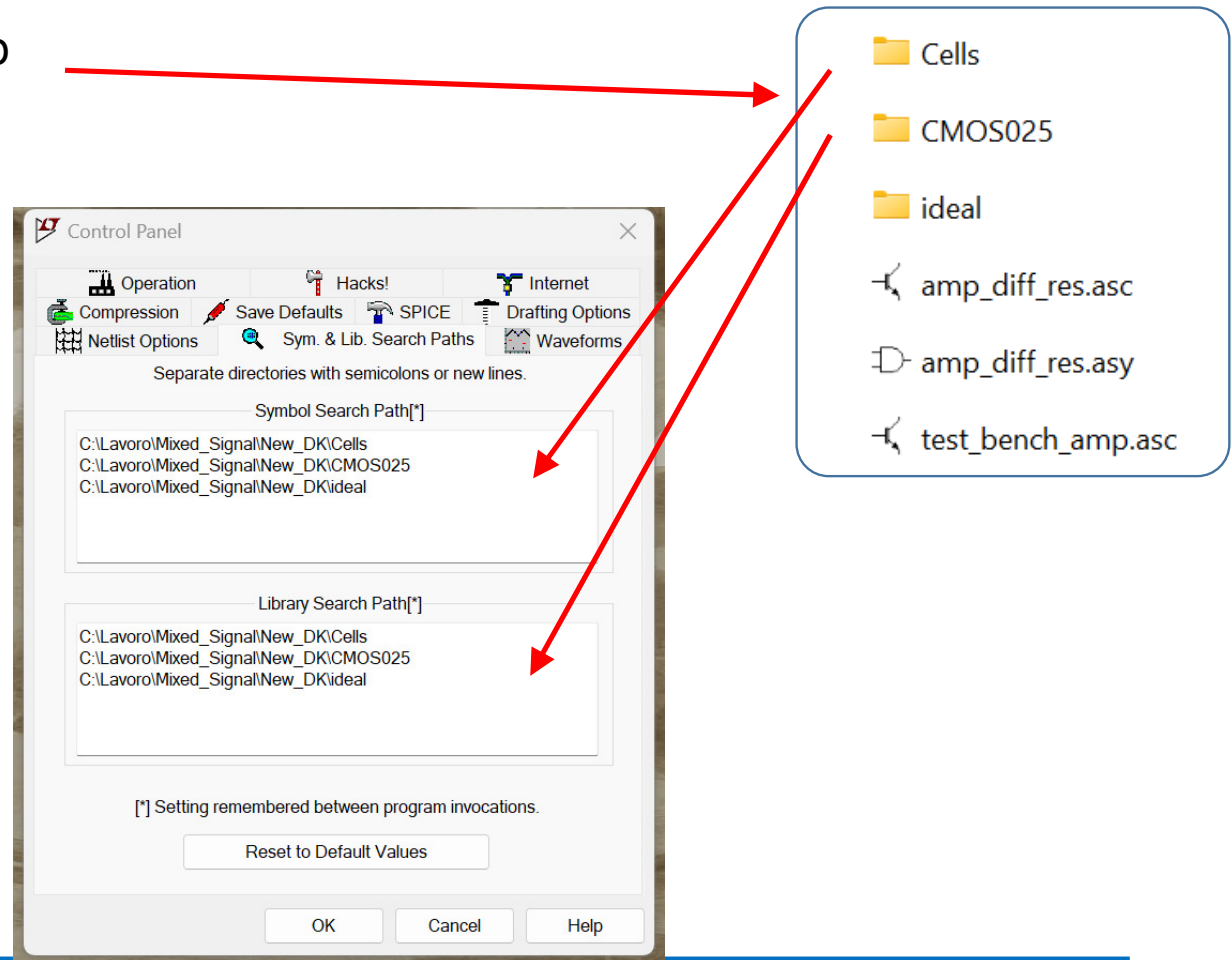
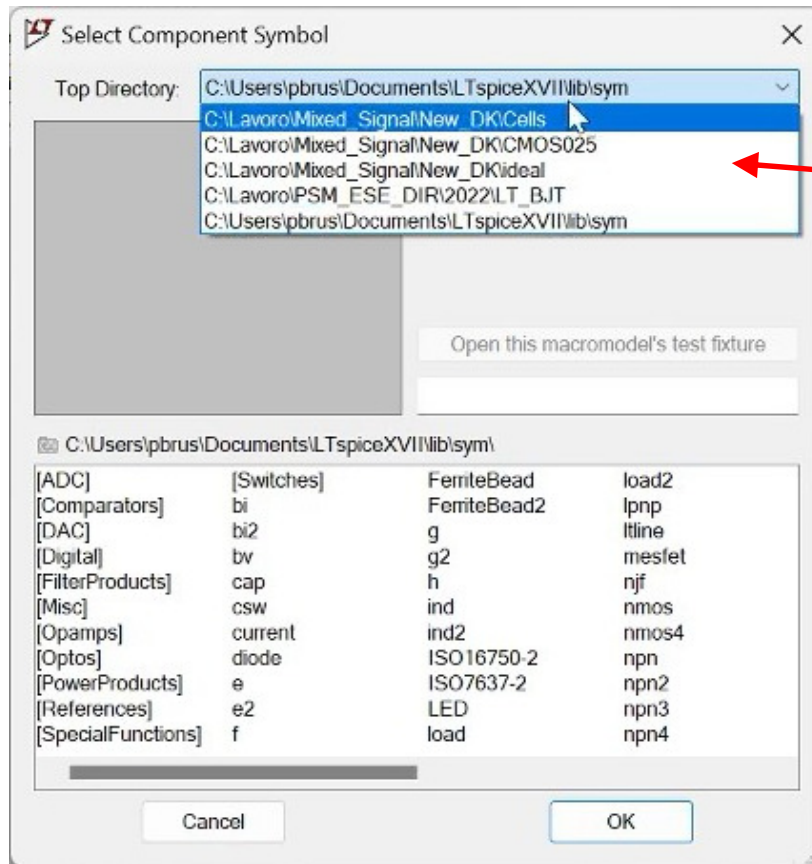


# Installation and Use of the LTSpice custom libraries

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~~N~~025 and CMOS~~P~~025)
- **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

