Simulazione di circuiti

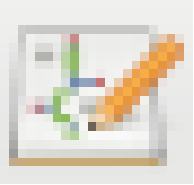
geda-gschem, gSpiceUI, GNU CAP

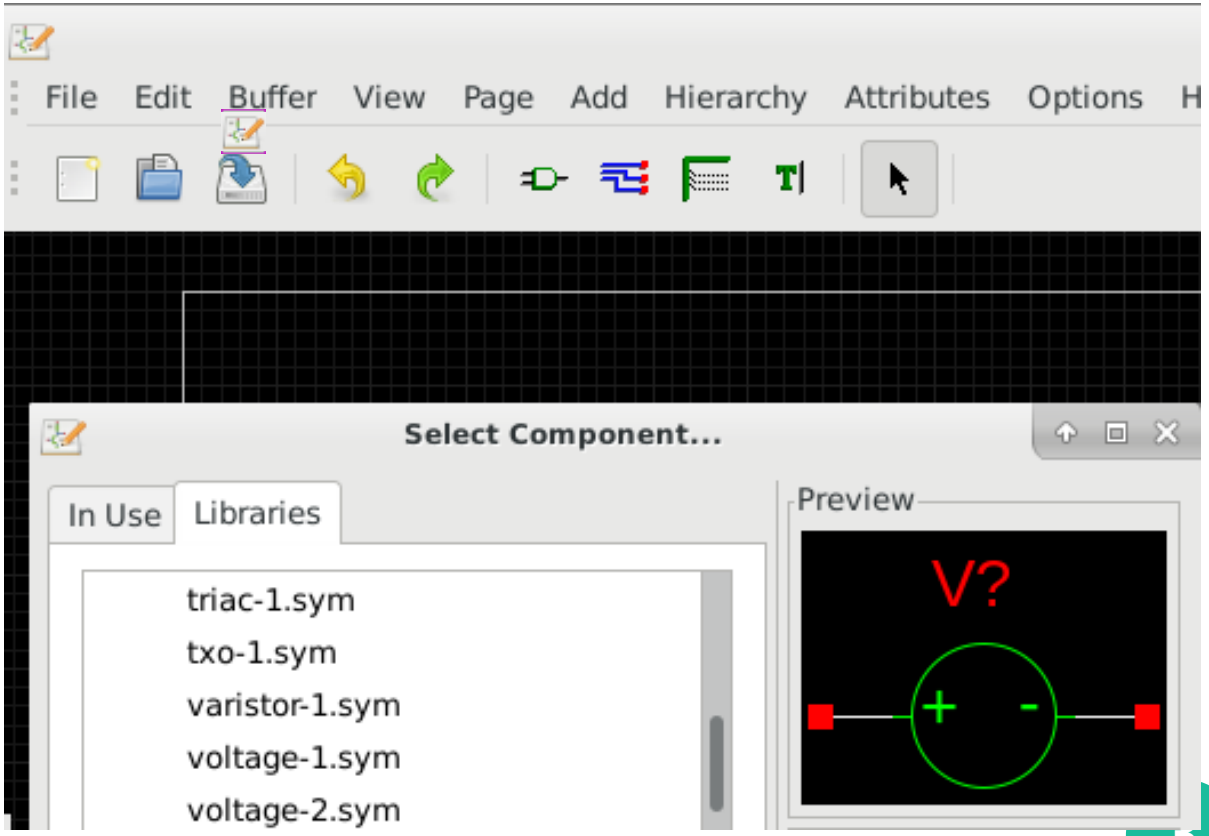
Tre software

- **geda-gschem**
  - per disegnare lo schema logico
- **gSpiceUI**
  - per eseguire la simulazione
- **GNUCAP**
  - è richiesto da gSpiceUI per la simulazione
- **nota: installare i precedenti programmi usando nomi tutti in minuscolo**

gschem

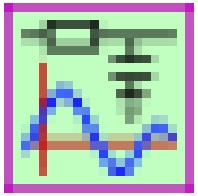
- **gschem**
  - per disegnare lo schema logico

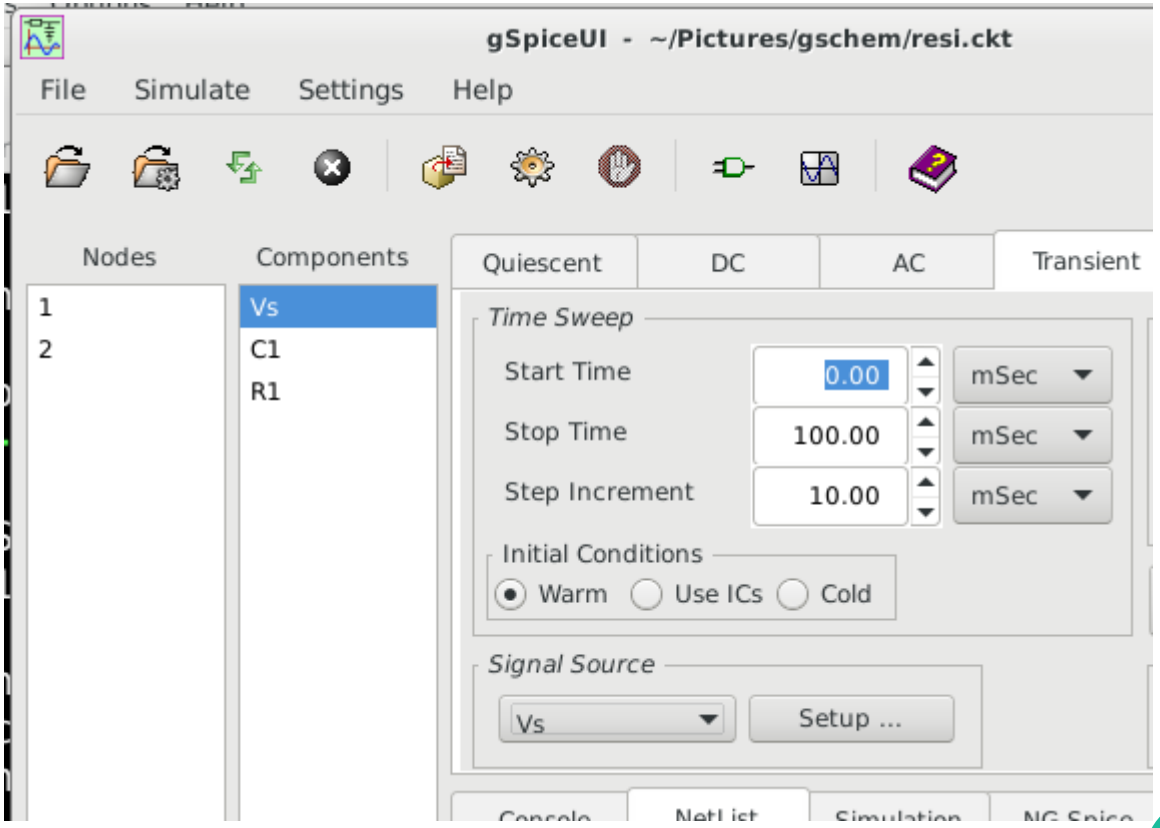




gSpiceUI

- **gSpiceUI**
  - per eseguire la simulazione (usando GNUCAP)



