EE 247B / ME 218 Discussion 13

Kieran Peleaux

SPICE Intro

Simulation

Program

with

Integrated

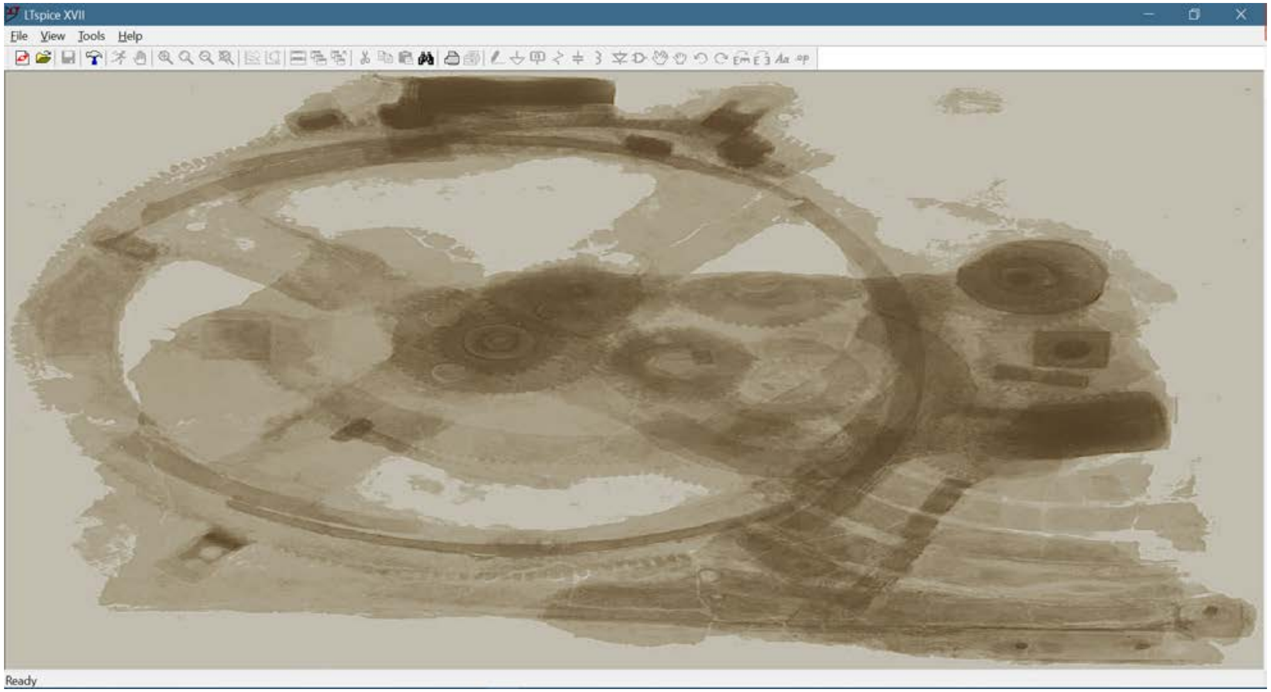
Circuit

Emphasis

- Software used for analog circuit simulation (originally intended for developing ICs)
- Developed @ UC Berkeley—version 1 released in 1973 (under public domain)
- Started out as a command-line tool
- Now multiple companies offer their own packaged versions of spice
  - ***LTspice***, *HSPICE, PSpice, TINA-TI*

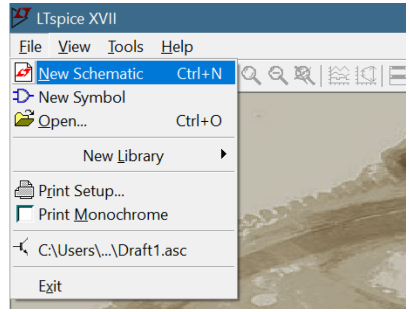
Getting Started with LTspice

- Analog Devices' (formerly Linear Technology's) free SPICE simulator
- Download the latest version from www.analog.com (LTspice XVII)
- Should see the window below upon opening Ltspice:

