Purdue University Department of Physics

Physics 53600
Electronics Techniques for Research

Now in PowerPoint!

Spring 2020 Semester

Prof. Matthew Jones

Simulating Circuits using SPICE

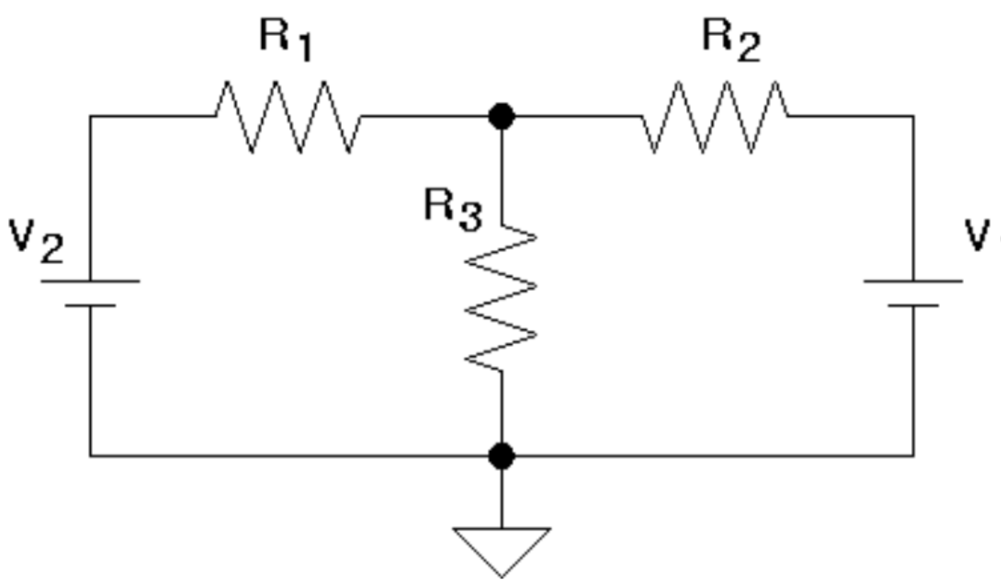
- After the invention of planar integrated circuits, analog designs became increasingly complex
- The time required to fabricate a design became much longer
- It was no longer possible to adjust component values to optimize circuit performance
- There was a need to accurately model analog electronic circuits in the design process

SPICE

- **S**imulation **P**rogram with **I**ntegrated **C**ircuit **E**mphasis
  - Initial concept developed in the 1960's
  - SPICE 1 released into public domain May 1972
  - SPICE 2 released in 1975
  - SPICE 3 released in March 1985
  - SPICE 3f.5 released in 1993
- Commercial versions
  - PSPICE (Cadence Design Systems)
  - HSPICE (Synopsys)
- Free spinoffs:
  - ADICE (Analog Devices)
  - LTspice (Liner Technologies → Analog Devices)
  - A few others

Defining Circuits

- SPICE performs numerical simulation of circuits
  - All components must have numerical values



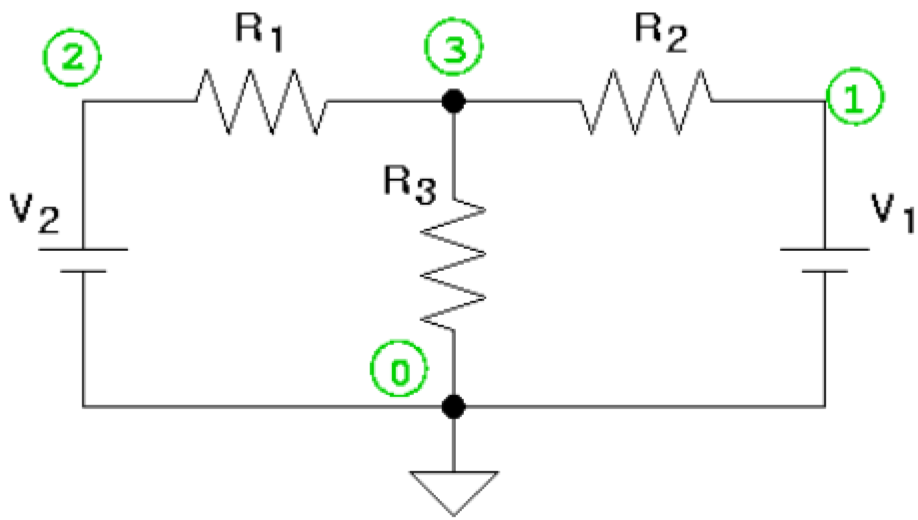
Component	Value
V_1	5 V
V_2	10 V
R_1	10 Ω
R_2	20 Ω
R_3	5 Ω

Defining Circuits

- Simulation of the circuit requires translating the schematic into a representation that can be interpreted by SPICE.
- This representation is called the *netlist*
- First, assign labels to each node in the circuit
- Then specify which components are connected to the nodes

Defining Circuits

- It is necessary to assign a ground node “0” in each disconnected part of the circuit
- Other nodes can have arbitrary labels



Defining Circuits

- The netlist consists of a list of all components and the nodes to which they are connected
  - Resistors: `Rxxxxxx N1 N2 <VALUE>`
  - Voltage sources: `Vxxxxxx N+ N- DC <VALUE>`
- This is the complete netlist:

```
V1 1 0 DC 5V
V2 2 0 DC 10V
R1 2 3 10
R2 1 3 20
R3 3 0 5
```

Assumes ohms

Defining Circuits

- Some minor additions are needed
  - A title for the circuit
  - What type of numerical analysis to perform?
  - The end

```
PHYSICS 536 EXAMPLE CIRCUIT
V1 1 0 DC 5V
V2 2 0 DC 10V
R1 2 3 10
R2 1 3 20
R3 3 0 5
.OP
.END
```

