

# 電路分析

- 目的：使用 python 的 pypspice 模組做電路模擬
- 實做：陽春的 low pass RC filter
  - 先試用免費的跨平台電路模擬軟體 qucs(圖形化使用者界面)
  - 後執行 python 的範例程式作比較
    - 安裝 (Linux 平台)
      - `sudo apt-get install libngspice0-dev`
      - `pip3 install pypspice`
        - matplotlib, numpy
- 結果
  - S domain 分析 (Laplace transform) 和 time domain 波形

# 低通濾波系統

(cutoff frequency @ about 160Hz)

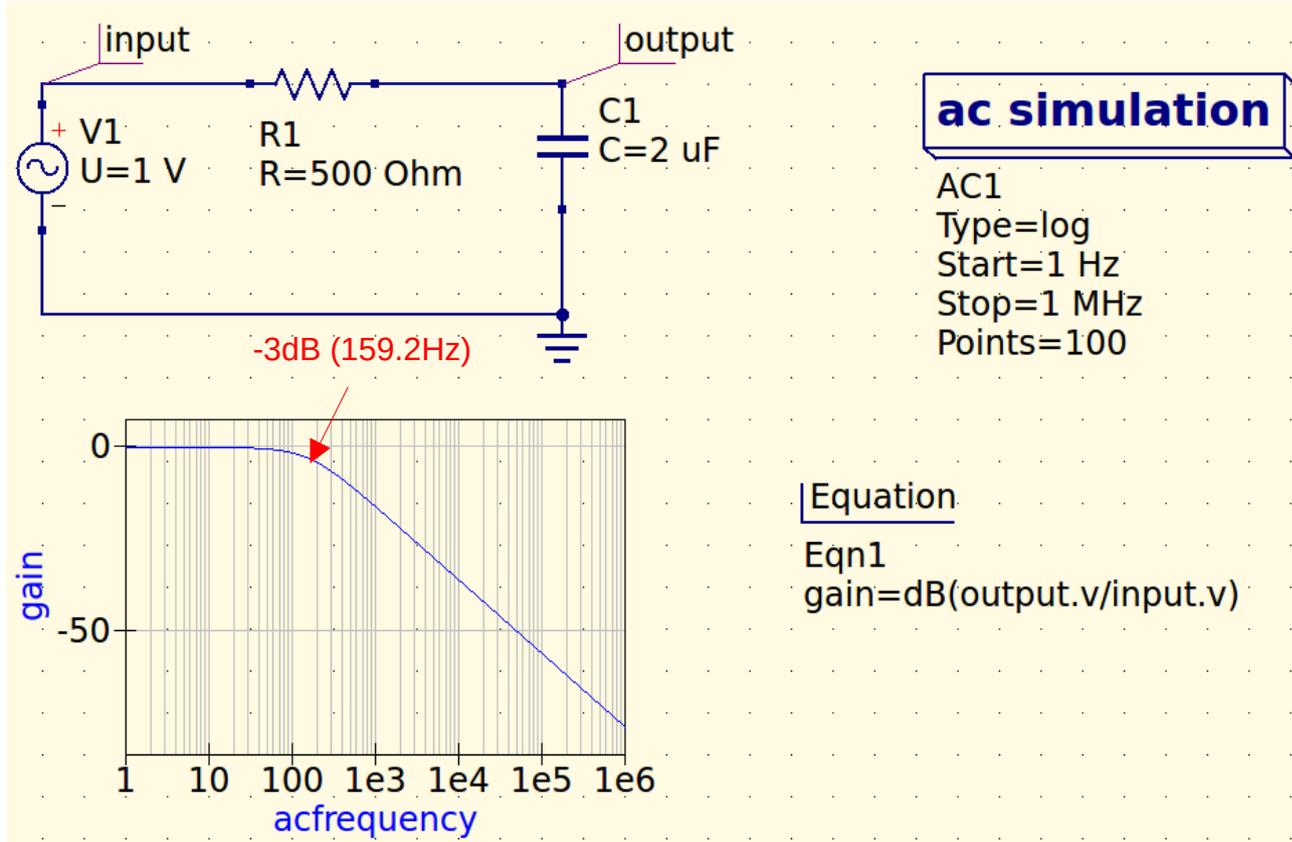
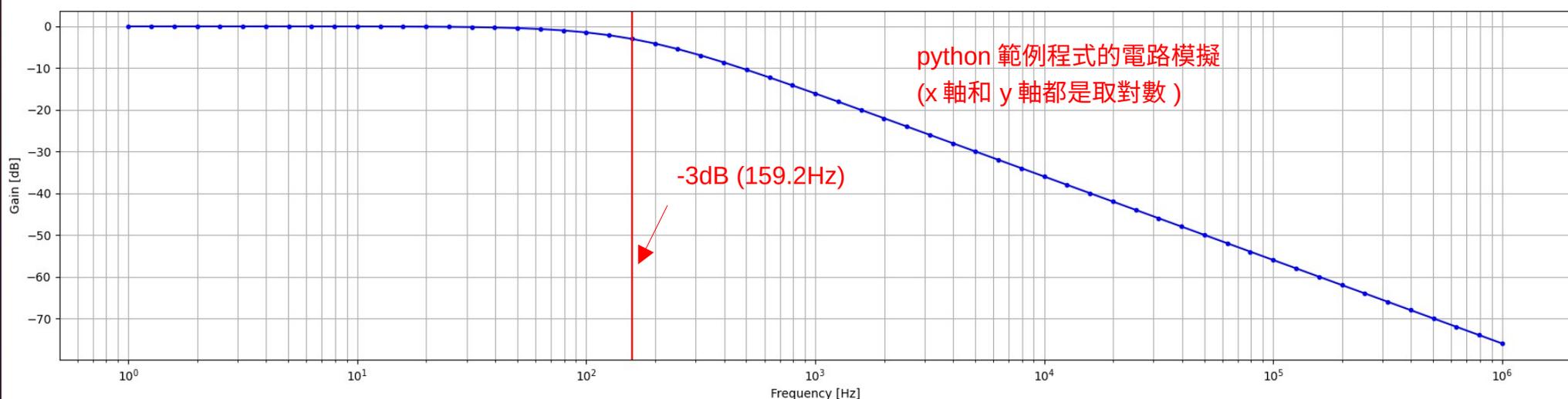
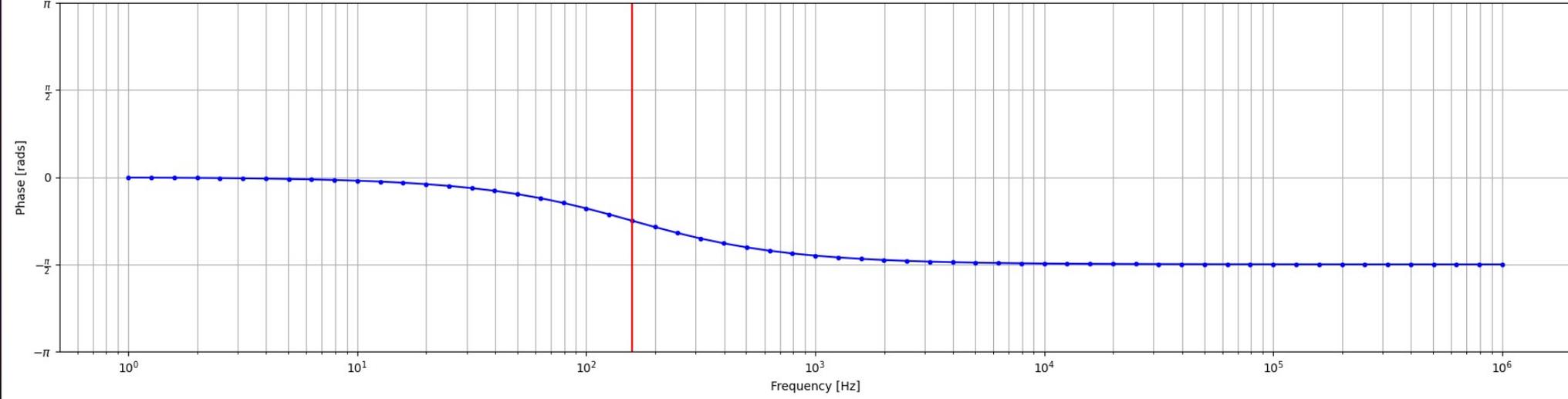