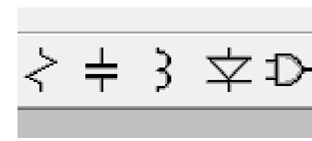


Schematic Capture

- ‘File > New Schematic’



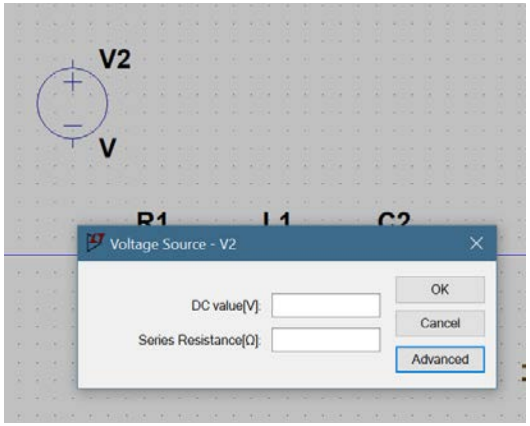
- To place components:
  - Click the desired component’s icon in the toolbar



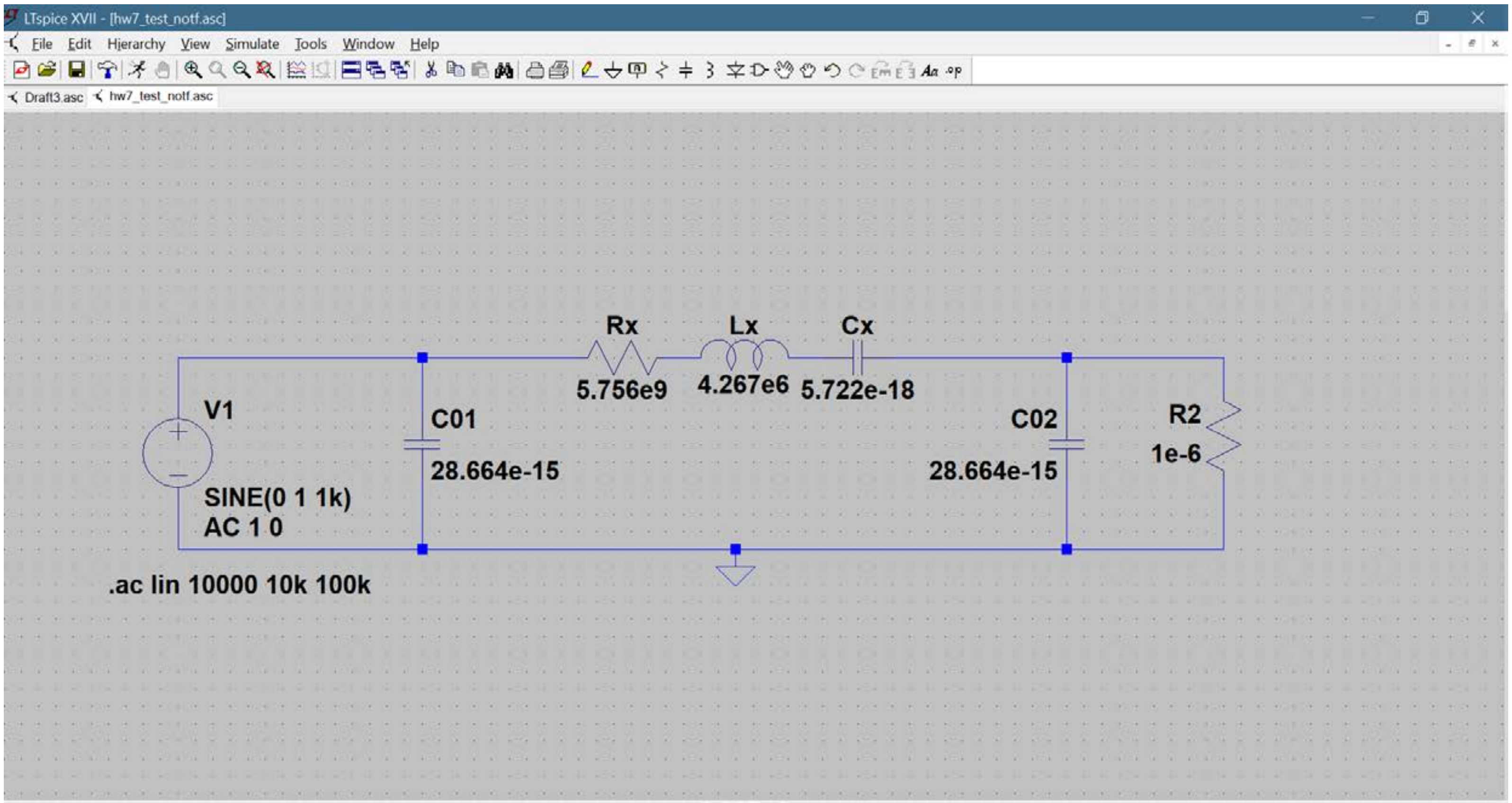
  - Or, use hotkeys (‘R’, ‘C’, ‘L’, ‘D’ or ‘F2’)
  - You can use ‘CTRL + R’ or ‘CTRL + E’ to rotate/mirror components before placing them
  - Click where you’d like your component to be placed
- Once placed, right click on component to edit values

