

LT Spice - Add New Components

[Home](#) | [Analysis](#) | [Help](#) | [Media](#) | [Links](#) | [Practical](#) | [Schematics](#) |
[Simulation](#) | [Updates](#) |

Please Note: All the information presented here is my own work and not from Linear Technology. This advice does not replace the information given by Linear Technology, the [LTWiki](#) or the [Yahoo LTspice User group](#) . Please also read my sites [general disclaimer](#).

Adding New Components using Windows

LTspice comes with a selection of components and models, but you may find on occasion there is a need to add new parts and models. You can design your own, but you may find that one has already been created.

The [yahoo LTspice user group](#) has a files section where you can download new components. You need to join the group first, then browse to the files section and lib (library) directory, screenshot below.

LTspice : Files - Mozilla Firefox

File Edit View History Bookmarks Tools Help

http://tech.groups.yahoo.com/group/LTspice/files/ Lib/Digit

openSUSE PCLinuxOS - Main Getting Started MyPCLinuxOS - Ex... The PCLinuxOS Ha...

LTspice : Files

Attachments

Files

Photos

Links

Database

Polls

Members

Calendar

Promote

Groups Labs (Beta)

Applications

Yahoo! Groups Tips

Did you know...

Show off your group to the world. [Share a photo of your group with us.](#)

