

# EE 105 | Discussion 8

Kieran Peleaux & Qianyi Xie

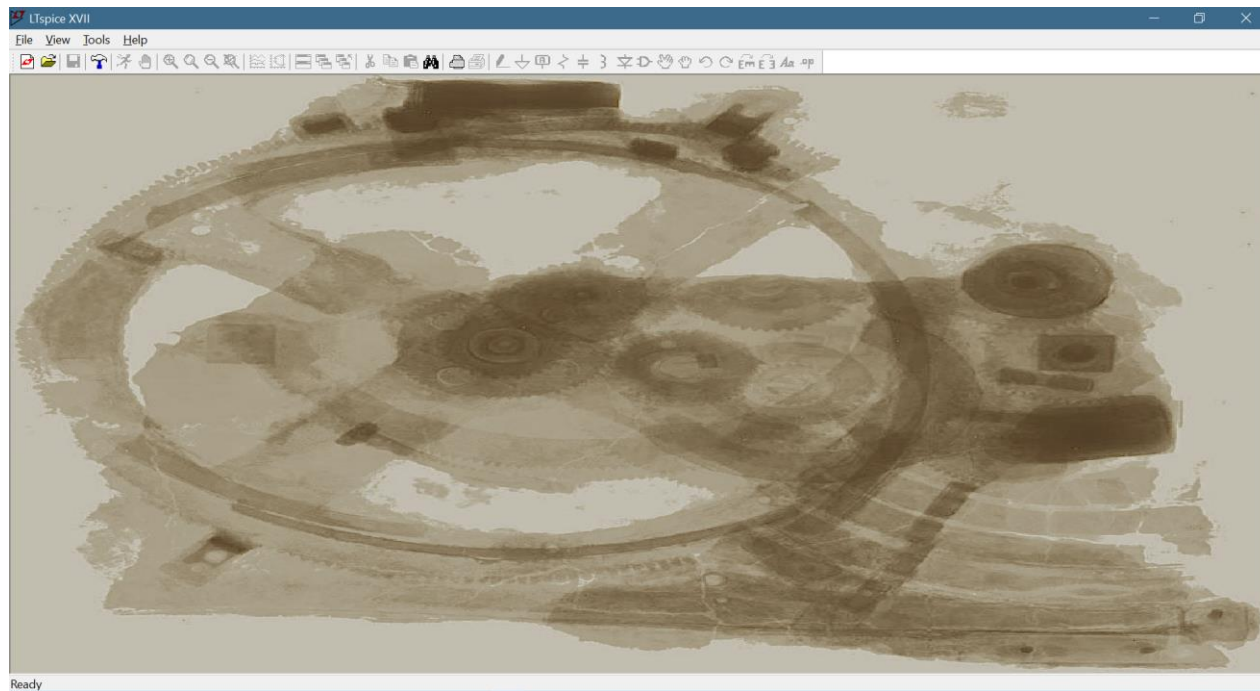
# Discussion Outline

---

- Intro to LT Spice

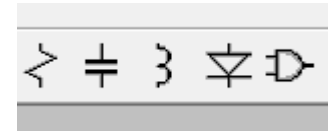
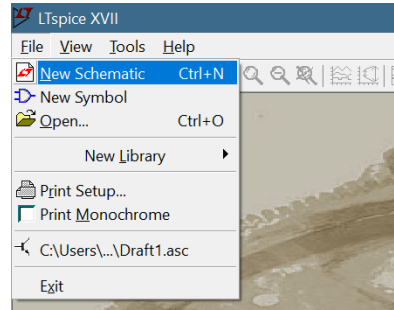
# Getting Started with LTspice

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

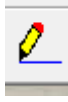
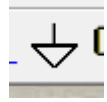



