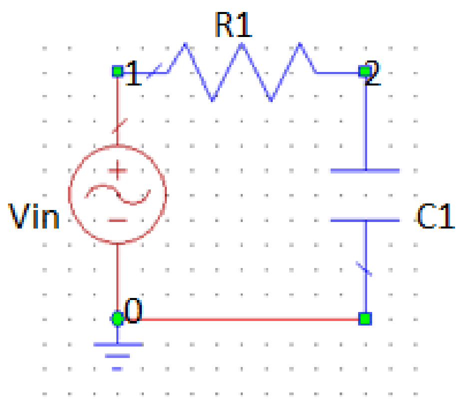
Discussion 11/29



Consider the following simple RC circuit. We know the voltage frequency response at node 2 should look like the following:

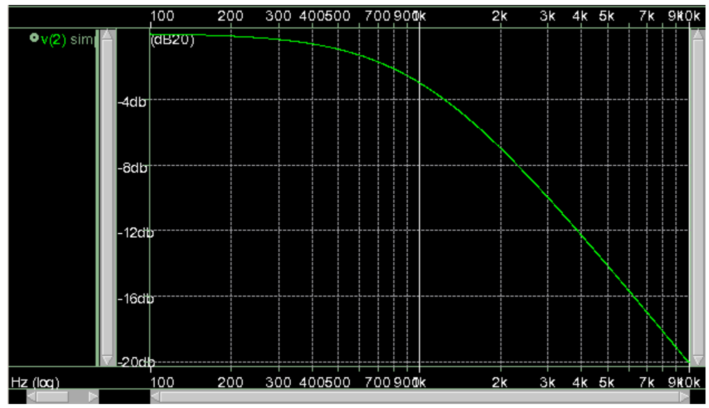
$$V_2(j\omega) = \frac{1}{1 + j\omega R_1 C_1}; \quad |V_2| = \frac{1}{\sqrt{1 + (\omega R_1 C_1)^2}}; \quad \tau_c = R_1 C_1$$

How do we simulate this in SPICE?

From the previous discussion, we first need to write a netlist.

The netlist for this example is as follows:

```
SIMPLE.SP
Vin   1     0     1     AC    1
R1    1     2     1000
C1    2     0     0.159uF
.op
.ac   dec   1000  100Hz 10KHz
.options post=2
.end
```



The figure shows the expected -3 dB point at 1 kHz.

The structure of this netlist can be broken down into two (maybe three) parts:

Part 0. The title – you MUST include a title! It helps for the title to be the actual title of the file.

Part 1. Components list – each component from the schematic can be represented in SPICE as follows.

- The first column is the name of the component. The first letter of the name tells SPICE what kind of component it is. The remainder of the name can be whatever you like.
- The second and third columns indicate the nodes of the circuit where the component is connected. They could be in either order, but it is best not to confuse yourself and maintain some order.
- The fourth column is the value of the component. Unless otherwise specified, SPICE will assume them to be in default units (ohms, farads). Note `R1` could be specified as either `1000` or `1K`. `C1` is specified using `u` to indicate microfarads. You can use `k, M, m, u, n, p`, and others.
- The last two columns apply only to the voltage source `V1`. This indicates `V1` is an AC voltage source (as opposed to a DC voltage source) and the last 1 may or may not actually be needed.

Part 2. Analysis – the remaining four lines tell SPICE what analyses to execute and how to save the output.

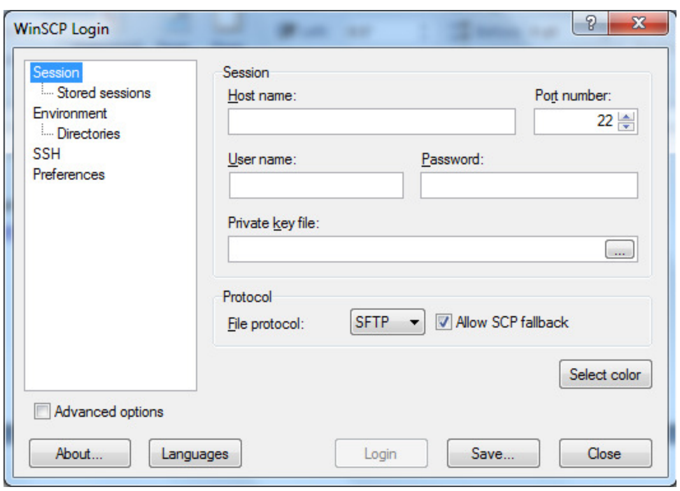
- `.op` is a DC operating point analysis. This is generally not necessary since other analyses tend to automatically include `.op`, but it doesn't hurt to include it.
- `.ac` tells SPICE to do the AC frequency sweep. This will give our frequency response (the figure).
- The `.options post=2` line tells SPICE to save output as a *.ac0 file (in our case, simple.aco).
- Finally, the `.end` line indicates the end of the SPICE file.