Best of Y! Groups

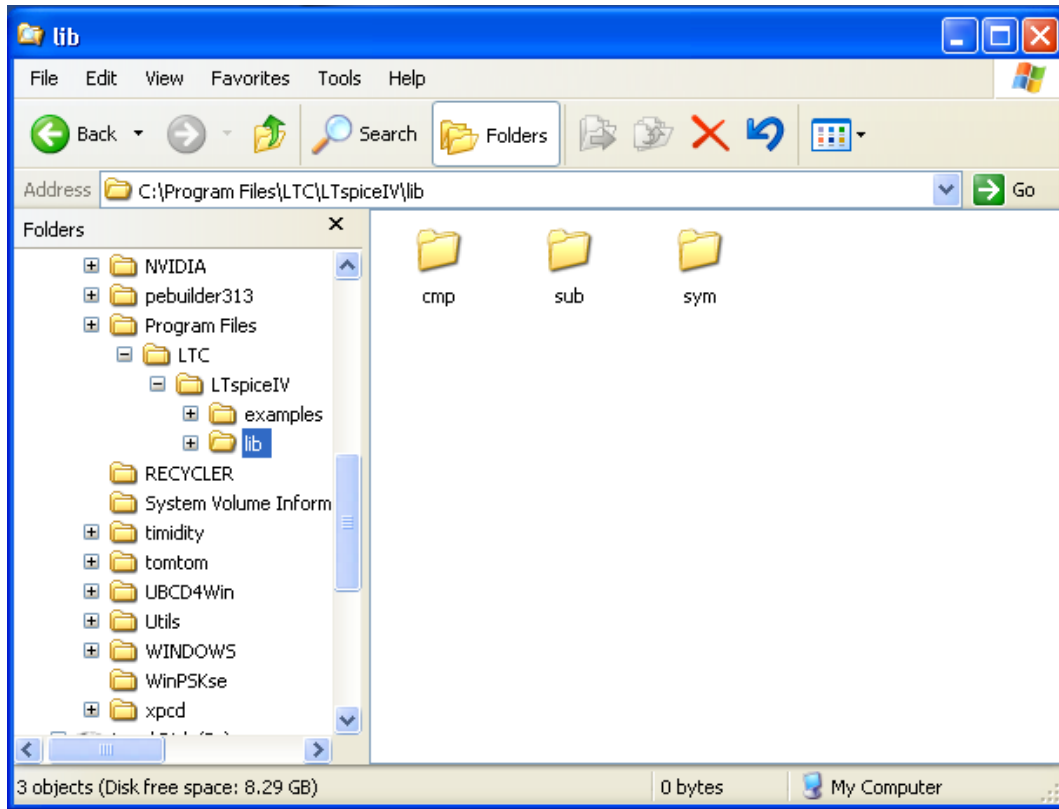
Check them

Files > Lib > Digital 74HCTxxx

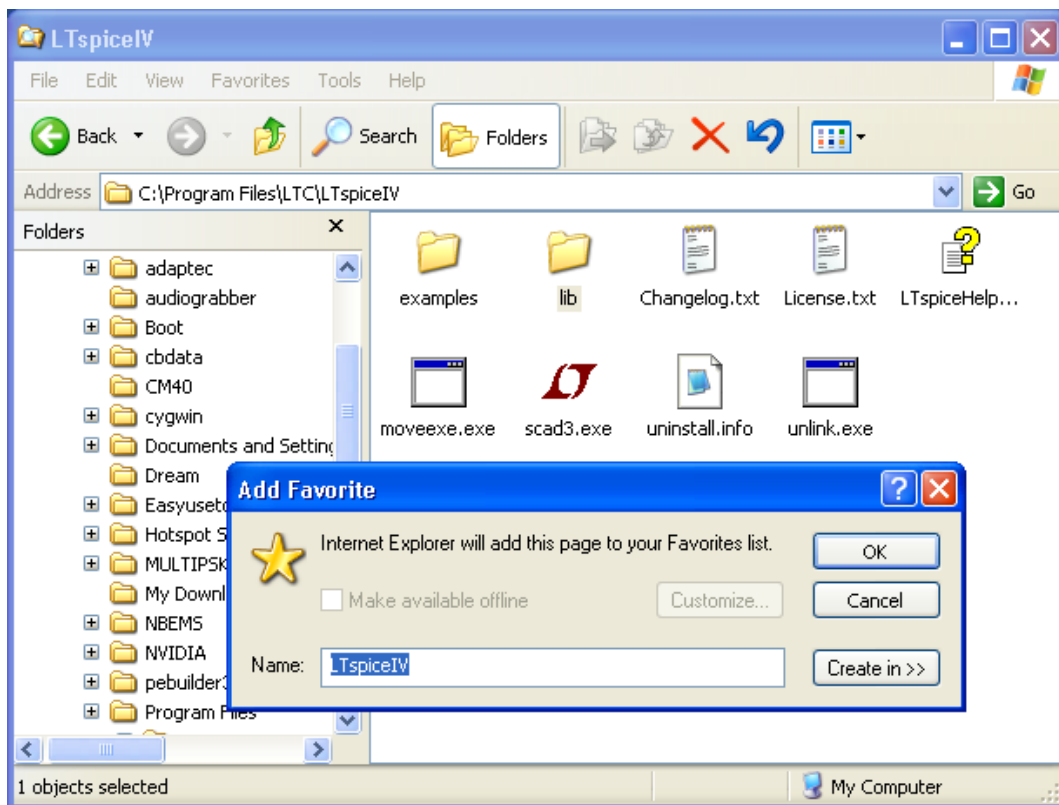
Name	Size	Creator	Created
74ACT Using the 74HCT library for 74ACT devices, a workaround.		helmutsennewald	Jun 28, 2005
74hct.lib 74hct.lib Library file v0.60 of the 74HCTxxx parts	112 KB	helmutsennewald	May 30, 2010
74HCT.zip Symbols of the 74HCTxxx parts, V1.04	66 KB	helmutsennewald	May 30, 2010
74HCT_Example.asc Circuit using digital 74HCTxxx parts	5 KB	helmutsennewald	Aug 21, 2003
74hct_Example.plt LTSPICE plot definition file for example.asc	1 KB	helmutsennewald	Aug 21, 2003
74HCT_Example.zip Requires 74hct.lib	6 KB	helmutsennewald	Dec 14, 2008
dview.lib Digital Signal Formatter/Viewer mdl	2 KB	helmutsennewald	Dec 14, 2008

Done

Download your desired files and save them to your computer. Next you need to place them in the required directories. On windows this location will generally be C:\Program Files\LTC\LTspiceIV\lib\ Open windows explorer and browse to the lib folder (screenshot).

