

ECE 3410 – Microelectronics I

SPICE Handout III

Subcircuits

The most powerful feature of SPICE is its ability to represent design hierarchies. A hierarchy is created through the *subckt* declaration, which uses the following syntax:

```
.subckt part_name n1 n2 n3 ...  
...  
.ends part_name
```

The list of nodes n1, n2, etc are local identifiers within the subnode definition. The same node names can be reused in the global design without conflict.

An instance of a subcircuit is declared using the letter 'X':

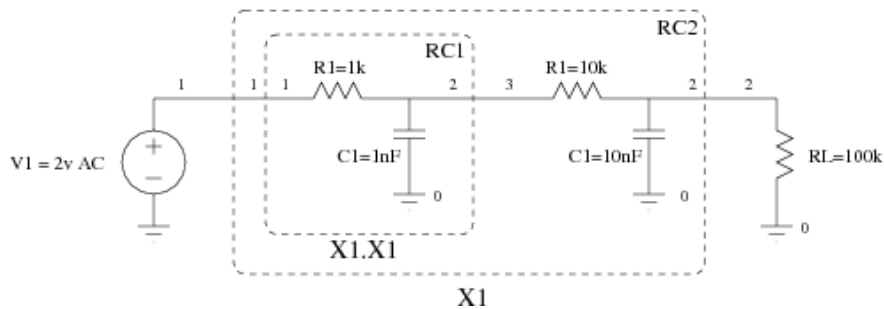
```
Xname n1 n2 n3 ... part_name
```

One subckt can be instantiated within another subckt. You can therefore describe complicated designs with relatively simple SPICE files.

Example:

```
SUBCKT example  
.probe  
  
.subckt RC1 1 2  
  R1 1 2 1kohm  
  C1 2 0 1nF  
.ends RC1  
  
.subckt RC2 1 2  
  X1 1 3 RC1  
  R1 3 2 10kohm  
  C1 2 0 10nF  
.ends RC2  
  
V1 1 0 AC 2V  
X1 1 2 RC2  
RL 2 0 100kohm  
  
.ac dec 10 1 10Meg  
.print ac v(2)  
.end
```

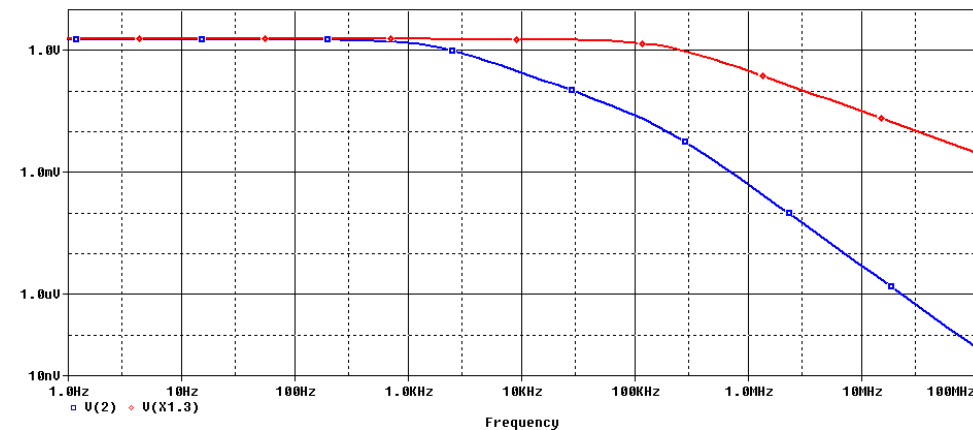
The above SPICE statements represent the circuit shown below:



Node 1 happens to have the same name inside the different subcircuits. To refer to a node within a subckt, use the notation Xname.node_name. For example, X1.X1.2 is connected to X1.3, and node X1.2 is connected to global node 2. Note also that X1.C1 is 10nF, but X1.X1.C1 is 1nF. Note that node 0 is ground throughout, including within subcircuits.

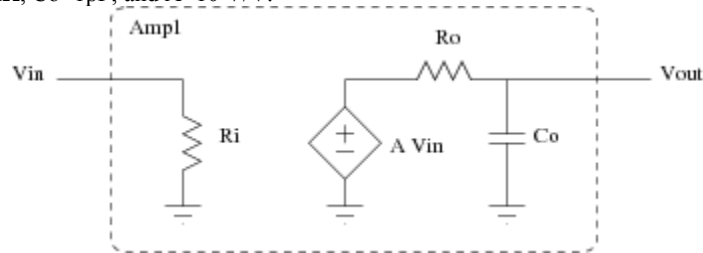
After simulating this circuit, a number of new traces appear in Probe's Add Trace menu. You can display the waveform $V(X1.3)$, which refers to the voltage at node 3 inside subcircuit X1. You can also look at $I(X1:1)$, which refers to the current entering X1 at its (local) node 1.

Displaying the waveforms $V(X1.3)$ and $V(2)$ with a log scale results in the following graph:

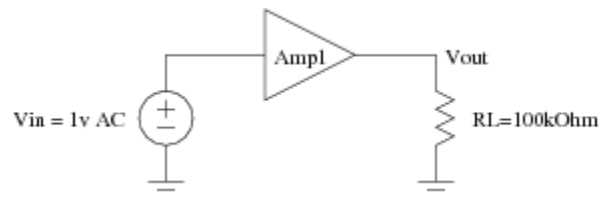


Exercises

- Design a SPICE subcircuit representing the circuit shown below. Title the subcircuit "Amp1". Use the values $R_{in}=10\Omega$, $R_o=10k\Omega$, $C_o=1pF$, and $A=10$ V/V.



- Simulate the Amp1 subcircuit in the circuit context shown below. Turn in a Bode plot of the transfer function (V_{out}/V_{in}).



- Simulate a cascade of two identical Amp1 modules, as shown below. Turn in your SPICE input file and a Bode plot showing the magnitude and frequency response of V_{out}/V_{in} .

