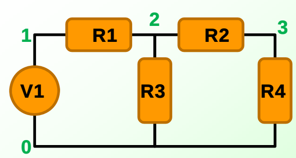
Quick SPICE Introduction

- SPICE = Simulation Program with Integrated Circuit Emphasis
- Netlist = text-based description of circuit
  - netlist does not use symbols or graphical elements
    - makes it simple to learn
    - but hard to visualize → usually need a companion schematic
  - netlist describes
    - circuit elements (resistors, capacitors, etc.)
    - power supplies, input voltages, bias currents, etc.
    - connections between circuit elements
    - analysis method; defines data to be calculated
- Example netlist

```
Example 1
V1 1 0 10       ← DC voltage supply
R1 1 2 10
R2 2 3 30
R3 2 0 200      ← Resistive elements
R4 3 0 4k
.PROBE
.TRAN 1 200 0 1 ←Analysis statements
.END
```

nodes, values



Op Amp SPICE Simulation

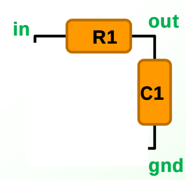
- Subcircuits: circuit block referenced as a single element
  - element line description
    - EX: Xopamp1 vin vref vdd gnd vout opamp741

X = subcircuit | unique element name | subcircuit nodes | subcircuit ID

  - subcircuit definition

```
.SUBCKT lpfilter in out
R1 in out 1k
C1 out gnd 1u
.ENDS
```

subcircuit nodes | subcircuit ID



In most SPICE simulators, can use numbers or text strings to define nodes

PSpice requires ground defined as node 0.

- Subcircuits for opamp simulation
  - commercial opamps often provide a model subcircuit netlist
    - include opamp model subcircuit in your SPICE file
    - "call" opamp model with single subcircuit element line

MICHIGAN STATE UNIVERSITY

Example Inverting Amp Using Subcircuits

```
* Main circuit
XOP467 Vpos Vneg VDD GND Vout OP467
*above line calls the subcircuit; node order defined in subcircuit
R1 Vin Vneg 100
R2 Vneg Vout 100k
* define Vin as AC
VS Vin agnd AC 1
 * define positive and negative supply and analog ground
Vpossupply VDD GND 10V
Vagnd agnd GND 5V
* connect Vneg op-amp input to agnd
V1 Vpos agnd 0V
Vg GND 0 0V
```

Elements: main part to edit; change to match your circuit

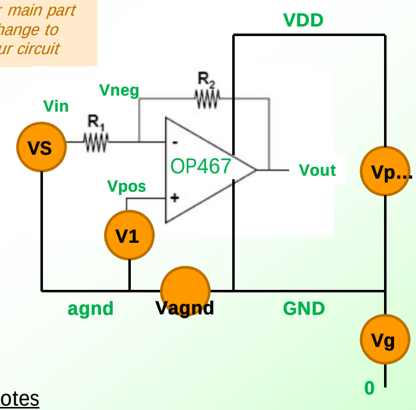
Voltages: may have to adjust to meet circuit requirements

```
* OP467 SPICE Macro-model
*                  non-inverting input
*                  |  inverting input
*                  |  |  positive supply
*                  |  |  |  negative supply
*                  |  |  |  |  output
*                  |  |  |  |  |
.SUBCKT OP467    1  2  99 50 27
<long subcircuit omitted>
.ENDS
```

Subcircuit: Just copy this; do not make any changes here

```
* define analysis type: .AC for frequency response, .TRAN for time response
.AC DEC 5 10 1MEG
.PROBE
.END
* End Main circuit
```

Analysis: Keep same for frequency response (can adjust frequencies if necessary)



Notes
* = comment line
first line of netlist is always circuit description
always end with .END

MICHIGAN STATE UNIVERSITY

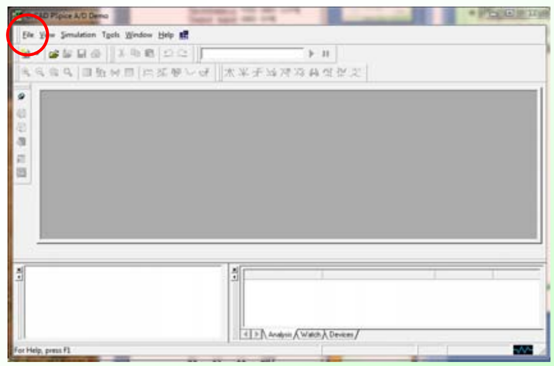
SPICE Analysis Types

- .AC: sweeps frequency for all defined AC sources
  - used for frequency response analysis, most common type for ECE445
  - must have at least one AC source in circuit
    - Example AC source: Vs 1 0 AC 1
      - source name, output node, reference node, AC (not DC), amplitude
  - Example .AC analysis statement
    - .AC DEC 10 100 1e6
      - DEC 10 = logarithmic range with 10pts per decade
      - 10 = starting frequency, 1e6 = stop frequency (1MHz)
- .TRAN: analyze circuit over time
  - generally for time-varying sources like sinusoidal signals
    - Example SIN source: Vs 1 0 SIN(2V 10mV 1k)
      - 2V = DC offset, 10mV = AC amplitude, 1k = frequency
  - Example .TRAN analysis statement
    - .TRAN 20u 20m
      - 20u = step time, 20m = stop time
- More details? http://www.uta.edu/ee/hw/pspice/
- Good SPICE manual: http://newton.ex.ac.uk/teaching/CDHW/Electronics2/userguide/sec1.html

