Programmi per la simulazione di circuiti

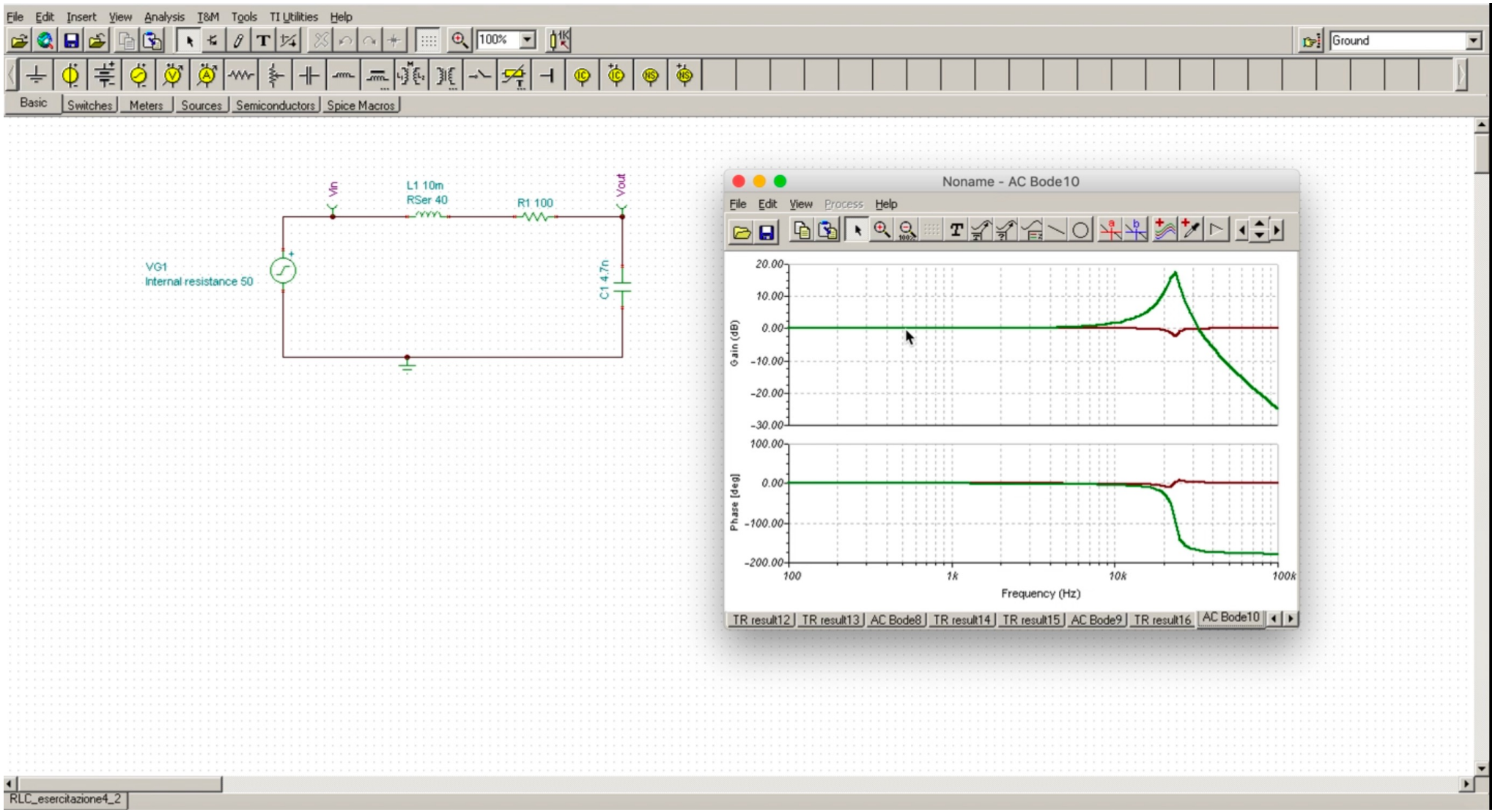
- **TINA-TI (Texas Instruments)**
  esiste versione gratuita, OS: Windows
  link: http://www.ti.com/tool/TINA-TI
  occorre registrarsi.
- **Pspice 9.1 educational**
  esiste versione gratuita, OS: Windows
  link: http://microwave.unipv.it/pages/campi_circuiti_I/PSpice.html
- **Ltspice**
  esiste versione gratuita, OS: Windows, Mac
  https://www.analog.com/en/design-center/design-tools-and-calculators/ltspice-simulator.html
  https://www.youtube.com/watch?v=6QzTZDm0guM
- **NGSPICE**
  esiste versione gratuita, OS: Ubuntu, Mac (installazione con fink)
  http://ngspice.sourceforge.net/
- **gEDA**
  esiste versione gratuita, OS: Linux, Mac (installazione con fink)
  http://www.geda-project.org/
  http://wiki.geda-project.org/geda:download
- **TinkerCAD**
  esiste versione gratuita, OS: su browser
  https://www.tinkercad.com/learn/circuits

