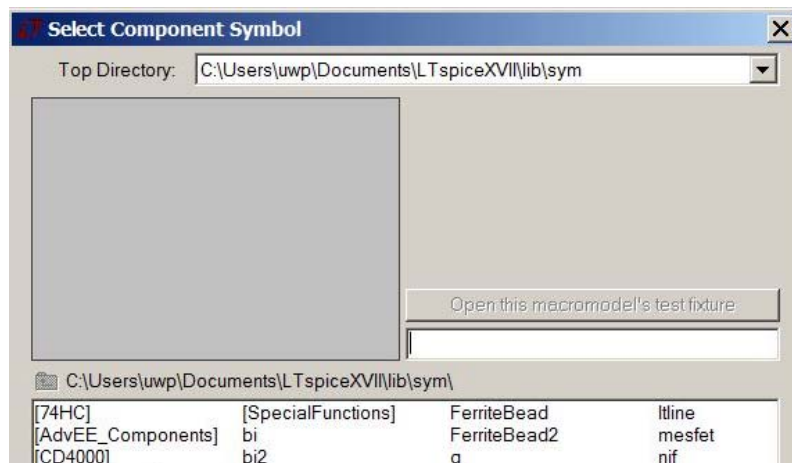# Installing the library for the PSpice simulator with the components of the course

Some of the analog components you need to simulate are not included in the libraries delivered with LTSpice. The missing parts are collected in the file 'AdvEE_Components.zip'. It contains a library with the model descriptions and the symbols for the circuit schematics. The different files have to be placed manually in the right directories below the main program directory.

First download and install the program. After the first call the standard libraries will be copied into the user space of the logged in user. The new library has to be added now.

1. Open a new schematic (in menu 'File' select 'New Schematic') then press 'F2'. The following window should appear (bottom cut away):

> **Select Component Symbol** ✕
>
> Top Directory: C:\Users\uwp\Documents\LTspiceXVII\lib\sym
>
> Open this macromodel's test fixture
>
> 📁 C:\Users\uwp\Documents\LTspiceXVII\lib\sym\
>
> | | | | |
> |---|---|---|---|
> | [74HC] | [SpecialFunctions] | FerriteBead | ltline |
> | [AdvEE_Components] | bi | FerriteBead2 | mesfet |
> | [CD4000] | bi2 | g | nif |

2. Record the file path in the top row under 'Top Directory'. Close the program.
3. Create a directory "X:\...\lib\sym\AdvEE_Components"
4. Unpack the files from "AdvEE_Components.zip" into this directory.
5. Shift or copy the file "AdvEE_Components.lib" to " X:\...\lib\sub"
6. Start LTSpice. To use the new library LTSpice has to detect it first. From menu select 'Tools' then 'Sync Release'. Confirm the first text box with 'Yes'. For the second text box it depends. Read it and choose. 'No' should work too.

Now you can select the circuit elements in the schematic and the models are visible for the simulator!