類比電子電路實作

ANALOG ELECTRONIC CIRCUIT PRACTICES

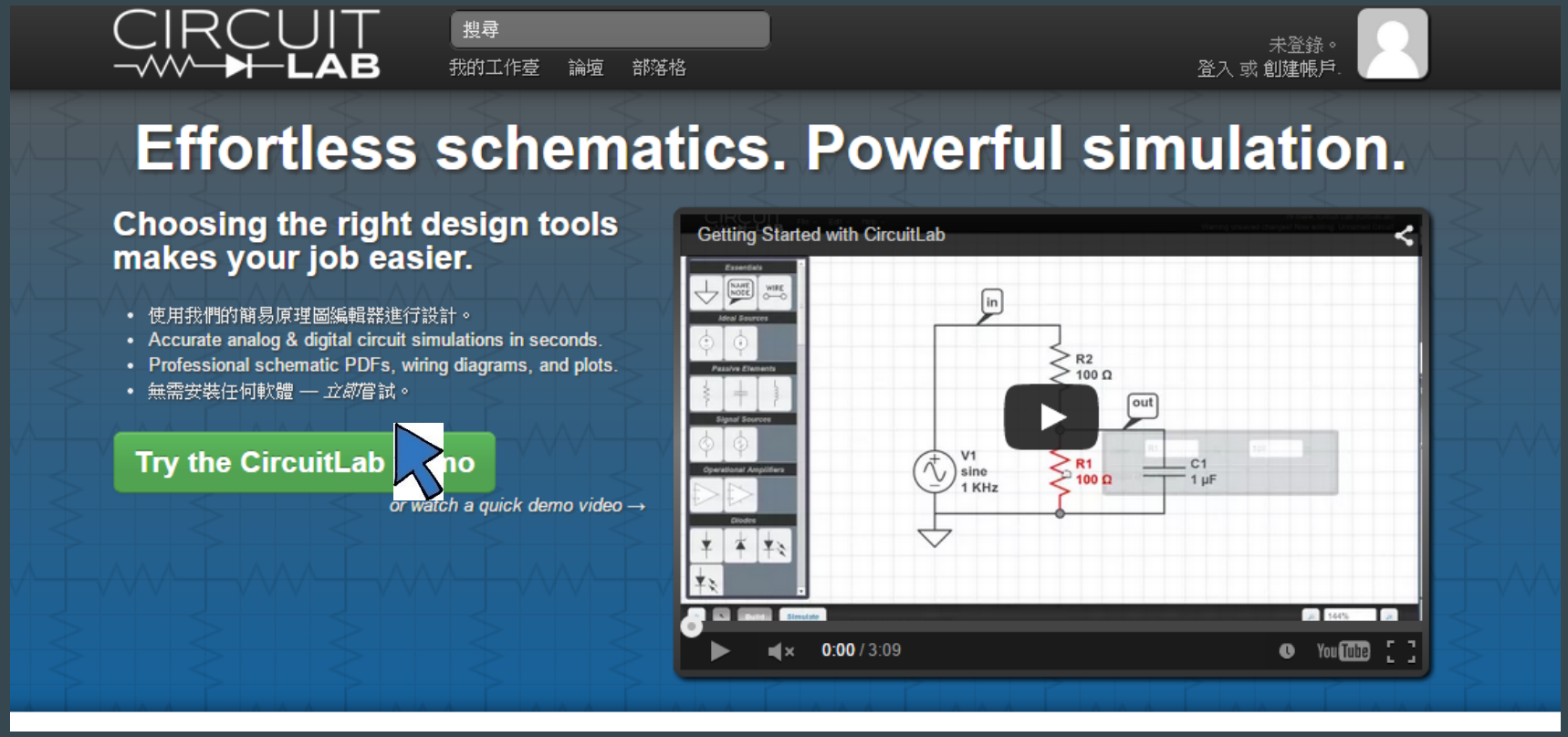
2017 WEEK 1

本周重點

