Simulations in the IBM design kit:

Start the Analog Design Environment (ADE). If you are viewing a schematic:
Tools → Analog Environment

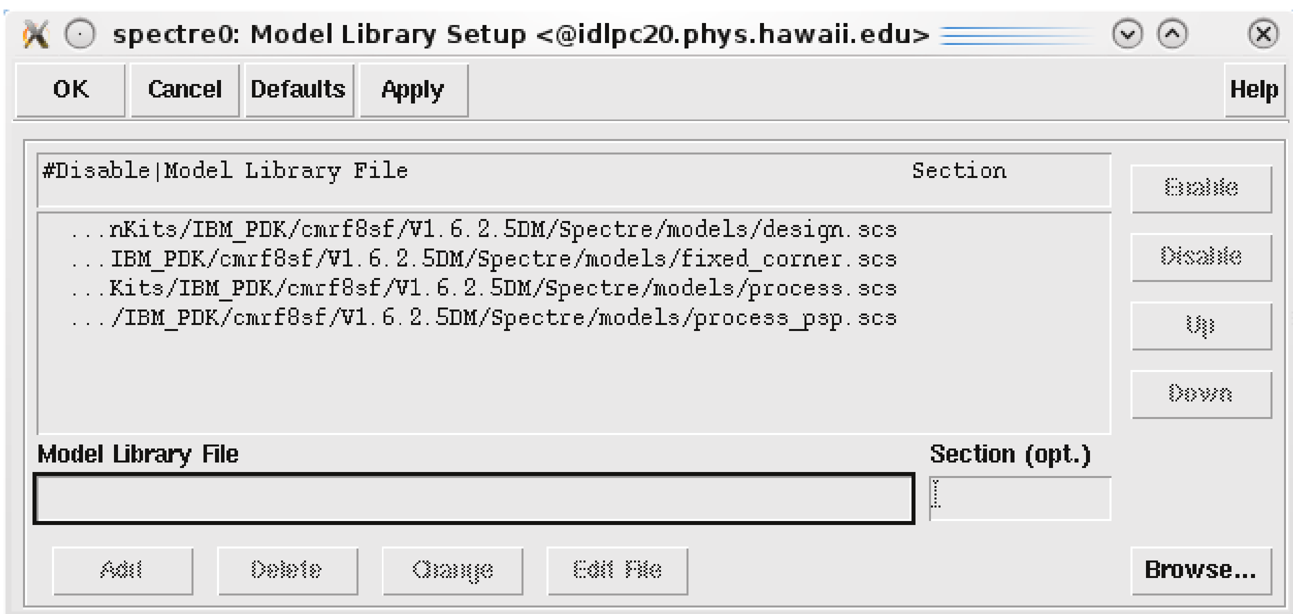
If you wish to start the ADE from the CIW:
Tools → Analog Environment → Simulation

Setup the design libraries:

1) **Setup → Model Libraries...**

2) Click **Browse...**

3) Select each model (order is **important**) and click **Add** after each one:
models/design.scs
models/fixed_corner.scs
models/process.scs
models/process_psp.scs



Some tips for using the visual simulation tools:

To plot outputs:
Outputs → To Be Plotted → Select On Schematic
Then click on which ever bus (wire) to monitor the voltage of. Clicking on a device or terminal of a device will monitor the current through the selected (ovals will appear at the device terminals indicating the current being monitored.

To select the simulation type:

Analyses → Choose

The *Stop Time* allows time selection. 20u = 20 microseconds. 50p = 50 picoseconds. You must use units otherwise seconds are the default.

Click on **Netlist And Run** (The green stop light)

The big, bad readme (PDF) about ADE is located at:
http://www.d.umn.edu/~htang/Cadence_doc/anasimhelp.pdf

How to add simulation primatives:

You can add sources (current sources, voltage sources, etc) to a schematic for simulation. These simulation only resources (*vpulse* for instance does not show up on the layout tool, only within the schematic editor and the ADE) are found under the **analogLib** library. The **analogLib** components are added as any other schematic item are within the schematic editor. The *vpulse* component is a useful cell for performing voltage sweeps.

When adding analogLib devices, you typically must include **Rise** and **Fall** time (1p is fine).

To perform **DC** sweeps you can either add a *vpulse* component or select the **DC** analysis sweep. When you sweep with DC using the **DC analysis**, it is useful to connect a voltage component to the input terminal and sweep a parameter from the component. For example, performing a sweep of the input to an INV, I would place a *vpulse* component connected to the IN port. I would select the **DC analysis** type. My sweep variable would be **Component Parameter**, with the **Component Name** the *vpulse* component (/V0 perhaps) and the **Paremeter Name** would be *dc* (a menu box will pop up asking you what you want to sweep.

Simulations from the command line / editing the netlist yourself:

To access the command line interpreter of Spectre type spectre at the command line. If you want to run spectre from the command line on a schematic, it is first very handy to export the netlist of the schematic. This is best done by running ADE and click the Simulation –> Netlist → Create menu items. The netlist is then created in the directory listed in the result window. Typically this directory will be similar to:

/home/$USER/simulation/$CELL_NAME/spectre/schematic/netlist/input.scs

This input.scs file can be edited in any text editor. Note, if you change the input.scs file, that change WILL NOT be reflected on the schematic, you must manually make the changes. If you want to run simulations on your netlist type:

spectre $NAME_OF_FILE

The results viewer can be opened from the CIW window (preferred if you already started icfb):

Tools → Analog Environment → Results Browser

or from the command line with the following command:

wavescan &

Annoying Issues:

When you exit ADE it will ask if you wish to save the state. Often times the state does not get saved. This means when you start a new simulation, you must manually input the simulation type, the models, the outputs to be plotted, etc...

The spectre format for the $NAME.scs file is very SPICE like, but not exactly so. Often times simple commands will work and some more complex ones, but it can be touchy. Also, performing various analysis types is best to read the documentation and see how Spectre wants it done. These problems become more apparent when you try to simulate multiple components strung together.

The results browser (wavescan) is a Java application and kind of slow. The output files aren't aways easy to open in a SPICE results browser.

If you accidentally miss putting a unit on a simulation time (for instance, set a step of 1us but a simulation length of 100s), Spectre will happily chug along all day long.

By default, all Spectre netlists are called input.scs, they are stored in different directories. Be careful copying and moving these files around, it is really easy to overwrite and confuse yourself.