Schematic Capture

- Place wires by clicking the pencil icon, or by pressing ‘F3’
- Don’t forget ground! (‘G’ or upside-down triangle icon)
- You can move components (wires won’t follow) or drag them after placing by using ‘F7’ or ‘F8’ respectively
- Sources & other components can be found in the symbol library, which you can access through the AND gate icon or ‘F2’
  - To change a DC source to AC or transient, right-click on the source, then click ‘Advanced’

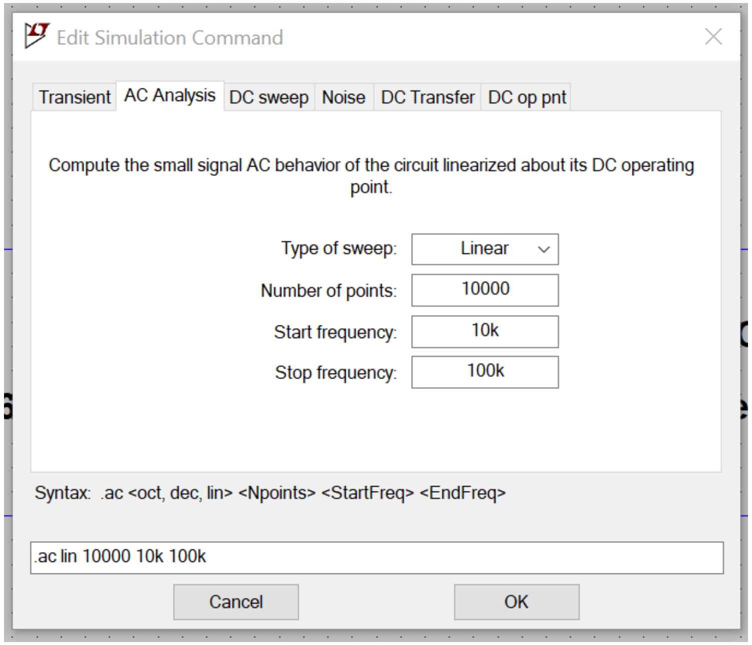


Schematic Capture



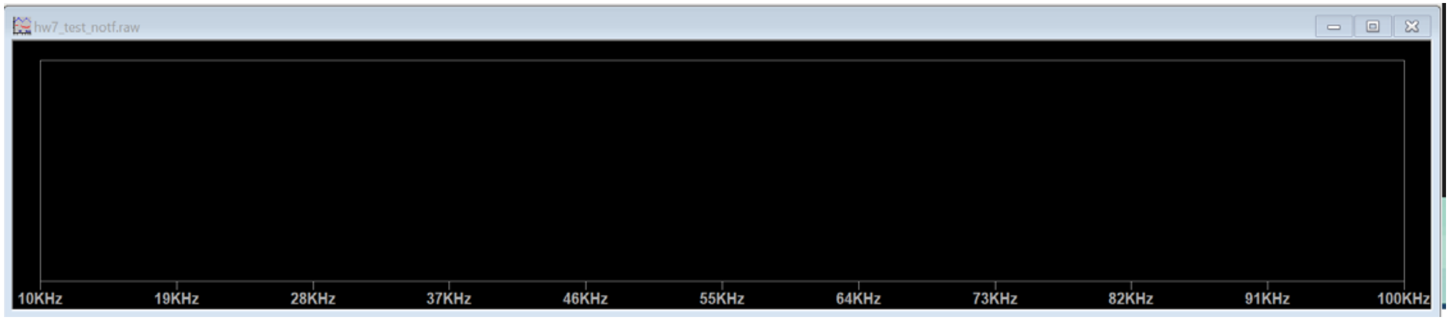
Simulation

- To simulate, go to ‘Simulate > Run’, or click the ‘Run’ icon
- For AC, select the ‘AC Analysis’ tab and choose your simulation parameters
- To change the simulation setup at any point, go to ‘Simulate > Edit Simulation Cmd’