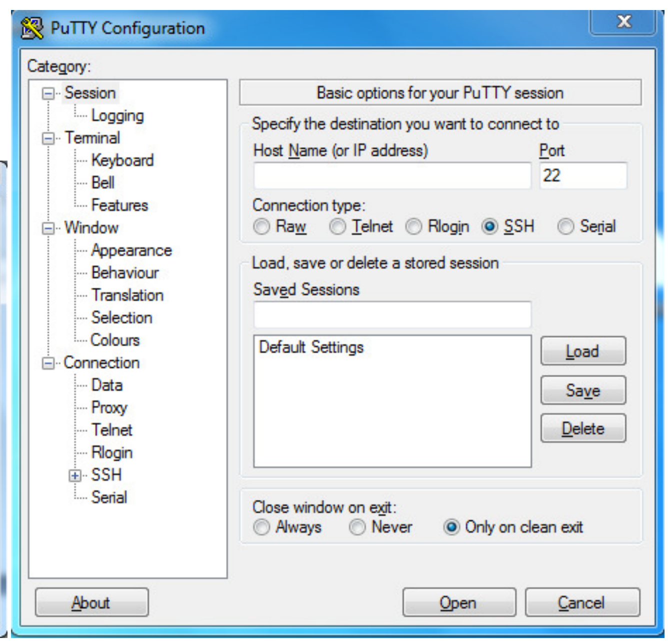
Reference SPICE manuals:
http://bwrc.eecs.berkeley.edu/Classes/IcBook/SPICE/UserGuide/overview_fr.html

Discussion 11/29

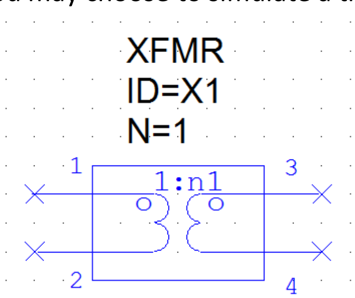
This is all well and good, but how do we actually run SPICE? Windows users will be spared the headache here.

1. You probably have written the SPICE netlist in Notepad or something on your own machine. To get it on a UNIX machine that can actually run HSPICE, you'll need to transfer it remotely. Grab something like WinSCP: http://winscp.net/eng/download.php.
2. After installation and execution, you'll see the login screen (left figure below). Use your instructional UNIX user/pass, and use "c199.eecs.berkeley.edu" as the Host Name. After this it works like a normal SFTP client – simply navigate to your file, navigate to your UNIX directory (perhaps "./ee245"), and drag and drop.
3. Install something like Xming: http://sourceforge.net/projects/xming/
4. And Putty: http://www.chiark.greenend.org.uk/~sgtatham/putty/
5. Run Xming first; it will show up running in your background or system tray.
6. Run Putty. You'll see something like the config screen below right. Type in the same Host Name, and then go to SSH->X11 and check the "Enable X11 forwarding" box.
7. Click Open and login using your instructional UNIX user/pass. Upon successful login, type "xterm". This will bring up a new command prompt. Navigate to your folder (perhaps "cd ee245") and run your SPICE file ("hspice simple.sp"). You may have to debug some, but when you see no error messages you can be assured it has simulated correctly.
8. List contents of your directory ("ls –a"). You should see a *.ac0 file ("simple.ac0"). Type "sx simple.ac0" to open the waveform viewer, where you can manipulate the graph, change axes scales, and add marker points, to obtain the figure on Page 1.





You may choose to simulate a transformer in SPICE as well.



