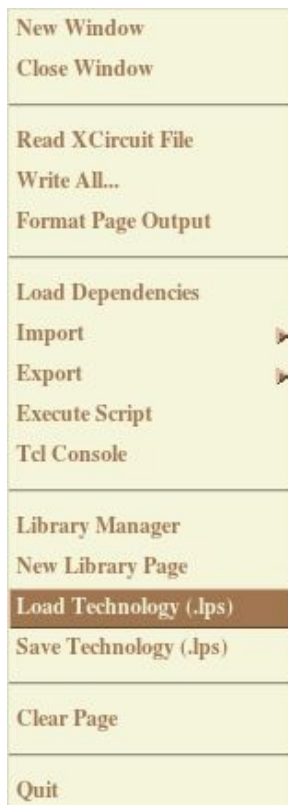Using symbol libraries

XCircuit has a small library of circuit symbols that by default is available within the program. Once the program is running, to see all the loaded libraries press a capital "L" letter. Or to select a library press a lower case "l" and the default library should be displayed. While viewing a library page pressing a lower case "l" again will select the next available library. Press ESC to close a library view and return to editing without selecting a symbol or click on a symbol in a library to select it and place it a drawing.

Loading a user library

A user of XCircuit can import a library that has already been created as a user library, modify the user library, or create a new library. I have created a library file named wwuXCircuitLib.lps that can be downloaded from the ENGR-355 class web page. This library has A and B sheet size borders for drawings, A/D converter, D/A converter, and several types of diodes.

If the wwuXCircuitLib.lps file is placed in the directory (i.e. folder) that will be used when creating a drawing using XCircuit, then it will appear in the file selection window and you will not need to enter the complete path to its location.

To load a library click on File on the top toolbar. A drop-down menu opens:

New Window
Close Window

Read XCircuit File
Write All...
Format Page Output

Load Dependencies
Import ▶
Export ▶
Execute Script
Tcl Console

Library Manager
New Library Page
Load Technology (.lps)   <—— Select Load Technology
Save Technology (.lps)

Clear Page

Quit

Then click on a displayed file to load or enter the path and filename:

The user library will need to be loaded each time you start XCircuit.

If you wish to create symbols of your own and place them in the user library do the following:

1) Draw an object in XCircuit
2) Select the object that you wish to add to the user library as a component.
3) Press the letter m on the keyboard. A dialog box like this will open:

4) Enter a name for the object. Do not put spaces in the name. Click Okay
5) Selecting the user library (by pressing the "l" key, possibly multiple times) you should see the
   new item in the library. You can select the new symbol and place it in a drawing.
6) To save the modified library, again select File on the main tool bar and then click on
   Save Technology (.lps)