DC operating point analysis

SPICE Output

```
Operating point information:

        Node     Voltage
        ----     -------
        V(3)     3.571429e+00
        V(2)     1.000000e+01
        V(1)     5.000000e+00

        Source   Current
        ------   -------

        v1#branch        -7.14286e-02
        v2#branch        -6.42857e-01

 Resistor models (Simple linear resistor)
 model                 R

      rsh              0
   narrow              0
      tc1              0
      tc2              0
     defw          1e-05

 Resistor: Simple linear resistor
 device               r3          r2          r1
 model                 R           R           R
resistance             5          20          10
         i         0.714      0.0714       0.643
         p          2.55       0.102        4.13

 Vsource: Independent voltage source
 device               v2          v1
      dc              10           5
   acmag               0           0
       i          -0.643     -0.0714
       p            6.43       0.357

CPU time since last call: 0.006 seconds.

Total CPU time: 0.006 seconds.
```

Voltages of each node
(electric potential with respect to ground)

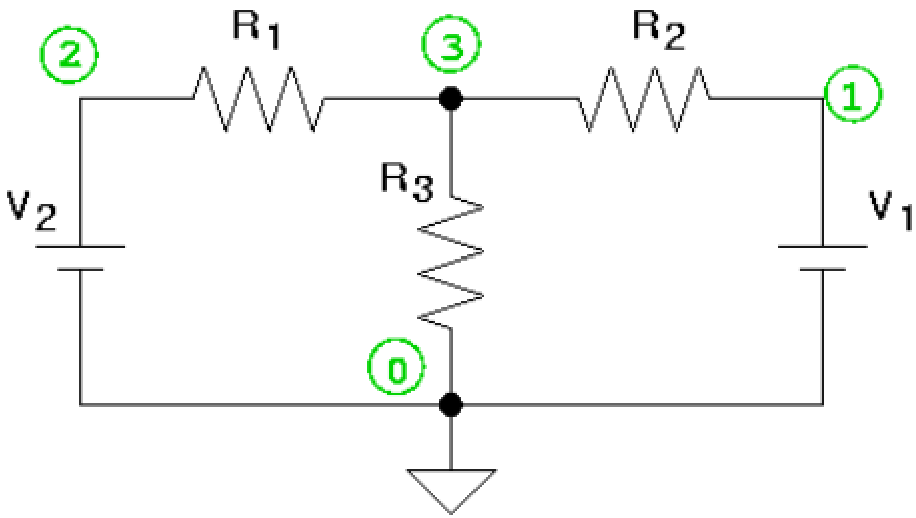
Current through each voltage source

Resistor models
(with temperature coefficients)

Current and power for each resistor

Current and power for each voltage source

SPICE Output



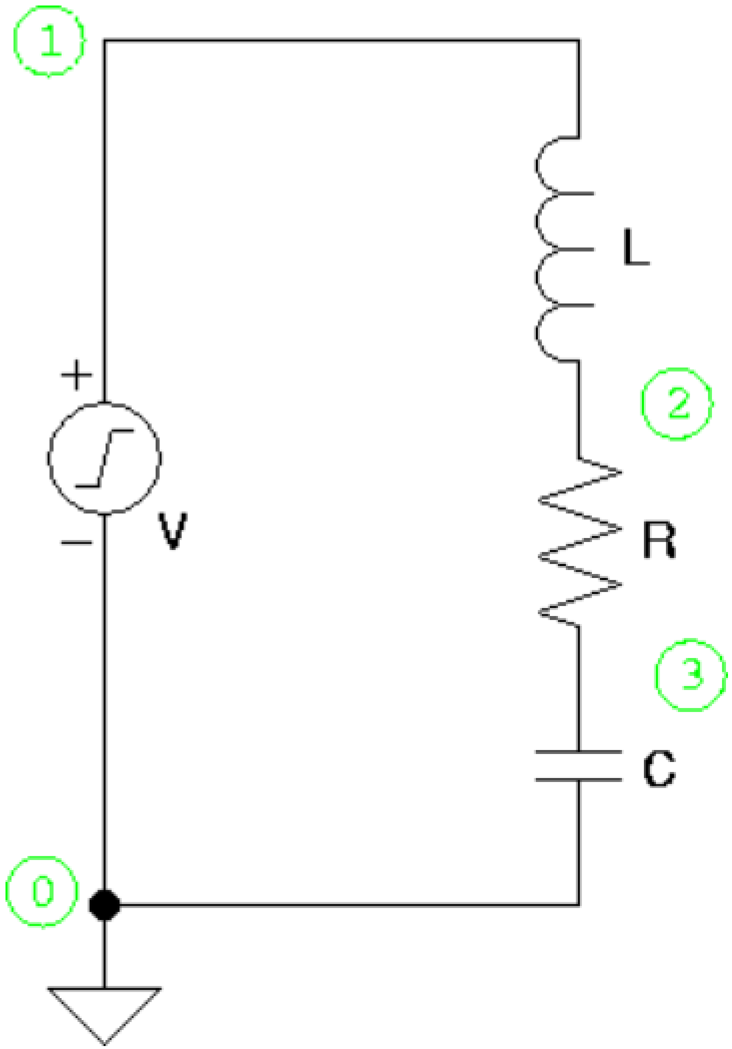
Component	Value	Current
V_1	5 V	71.4 mA
V_2	10 V	643 mA
R_1	10 Ω	643 mA
R_2	20 Ω	71.4 mA
R_3	5 Ω	714 mA

