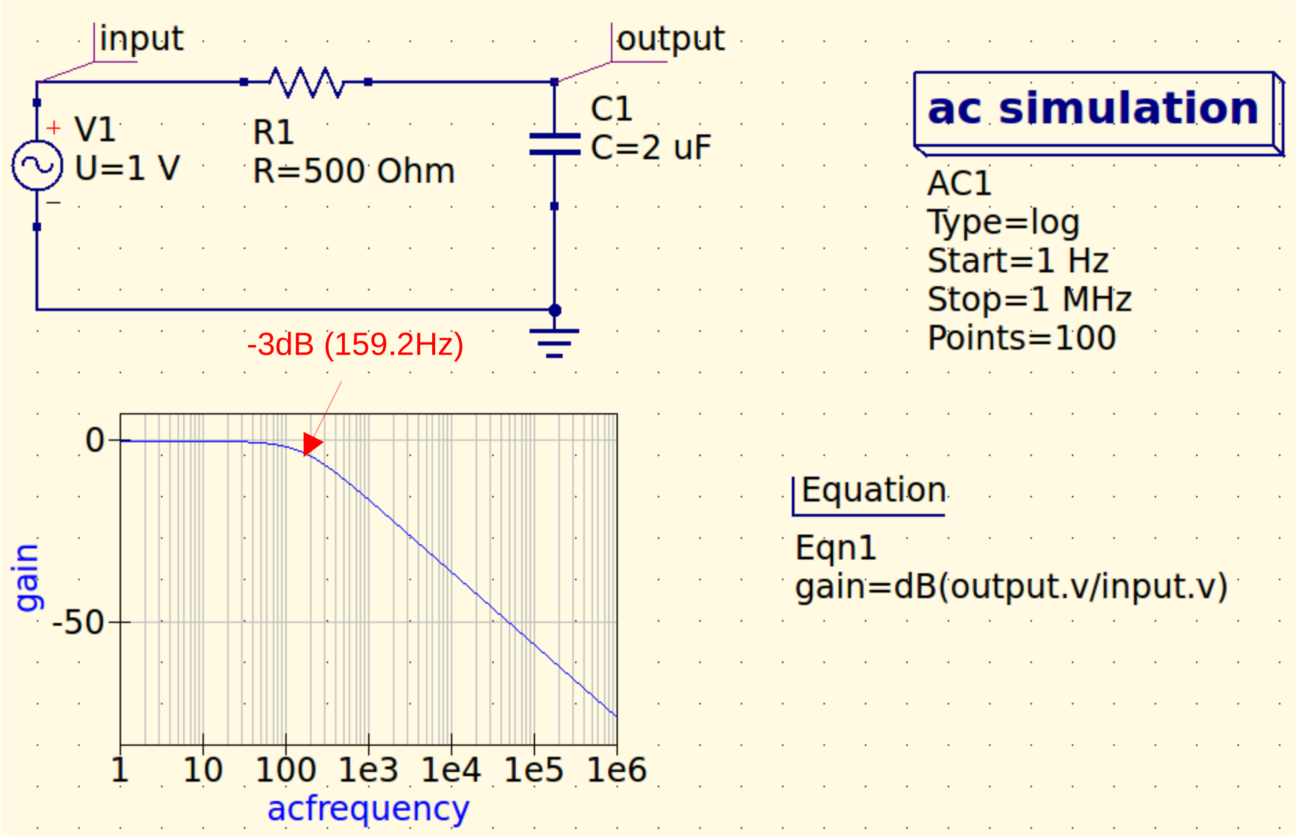
電路分析

- 目的：使用 python 的 pyspice 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 pyspice
        - 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)



