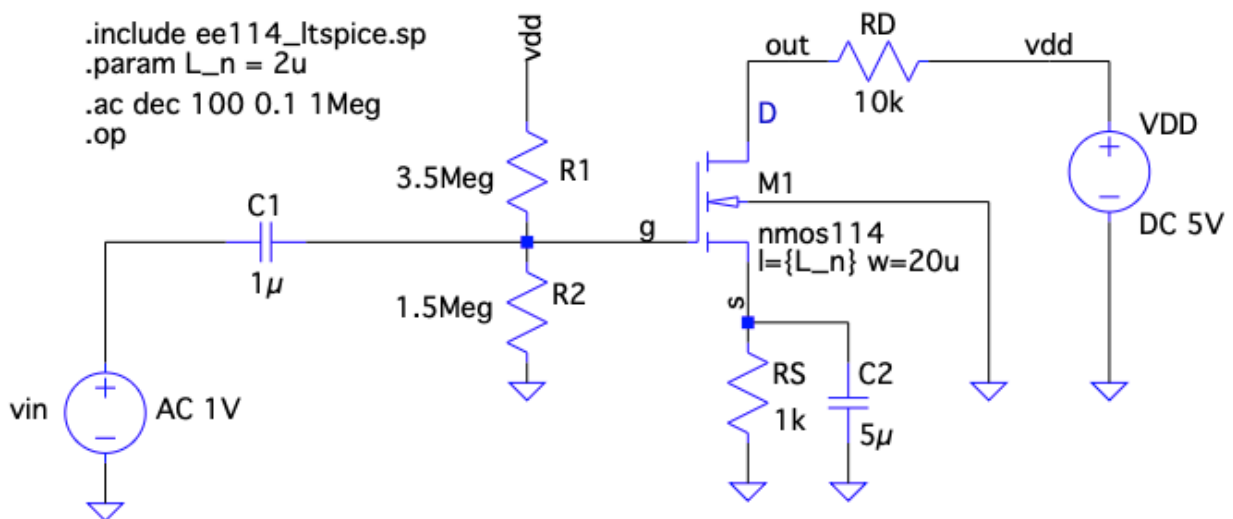The main goal of this lab. is to learn how to use Python to post process LTspice simulations.

Pre-Lab

Create the following LTSpice circuit and simulate it. Make to save the binary raw file of the AC simulation

1. Install Anaconda Python Distribution (link)

2. Install the python's PyLTSpice version 3.1 package

   ```
   pip install PyLTSpice==3.1
   ```

3. Install the python's control package

   ```
   pip install control
   ```

Lab

In today lab we'll learn how to use Python for:

- solving systems of equations
- creating Bode plots
- post-processing LTspice's simulation results

Please download the following python files provided on canvas

- *systemEquation.py*
- *Bode.py*
- *Plot_LTSim.py*