MICHIGAN STATE UNIVERSITY

OrCAD PSpice

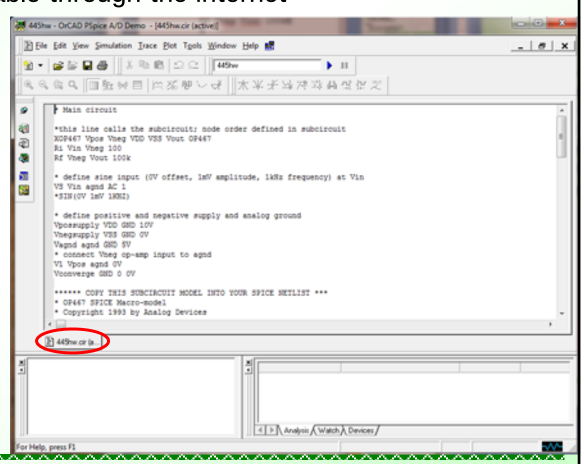
- PSpice: version of SPICE by Cadence
  - available in DECS labs
  - free student version available through the internet
- Create new file
  - File >> New >> Text File



MICHIGAN STATE UNIVERSITY

OrCAD PSpice

- PSpice: version of SPICE by Cadence
  - available in DECS labs
  - free student version available through the internet
- Create new file
  - File >> New >> Text File
- Type in netlist
  - save as .cir file

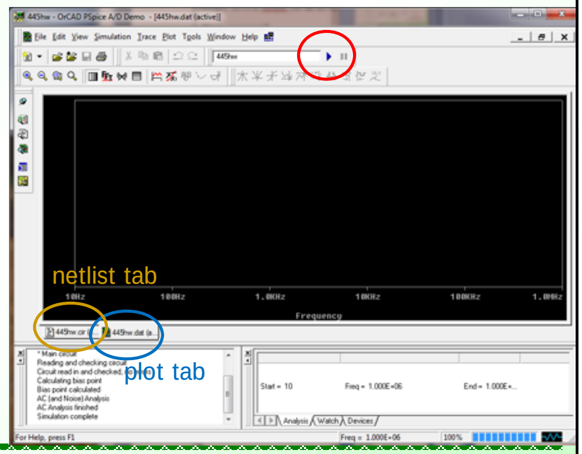


MICHIGAN STATE UNIVERSITY

OrCAD PSpice

- PSpice: version of SPICE by Cadence
  - available in DECS labs
  - free student version available through the internet
- Create new file
  - File >> New >> Text File
- Type in netlist
  - save as .cir file
- Run simulation
  - type specified in netlist
  - check .out file if errors

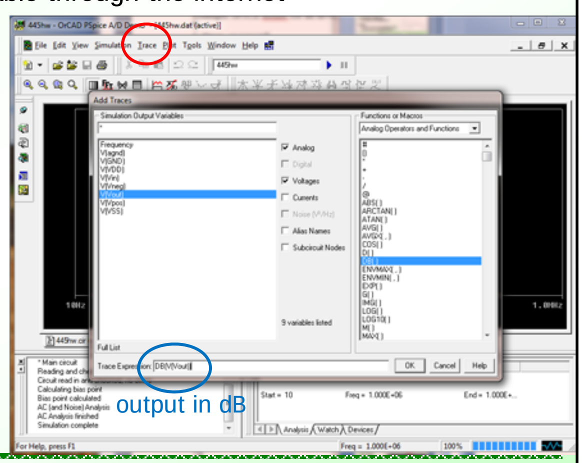
```
* define analysis type: .AC for frequency response,
.Probe
.AC DEC 5 10 1MEG
*.TRAN 3m

.END
```



MICHIGAN STATE UNIVERSITY

OrCAD PSpice

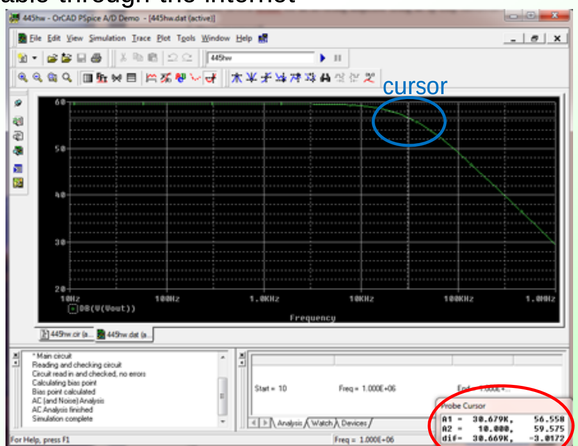
- PSpice: version of SPICE by Cadence
  - available in DECS labs
  - free student version available through the internet
- Create new file
  - File >> New >> Text File
- Type in netlist
  - save as .cir file
- Run simulation
  - type specified in netlist
  - check .out file if errors
- Add trace
  - Trace >> Add Trace
  - DB(V(Vout))



MICHIGAN STATE UNIVERSITY

OrCAD PSpice

- PSpice: version of SPICE by Cadence
  - available in DECS labs
  - free student version available through the internet
- Create new file
- Type in netlist
- Run simulation
- Add trace
  - Trace >> Add Trace
  - DB(V(Vout))
- Add cursor
  - Trace >>Cursor>>Display
  - -3dB frequency = 30.7kHz



- Questions?