- All the current flowing from V1 must pass through R2
- All the current flowing from V2 must flow through R1
- Current flowing through R3 must be the sum of these currents
- SPICE reports the current flowing *into* the positive node of voltage sources.

```
Source    Current
------    -------

v1#branch        -7.14286e-02
v2#branch        -6.42857e-01
```

Transient Analysis



Component	Value
V_1	0 V for t<0, 10 V for t> 0
L_1	470 µH
C_1	5.4 nF
R_1	100 Ω

Netlist:

```
RLC CIRCUIT
V1 1 0 PULSE(0 10)
L1 1 2 470UH
C1 3 0 5.4NF
R1 2 3 100
.PRINT TRAN I(V1)
.TRAN 1US 100US
.END
```

Transient Analysis

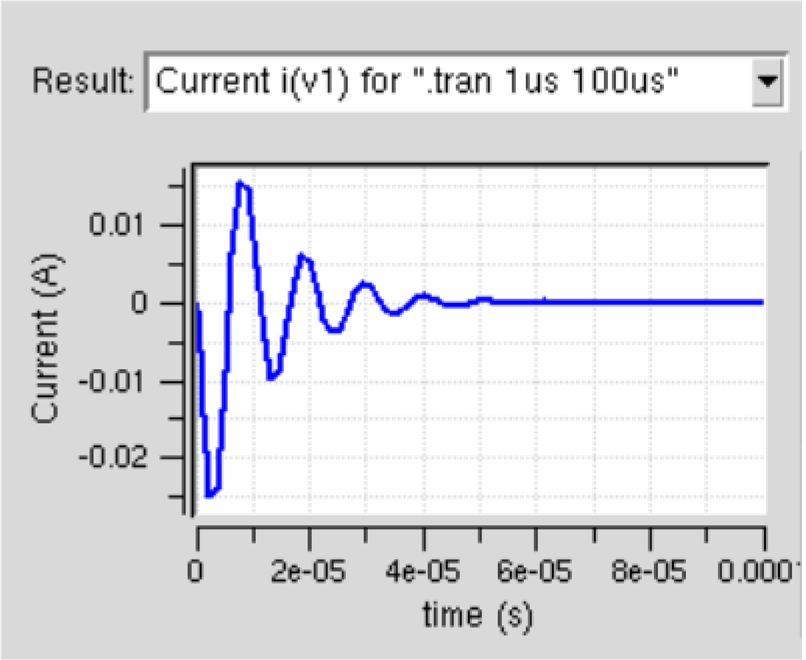
- What range of resistances will produce oscillations?

$$\frac{R^2}{4L^2} < \frac{1}{LC}$$

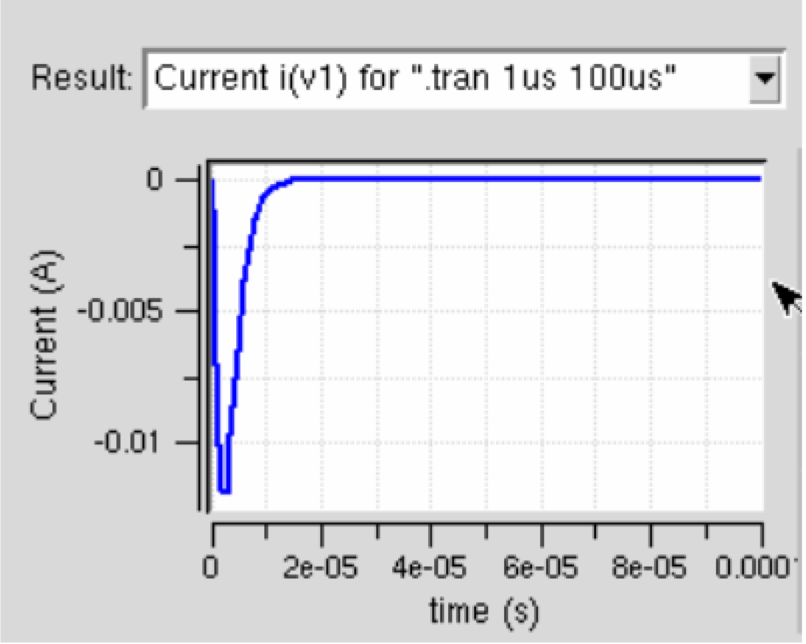
- Expect oscillations for $R < 590\ \Omega$
- Oscillation frequency is $f = \frac{\omega}{2\pi} = \sqrt{\frac{1}{LC} - \frac{R^2}{4L^2}}$
- When $R = 100\ \Omega, f = 101\ kHz$

Transient Response

R = 100 Ω



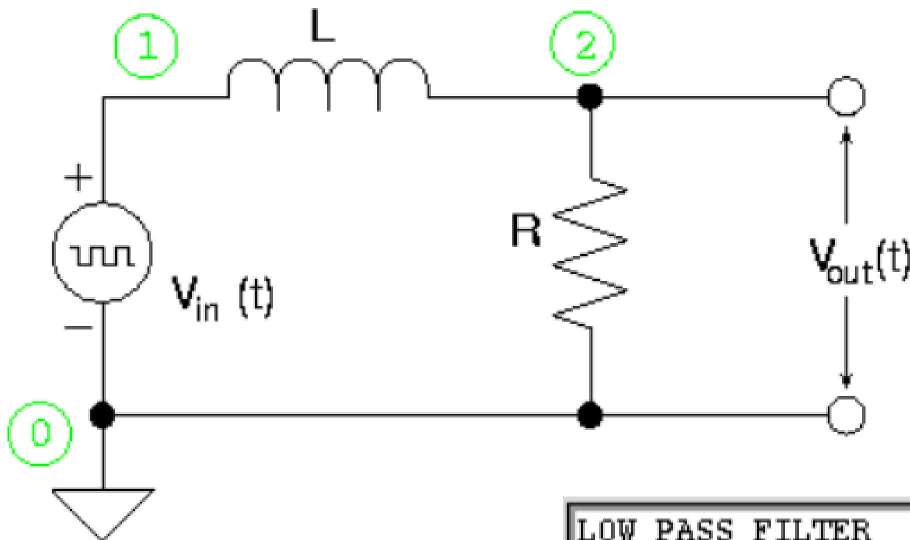
R = 600 Ω



Remember that the current is measured flowing INTO the positive node of the voltage source

