Running HSPICE Interactively

For a full discussion, refer to Chapter 5, Using Interactive Mode. Interactive mode enables you to use these HSPICE commands at the HSPICE prompt to help you simulate circuits interactively:

ac [...statement]	cd
dc [...statement]	edit
help	info outflag
input	list [lineno]
load filename	ls [directory]
measure [statement]	op
print <tran/ac/dc>,v/vm/vr/vi/vp/vdb>	pwd
quit	run
save *netlist/command filename*	set outflag *true*	*false*
tran [...statement]	

Starting Interactive Mode

To invoke the interactive mode, enter:

```
hspice -I
```

You can also use the `help` command at the HSPICE prompt for an annotated list of the commands supported in the interactive mode.

The interactive mode also supports saving commands into a script file. To save the commands that you use and replay them later, enter:

```
hspice> save command filename
```

Running a Command File in Interactive Mode

To run the command you have saved in a command file, enter:

```
hspice> -I -L filename
```