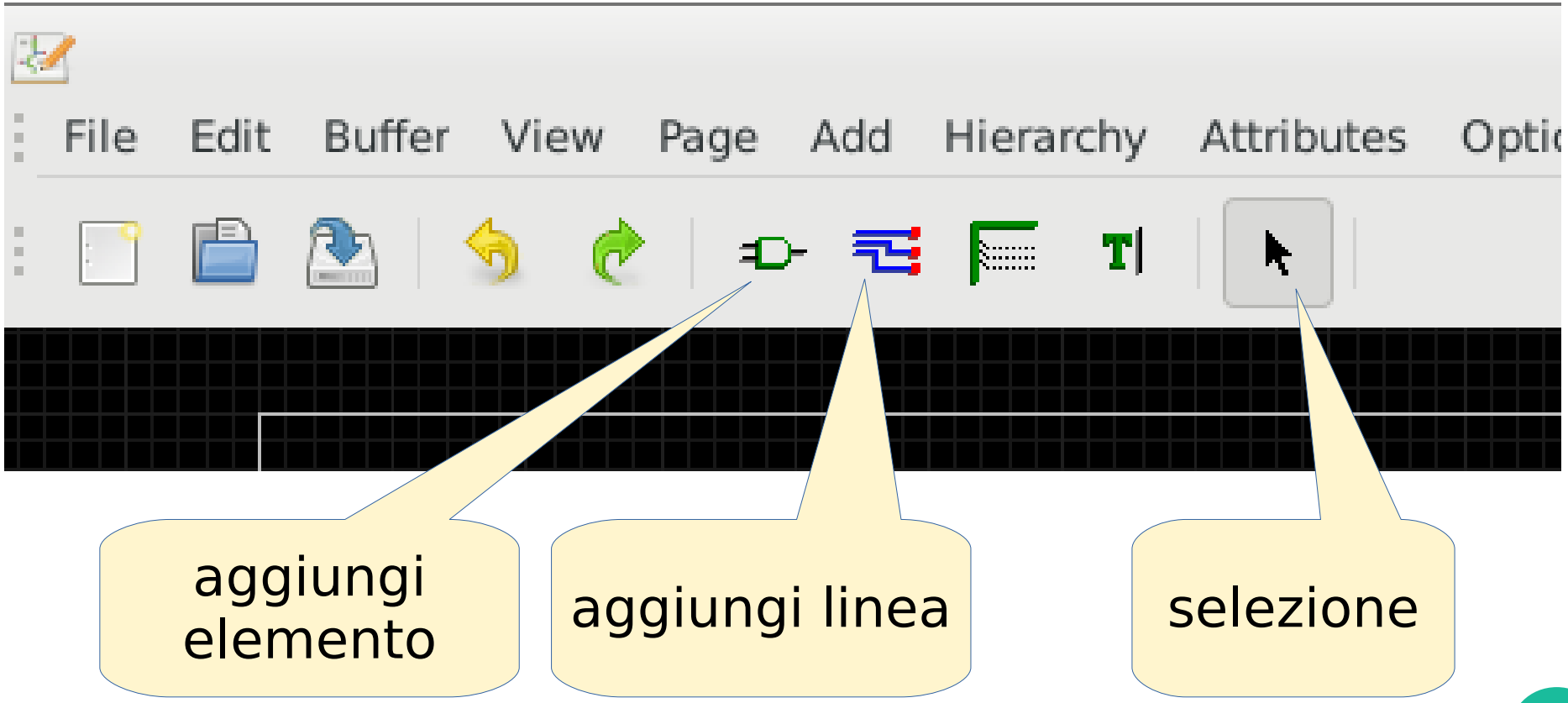


Esempio d’uso

Lo schema

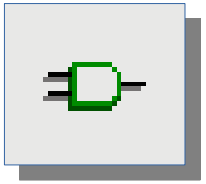
gschem (Strumenti)

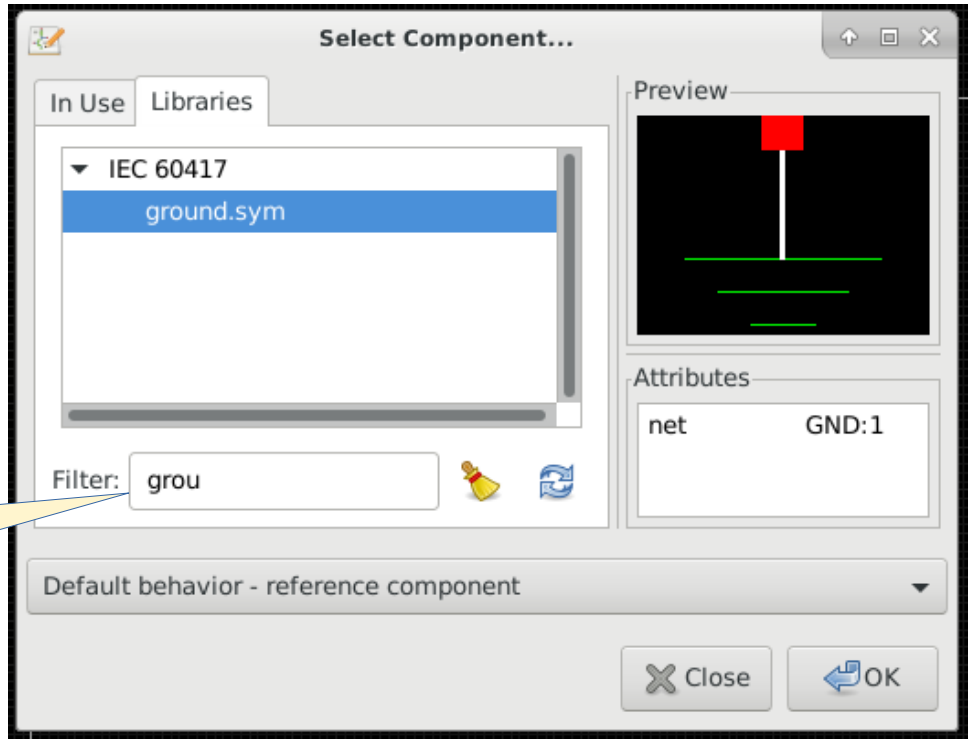
- **gschem si può avviare dal terminale**



Add component

- **Strumento:**
  - per trovare un particolare componente si può usare il filtro
  - digitare ‘ground’ nel filtro