A Low-Pass Filter Example

- Inductors pass low frequencies (just a wire for DC)
- Capacitors pass high frequencies



Component	Value
$V_{in}(t)$	+- 1 V square wave, f=250 kHz
L	10 μH
R	63 Ω

```
LOW PASS FILTER
VIN 1 0 PULSE(-1 1 0.0 1NS 1NS 2US 4US)
L1 1 2 10UH
R1 2 0 63
.PRINT TRAN V(2) V(1)
.TRAN 10NS 8US
.END
```

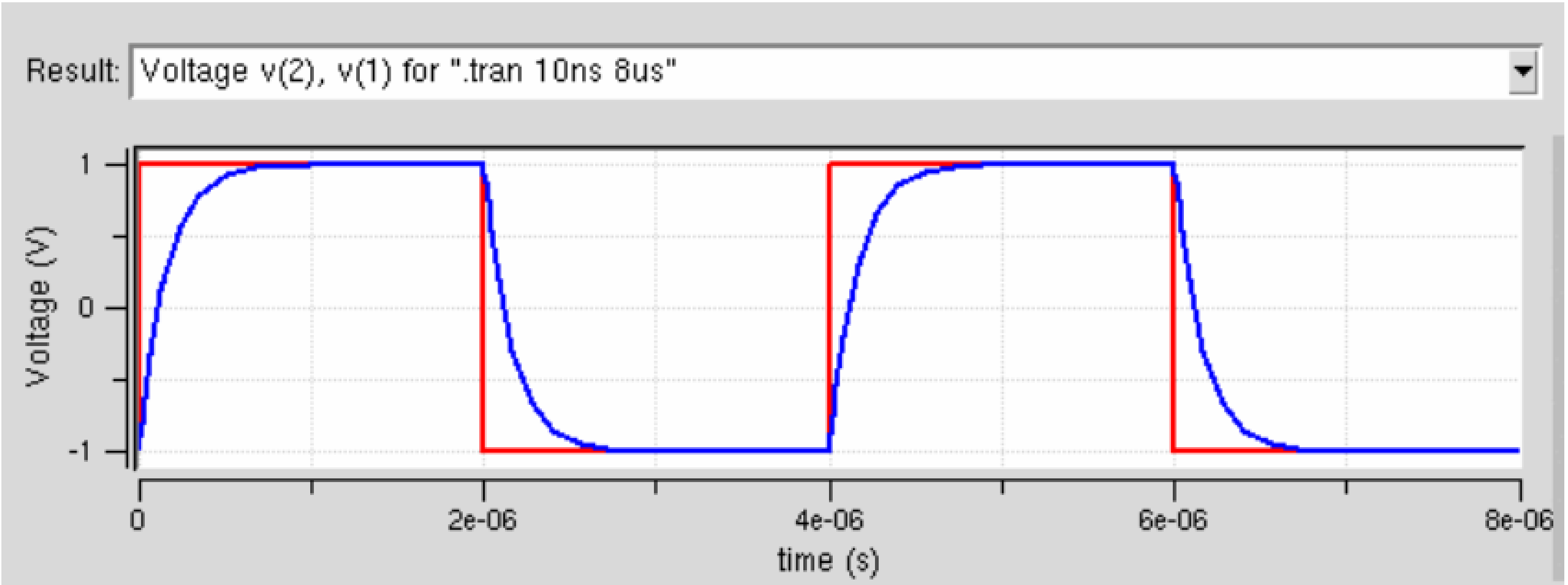
A Low-Pass Filter Example

- Voltage source defined using

```
VIN <N+> <N-> PULSE(V1 V2 TD TR TF PW PER)
```

V1	initial voltage (volts)
V2	pulsed voltage (volts)
TD	delay time (seconds)
TR	rise time (seconds)
TF	fall time (seconds)
PW	pulse width (seconds)
PER	period of waveform (seconds)

A Low-Pass Filter Example



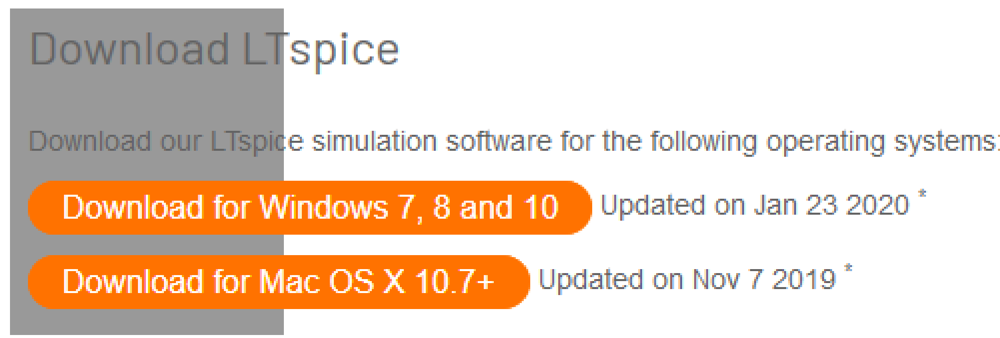
- The inductor blocks the high frequency components and rounds off the sharp corners.

