EE 105 | Discussion 2

Kieran Peleaux & Ali Ameri

Discussion Outline

- Overview of SPICE
  - What is SPICE?
  - SPICE Workflow
  - Netlist Syntax
    - Topology & Analysis
  - Running a SPICE Simulation
  - Viewing Simulation Results
  - Beyond the Basics

- Opamp practice

What is SPICE?

Simulation

Program

with

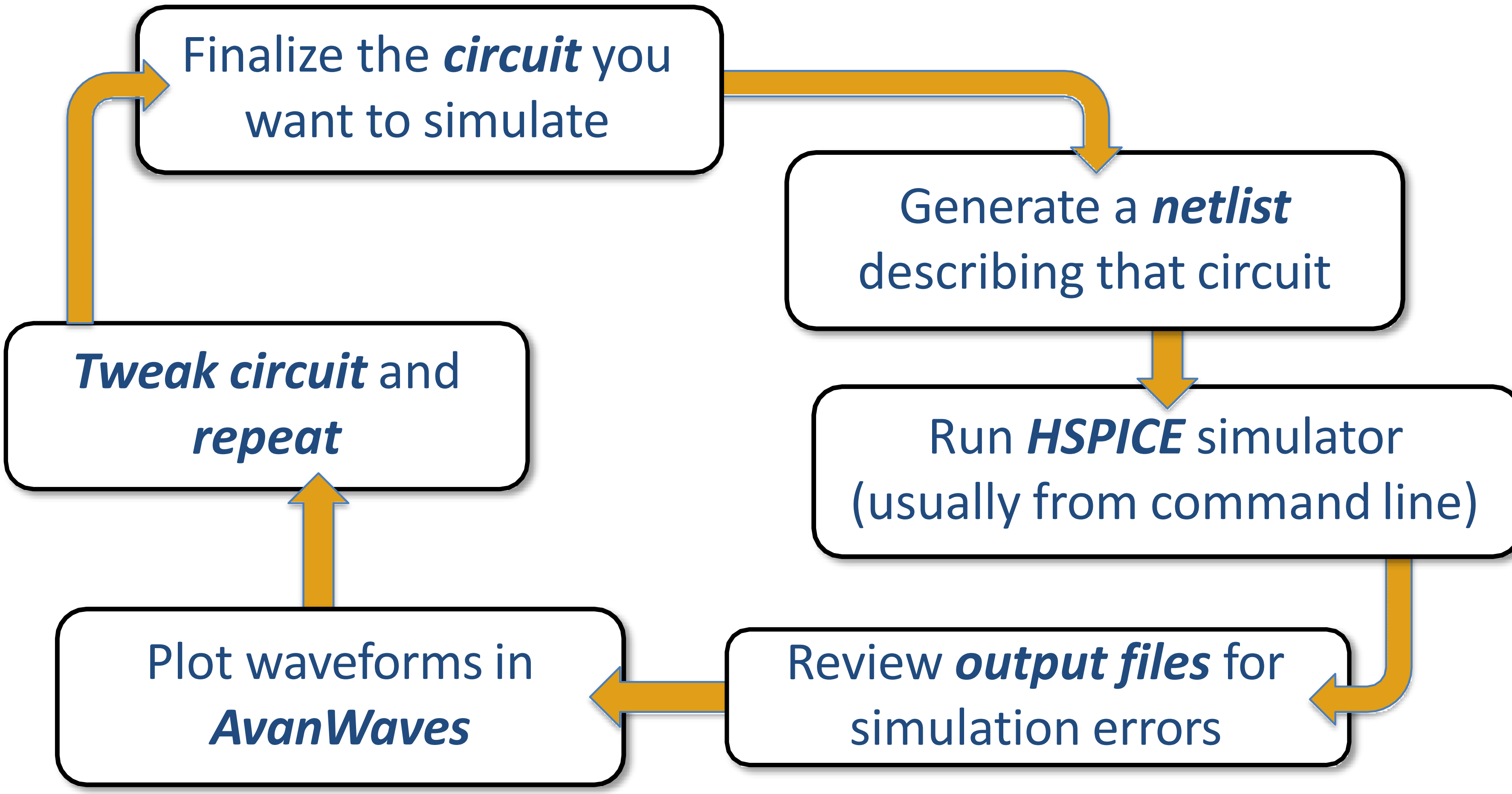
Integrated

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*

SPICE Workflow



What is a Netlist?

