This document contains information about running PSpice on a Macintosh II.

PSpice, being a circuit analyzer, does not interact with the user. Very little about its operation is different than the operation of PSpice on any other machine for which we make a PSpice.

To run PSpice:

- double click on the PSpice application;
- select open from the File Menu;
- choose an input (circuit net list) file;
- choose an output file.

To abort PSpice, either:

- click on the close box;
- select Quit from the file menu
- type a . (period) while holding down the Apple key.

Note that once a circuit file has been run (without errors), you may double click on its icon to re-run it. This enables you to run long analyses overnight. Read under Extended Capabilities for more information on this.

If you run PSpice under a shell (the MPW shell, for example), be aware that typing:

```
PSpice example1.cir example1.out
```

will not behave as typing the same line on an IBM PC would. On the Macintosh, all files passed to PSpice are treated as input files, whereas on the PC, you may specify the output filename along with the input filename. On the Macintosh, both example1.cir AND example1.out will be treated as input files.

Extended Capabilities

PSpice's Macintosh features are limited, due to the nature of the application:

- You may move the window around on the screen
- You may make the window shrink down to nothing by clicking on the Zoom box while holding down the Option key. (The next click on the Zoom box restores the window.)
- You may make the display appear in color by selecting Color Monitor in the Color Menu.
- You may select About PSpice under the Apple Menu to get the version number and your ID number.
- You may define a list of folders in thich to search for Libraries. See under Libraries later on.
- You may (optionally) have PSpice change the Finder information for a file if it completes a simulation without errors such that you may Shift-Click on a number of files to re-simulate. This is useful if you want to make a change to a component value after you have simulated once. Select Miscellaneous under the File Menu for more information.

Application Memory Size

PSpice is set up to run in a 1024K region. This will enable you to simulate a circuit of about 200 transistors. If you are running under the MultiFinder and PSpice indicates that it ran out of memory, you can either run under the Finder, or you can increase the suggested size of the CurrentPSPice application. You may change the memory size allotted to PSpice by single-clicking on CurrentPSpice (not PSpice), then typing an Apple + I to get the Information Box on the screen. The suggested size appears in a box in the lower right. Try increasing by 256K increments.

Libraries

We have incorporated an indexing scheme into the library search mechanism to speed up access. For each library XXX.LIB, there is a corresponding XXX.IND which has information concerning the members of the library, such as the offset from the beginning where each .MODEL and .SUBCKT may be found. The XXX.IND files are automatically rebuilt whenever an inconsistency is encountered.

You may define a sequence of folders in which to search for libraries you circuit may need. The Libraries under the File Menu enables you to define and delete, as well as to view, the list of folders in which libraries are searched. When PSpice encounters a .LIB statement, it will first search the folder in which the circuit files is in; second, the list of folders you specified, in the order you specified; last, PSpice will search the PSpice folder. Select Libraries under the File Menu for more information.

The Library search path, along with window placement information is saved in a file named PSpicePreference in the root folder on your hard disk. The Library information will become invalid if you restore a backed-up system from tape. In this case, you must delete the PSpicePreference file or your simulations will fail.