Using LTspice

- LTspice was developed by Linear Technologies and was used in-house for IC and power supply circuit design
- In addition to circuit simulation, it provides a graphical schematic entry interface
- In July 2016, Analog Devices acquired Linear Technologies (amicably) for $14.8 billion in cash and stock
- https://www.analog.com/en/design-center/design-tools-and-calculators/ltspice-simulator.html

Using LTspice

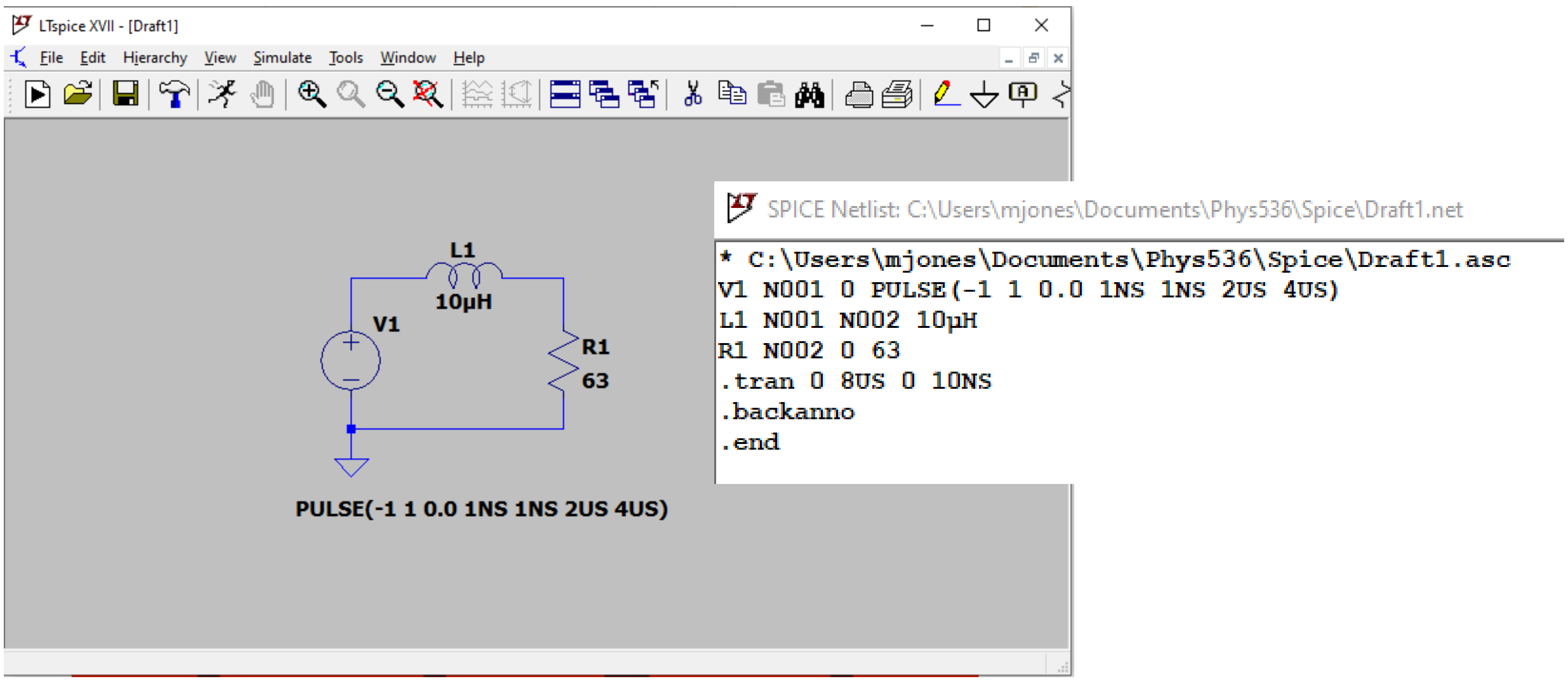
- Please locate and install LTspice



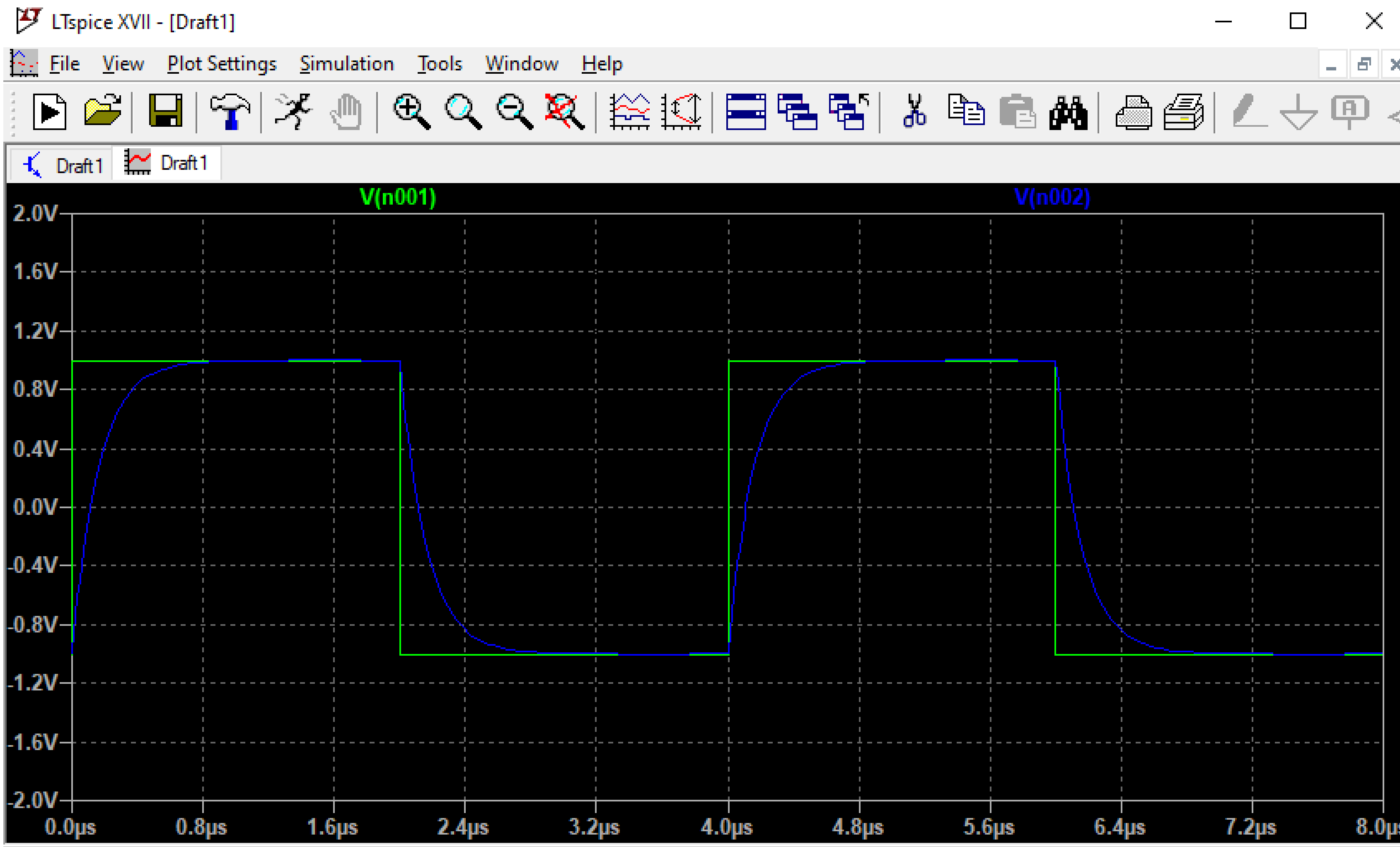
It should be installed on the computers in the lab.

Using LTspice

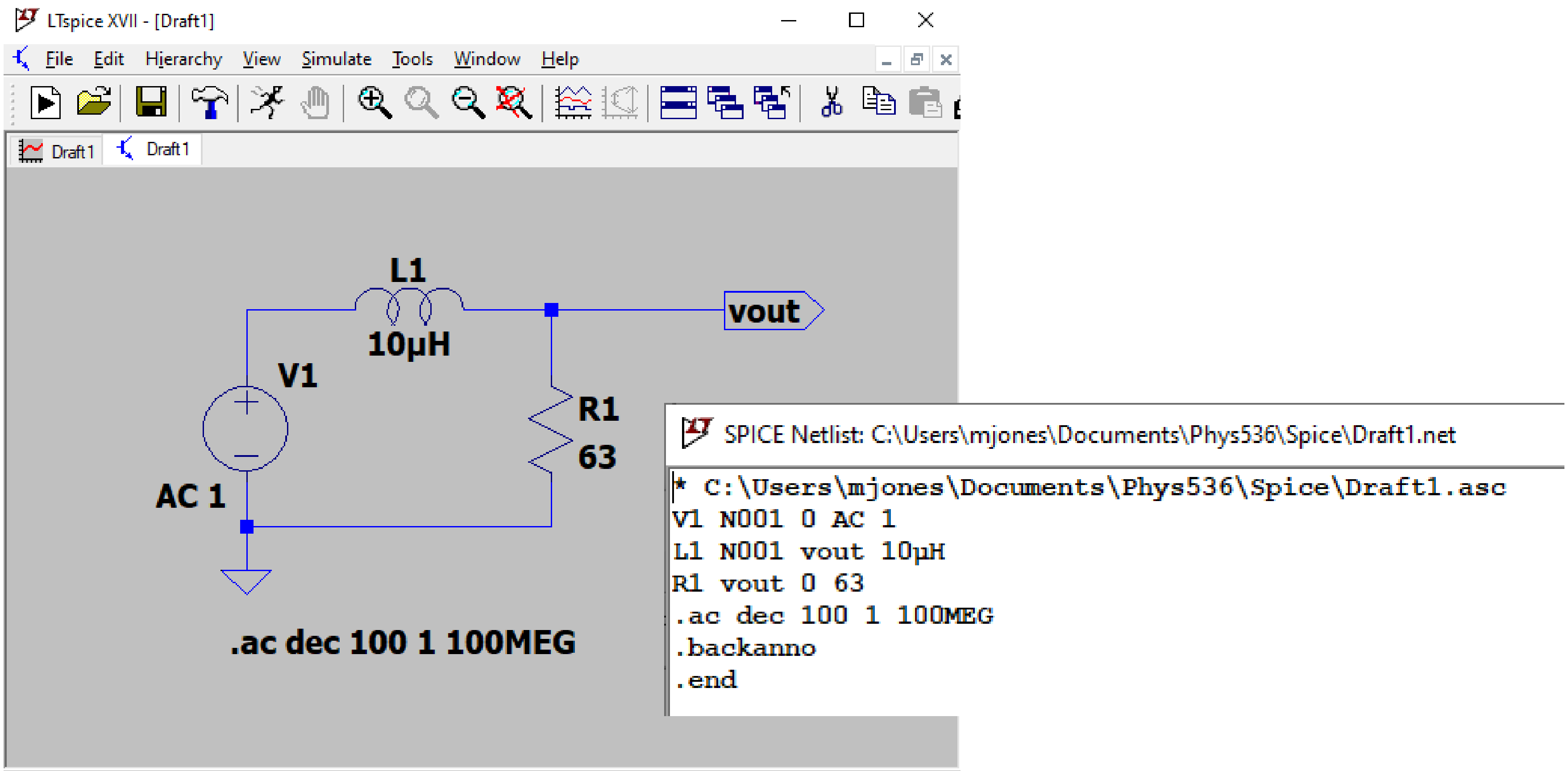
- There is lots of documentation and tutorial videos. Just ask señor google…