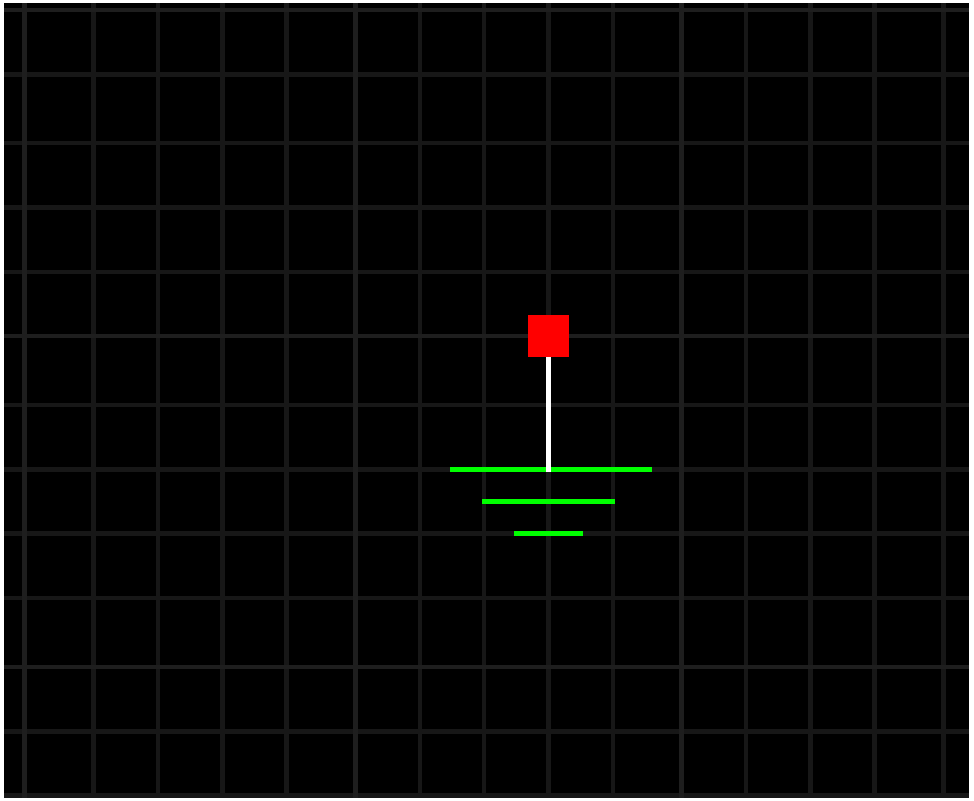




filtro

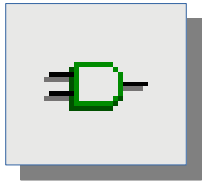
Zoom

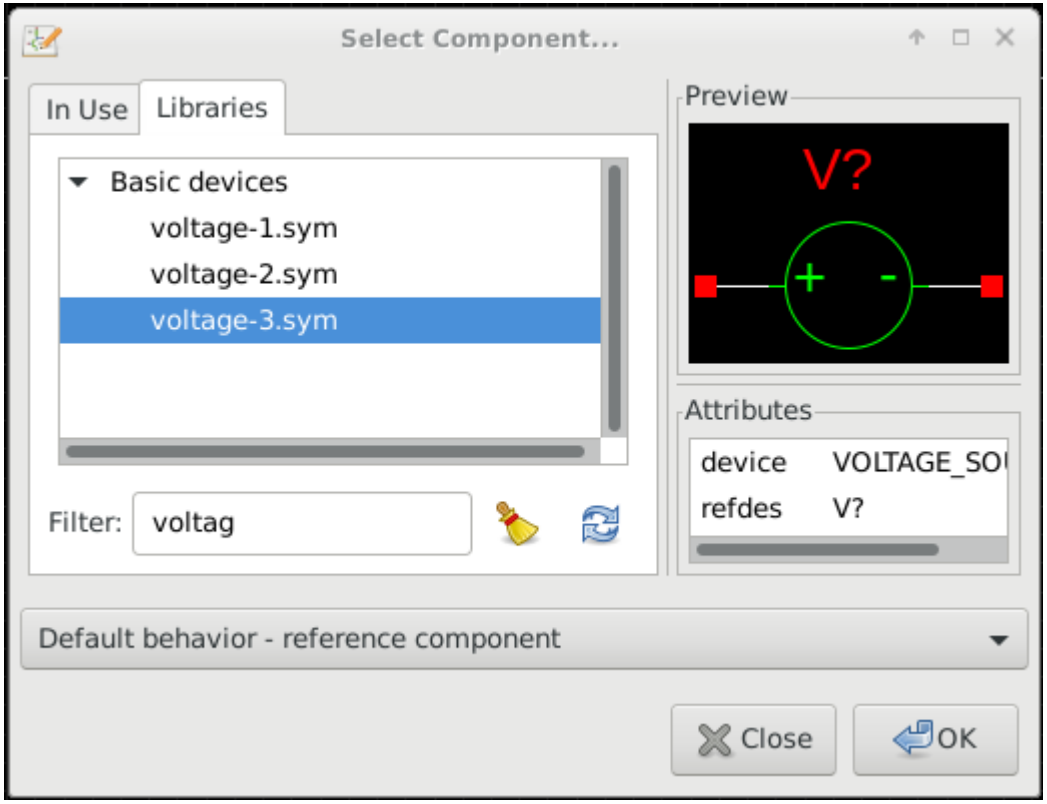
- **Per eseguire lo zoom**
  - ruotare in avanti la rotella del mouse



Voltage source

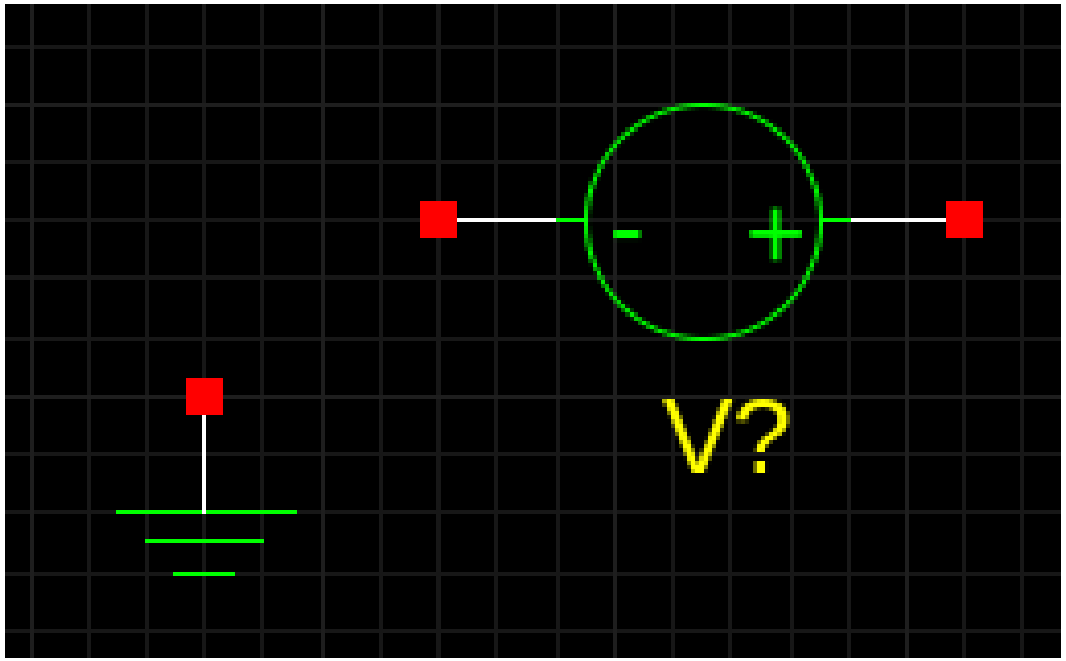
- **Componenti:**
  - Cercare nel filtro anche "voltage"





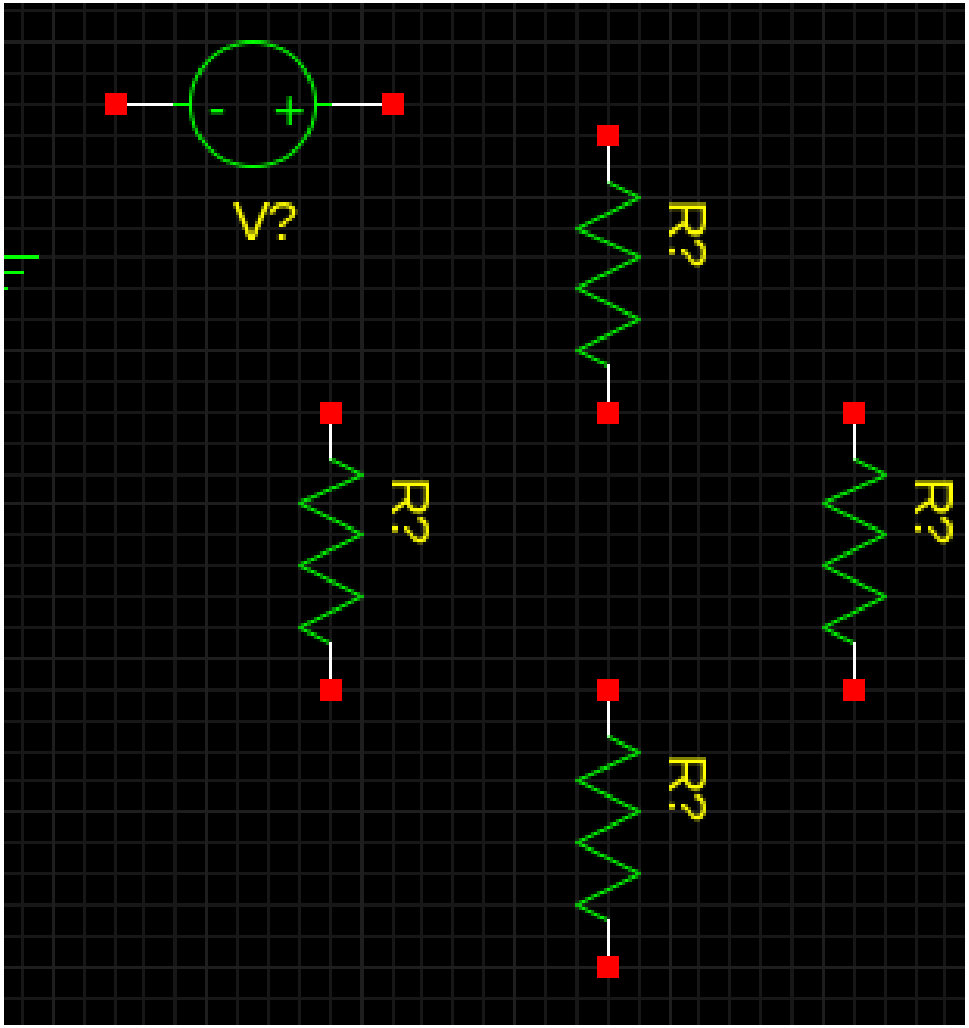
Rotazione

- **Quando è necessario ruotare un elemento**
  - prima ancora di fissarlo
  - Fare due click sulla rotella del mouse,