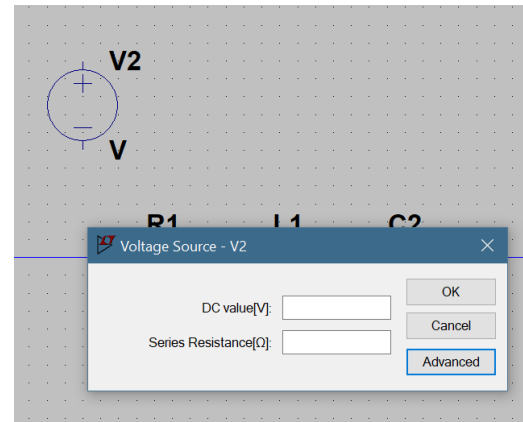
# Schematic Capture

- ‘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

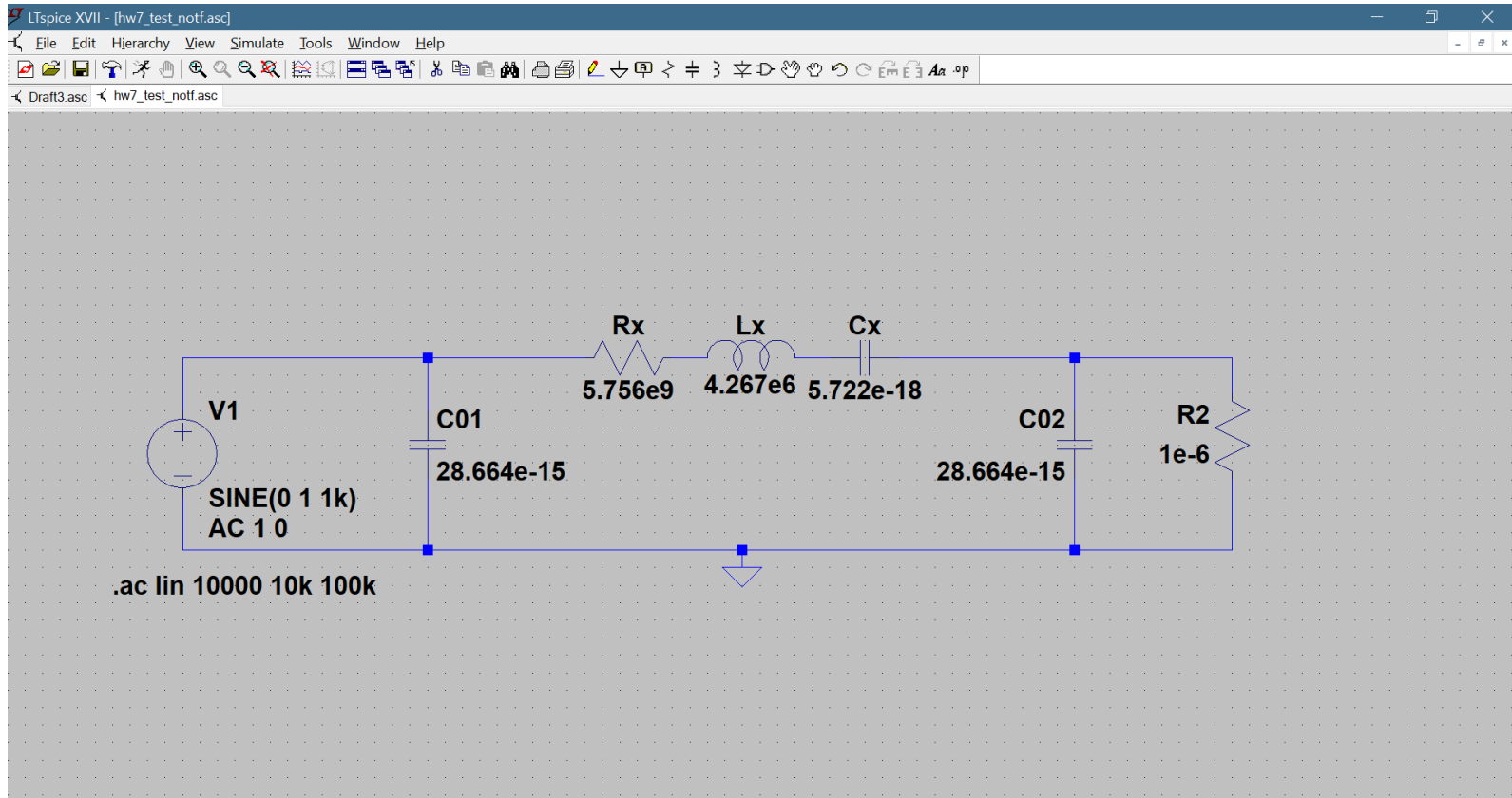


# Schematic Capture


- 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'