- A simple text file that contains a ***circuit description*** and ***analysis options***
- Different circuit elements specified by unique letters
- Circuit topology is defined by:
  - giving each node a unique name
  - assigning elements between these nodes

Netlist Syntax

- Filename ends in .sp
  - E.g., `mycircuit.sp`
- First line is always a comment!
- Not case sensitive
  - `vs = Vs = VS = vS`
- Last line must be `.end`
- Other than first/last line, order doesn't matter

```
EE105 SPICE Tutorial Example 1 - Simple RC Circuit
vs vs gnd PWL(0s 0V 5ms 0V 5.001ms 5V 10ms 5V)
r1 vs vo  1k
c1 vo gnd 1uF
.tran 0.01ms 10ms
.option post=2 nomod
.end
```

Netlist Syntax | Topology

- Each line denotes a different circuit element
  - The 1st character defines the type of element and the following characters denote the name of the element
  - The 2nd term is the first node the element is connected to
  - The 3rd term is the second node the element is connected to (more nodes will follow for devices with >2 terminals)
  - The last term is the element value/properties (can use prefixes `f`, `p`, `n`, `u`, `m`, `k`, `meg`, `giga` & `tera` to denote magnitude)

```
EE105 SPICE Tutorial Example 1 - Simple RC Circuit
vs vs gnd PWL(0s 0V 5ms 0V 5.001ms 5V 10ms 5V)
r1 vs vo  1k
c1 vo gnd 1uF
.tran 0.01ms 10ms
.option post=2 nomod
.end
```

Netlist Syntax | Topology

- Circuit nodes & elements can have the same name
- `gnd` is a standard name for global ground (can also use `0`)
- Order of nodes matters for things like sources!

```
EE105 SPICE Tutorial Example 1 - Simple RC Circuit
vs vs gnd PWL(0s 0V 5ms 0V 5.001ms 5V 10ms 5V)
r1 vs vo  1k
c1 vo gnd 1uF
.tran 0.01ms 10ms
.option post=2 nomod
.end
```

Netlist Syntax | Topology

- What does this circuit look like? Draw using labels that match the netlist.

```
EE105 SPICE Tutorial Example 1 - Simple RC Circuit
vs vs gnd PWL(0s 0V 5ms 0V 5.001ms 5V 10ms 5V)
r1 vs vo  1k
c1 vo gnd 1uF
.tran 0.01ms 10ms
.option post=2 nomod
.end
```

Netlist Syntax | Analysis

- Line 5 tells HSPICE to perform a transient analysis from time t = 0 ms to t = 10 ms in steps of 10 μs
- Line 6 tells HSPICE to generate waveform files necessary for viewing in *awaves* while not including model info in the output

```
1   EE105 SPICE Tutorial Example 1 - Simple RC Circuit
2   vs vs gnd PWL(0s 0V 5ms 0V 5.001ms 5V 10ms 5V)
3   r1 vs vo  1k
4   c1 vo gnd 1uF
5   .tran 0.01ms 10ms
6   .option post=2 nomod
7   .end
```

Simulating in HSPICE

- To simulate the circuit, simply run the command below in a UNIX terminal

```
hspice mycircuit.sp > mycircuit.lis
```