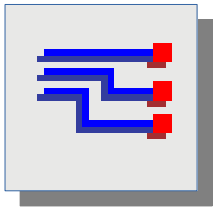


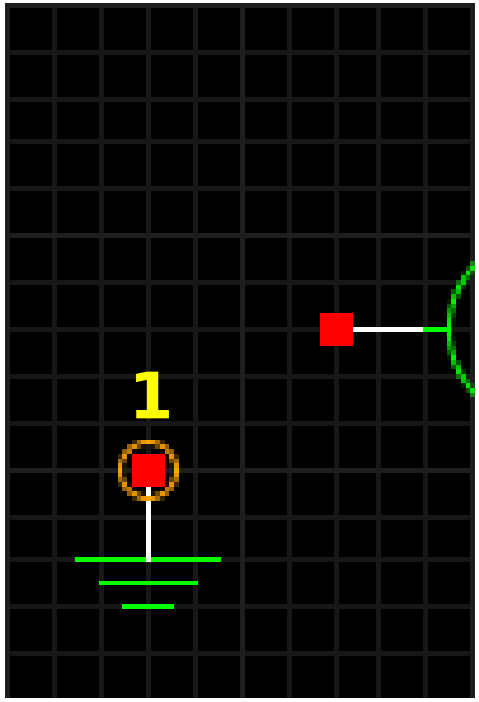
Resistor

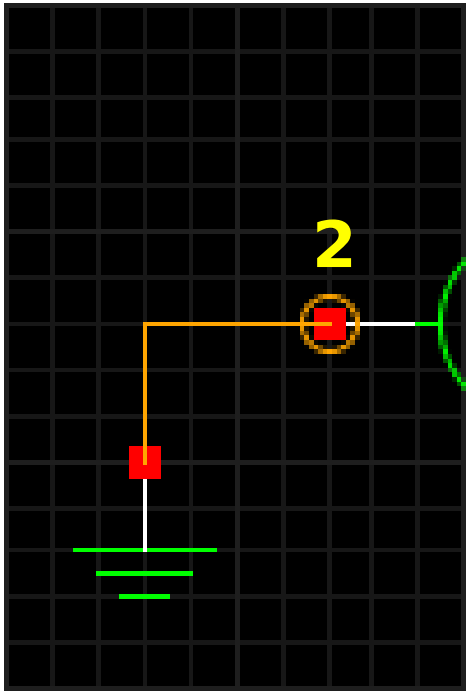
- **Cercare "resistor" nel solito filtro**
- **Per ruotarlo**
  - Cliccare tre volte la rotella del mouse

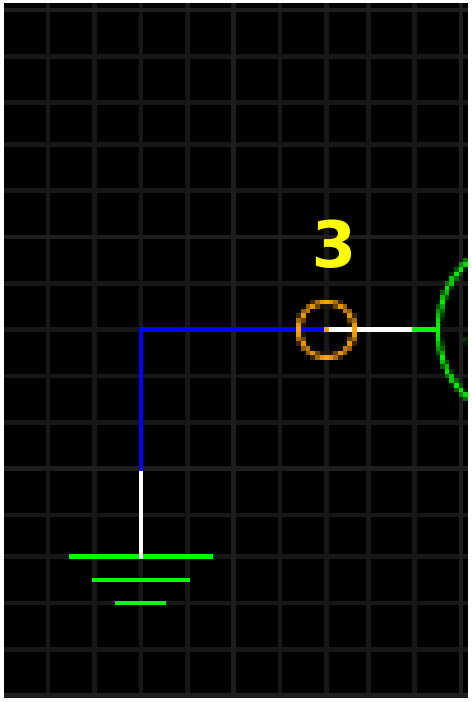


Add nets mode

- **strumento linee:** 
- **eseguire poi una sequenza di tre click**
  - 1.sinistro, 2.sinistro, 3.destro