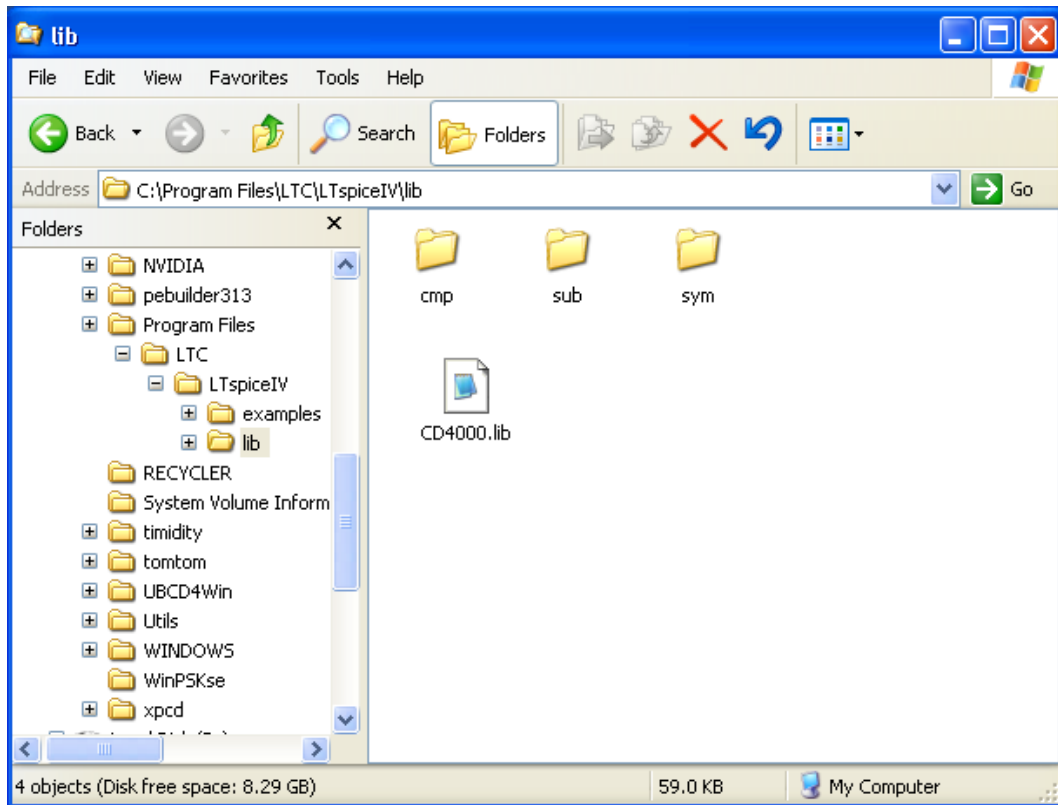


Inside C:\Program Files\LTC\LTspiceIV\lib will be three more folders, cmp, sub and sym. It is a good idea to bookmark this location for future use. Click Favorites and add a suitable name.

