Getting Started with HSPICE

Simulating a CMOS inverter

We will learn how to simulate a CMOS inverter using HSPICE, a circuit simulator widely used in industry and academia. Synopsys' HSPICE is installed on USF research computing cluster.

Step 1: Connecting to USF Research Computing Cluster

There are several ways to connect to the cluster. Use one of the following ways.

(a) Using ssh (secure shell) on a USF Linux machine

1. Open a terminal, you can run ssh with the following command on the command line. Substitute your netid in the place of "`<your_netid>`" in the command. Type the command and hit Enter.

   ```
   prompt% ssh -Y -l <your_netid> sc.rc.usf.edu
   ```

2. If you see a message similar to the following, type `yes`. Usually you will see this message when are you connecting to the cluster for the first time from a computer.

   ```
   The authenticity of host 'sc.rc.usf.edu (131.247.250.113)' can't be established.
   RSA key fingerprint is SHA256:DqJU3QpfKoFyUMLiBzi/tZwzxnwtzcXUSn/hqwzd1OU. Are
   you sure you want to continue connecting (yes/no)?
   ```

3. Enter your netid password. You will now be logged into the cluster.

(b) Using Mobaxterm on a USF Windows machine

1. Invoke Mobaxterm from **Start** -> **Mobaxterm**. If Mobaxterm is not installed, download the **portable edition** from `http://mobaxterm.mobatek.net/download-home-edition.html`.

2. To start the ssh session, click on **SSH** in the top menu. Then set **Remote host: sc.rc.usf.edu**.

3. Select **Specify username** and use your netid to login.

(c) Using your own laptop

- **Windows**

  Download the **portable edition** from `http://mobaxterm.mobatek.net/download-home-edition.html` and then follow the steps in procedure (b) above.

- **Linux (Ubuntu, CentOS, etc.)**

  Follow instructions under (a) above.

- **MacOS**

  Install xQuartz from `https://www.xquartz.org/`

  After installing xQuartz, invoke it from **Applications** folder. Then Right click on the icon and then click on **Terminal**. This will open up a terminal window, follow instructions under (a) above.

Step 2: Modify .bashrc startup file

This step is done only once and after that every time you connect to the cluster, HSPICE is included in your path.

1. Open `.bashrc` file in an editor. You can use any text editor (eg., emacs, pico). We will use `vi` editor.

   ```
   prompt% vi ~/.bashrc
   ```

2. Add the following two lines (first one is for HSPICE and second for sx) in the `.bashrc` file.

   Go to the end of the file using down arrow. Press 'o' and then type these lines.

   ```
   module add apps/synopsys/hspice/F-2011.09-SP2
   module add apps/synopsys/sx/C-2009.03-SP1
   ```

   Make sure you hit Enter at the end of the second line.

3. Press Esc then press Shift + ':', then type `wq` to save and exit.

4. Source the `.bashrc` file.

   ```
   prompt% source ~/.bashrc
   ```

5. Check if `hspice` is in the path with `which` command. The computer will echo the full path of `hspice`.

   ```
   prompt% which hspice
   /apps/synopsys/hspice/F-2011.09-SP2/hspice/amd64/hspice
   ```

Step 3: Copy inverter spice netlist and technology model file

1. Create a working directory.

   ```
   prompt% mkdir inv
   ```

2. Change to the directory.

   ```
   prompt% cd inv
   ```

3. Copy files to current directory. Note the "." at the end of each command.

   ```
   prompt% cp /shares/cda4213_001/hspice_tutorial/inv.sp .
   prompt% cp /shares/cda4213_001/hspice_tutorial/mosistsmc180.sp.txt .
   ```

4. Check if the files are copied by listing the folder contents. You should see the two files `inv.sp` and `mosistsmc180.sp.txt` listed.

   ```
   prompt% ls
   inv.sp mosistsmc180.sp.txt
   ```

Step 5: Simulate the inverter netlist with HSPICE from command line

Invoke HSPICE with `inv.sp` as input file and `inv.lis` as the output file. If successful, an info message will be displayed as shown below.

```
prompt% hspice -i inv.sp -o inv.lis
>info:  ***** hspice job concluded
```

Step 6: View the simulator output

The voltage waveform data is written to `inv.tr0` file. Invoke sx with this file as input as follows.

```
prompt% sx inv.tr0
```

A sx window will open up. In the middle subwindow on left expand `D0:inv.tr0` and click on `toplevel`. Click on `v(in)` and `v(out)` to add them to the waveform window. Verify that the inverter is working correctly.

Step 7: Saving the waveforms as an image

In the sx window, choose **WaveView** then **Dump Screen**, enter the filename `inv.png` and click **Save**. The image file can viewed with `display` command.

```
prompt% display inv.png
```