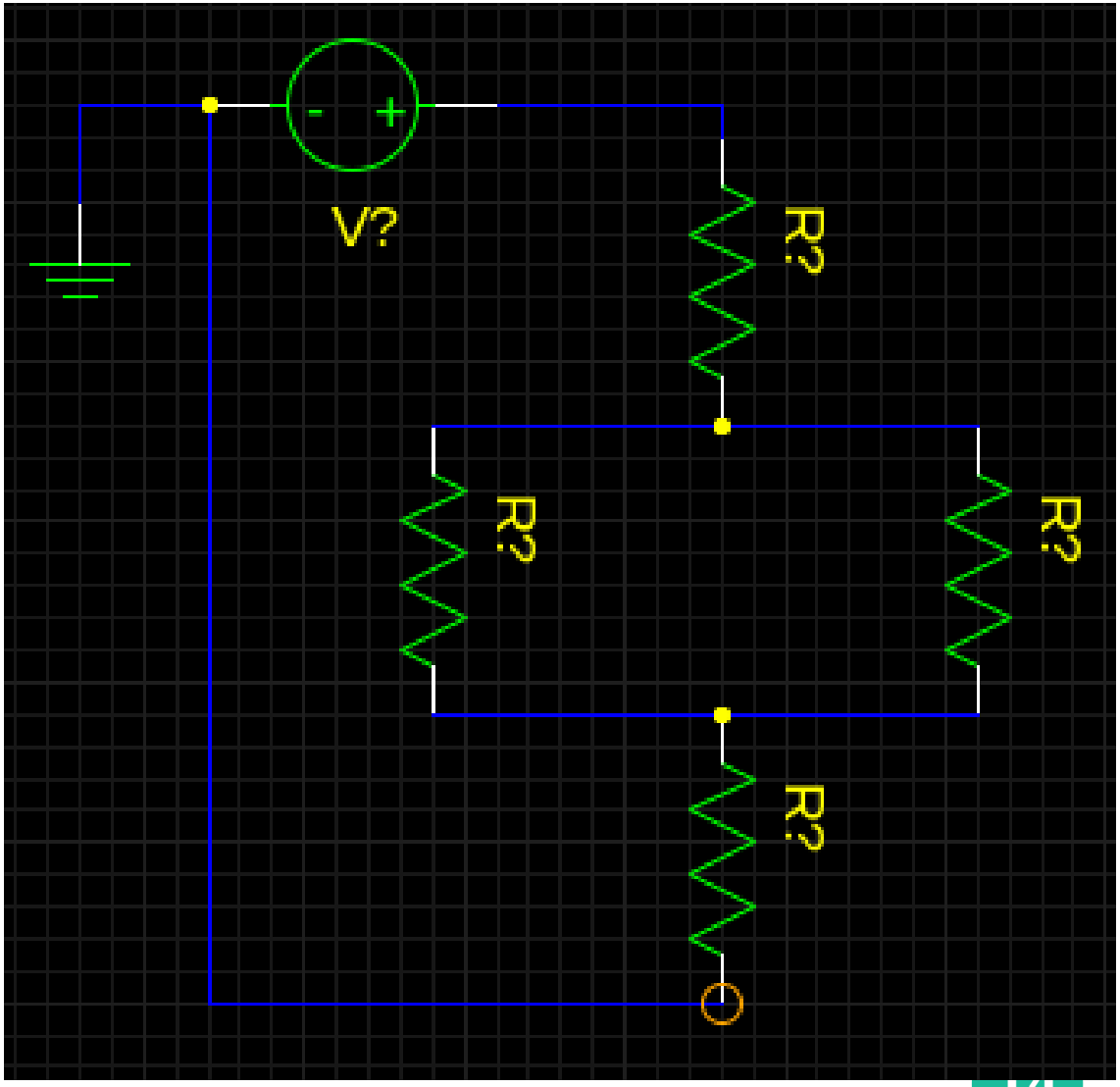






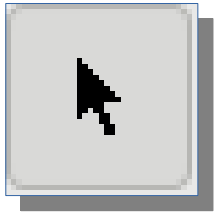
Add nets mode

- **risultato**



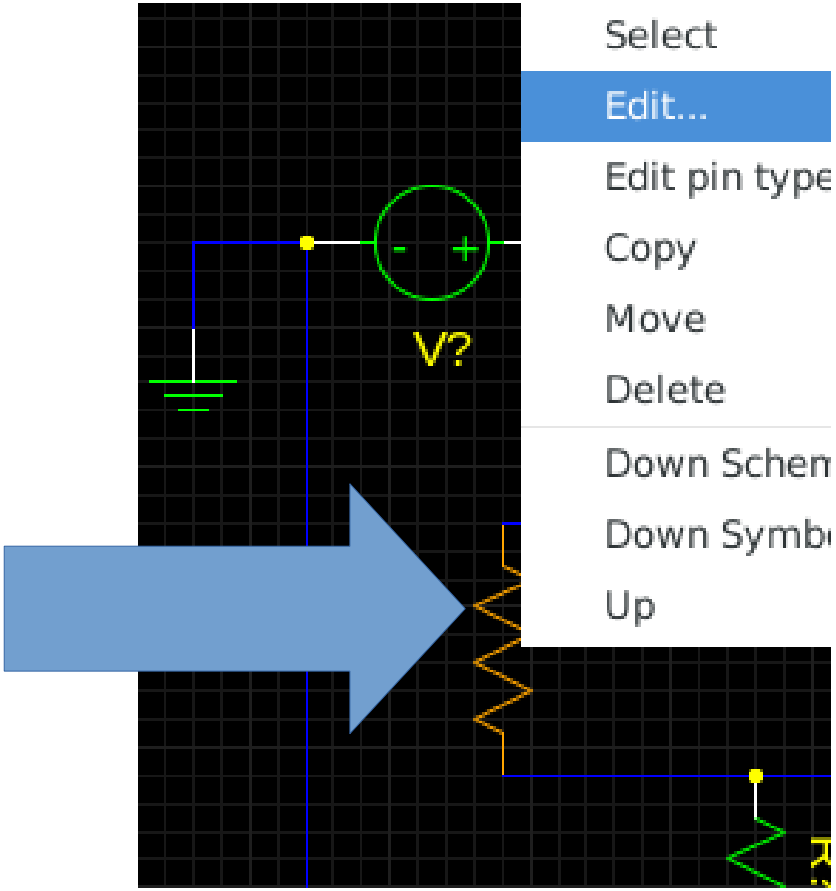
Select mode

- **Strumento**
  - modificare il **nome** e il **valore** di una resistenza



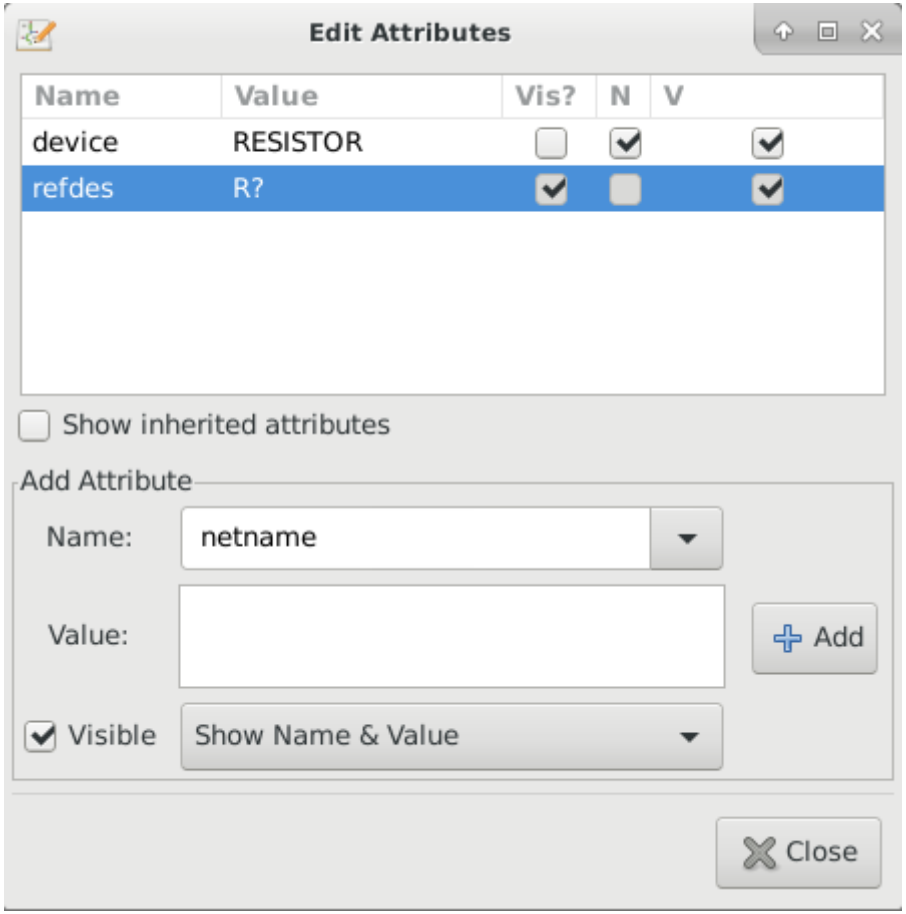
Edit…

- Usare il tasto destro sopra l’elemento
- menù Edit…



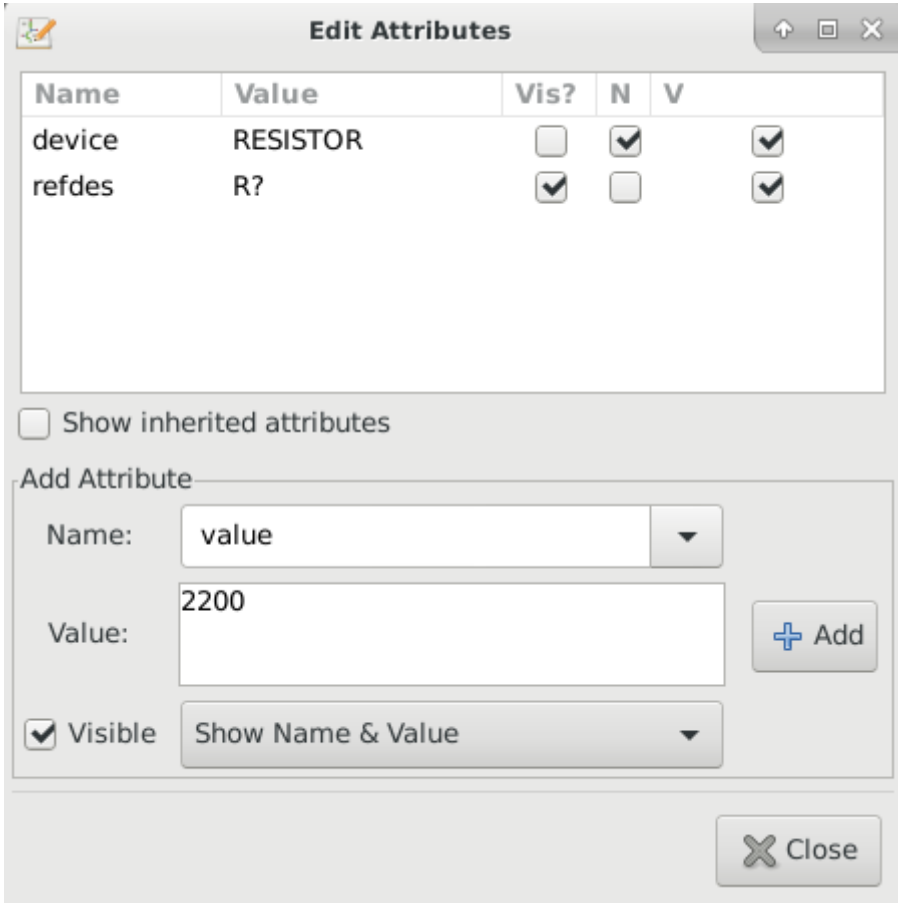
Edit...

- **in alto**
  - Modificare il nome ‘R?’ con doppio click
  - scrivere ‘R2’
  - premere ‘invio’