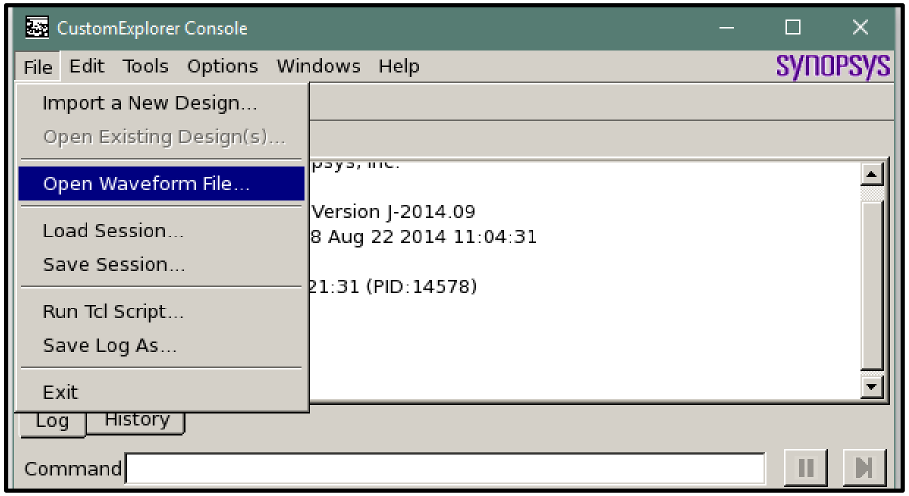
- This will run the simulation and store the outputs in `mycircuit.lis`
- You can open `mycircuit.lis` and check for errors/operating points, but most of the time we'll be interested in looking at plots of voltages and currents

Viewing Simulation Results

- To run *awaves*, use the command below (make sure you have X11 enabled!)