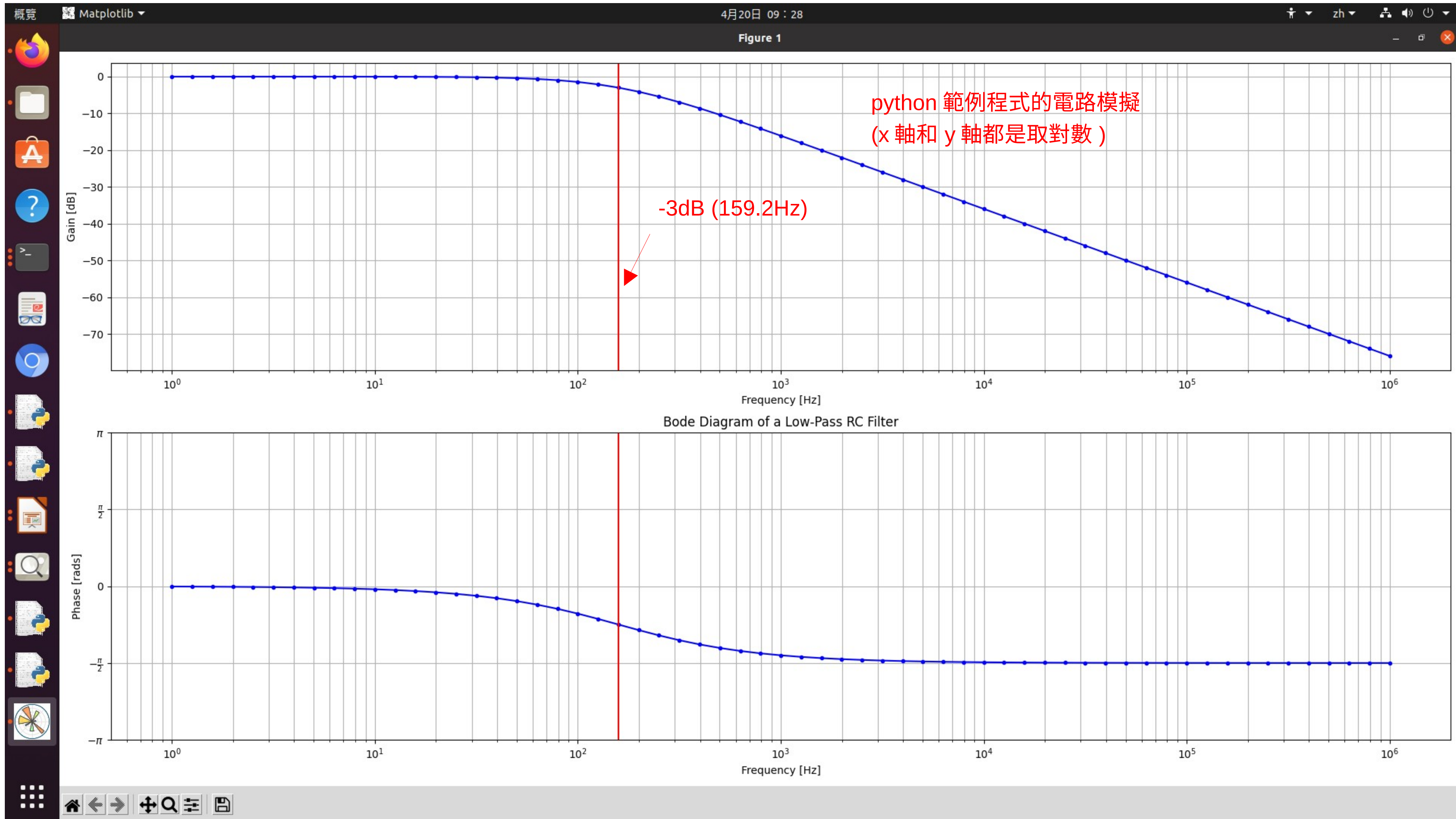
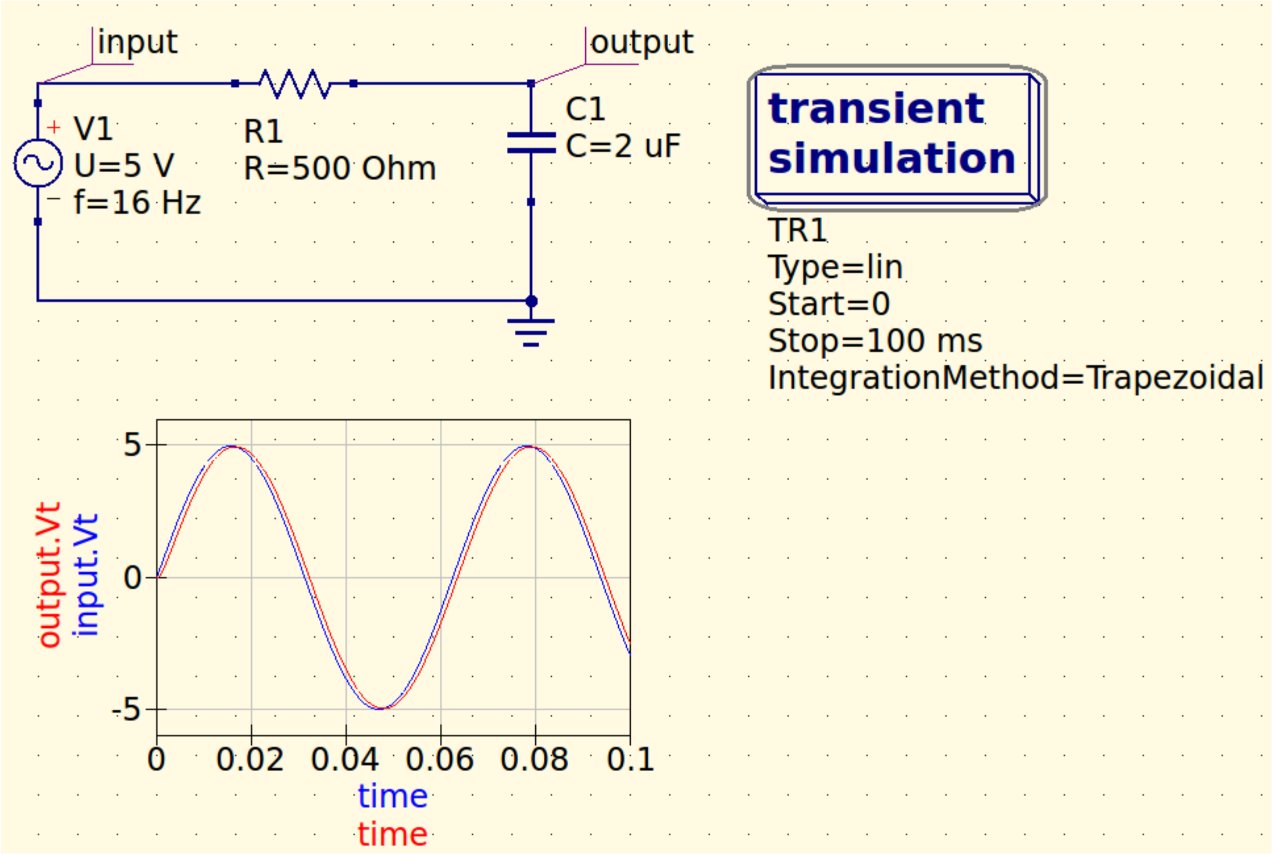