The netlist for a transformer is:

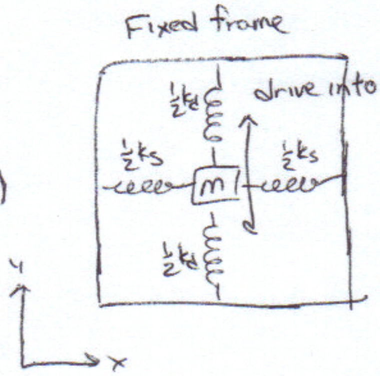
```
Lin   1     2     1
Lout  3     4     N
K     Lin   Lout  1
```

Transformers are just represented by inductors and a coupling factor K. For ideal transformers its value should equal 1. The ratio of inductances 1:N is the turns ratio.

Discussion 11/29

Gyroscopes

(a horrendously brief overview)

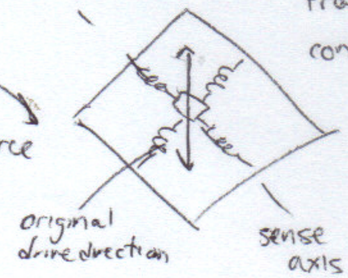


rotation

generates Coriolis Force

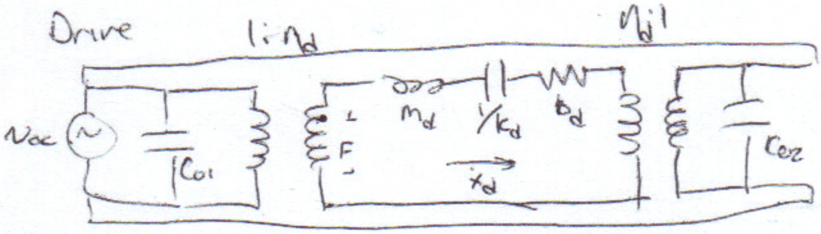
$F_c = m_s(2\dot{x}_d \times \Omega)$

sense mass, drive velocity, rotation



Frame rotated
conservation of momentum
=> motion splits into drive & sense directions

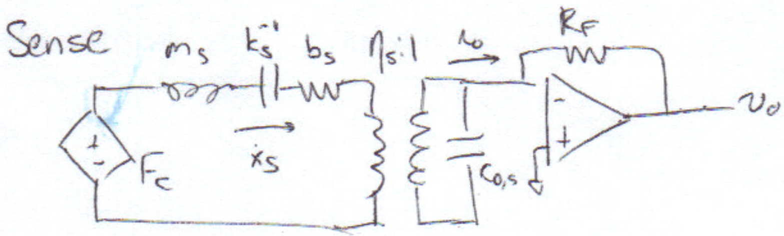
Equivalent circuits



C_{o1}/C_{o2} are comb drive nominal capacitances

$\eta = \left|V_p \frac{\partial C}{\partial x}\right|$ is turns/coupling (electromechanical coupling)

We drive the mechanical LCR @ resonance with sinusoidal input. (Much like comb drive resonator)



$C_{o,s}$ is parallel plate nominal output capacitance

We convert the mechanical Coriolis Force to an output current.

What is generally important is the device transfer function. For gyros, the input is Ω & output is i_o. So the gyro converts rotation (Ω) to a current (i_o) that we can work with in a circuit (such as the inverting amp) to convert to a voltage.

Transfer Function $y = Hx \Rightarrow H = \frac{y}{x}$

$\frac{i_o}{\Omega}$ = (break into known pieces)

coupling parameter between "currents"

ohm's law (for mech ckts)

coriolis force

$$\frac{i_o}{\Omega} = \left(\frac{i_o}{\dot{x}_s}\right)\left(\frac{\dot{x}_s}{F_c}\right)\left(\frac{F_c}{\Omega}\right) = \eta_s \cdot \frac{1}{Z_{mech}} \cdot 2\dot{x}_d m_s$$

$$\frac{\dot{x}_d}{v_{ac}} = \frac{\dot{x}_d}{F}\frac{F}{v_{ac}} = \frac{1}{Z_{m,d}} \cdot \eta_d$$

$$\boxed{\Rightarrow \frac{i_o}{\Omega} = \frac{2\eta_d \eta_s m_s}{Z_{m,s} \cdot Z_{m,d}}}$$

Ref. C. Nguyen Module 15