

# SPICE BASICS

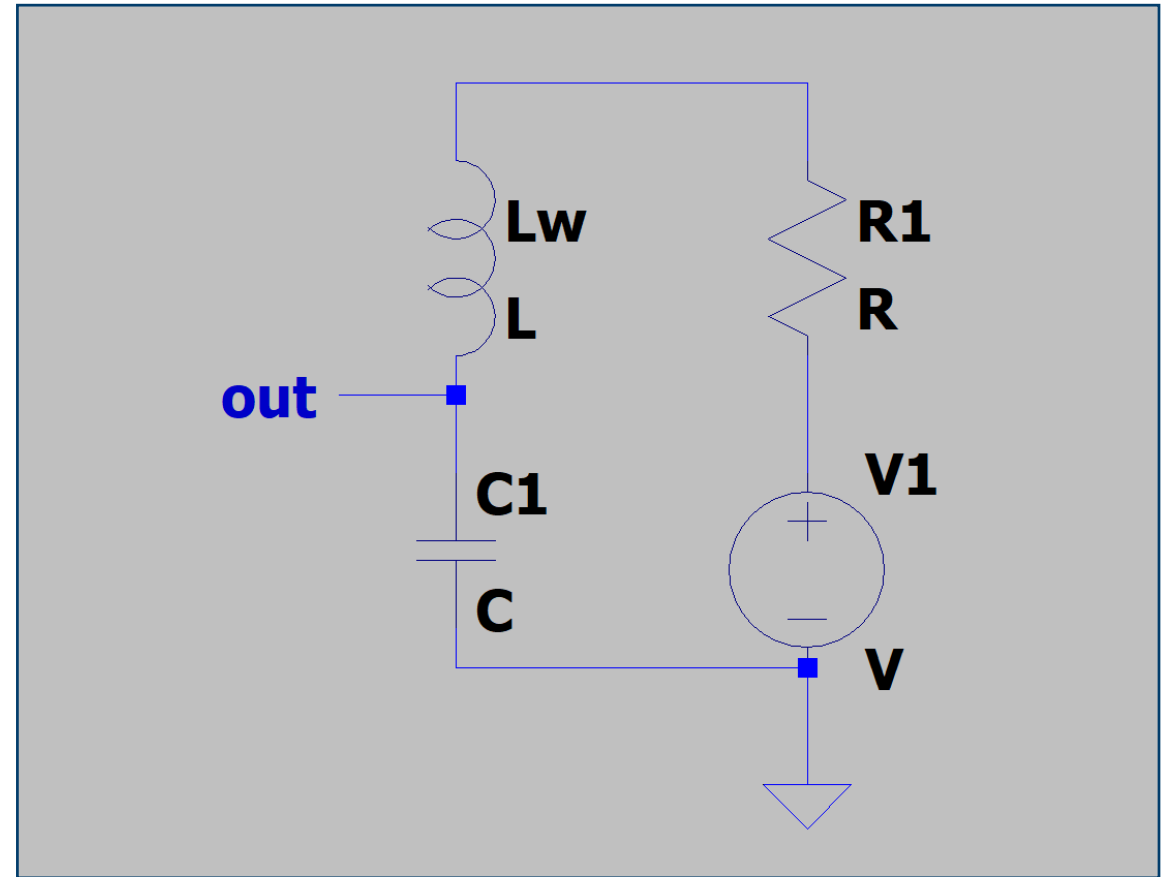
## Key Component Types

| Prefix | Component   | Example       | Notes                           |
|--------|-------------|---------------|---------------------------------|
| R      | Resistor    | R1 in n2 1k   | 1 k $\Omega$ between node 1 & 2 |
| C      | Capacitor   | C1 out 0 10u  | Node 0 = ground                 |
| L      | Inductor    | Lw n2 out 50n | 1 mH                            |
| V      | Voltage src | V1 in 0 DC 5  | DC or AC                        |

# SPICE BASICS

## Netlist vs Schematic

```
V1 in 0 DC 5  
R1 in n2 1k  
Lw n2 out 50n  
C1 out 0 10u  
.end
```



# SPICE BASICS

## Noise Analysis

```
.NOISE V(<OUT_NODE>) <INPUT_SOURCE> <SWEEP_TYPE> <NPOINTS> <FSTART> <FSTOP>
```

|                  |   |
|------------------|---|
| V(<out_node>)    | Output node for noise calculation                   |
| <input_source>   | Reference source for input-referred noise           |
| <sweep_type>     | Frequency sweep style (dec, oct, lin)               |
| <Npoints>        | Points per decade/octave or total                   |
| <Fstart> <Fstop> | start and stop frequencies ( <b>1 Hz to 1 GHz</b> ) |

# SPICE BASICS

## Time-domain Transient Analysis

```
.TRAN <TSTEP> <TSTOP> [TSTART [DTMAX]]
```

|          |                           |
|----------|---------------------------|
| <Tstep>  | Plotting increment        |
| <Tstop>  | Total simulated time      |
| [Tstart  | Time to start saving data |
| [dTmax]] | Maximum time step         |

# SPICE BASICS

## Frequency Response (Gain & Phase)

.AC <TYPE> <POINTS> <START\_FREQ> <STOP\_FREQ>

|              |  |
|--------------|--|
| <type>       | Sweep spacing (dec, oct, lin)                                  |
| <points>     | Number of frequency points per decade/octave or total (linear) |
| <start_freq> | Starting frequency   |
| <stop_freq>  | Ending frequency   |

# SPICE BASICS

## Parameter Sweep

```
.STEP PARAM <PARAM_NAME> <START> <STOP> <INCREMENT>
```

<param\_name>                      Name from the parameter definition

<start>                              Start value of the sweep

<stop>                                End value of the sweep

<increment>                        Sweep increment

# SPICE CHALLENGE

- Make Teams with minimum one Github Account and minimum one IC Designer
- Install LTSpice
- Clone Repo <https://github.com/padiro/co-simulation-workshop>
- Import netlist into LTSPICE
- Fix errors in netlist
- Find the resistance value for R1
- Find the voltage noise power spectral density at 1kHz
- Submit solution via Pull Request on Github