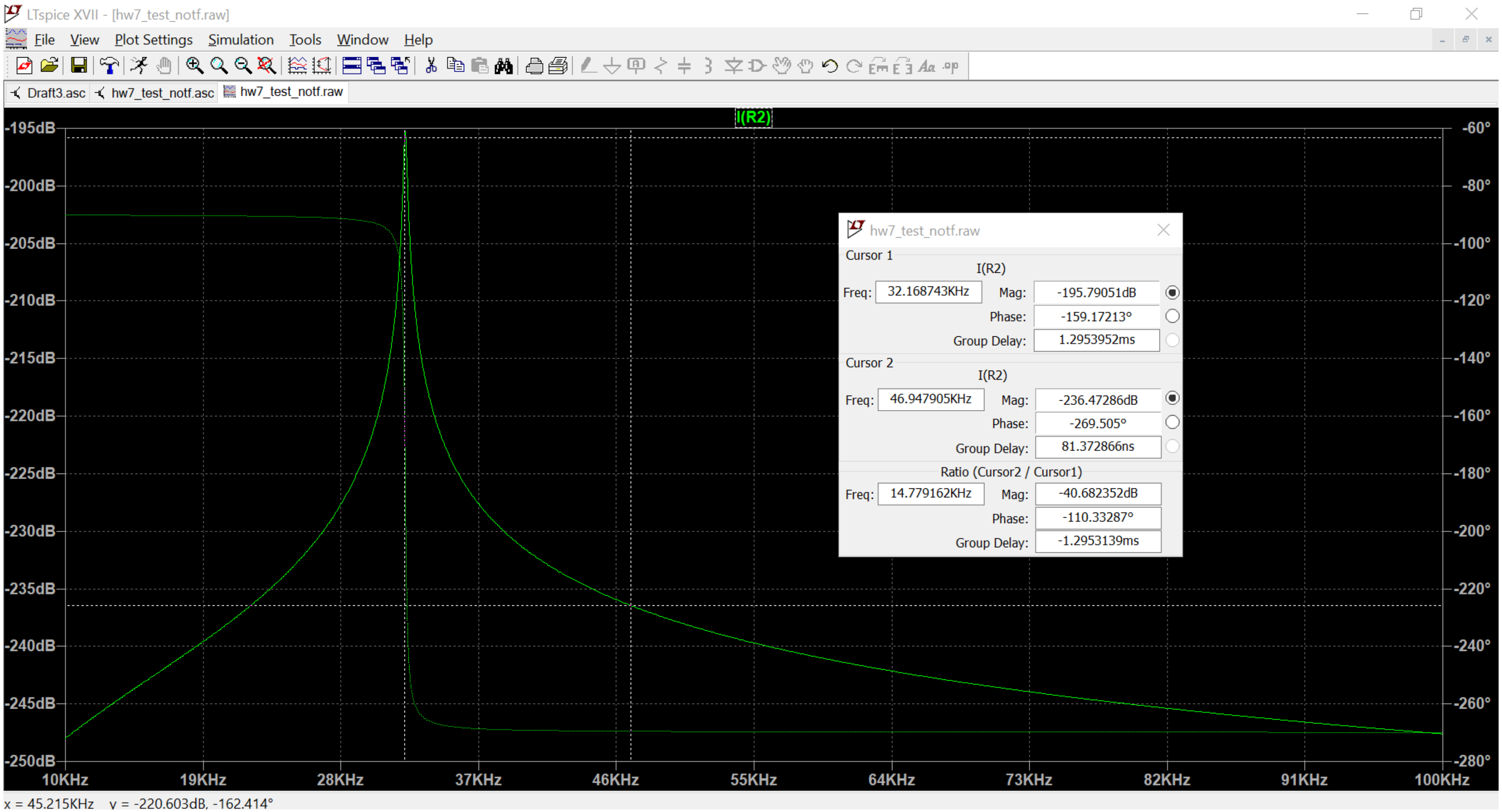
Simulation

- If there are no errors, a new window should pop up



- Right-click anywhere in this window and select ‘Add Traces’
  – Select the parameter you want to plot (current through R2 in this case)
  – Alternately, go back to the schematic window, hover over the component whose current you’d like to plot, and click when the current probe icon appears
- Click twice on the trace name to generate cursors

Simulation


LTspice XVII - [hw7_test_notf.raw]
File View Plot Settings Simulation Tools Window Help
Draft3.asc hw7_test_notf.asc hw7_test_notf.raw
I(R2)
-195dB
-200dB
-205dB
-210dB
-215dB
-220dB
-225dB
-230dB
-235dB
-240dB
-245dB
-250dB
-60°
-80°
-100°
-120°
-140°
-160°
-180°
-200°
-220°
-240°
-260°
-280°
10KHz 19KHz 28KHz 37KHz 46KHz 55KHz 64KHz 73KHz 82KHz 91KHz 100KHz
hw7_test_notf.raw
Cursor 1
I(R2)
Freq: 32.168743KHz
Mag: -195.79051dB
Phase: -159.17213°
Group Delay: 1.2953952ms
Cursor 2
I(R2)
Freq: 46.947905KHz
Mag: -236.47286dB
Phase: -269.505°
Group Delay: 81.372866ns
Ratio (Cursor2 / Cursor1)
Freq: 14.779162KHz
Mag: -40.682352dB
Phase: -110.33287°
Group Delay: -1.2953139ms
x = 45.215KHz y = -220.603dB, -162.414°

Netlists

- Every SPICE circuit is represented by a netlist
- This is essentially code that describes a circuit, which the SPICE interpreter analyzes when it performs simulations
- To see the netlist corresponding to your schematic, go to ‘View > Spice Netlist’
- Can use schematic capture for this homework, but learning netlisting is encouraged!

SPICE Netlist: C:\Users\kieran.peleaux\Documents\LTspiceXVII\hw7_test_notf.net

```
* C:\Users\kieran.peleaux\Documents\LTspiceXVII\hw7_test_notf.asc
C01 N001 0 28.664e-15
Cx N004 N003 5.722e-18
C02 0 N004 28.664e-15
Rx N002 N001 5.756e9
Lx N002 N003 4.267e6
R2 0 N004 1e-6
V1 N001 0 SINE(0 1 1k) AC 1 0
.ac lin 10000 10k 100k
.backanno
.end
```