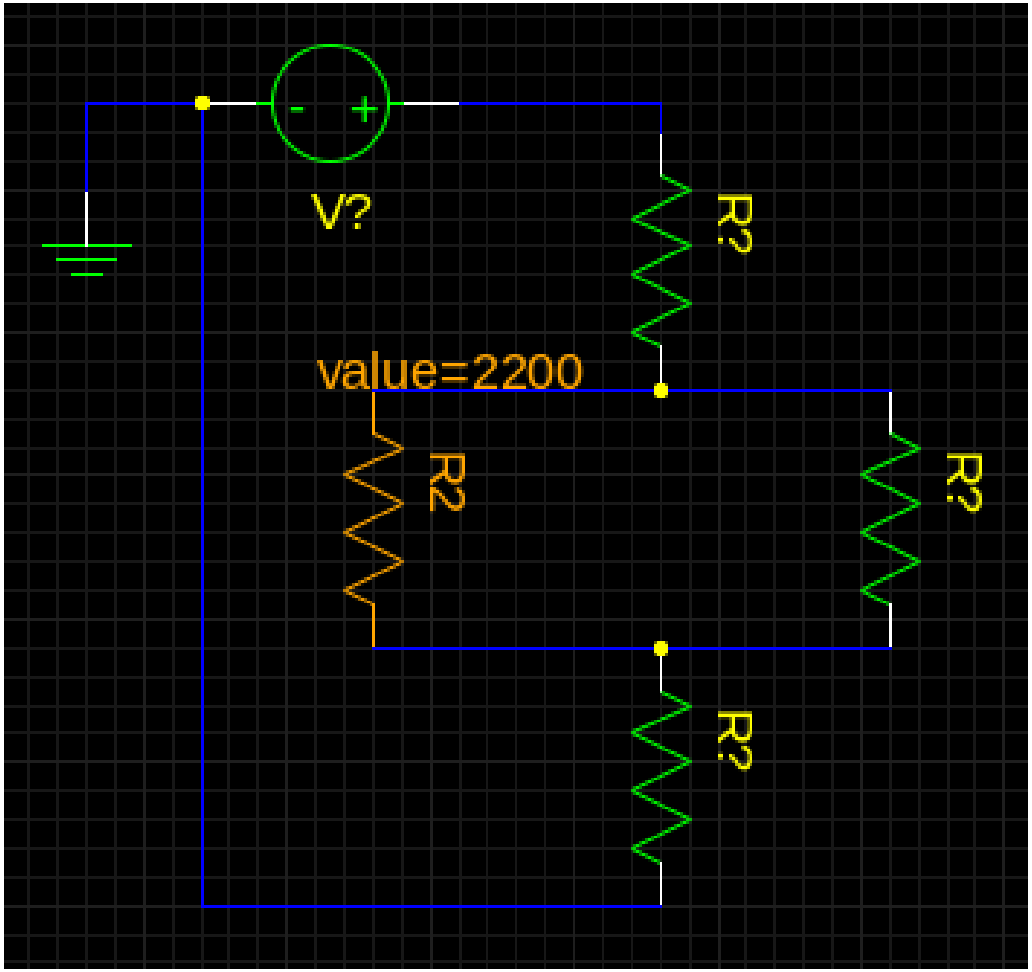
Edit...

- **in basso**
  - Selezionare nel menù a tendina Name: ‘value’
  - Digitare 2200
  - Premere ‘Add’
- **chiudere**



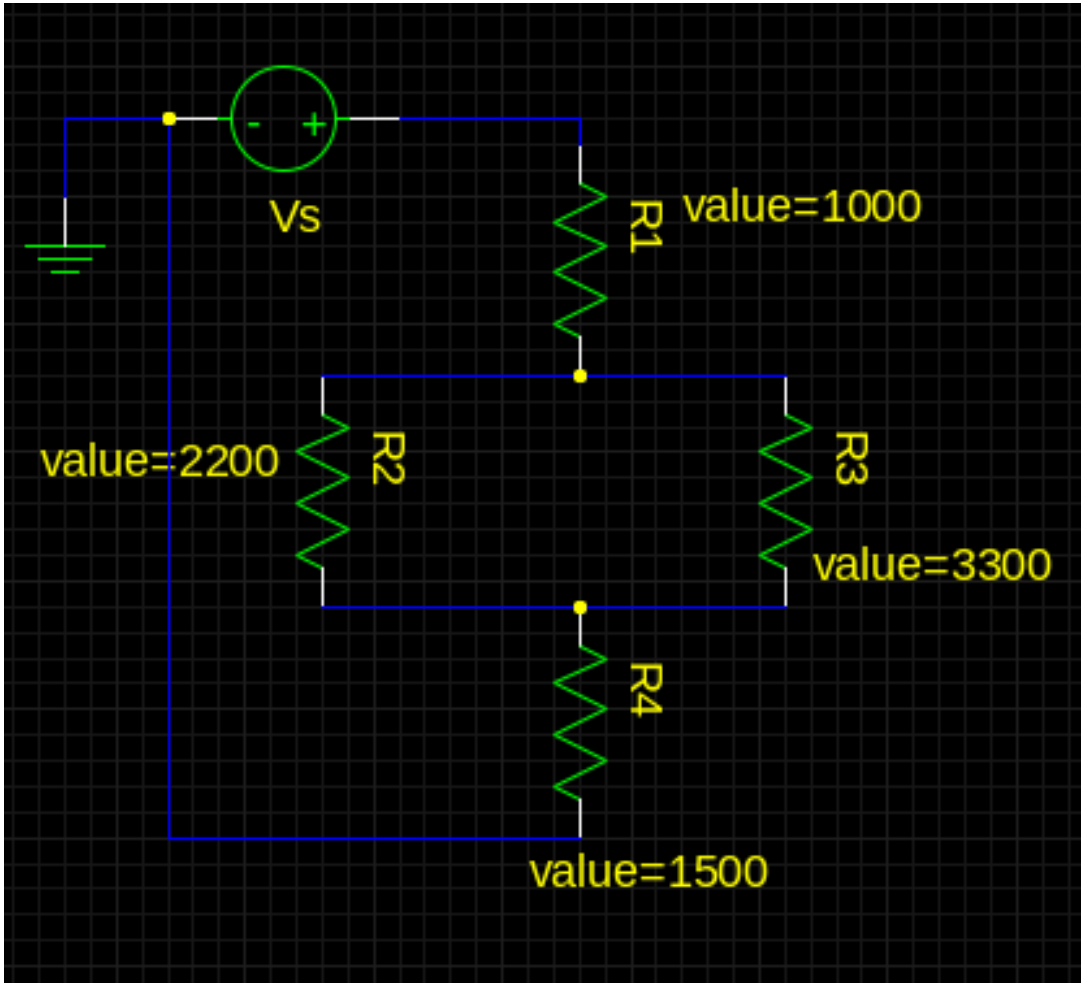
Edit...

- **risultato**
  - è possibile spostare il testo ‘value’
  - cliccare (su ‘value’) una sola volta
  - trascinare la parola ‘value’ dove si preferisce



Edit...

- **Risultato**
  - in questo caso la scelta di ‘value’ su ‘Vs’ non è importante
  - Il generatore sarà modificato dopo, durante la simulazione, usando gSpiceUI



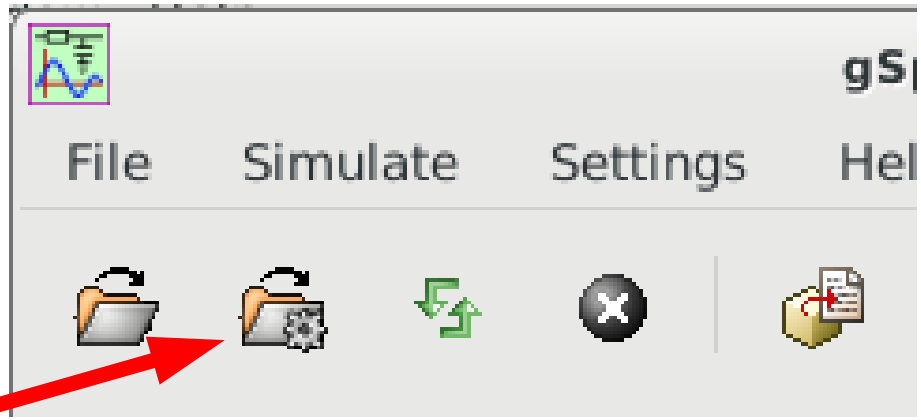
salvare

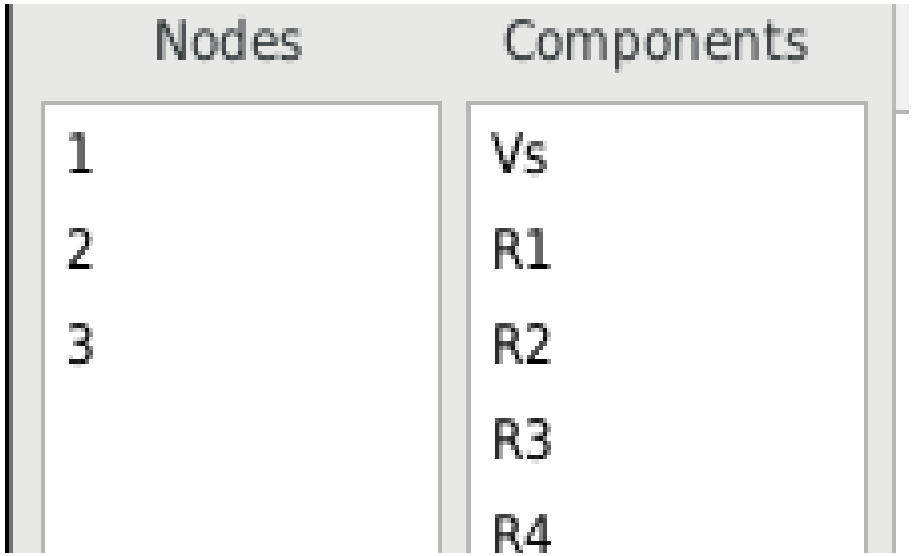
- **salvare il proprio schema sul Desktop**
  - esempio.sch