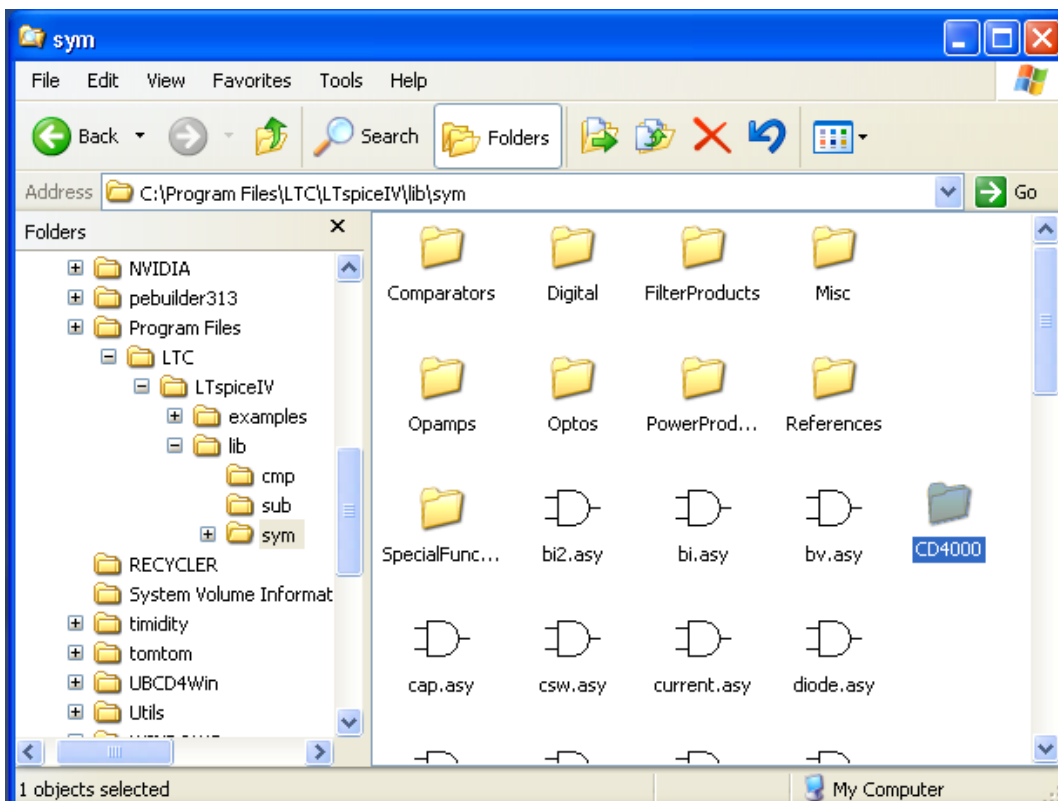


To add for example the CMOS4000 library, download the files CD4000.lib and CD4000.zip. The lib folder can be placed anywhere inside lib directory. Many library files are kept in the sym folder but as long as an include statement (`.inc CD4000.lib`) is

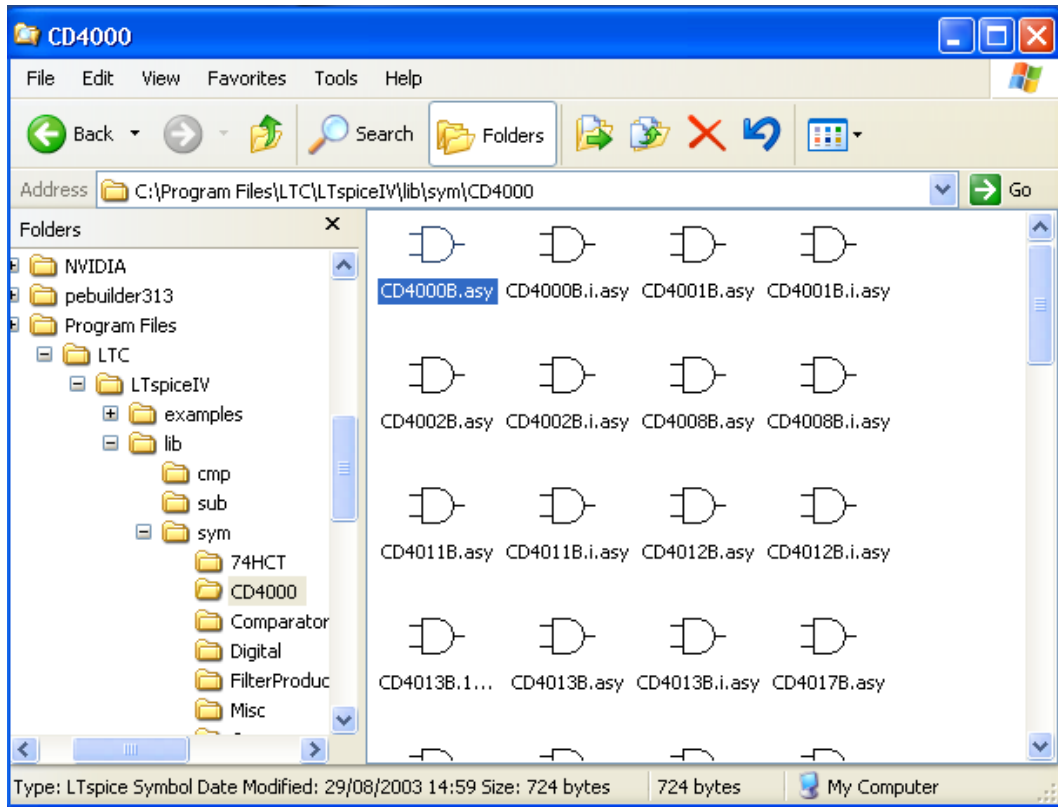
placed on the schematic they will be found as the search is recursive through the directory structure. (Screenshot below.)