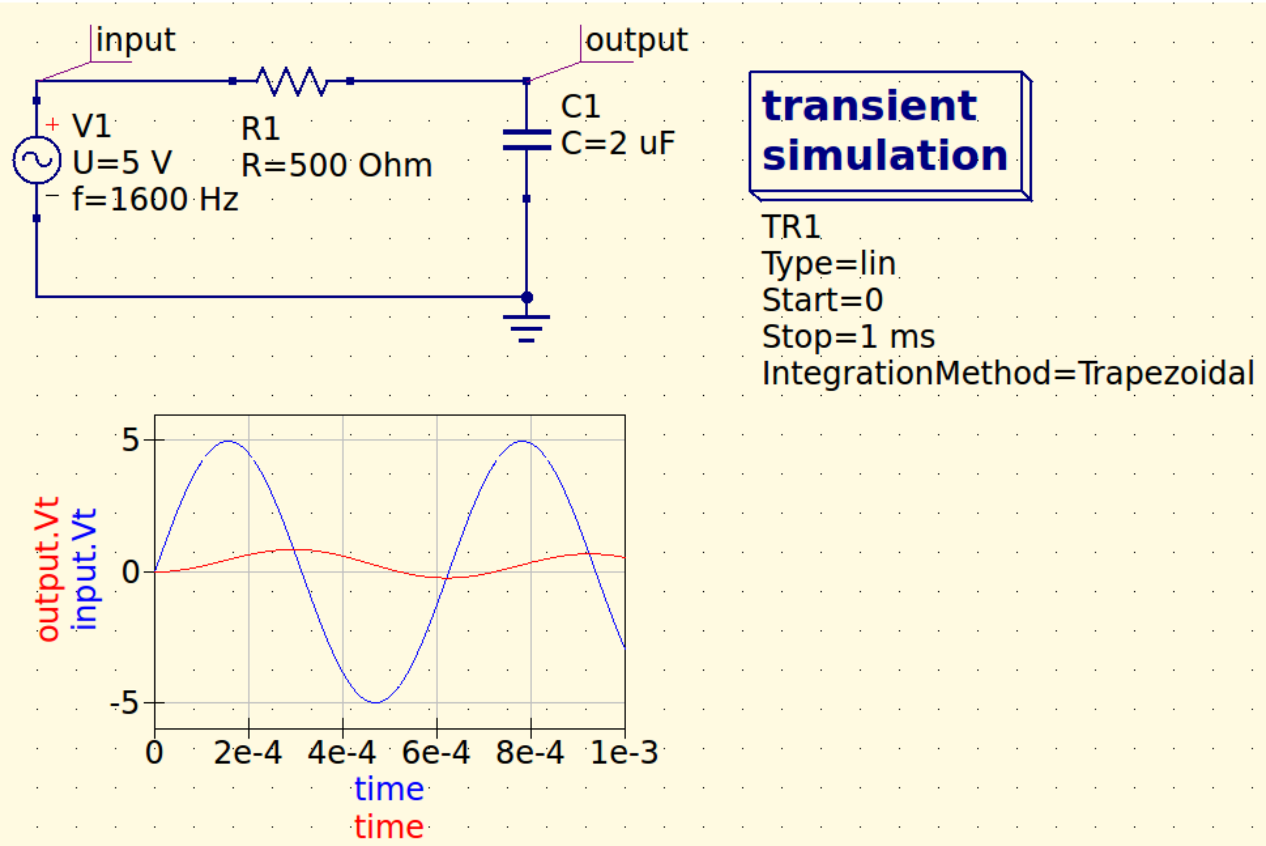