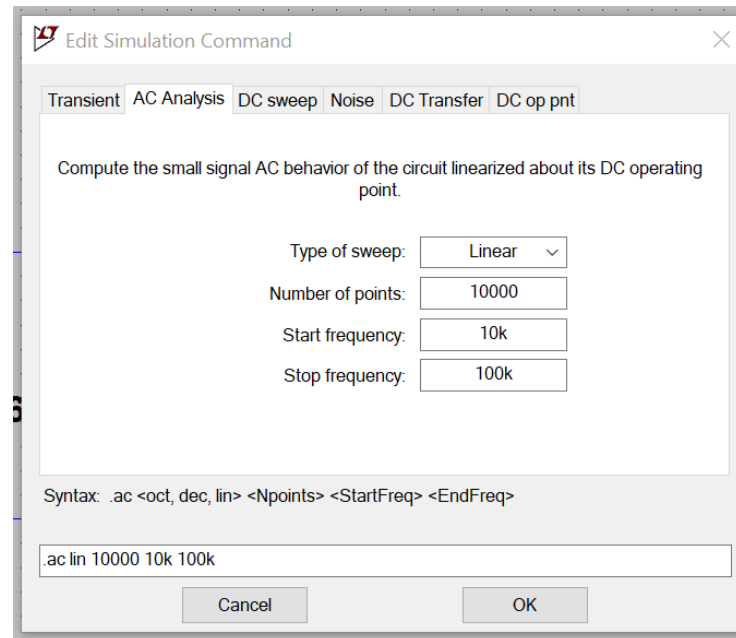


# Schematic Capture



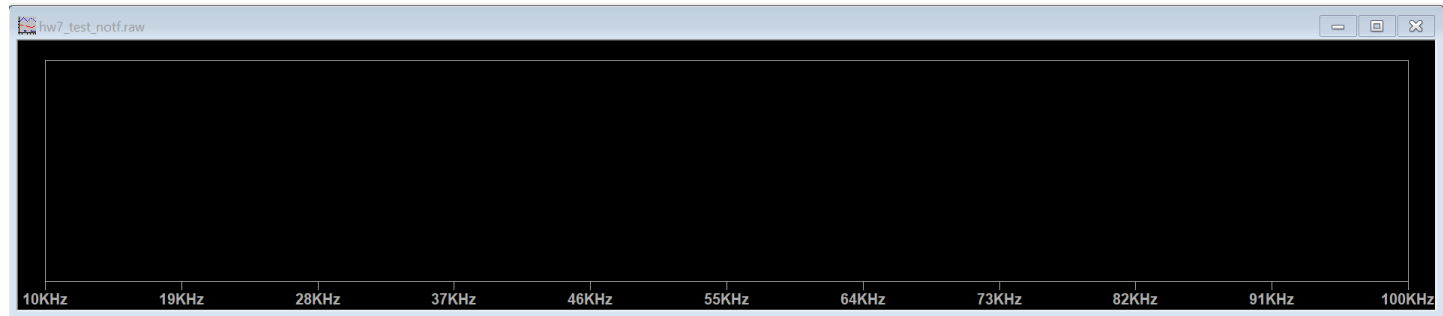
# Simulation

- 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

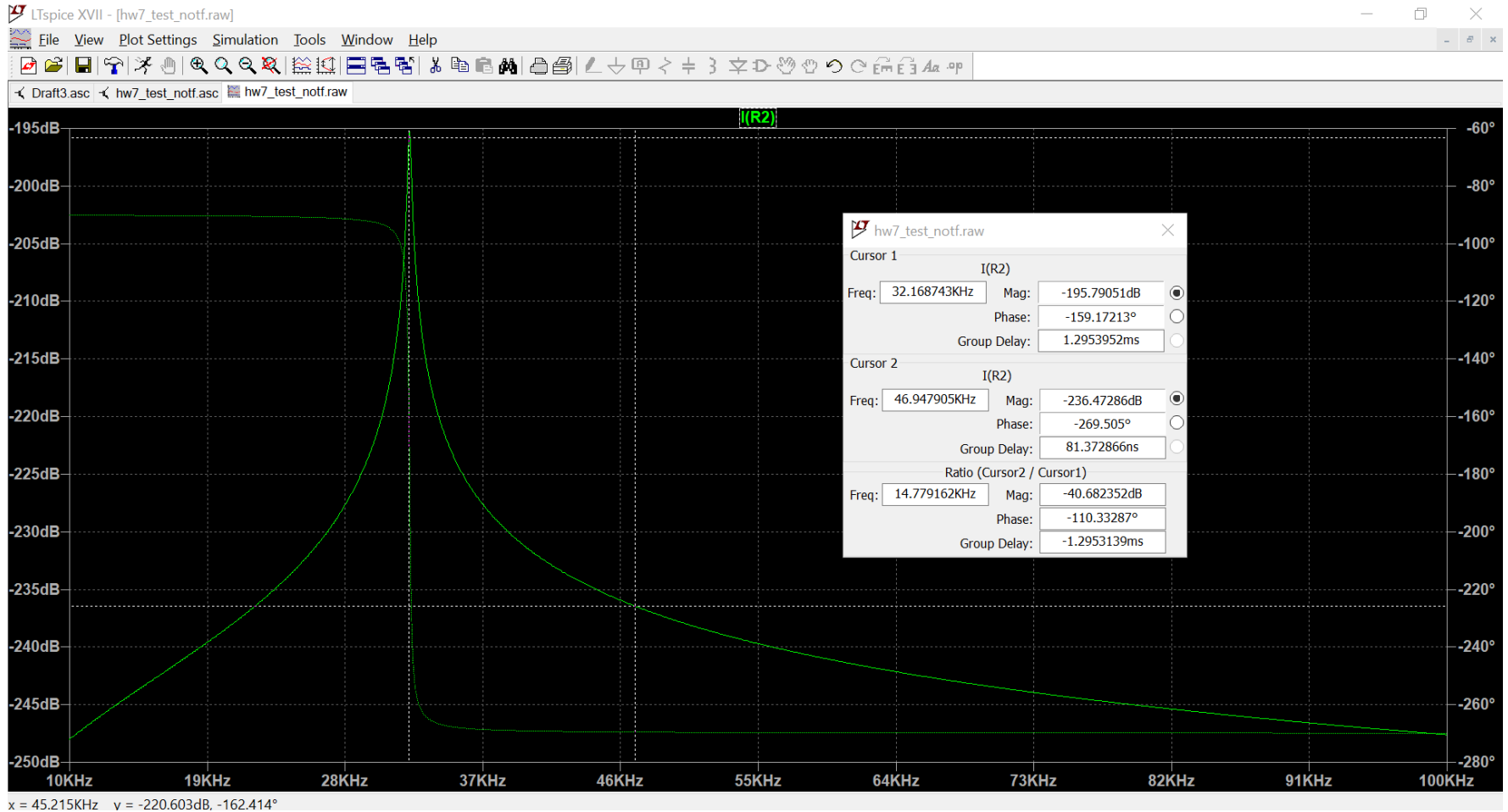