La simulazione

gSpiceUI

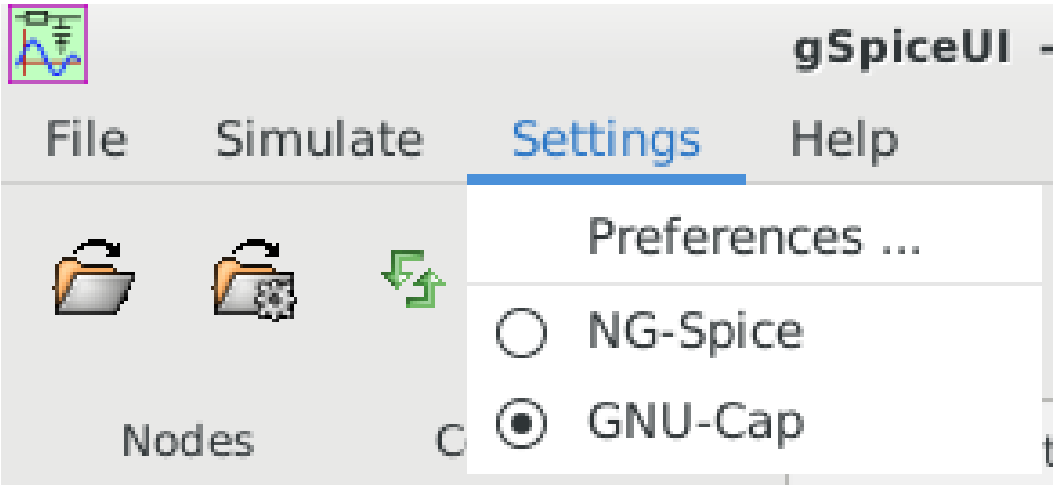
- **In un nuovo terminale, digitare**
  - **`gspiceui`**
- **Usare la seconda icona per aprire nel Desktop esempio.sch**
- **Ora si vedrà apparire l'elenco dei componenti**





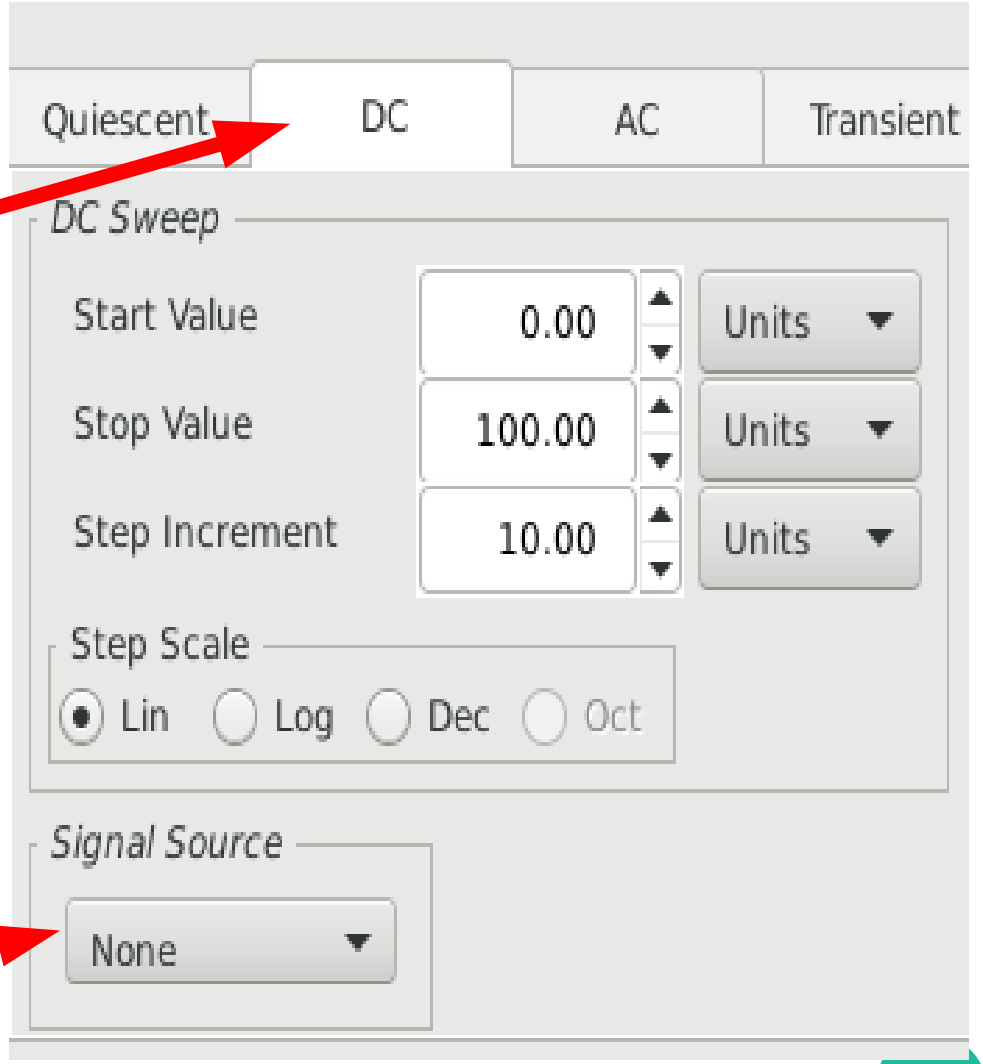
GNUCAP

- **Nel menù Setting si dovrebbe trovare selezionato un software simulatore**
- **GNUCAP è un software libero**



