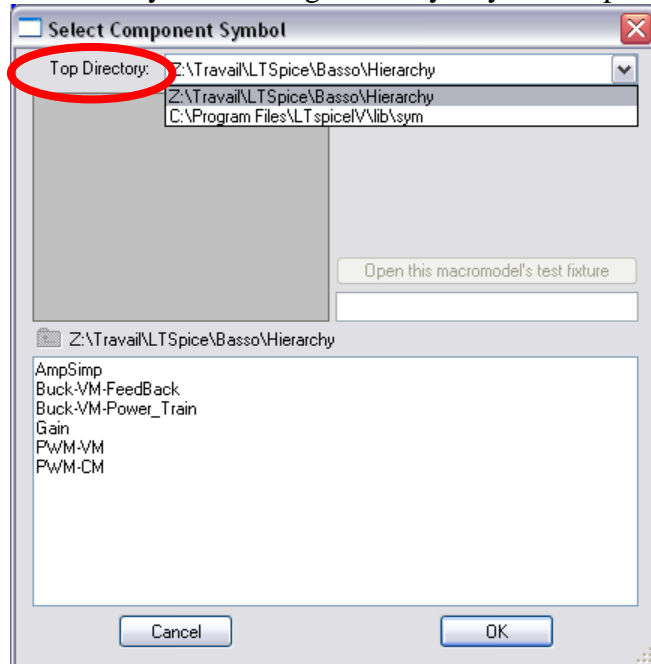


Using LTSpice Average PWM Model

Article I. First method : Hierarchy schematic

1. You need to add the files included in the Base.Zip in your working directory.
2. Create a new schematic in your working directory.
3. To add one model : Go to Edit->Component or F2.
4. Choose your working directory as your “Top directory”.



5. Choose the model you want.

Remarks :

One advantage of that Hierarchy method is that you have access to the model schematic, just double-click on the symbol. If something goes wrong with the results, you can plot the model internal voltages and currents.

Parameters used for the simulation are always the ones in the high level schematic.

Article II. Second Method : Using the Library.

To add the models in the library you need to :

1. Create a sub-directory called “PWM” in the LTSpice symbol library directory : ...\\lib\\sym
2. UnZip sym.zip in the LTSpice library directory ...\\lib\\sym\\PWM\\
3. Create a sub-directory called “PWM” in the LTSpice sub-circuit library directory : ...\\lib\\sub
4. UnZip sub.zip in the LTSpice library directory ...\\lib\\sub\\PWM\\

To use these models :

1. In your schematic, go to Edit->Component or F2.
2. Go in the sub-directory “PWM” and make your choice.

Remarks :

As models are in your library, you don't need to add them each time you start a new project (regard to the hierarchy method) but you can not edit the internal schematic.