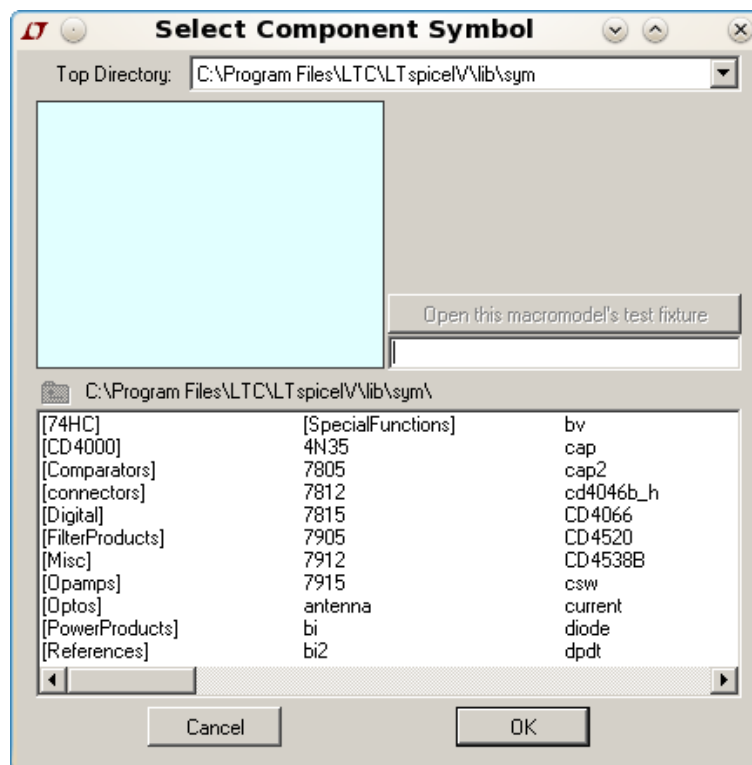
Now it is also necessary to add the symbols for the CMOS4000 series. The symbols must be placed in the sym folder. The symbols are compressed into a zip file, copy CD4000.zip into the sym folder and extract it, a new sub folder called CD4000 will be created.



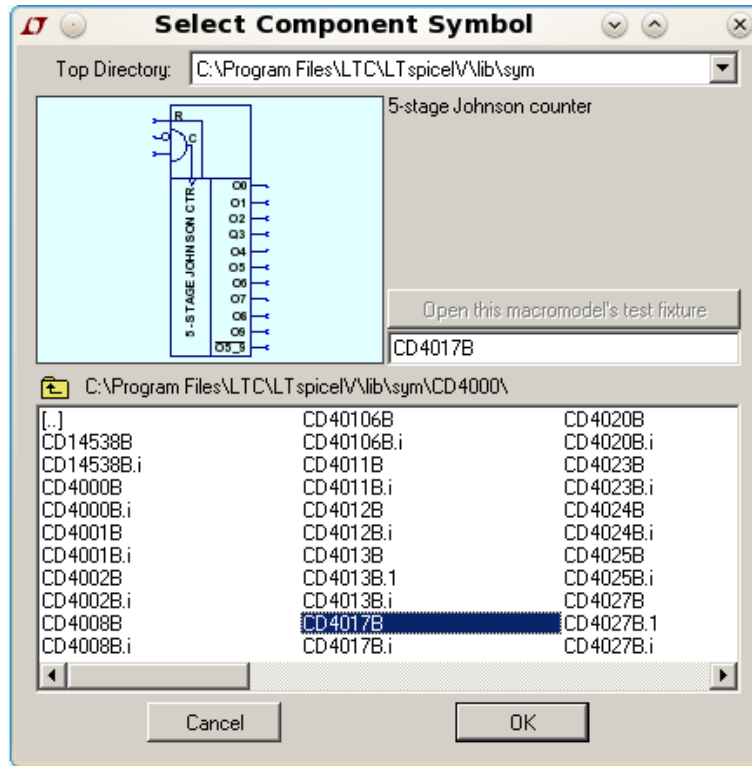
Inside the CD4000 folder, the symbols of many of the 4000 series IC's have been created.



Using the new components is easy, start a new schematic, click parts (or press F2) and you should now see the new CD4000 library (screenshot below):-



Selecting the [CD4000] folder should now display many of the CMOS4000 series IC's. Clicking the model should also preview the symbol.



Once created remember to add the include statement.

Important ! If designing circuits for other people then all component libraries, models and any custom symbols need to be included in the same file otherwise they will get a component not found error.