- If there are no errors, a new window should pop up



- 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

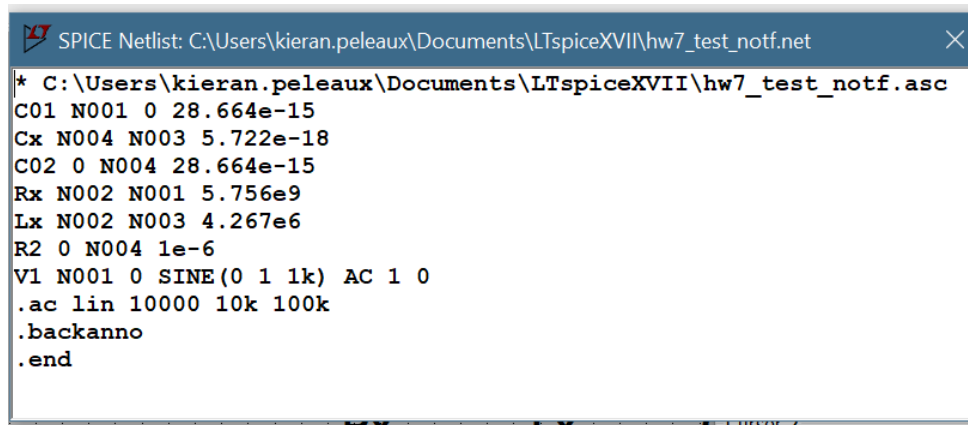


# Simulation



# 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
```