Figure 1
python 範例程式的電路模擬
(x 軸和 y 軸都是取對數)
-3dB (159.2Hz)
Gain [dB]
Frequency [Hz]
Bode Diagram of a Low-Pass RC Filter
Phase [rads]
Frequency [Hz]

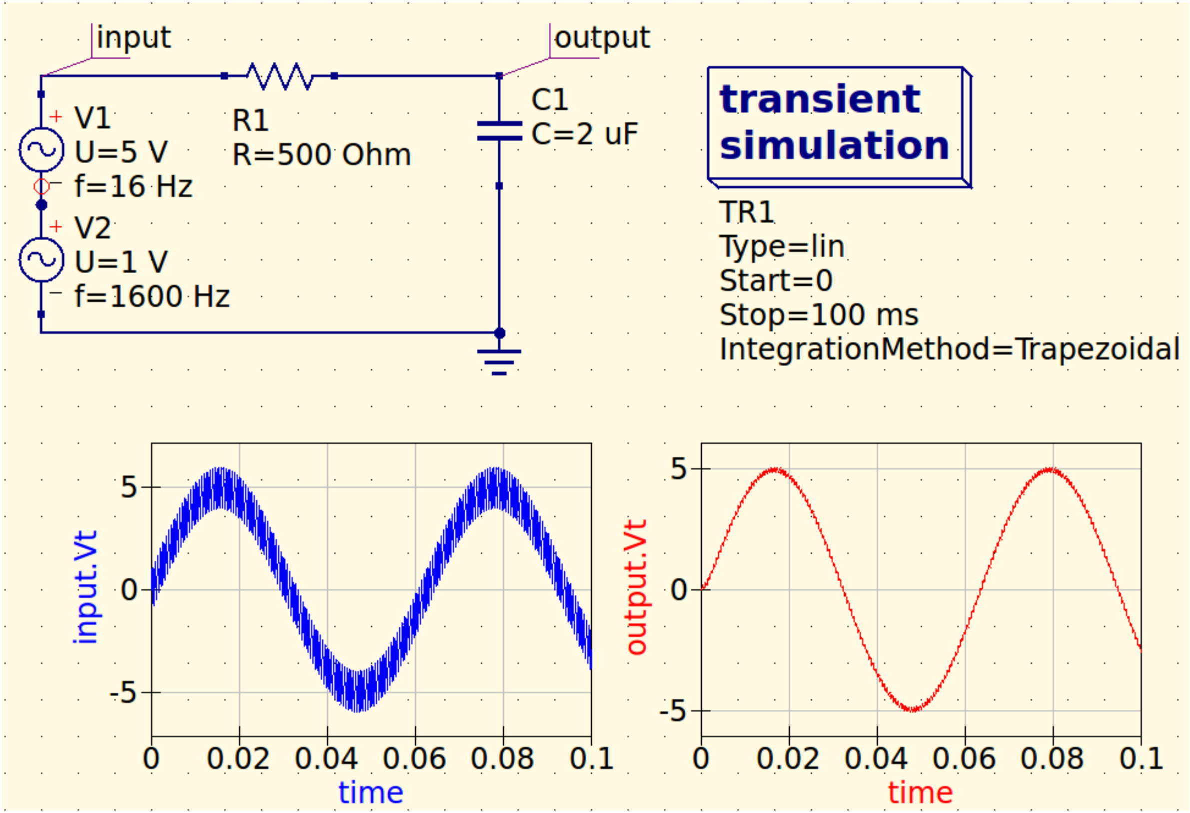
低通濾波測試 A (16Hz 正弦波)