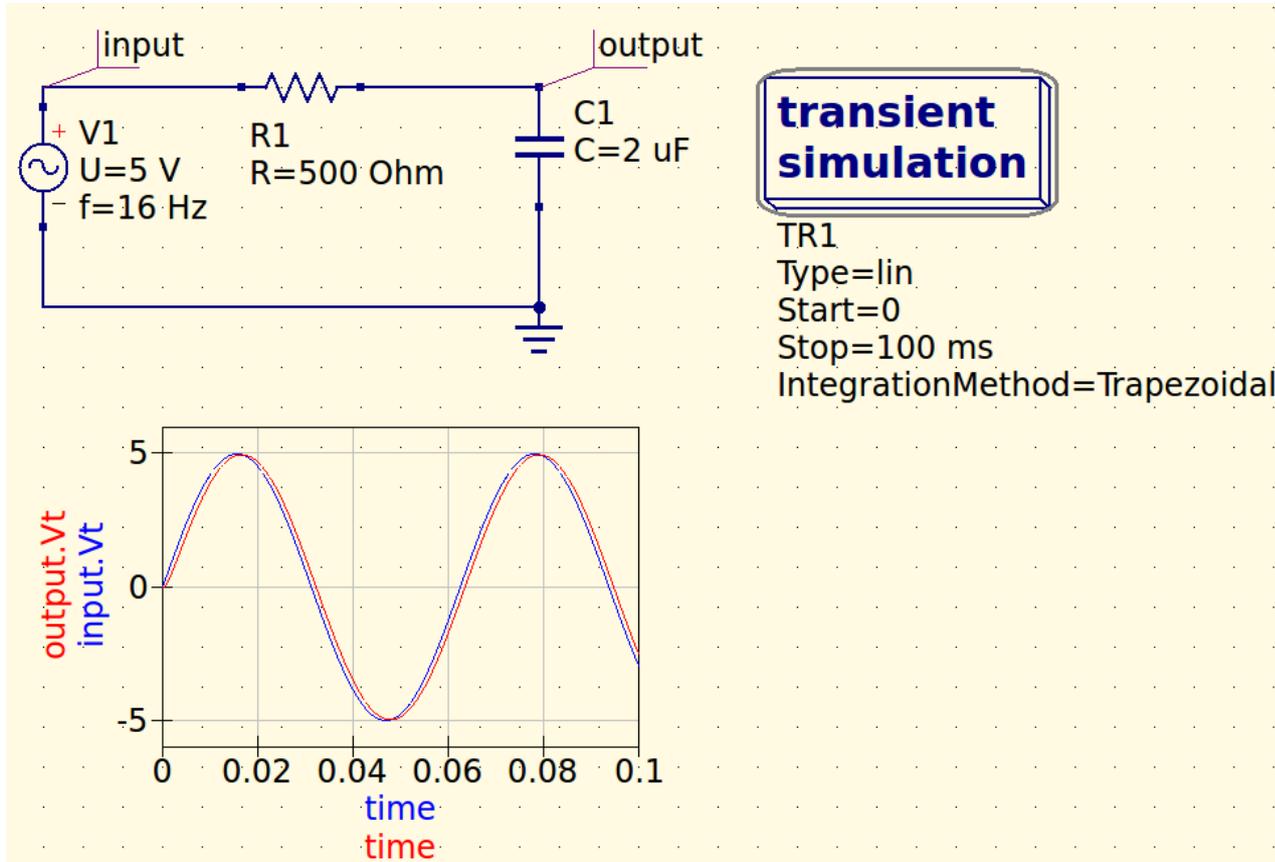
Figure 1



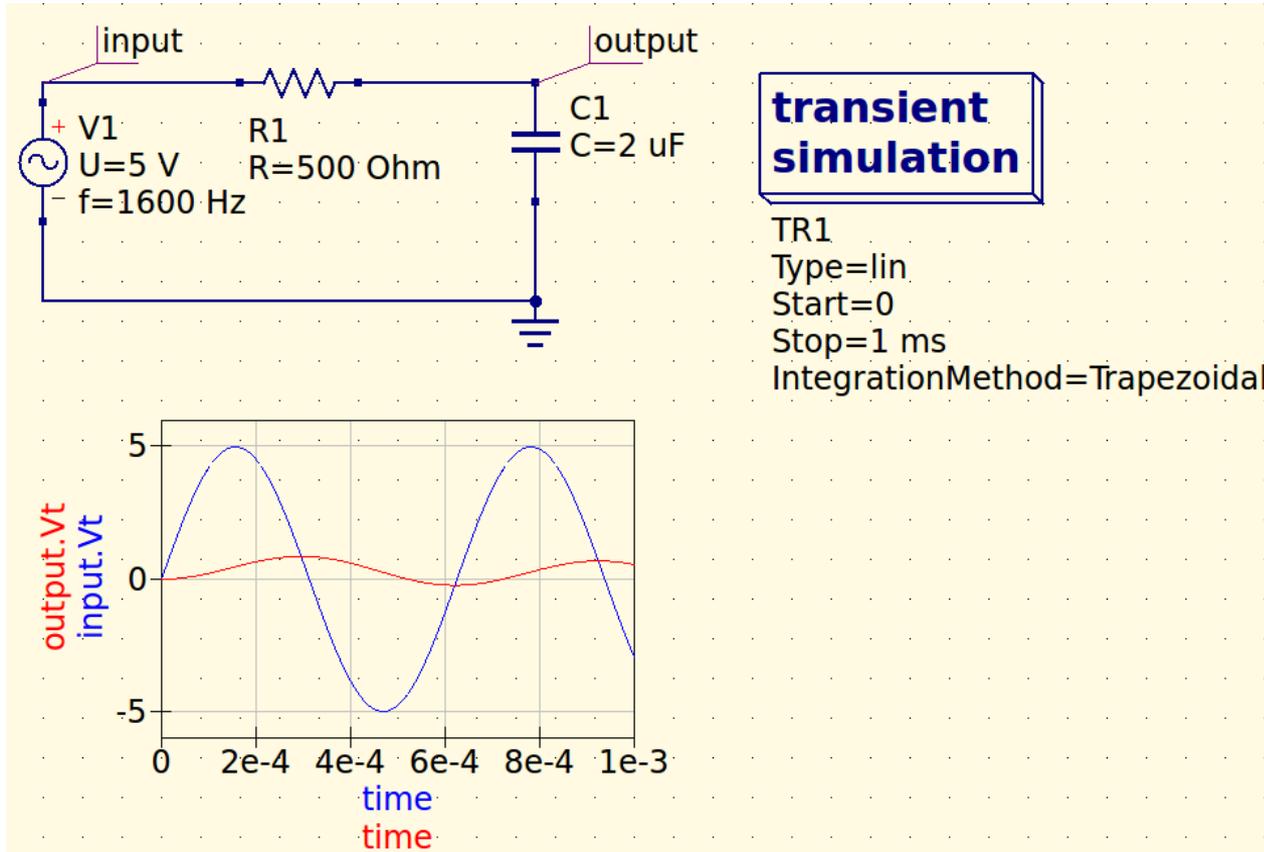
Bode Diagram of a Low-Pass RC Filter



