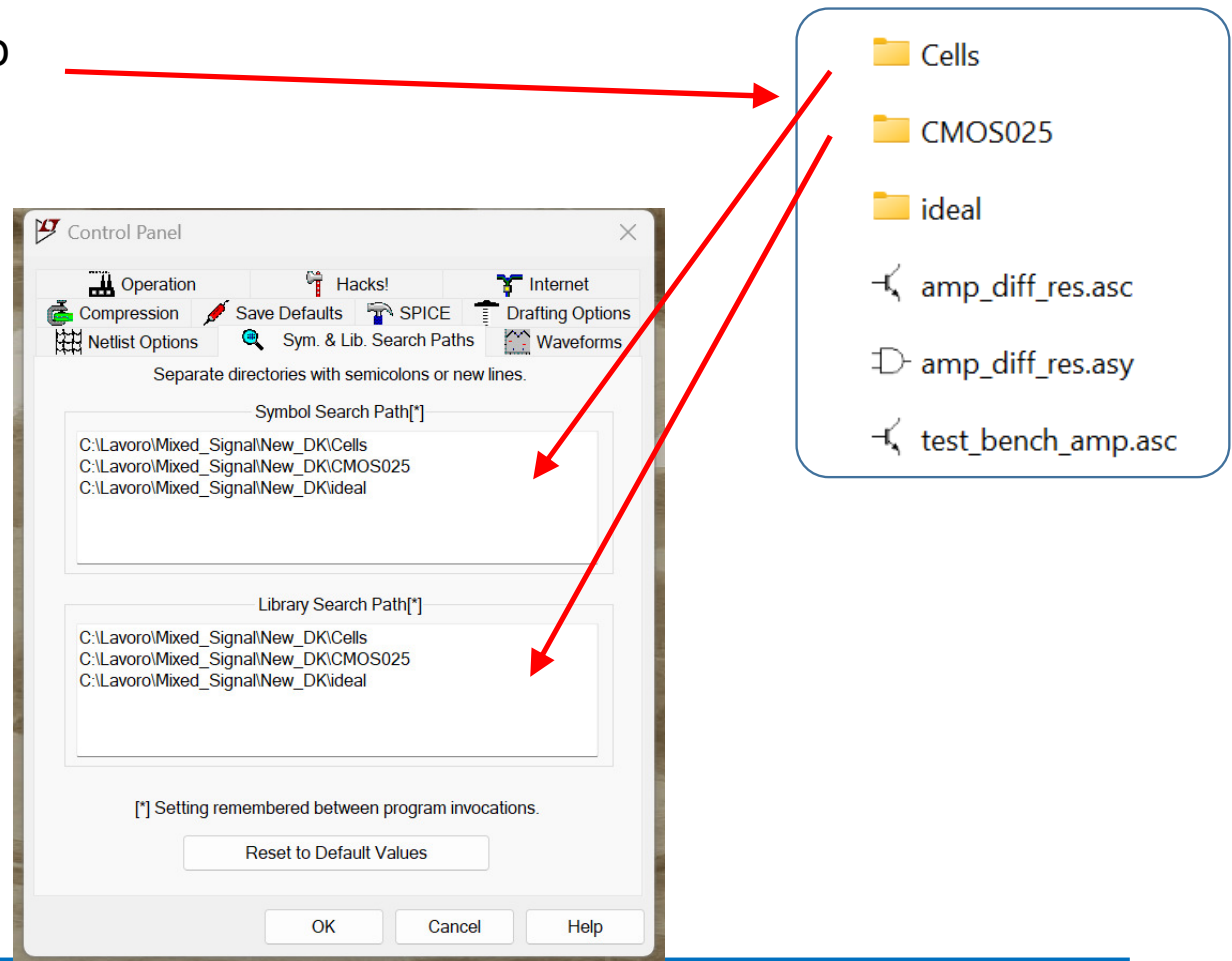
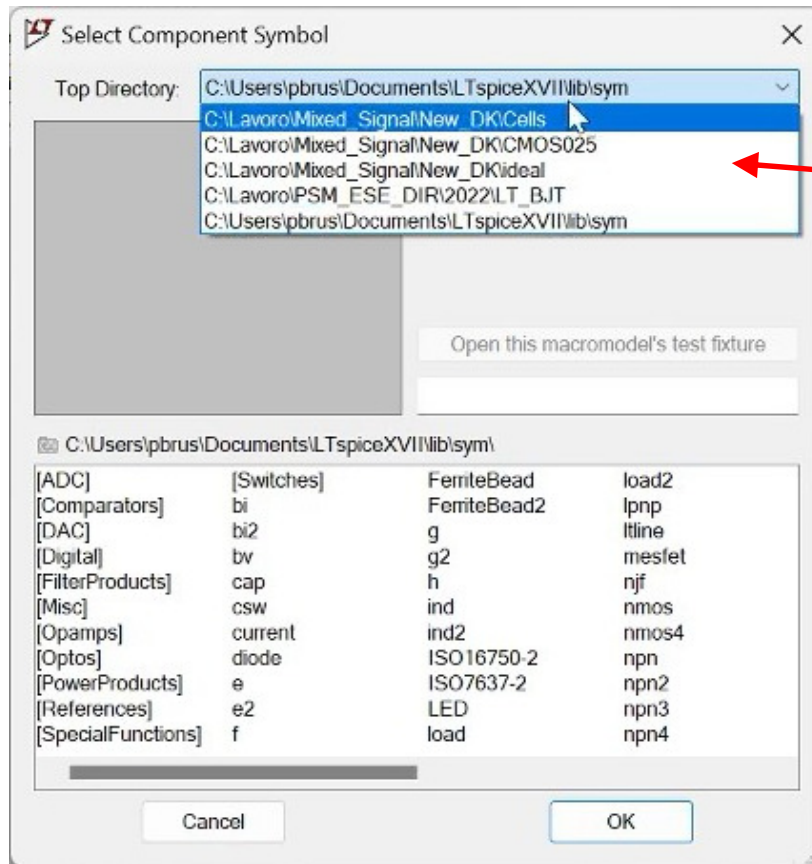


Installation and Use of the LTSpice custom libraries

1. Uncompress the New_DK.zip file into a directory, possibly with a not too long path (e.g. c:\Users\my_home\Mixed_Signal)
2. Launch LTSpice
3. Open the LTSpice control panel, "Sym & Lim Search path"
4. Add the path of the tree directories "Cells", "CMOS025", "Ideal" to both the "symbol search" and "library search" fields



How to include a cell into a schematic view



Simply operate as usual, by command "F2".

In the window that opens, now the added libraries appears in addition to the standard LTSpice Libs

- **CMOS025** : components of the process (CMOS^N025 and CMOS^P025)
- **Cells**: Some CMOS cells useful for the exercises (e.g. pass-gate)
- **Ideal**: Ideal components to be used to simplify simulations.

How to set the L and W parameter of MOSFETS

After right-clicking on the MOSFET of interest, this window opens

Simply edit this line. Be careful not to delete the "u" suffix (microns).

Note: It is not necessary to put the ".lib " command into LTSPICE schematic view