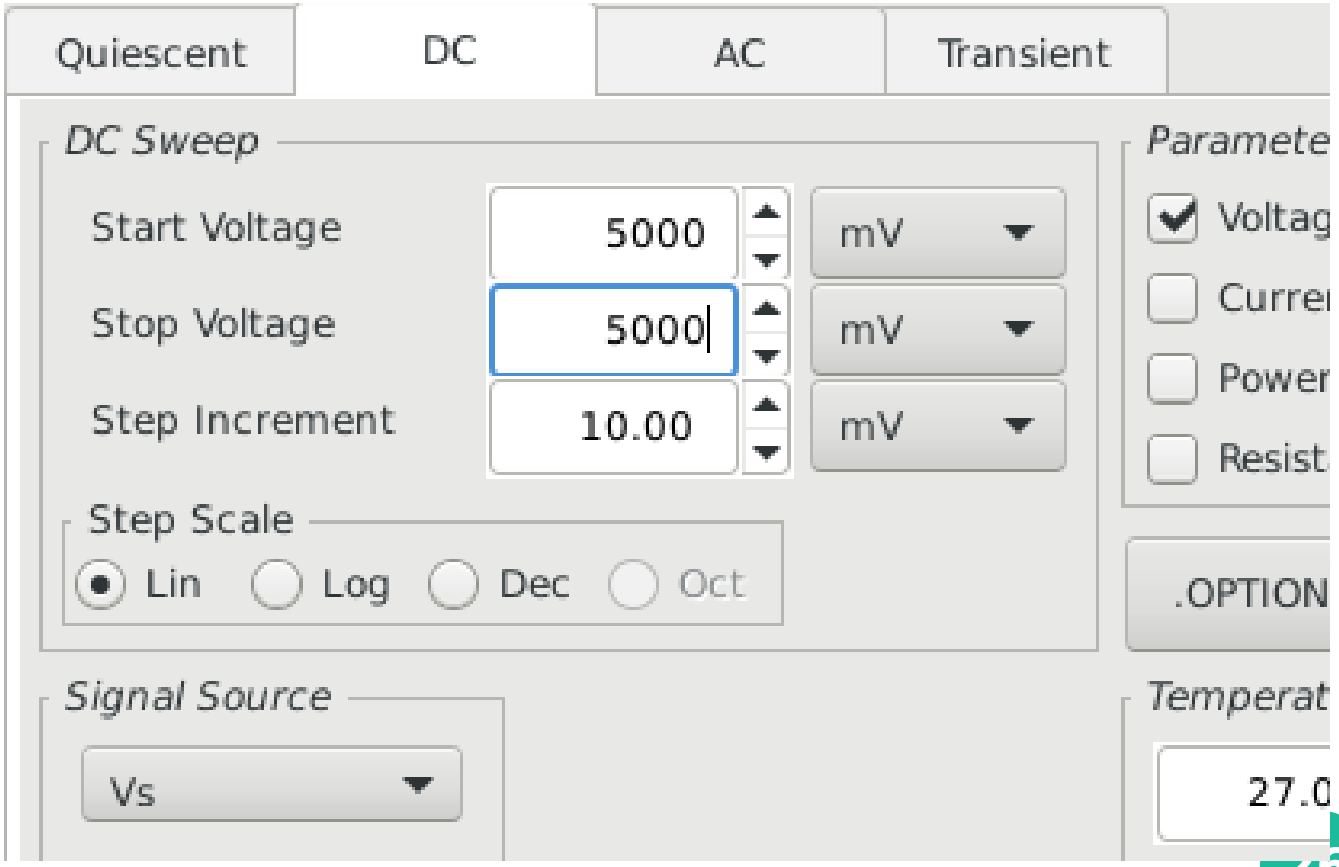
DC Sweep

- **DC Sweep**
  - è un tipo di simulazione
  - Permette di usare un generatore di tensione continua
- **in basso si deve selezionare 'Signal Source'**
  - scegliere 'Vs'



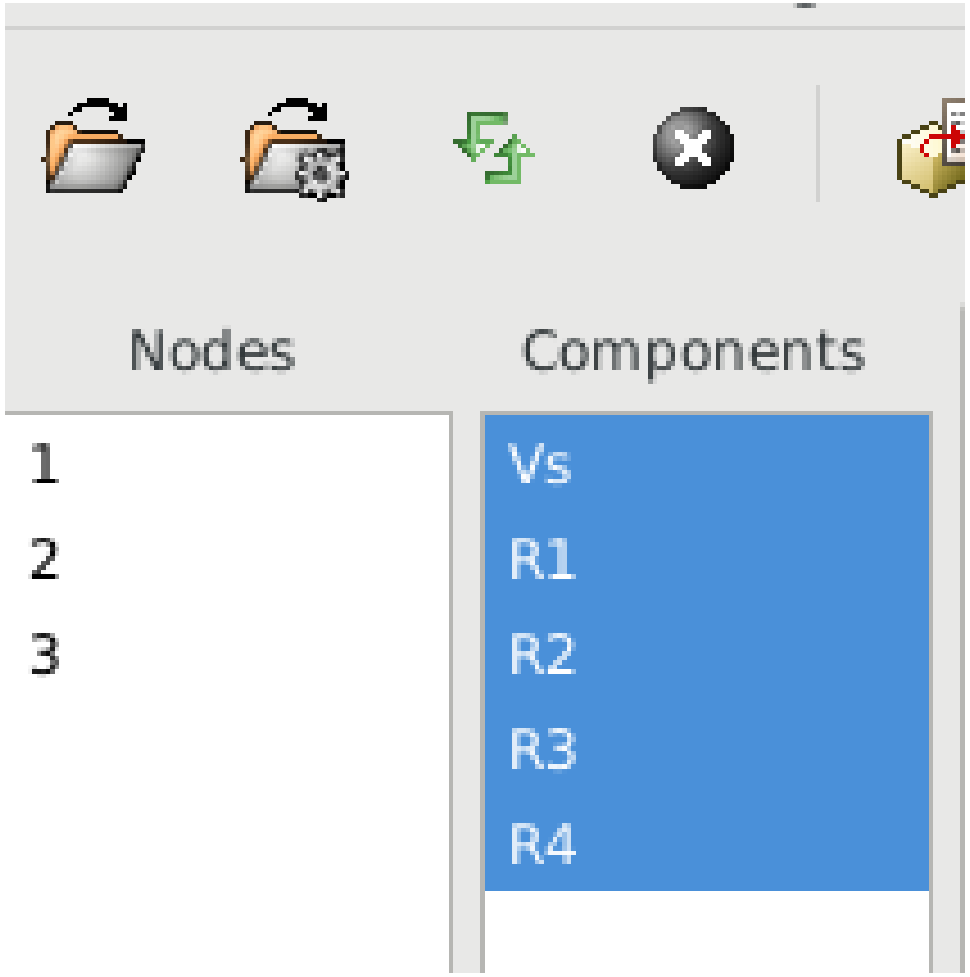
DC Sweep

- **in alto digitare su Start e Stop**
  - 5000
  - 5000



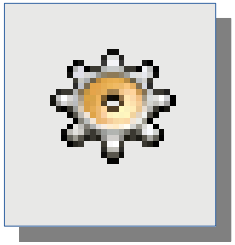
Componenti

- **Selezionare tutti i componenti di proprio interesse**
- **Per farlo**
  - cliccare sui componenti tenendo premuto anche CTRL

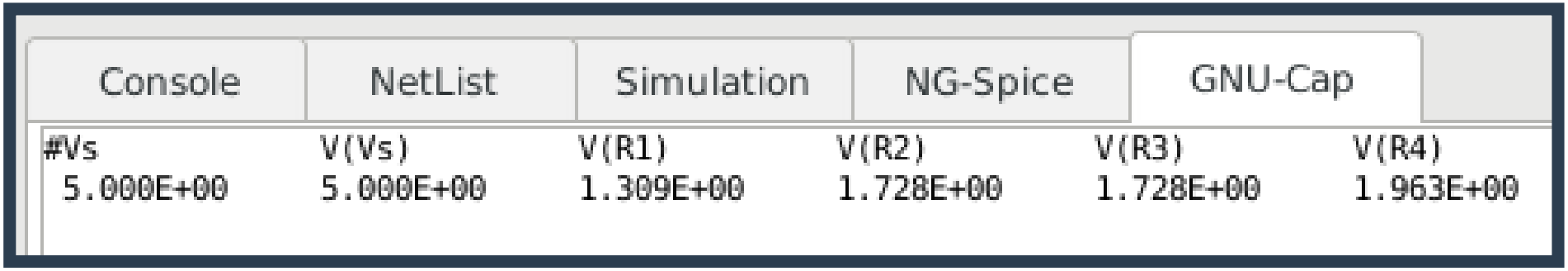


Avviare la simulazione

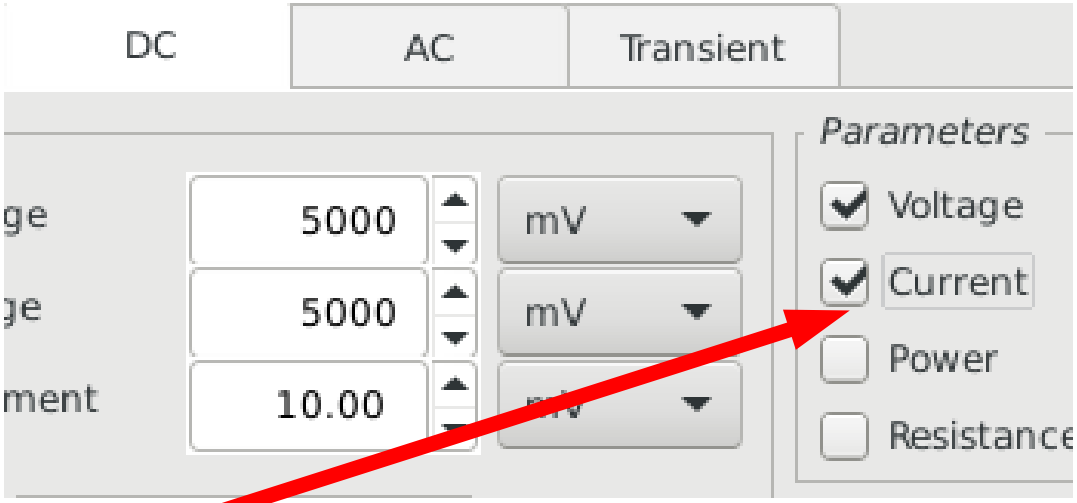
- **Premere la ruota con ingranaggi**



- **Per il risultato**
  - cliccare in basso sulla scheda GNUCAP



- **Oltre alla tensione è possibile simulare anche la corrente**
  - spuntare ‘Current’



- **Simulare di nuovo**

Copyright

- **Fabio Proietti (c) 2018**
- **Licenza:**
  - http://creativecommons.org/licenses/by-sa/4.0/