Using LTspice

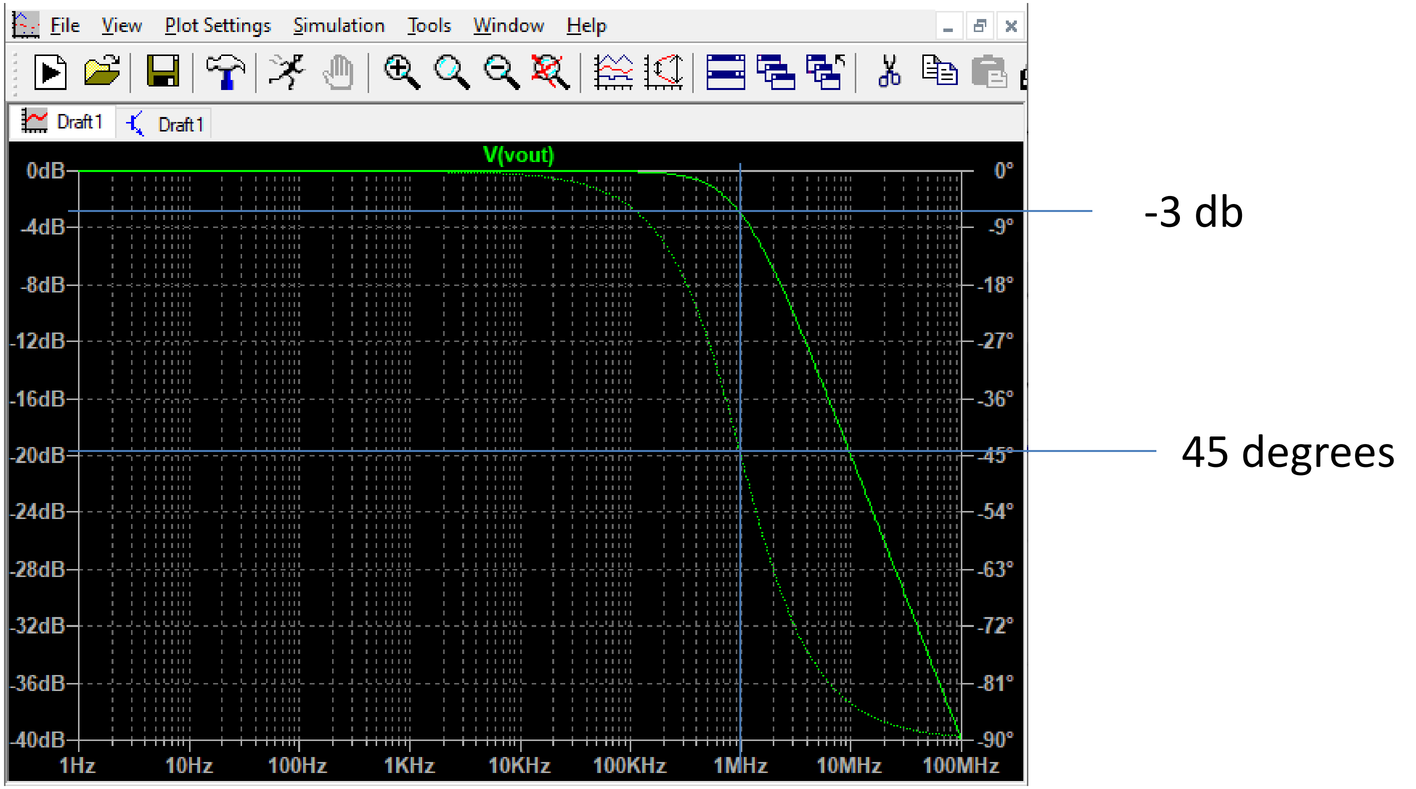


Frequency Analysis



The AC analysis will scan the frequency of a voltage source and plot the ratio voltages and the relative phase