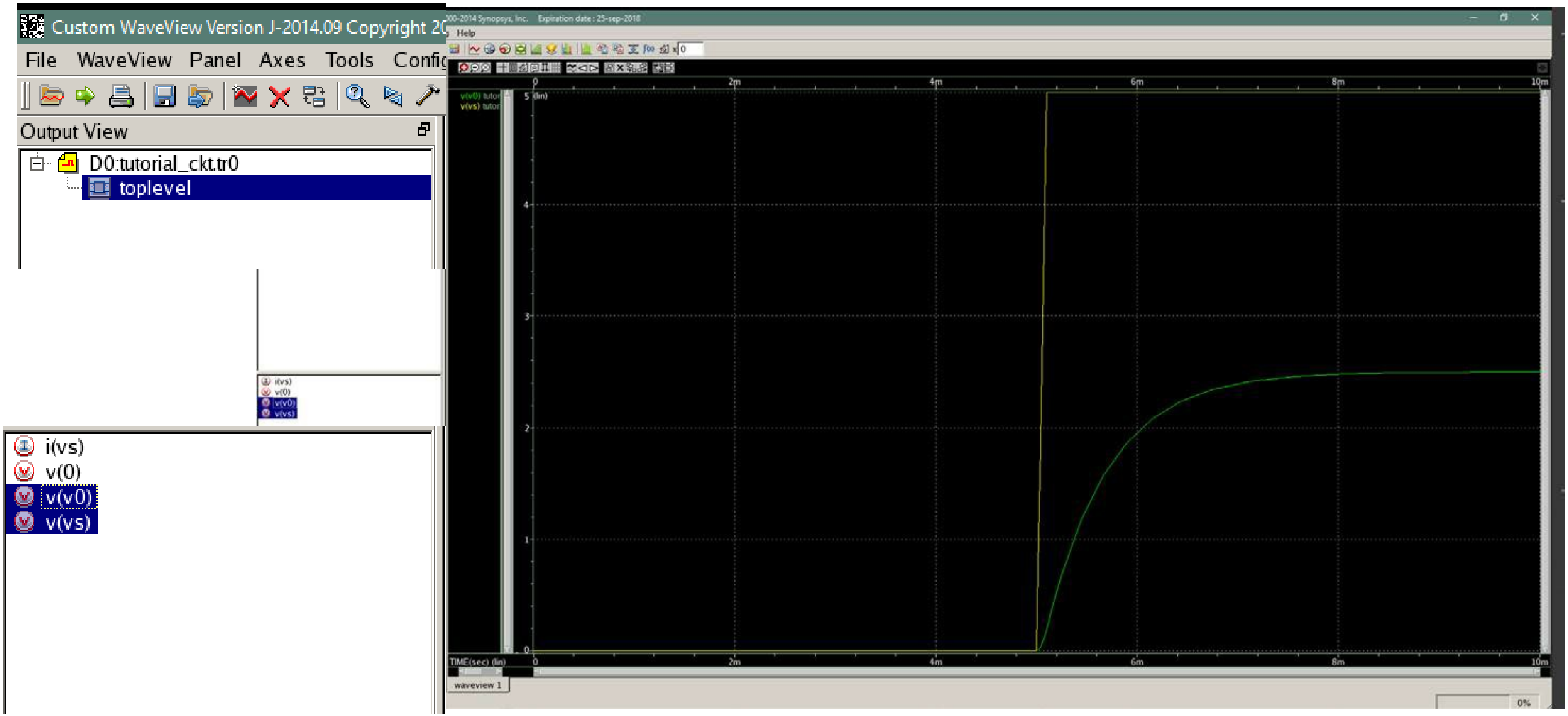
```
awaves &
```

- This will run the *awaves* software and leave your terminal free to use
- Click "Open Waveform File" & navigate to the file called `mycircuit.tr0`



Viewing Simulation Results

- This will open up the Custom WaveView window, where you can add traces to view



Beyond the Basics

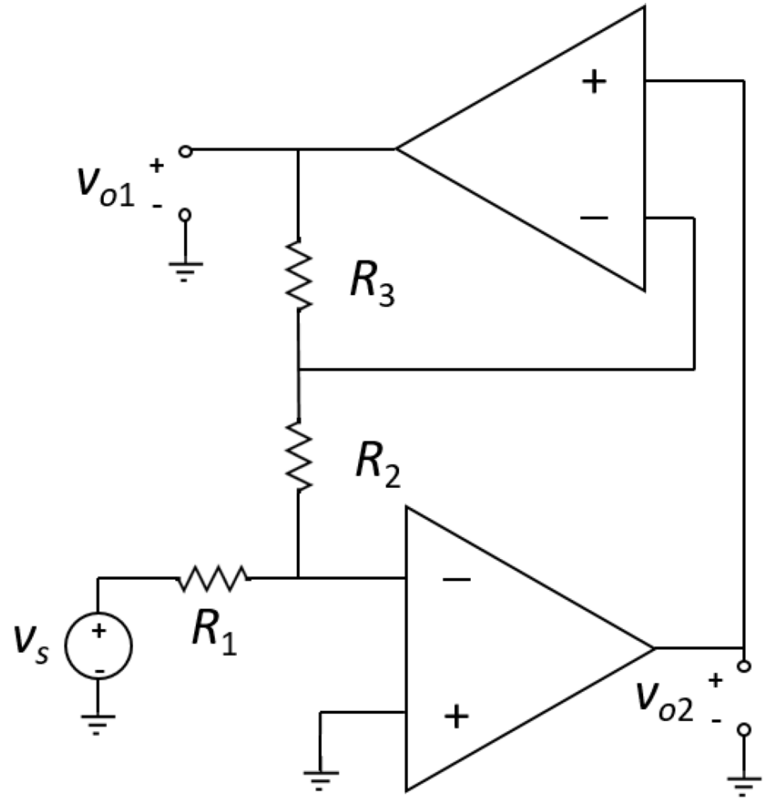
- Can perform many different type of analyses
  - AC (`.ac`), DC (`.dc`), transfer function (`.tf`), DC operating point (`.op`)
- For nonlinear devices (diodes, MOSFETs), must define a model to specify device parameters
  - Model names cannot start w/ a number!

```
EE105 SPICE Tutorial Example 2 - Simple Diode Circuit
.model tut_diode d (is=1e-14 vj=0.6 rs=10)
vs vs gnd 5V
rs vs vd  5k
d1 vd gnd tut_diode
.op
.end
```

Ideal Opamps

- Find an expression for v_{o1} and v_{o2}:

- Two key properties of ideal opamps:
  - Infinite input resistance
  - Infinite gain
- Follow the currents



Ideal Opamps

- $i_1 = v_s/R_1$
- $v_{o1} = -(R_2+R_3)i_1 = -v_s(R_2+R_3)/R_1$
- $v_{o2} = -(R_2)i_1 = -v_s(R_2/R_1)$