# 低通濾波測試 A (16Hz 正弦波)

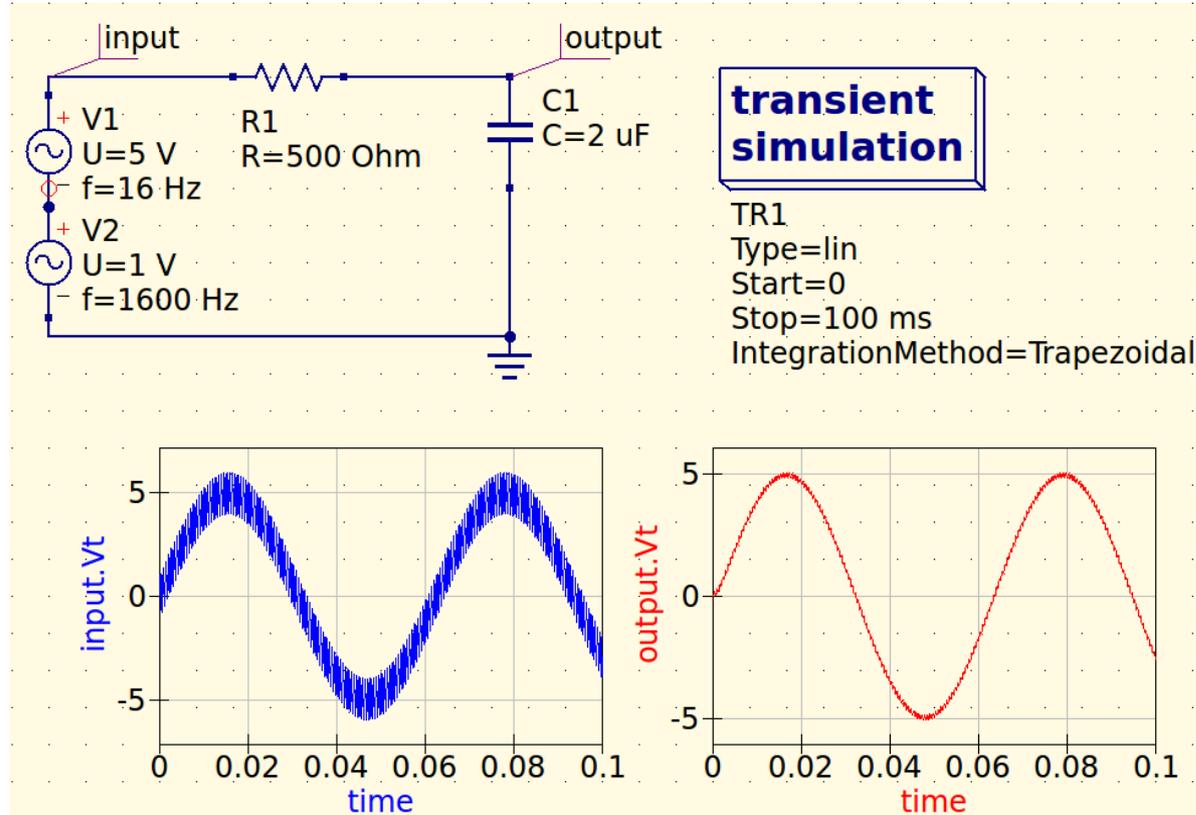


# 低通濾波測試 B (1600Hz 正弦波)



# 低通濾波測試 C

(1600Hz + 16Hz 正弦波)

