Installing LTSpice

The software can be downloaded from the following address:

http://www.linear.com/designtools/software/

On the same page you will also find tutorials in pdf or video formats.

Extensions of important LTSpice files and their significance:

- *foo.asy* – symbol associated with a subcircuit, it must bas saved in the same folder with the corresponding *.asc* file
- *foo.asc* – schematic of project top level file
- *foo.lib* – model library (Spice primitives or netlist description of subcircuits)

Contents of the *ltspice.zip* archive:

- SimboluriLTspice – a folder containing the bipolar (*npn.asy* and *pnp.asy*) and MOS (*nmosb.asy* and *pmosb.asy*) transistor symbols used throughout the lab schematics;
- FisierConfigurare – the configuration file (scad3.INI) that redefines the keyboard shortcuts of LTSpice;
- LibrariiLTSpice – a folder with model libraries for bipolar (bipolar.lib) and MOS (*180nmL7.lib*, *90nmL7.lib*) transistors.

How to use the contents of the *ltspice.zip* archive

The symbols in the folder SimboluriLTspice will be copied (overwritten if required) into the folder *C:\Program Files\LTC\LTspiceIV\lib\sym* (this is the default symbol location).

The configuration file (*scad3.INI*) will be copied into the folder *C:\Windows* (overwrite if exists).

The libraries from the folder LibrariiLTSpice will be copied into the active project folder or working directory (this is the location from where schematics foo.asc are opened).

Specification of the library and the transistor symbol association with a preferred model (for new projects):

- choose one of the symbols *npn*, *pnp*, *nmosb* or *pmosb* from the **Edit\Components** menu;
- on the chematic left click to place as many components as necessary on the sheet, then right click to reactivate editing mode;
- change the names of the components from *npn*/*pnp* in *npnver*/*pnplat* and from *nmosb*/*pmosb* in *n018*/*p018* for AIC and *n90*/*p90* for SDIC;
- choose the transistor parameters (W,L, As, Ad, Pd, Ps, M);
- copy the libraries *bipolar.lib*/180nmL7.lib/90nmL7.lib into the working directory;
- the libraries can be included in a schematic by using the Spice command **.lib .include** from the **Edit\Spice Directive** menu (e.g. **.lib 180nmL7.lib**).

Now the simulator should be able to find the transistor models and run your simulations without library related errors.