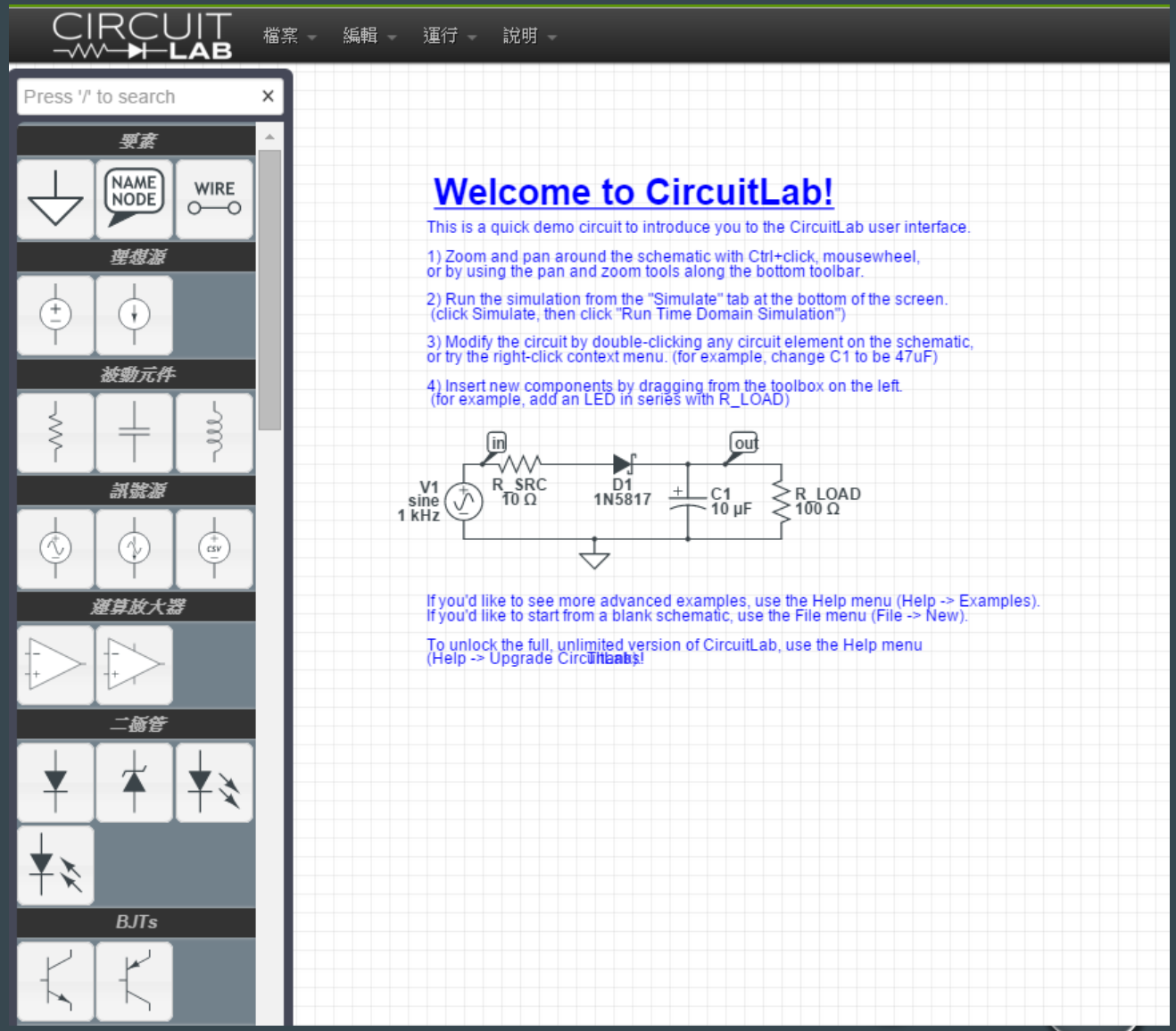
- 了解 Circuit Lab 的基本使用方法
  - 拖拉元件
  - 更改電容值
  - 設立節點
  - 模擬電路