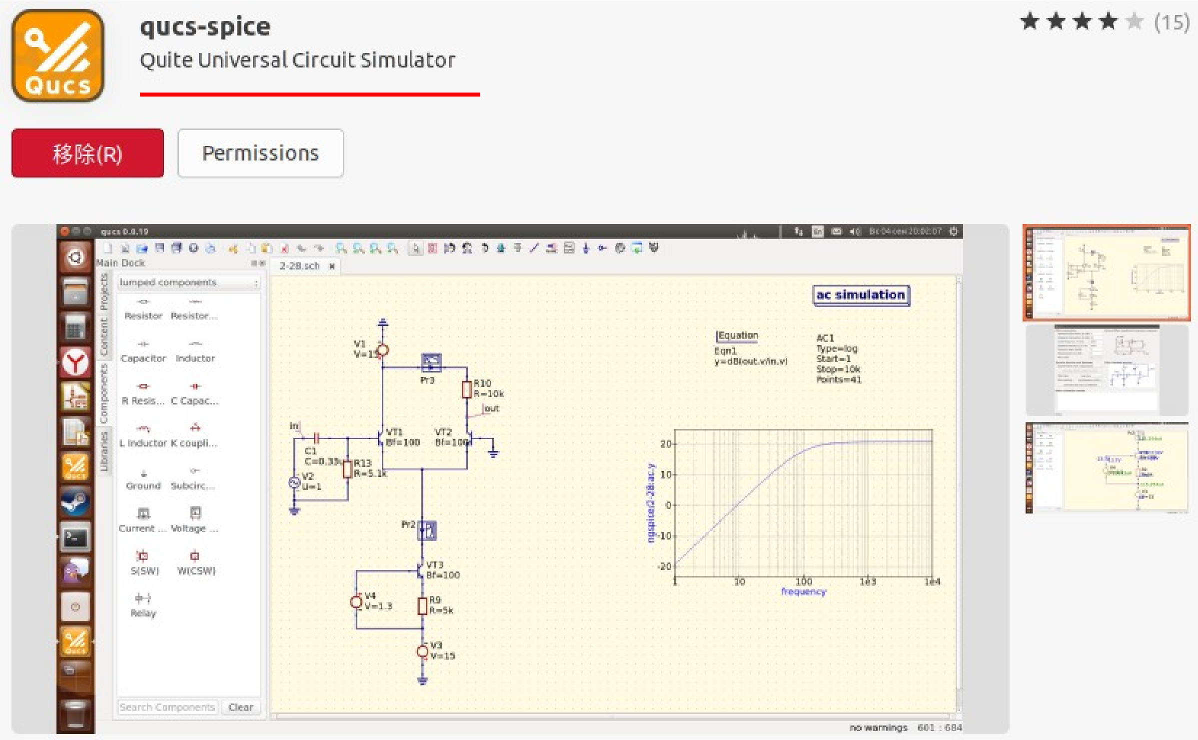
低通濾波測試 B (1600Hz 正弦波)



低通濾波測試 C
(1600Hz + 16Hz 正弦波)



電路模擬軟體 qucs(Linux)



Qucs 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.

pyspice 的範例程式

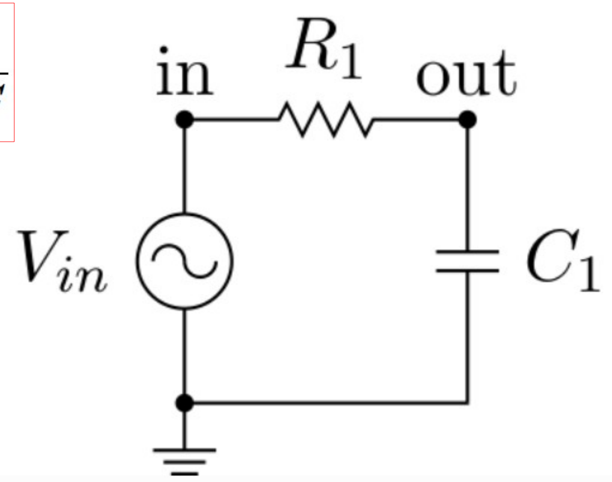
```
import math
import numpy as np
import matplotlib.pyplot as plt


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)
```