Frequency Analysis

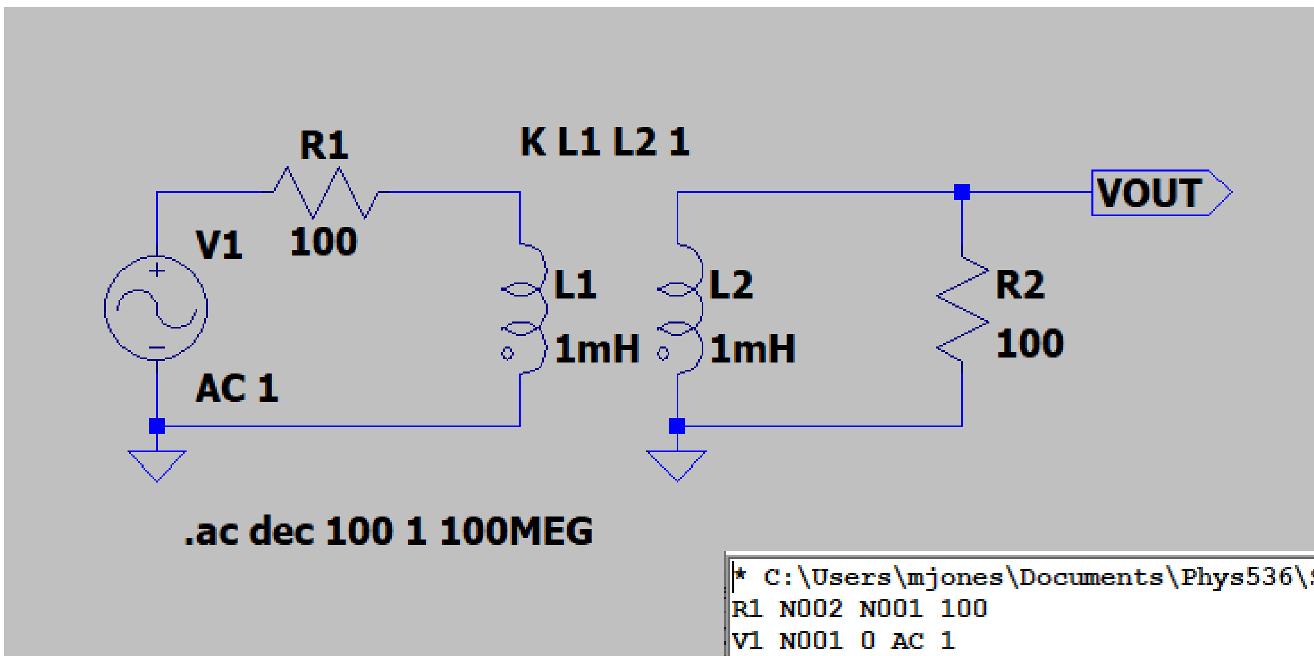


Expected cutoff frequency,

$$f_{-3\ db} = \frac{R}{2\pi L} = 1\ MHz$$

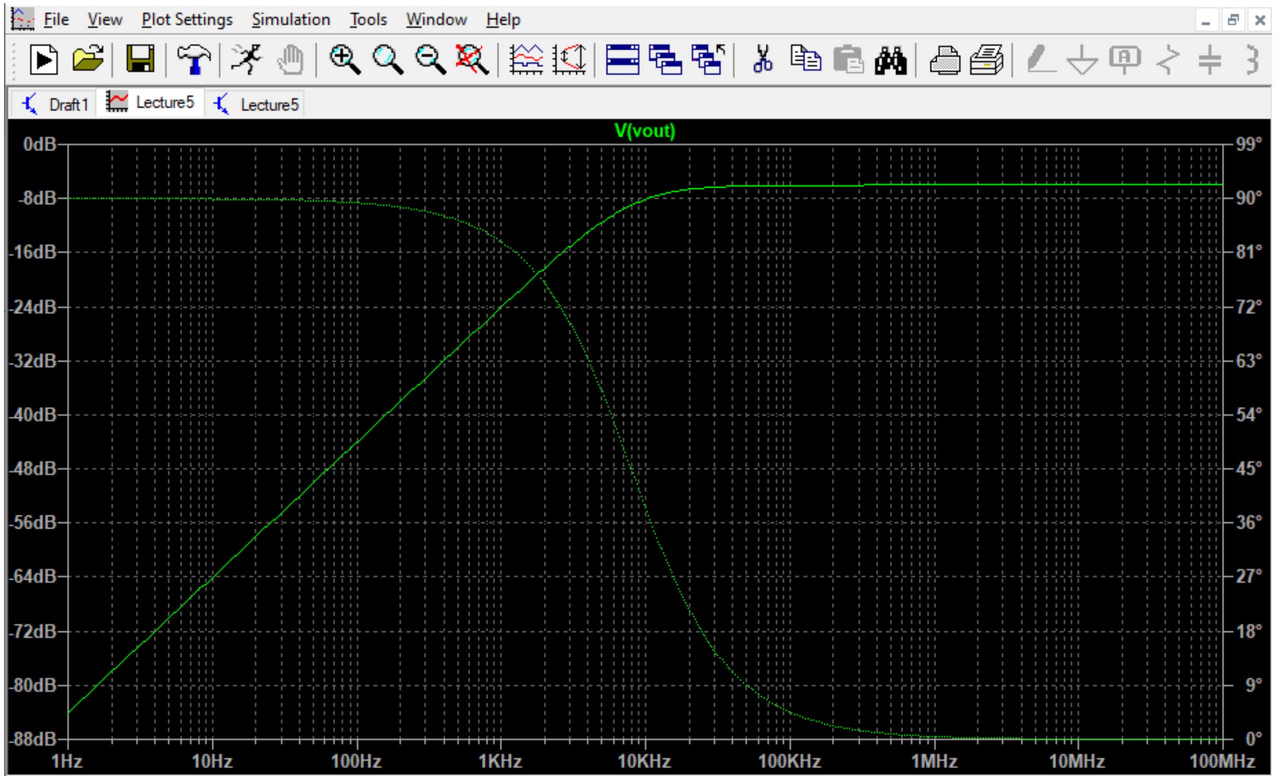
Transformers

- Transformers are described by individual inductors with a specified coupling




```
* C:\Users\mjones\Documents\Phys536\Spice\Lecture5.asc
R1 N002 N001 100
V1 N001 0 AC 1
L1 N002 0 1mH
L2 VOUT 0 1mH
R2 VOUT 0 100
K L1 L2 1
.ac dec 100 1 100MEG
.backanno
.end
```

Transformers



Notes:

- Make sure to define ground on BOTH sides of the transformer
- Alternatively connect both sides by a LARGE resistor