CIRCUIT LAB

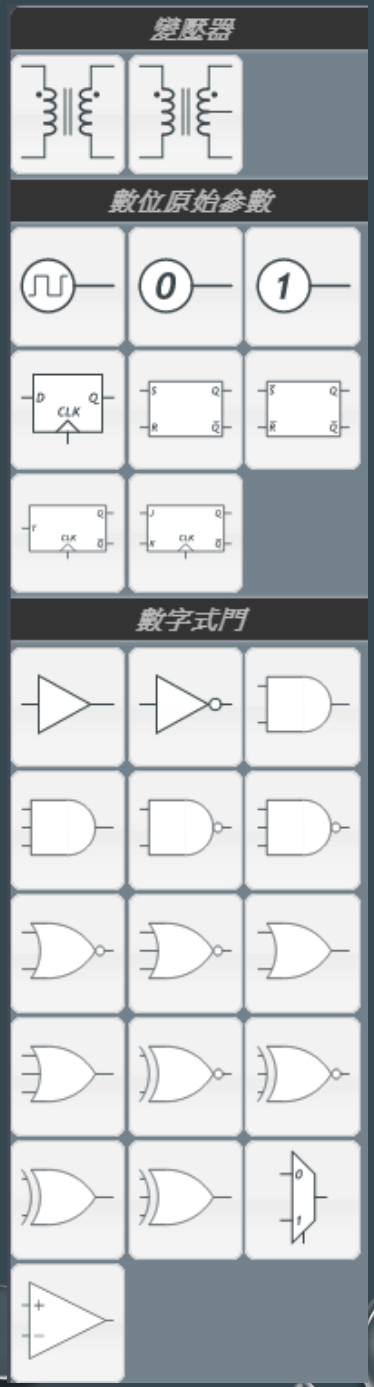
https://www.circuitlab.com/



EDITOR

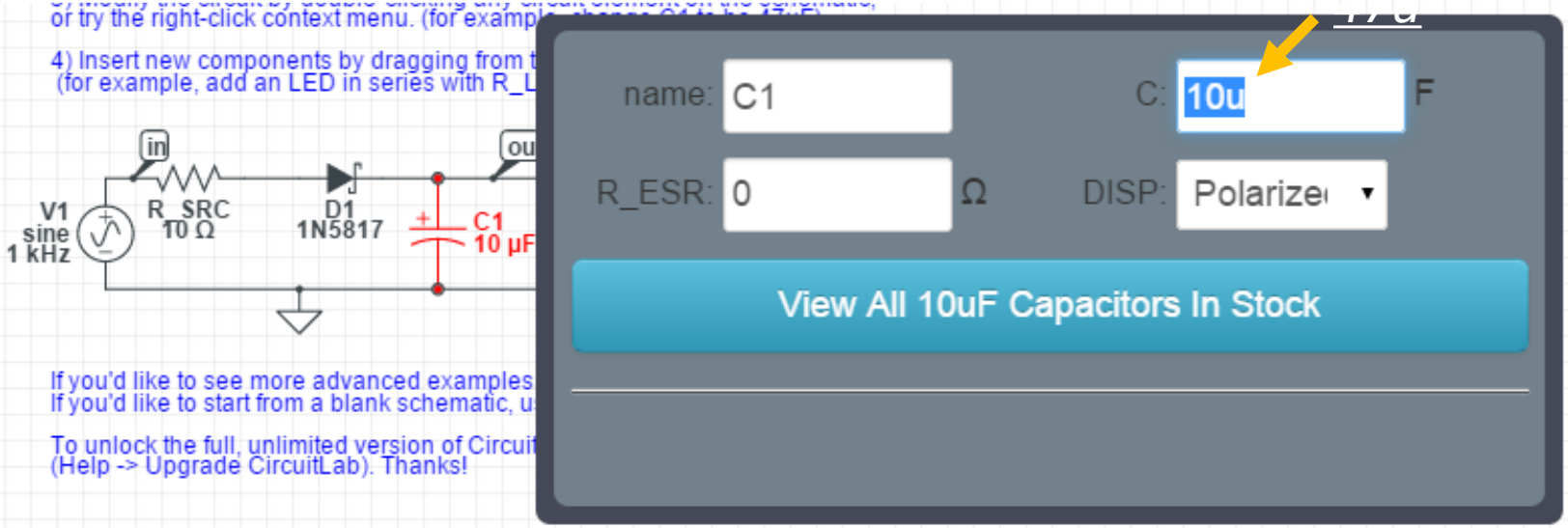

CIRCUIT LAB
檔案
編輯
運行
說明
Press '/' to search
要素
NAME NODE
WIRE
理想源
被動元件
訊號源
運算放大器
二極管
BJTs
Welcome to CircuitLab!
This is a quick demo circuit to introduce you to the CircuitLab user interface.
1) Zoom and pan around the schematic with Ctrl+click, mousewheel, or by using the pan and zoom tools along the bottom toolbar.
2) Run the simulation from the "Simulate" tab at the bottom of the screen. (click Simulate, then click "Run Time Domain Simulation")
3) Modify the circuit by double-clicking any circuit element on the schematic, or try the right-click context menu. (for example, change C1 to be 47uF)
4) Insert new components by dragging from the toolbox on the left. (for example, add an LED in series with R_LOAD)
in
out
V1
sine
1 kHz
R_SRC
10 Ω
D1
1N5817
C1
10 µF
R_LOAD
100 Ω
If you'd like to see more advanced examples, use the Help menu (Help -> Examples).
If you'd like to start from a blank schematic, use the File menu (File -> New).
To unlock the full, unlimited version of CircuitLab, use the Help menu
(Help -> Upgrade CircuitLab!)