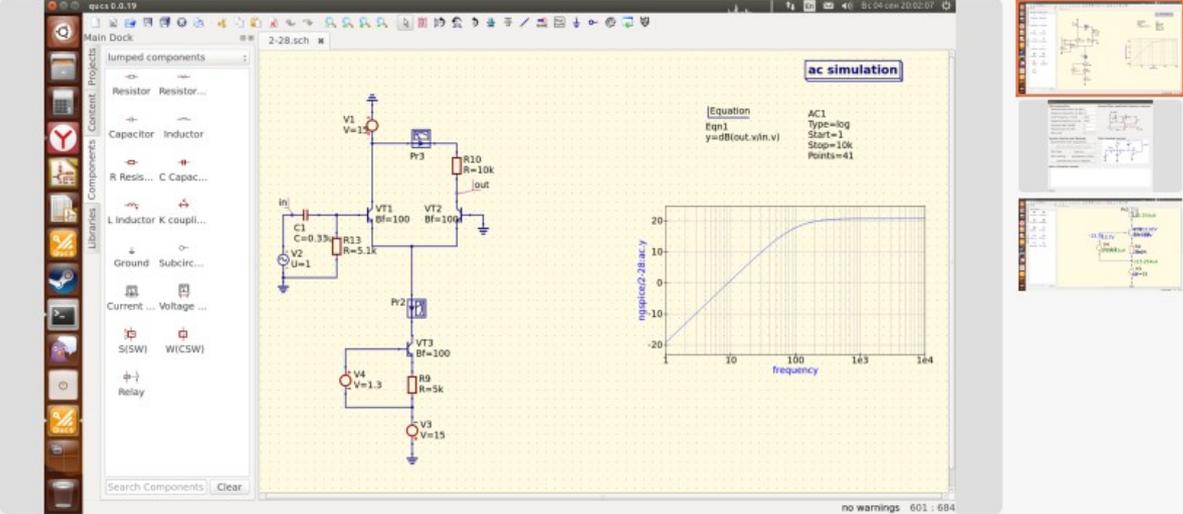


# 電路模擬軟體 qucs(Linux)

 **qucs-spice**  
Quite Universal Circuit Simulator

★★★★★ (15)

