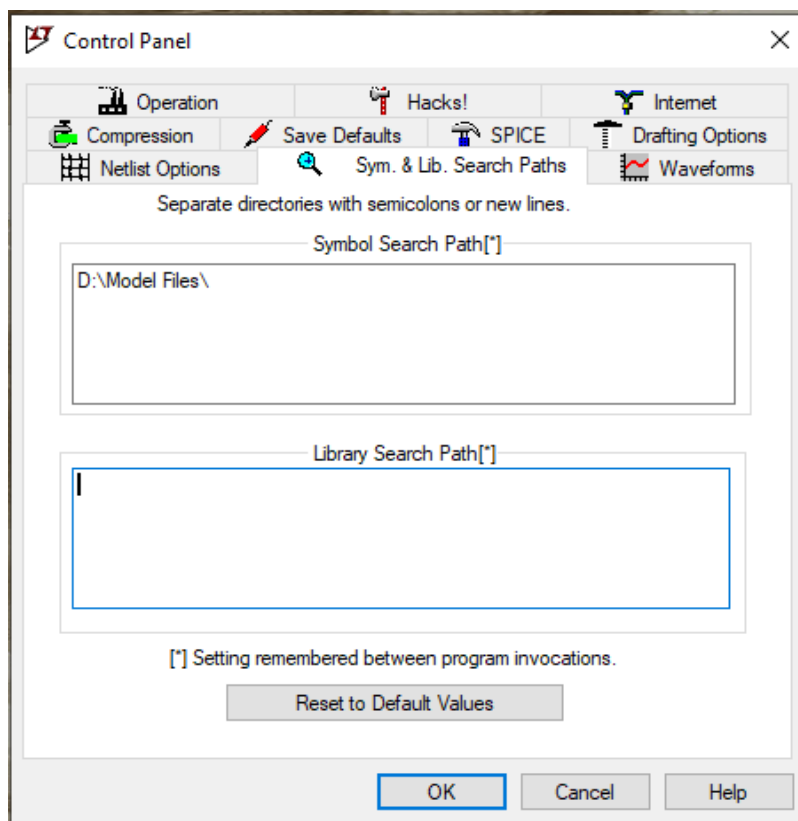
        - Click OK, now the component library should show the component.

- **To import the library file (.lib)**, Open Schematic in LTspice, write the SPICE directive as follows:
  - .lib <" Path for .lib file">

  - After including the .lib and .asy files, LTspice allows to simulate the schematic with new components.

Note: In this method, one needs to edit the symbol attributes and delete the default model file name from there. The steps are as follows:

  - Open the symbol file (.asy) with LTspice > Press <CTRL+A>
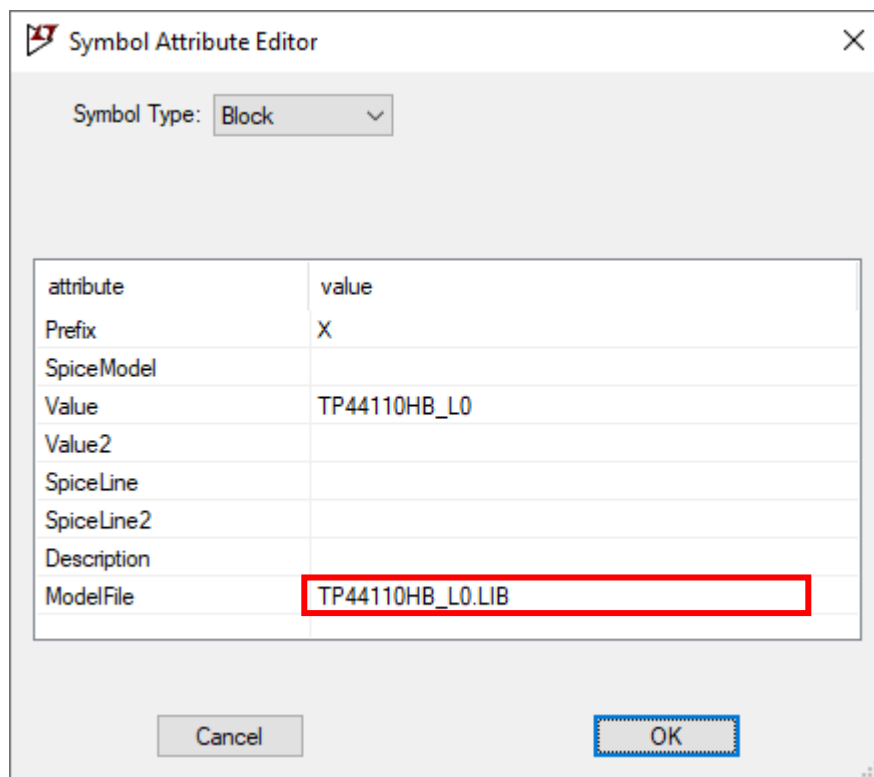
  - Open the Symbol Attribute Editor, double click on the "value of ModelFile" as shown in the above picture. It should allow to edit the value of ModelFile. The value should be deleted by user; click OK and save the symbol file.
  - Open the schematic and run the simulation.
  - Please note that this edit in the symbol attributes need not to be done in Method-1.

Note: The provided LTspice model supports the temperature dependent simulations. The temperature of the LTspice simulation environment can be changed in the LTspice tool by editing the global temperature as follows:

  - For example, to make the global temperature as 50 °C. Write SPICE directive ".**temp 50**". Default global temperature in LTspice simulation is 25 °C.