AVAILABLE ELEMENTS

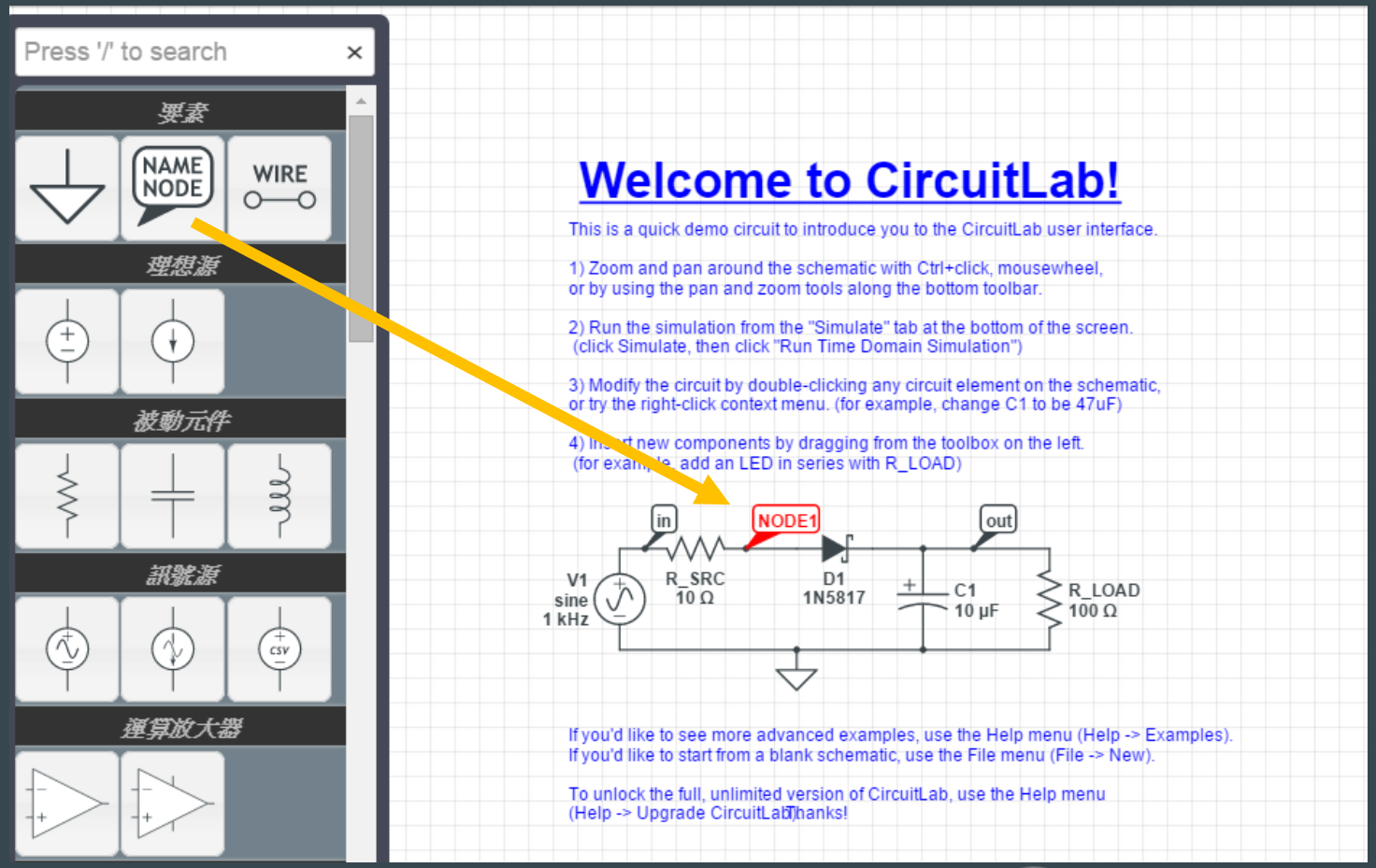


更改電容值

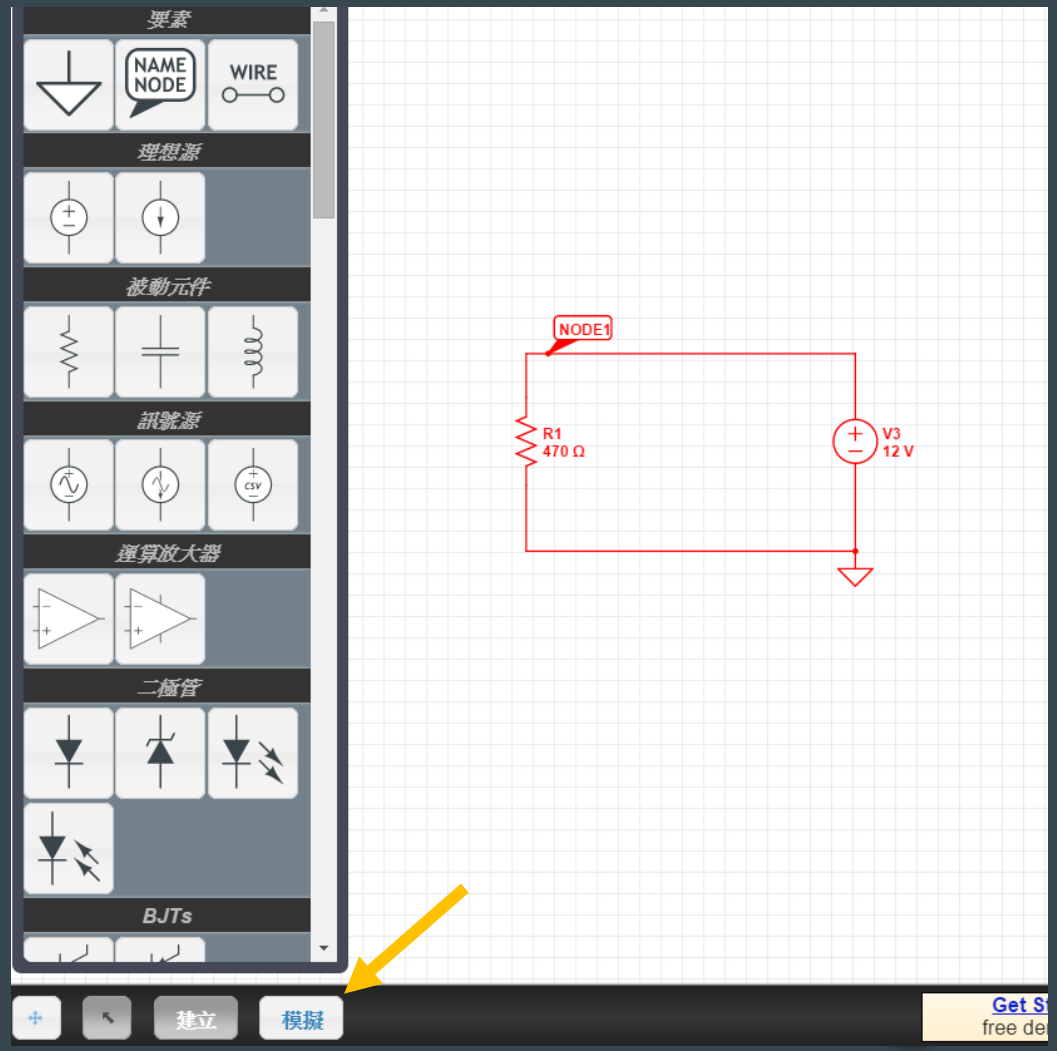
- double click the element



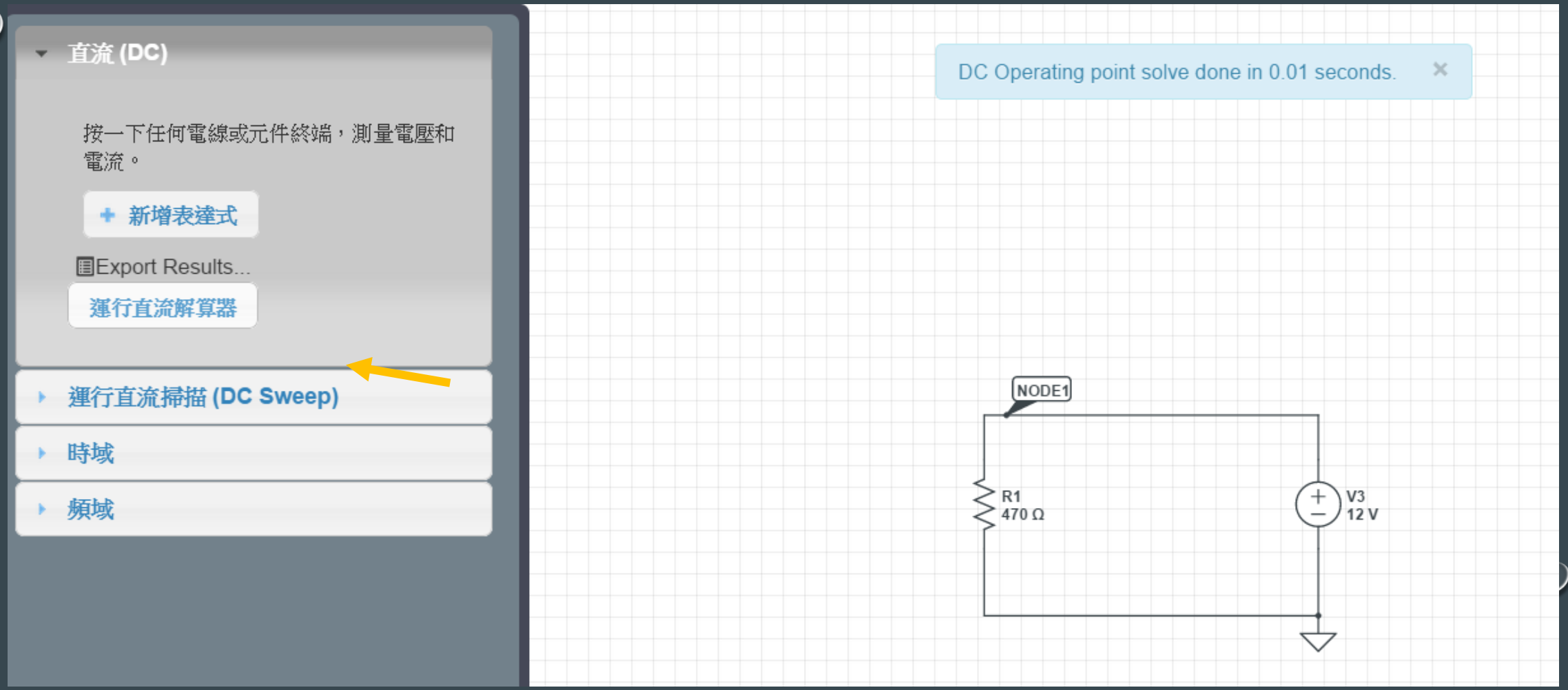
設立節點


Press '/' to search
元素
NAME NODE
WIRE
理想源
被動元件
訊號源
運算放大器
Welcome to CircuitLab!
This is a quick demo circuit to introduce you to the CircuitLab user interface.
1) Zoom and pan around the schematic with Ctrl+click, mousewheel, or by using the pan and zoom tools along the bottom toolbar.