移除(R) Permissions



The screenshot displays the QUCS software interface. On the left is a component library with categories like Resistor, Capacitor, Inductor, and Voltage source. The main workspace shows a circuit diagram with components labeled V1, V2, C1, R13, Pr3, Pr2, Pr1, R10, V4, R9, V3, VT1, VT2, and VT3. An AC simulation plot is shown on the right, with the equation  $y=dB(out.v/in.v)$  and parameters: AC1 Type=log, Start=1, Stop=10k, Points=41. The plot shows a frequency response curve. The status bar at the bottom indicates 'no warnings 601 : 684'.

Qucs is an integrated circuit simulator which means you are able to setup a circuit with a graphical user interface (GUI) and simulate the large-signal, small-signal and noise behaviour of the circuit. After that simulation has finished you can view the simulation results on a presentation page or window. Supports spice simulators.

8.6.1. Low Pass Rc Filter

8.6.2. RLC Filter

8.7. Kirchhoff's circuit laws

8.8. Ohm's Law

8.9. Simulation using External Sources

8.10. NgSpice Interpreter

8.11. Operational Amplifier

8.12. Passive

8.13. Power Supplies

8.14. Relay

8.15. Resistor

8.16. Spice Netlist Parser Bootstrap Example

8.17. Kicad Netlist Parser Example

8.18. Switched Power Supplies

8.19. Transformer

8.20. Transistor

8.21. Transmission Lines

9. Example Wish List

10. Design Notes

11. API Documentation

12. Development & Community

13. How to Refer to PySpice ?

# pyspice 的範例程式

```
import math
import numpy as np
import matplotlib.pyplot as plt
```

```
import PySpice.Logging.Logging as Logging
logger = Logging.setup_logging()
```

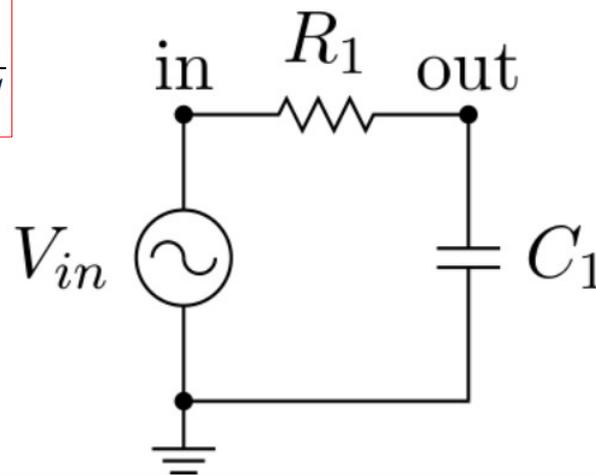
```
from PySpice.Plot.BodeDiagram import bode_diagram
from PySpice.Spice.Netlist import Circuit
from PySpice.Unit import *
```

$$\frac{V_{out}}{V_{in}} = \frac{\frac{1}{sC}}{\frac{1}{sC} + R} = \frac{1}{1 + sRC}$$

$$s = \omega i = 2 \pi f i$$

$$\sqrt{(2 \pi f RC)^2 + 1^2} = \sqrt{2}$$

$$f = \frac{1}{2 \pi RC}$$



```
circuit = Circuit('Low-Pass RC Filter')

circuit.SinusoidalVoltageSource('input', 'in', circuit.gnd, amplitude=1@u_V)
R1 = circuit.R(1, 'in', 'out', 1@u_kΩ)
C1 = circuit.C(1, 'out', circuit.gnd, 1@u_uF)
```