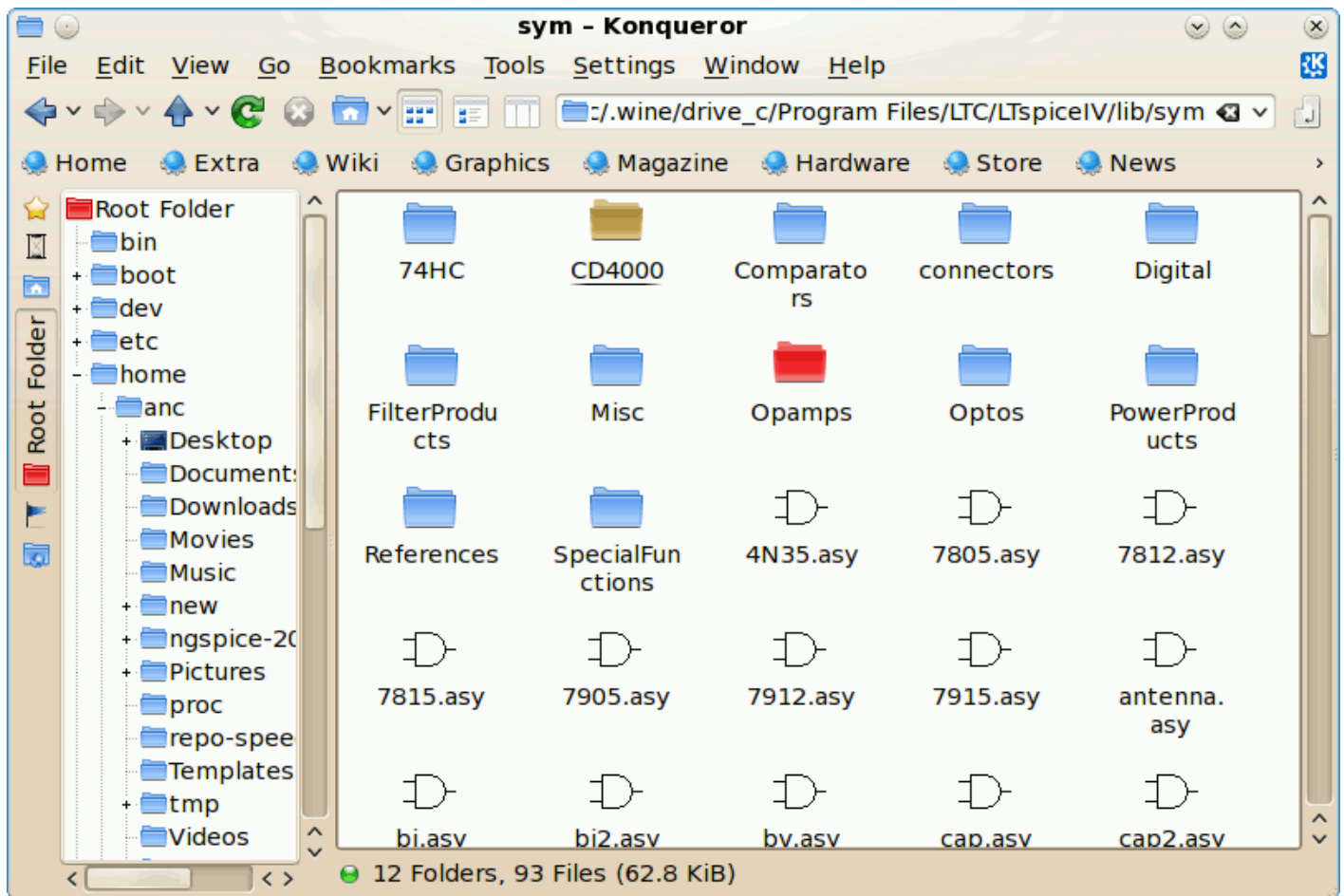
The following example circuit is an example using the CMOS 4000 library and LTspice : [1 Hour Timer](#)

Adding New Components using Linux

If you are running LTspice from linux then installing new components is the same as for windows. The main difference is the location of LTspice.

LTspiceIV runs perfectly as long the wine program is installed. Wine is software that allows many windows programs to run on linux. A hidden .wine folder will be created for each user on the system. Download the component files to `~/.wine/drive_c/Program Files/LTC/LTspiceIV/lib/cmp`

You can bookmark this location in konqueror or nautilus or simply copy using the terminal. A screenshot using konqueror file manager is below:



Adding New Discrete Components

To add new transistors [download this file](#) and save in the cmp directory. This file adds new European transistor e.g. BC107 series, BC547 series and many of the circuits on Circuit Exchange. [Please note](#) this file will overwrite existing model files in LTspice.

Shell Script for linux Users

The following bash script is designed to make life easier when adding new components to LTspice. It is a simple shell script that can be modified and places all component, library, model and subcircuit files into the relevant directory in LTspice. My download location is /media/share/electronics/ltspice Just replace this with your download location for your system.

[Transfer shell script](#)

Just download the script and execute it with `./transfer.sh`

[Return to LTspice Index](#)