2) Run the simulation from the "Simulate" tab at the bottom of the screen. (click Simulate, then click "Run Time Domain Simulation")
3) Modify the circuit by double-clicking any circuit element on the schematic, or try the right-click context menu. (for example, change C1 to be 47uF)
4) Insert new components by dragging from the toolbox on the left. (for example, add an LED in series with R_LOAD)
in
NODE1
out
V1
sine
1 kHz
R_SRC
10 Ω
D1
1N5817
C1
10 µF
R_LOAD
100 Ω
If you'd like to see more advanced examples, use the Help menu (Help -> Examples).
If you'd like to start from a blank schematic, use the File menu (File -> New).
To unlock the full, unlimited version of CircuitLab, use the Help menu (Help -> Upgrade CircuitLab). Thanks!

SIMULATE


要素
NAME NODE
WIRE
理想源
被動元件
訊號源
運算放大器
二極管
BJTs
NODE1
R1
470 Ω
V3
12 V
建立
模擬
Get St
free de

SIMULATE


直流 (DC)
按一下任何電線或元件終端，測量電壓和電流。
新增表達式
Export Results...
運行直流解算器
運行直流掃描 (DC Sweep)
時域
頻域
DC Operating point solve done in 0.01 seconds.
NODE1
R1
470 Ω
V3
12 V

SIMULATE

- 按下電線或元件終端，得知電壓與電流