Importing ngspice data to MATLAB with the “wrdata”

Use the wrdata command in your spice file

You can save you ngspice simulation data in a tab delimited file with `wrdata`. Use the following command in the end of your simulation:

```
wrdata <filename> v (<nodename>)
```

- Notice that the `<filename>` argument does not require the file extention. Your file will be automatically saved as `<filename>.data` file with the function values `v (<nodename>)` in separate columns.

- In your output file, the functions will be ordered as you have specified in the command. Each function is preceded by a column with the simulation variable. Usually all the odd numbered columns will have the variable, and all the even columns will have the function values. For example, if you write the following command after your transient simulation:

  ```
  wrdata tran v(nvout) v(nvz) v(nvx) v(nvsig) v(nvy) ;
  ```

  —you will find a file named “tran.data” with the simulation results in your working directory. It will have timing information (time is the variable in transient simulations) in all the odd numbered columns. The functions `v(nvout) v(nvz) v(nvx) v(nvsig) v(nvy)` will be in the 2nd, 4th, 6th, 8th, and 10th column respectively. You can import the 1st column for the timing information and ignore the other odd numbered columns, because they will have the same values as column 1.

- If you have multiple simulations inside a control block, you can save separate files with `wrdata` after the end of each simulation. Here is a sample control block for lab 1:

  ```
  .control
  ac dec 10 100 1e6
  plot xlog db(v(nvx))
  plot xlog db(v(nvy))
  plot xlog db(v(nvz))
  plot xlog db(v(nvout))
  hardcopy dbx.ps xlog db(v(nvx))
  hardcopy dby.ps xlog db(v(nvy))
  hardcopy dbz.ps xlog db(v(nvz))
  hardcopy dbout.ps xlog db(v(nvout))
  wrdata ac db(nvx) db(nvy) db(nvz) db(nvout)

  tran 10ns 200us
  plot xlimit 0 200u v(nvout) v(nvsig) v(nvx) v(nvy)
  plot xlimit 0 200u v(nvx)
  ```

```
hardcopy tran.ps xlimit 0 200u v(nvout) v(nvsig)
wrdata tran v(nvout) v(nvz) v(nvx) v(nvsig) v(nvy)

tran 10ns 2ms
fft v(nvout)
plot xlimit 10e3 500e3 db(v(nvout))
hardcopy fft.ps xlimit 10e3 500e3 db(v(nvout))
wrdata fft v(nvout)
.endc
```

- You will see 3 files: ac.data, tran.data and fft.data in your working directory after you have run

  ```
  ngspice -b <spice-filename.sp>
  ```

  You can import data from these files to MATLAB.

Importing in MATLAB

Here is an example to produce the `fft` plot from the fft.data file:

- Import the data:

  ```
  fft = importdata('fft.data')
  ```

  —This will create an array called `fft` with variable `frequency` in the 1st column. The function `v(nvout)` has real and imaginary parts. The real part is in the 2nd column and the imaginary part is in the 3rd column.

- Notice that the imported array does not have any "labels". You will need to check with your `wrdata` command to keep track of the functions.

- Extract the vectors from the array `fft` with the following commands:

  ```
  frequency = fft(:,1);
  real = fft(:,2);
  img = fft(:,3);
  ```

- Now plot with

  ```
   semilogx(frequency,20*log10(real+i*img));
  ```

Your plot should look like this:

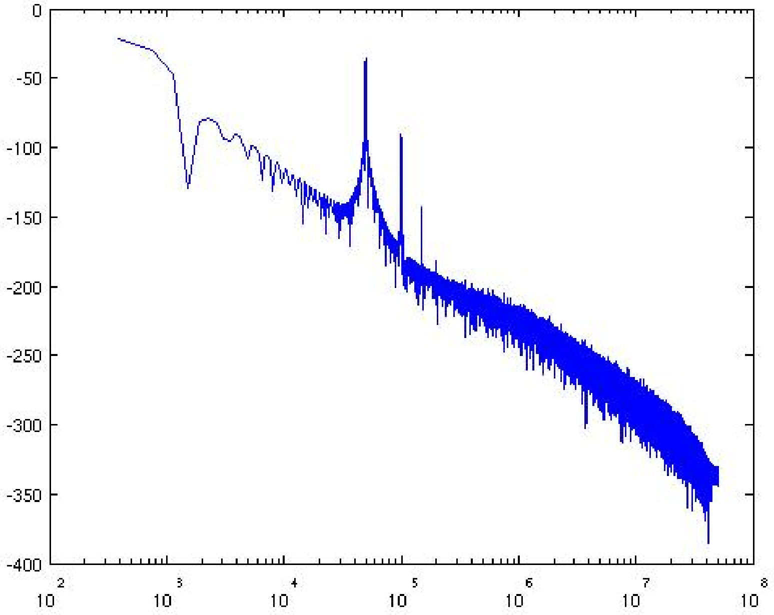

Figure 1: fft plot