- Per versioni Windows su Linux (e Mac)
  https://www.winehq.org/

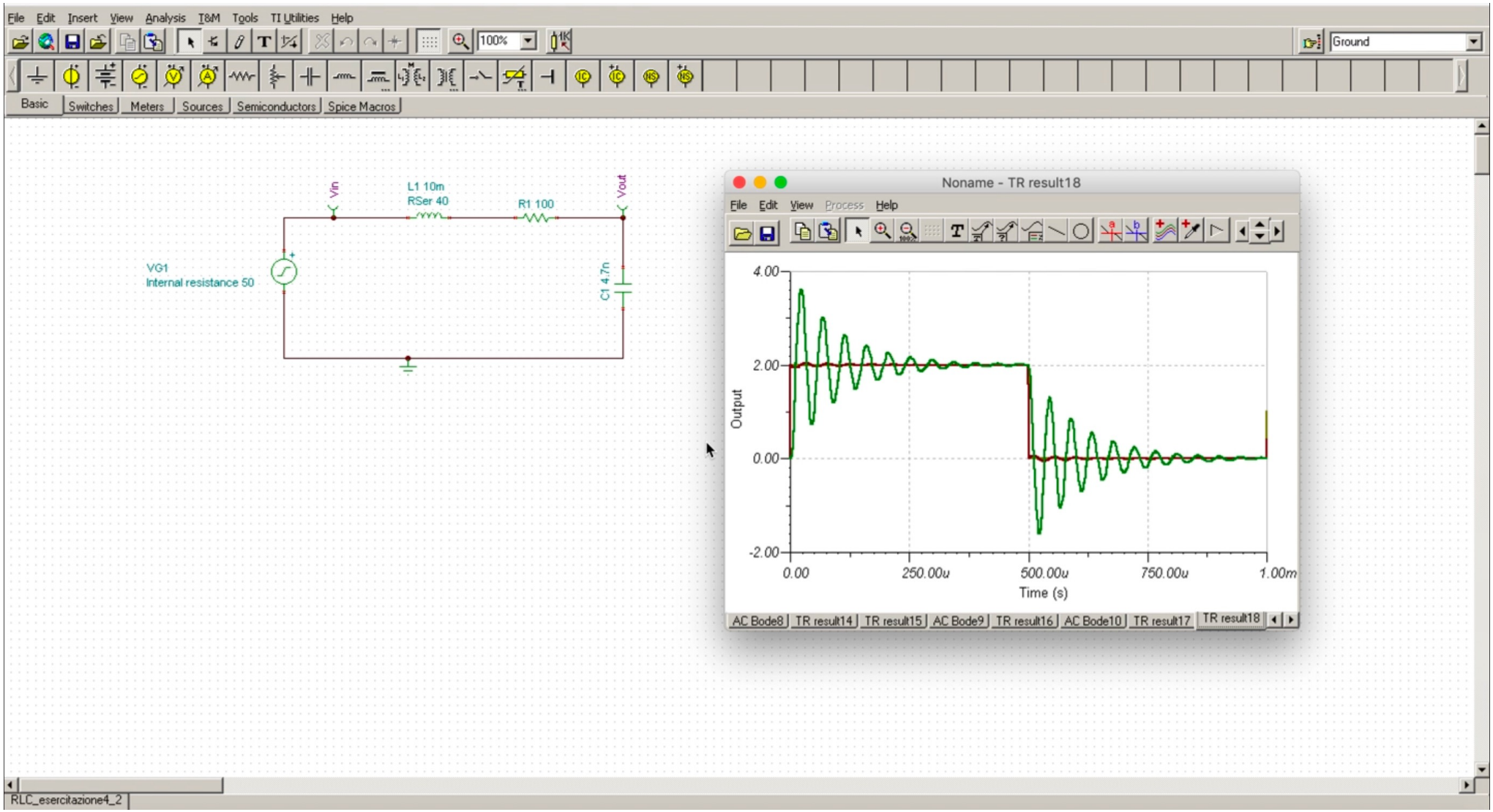
Simulazione circuiti e strumentazione

- Fra tutti i programmi disponibili gratuitamente, uno si è mostrato di particolare efficacia didattica, TINA-TI, messo a disposizione da Texas Instruments previa registrazione sul sito. Infatti TINA-TI non solo possiede una interfaccia grafica molto ben sviluppata, ma consente anche la possibilità di simulare la strumentazione presente in laboratorio, come l'oscilloscopio ed il generatore di segnale. Si mostrano alcuni esempi nelle slides seguenti.

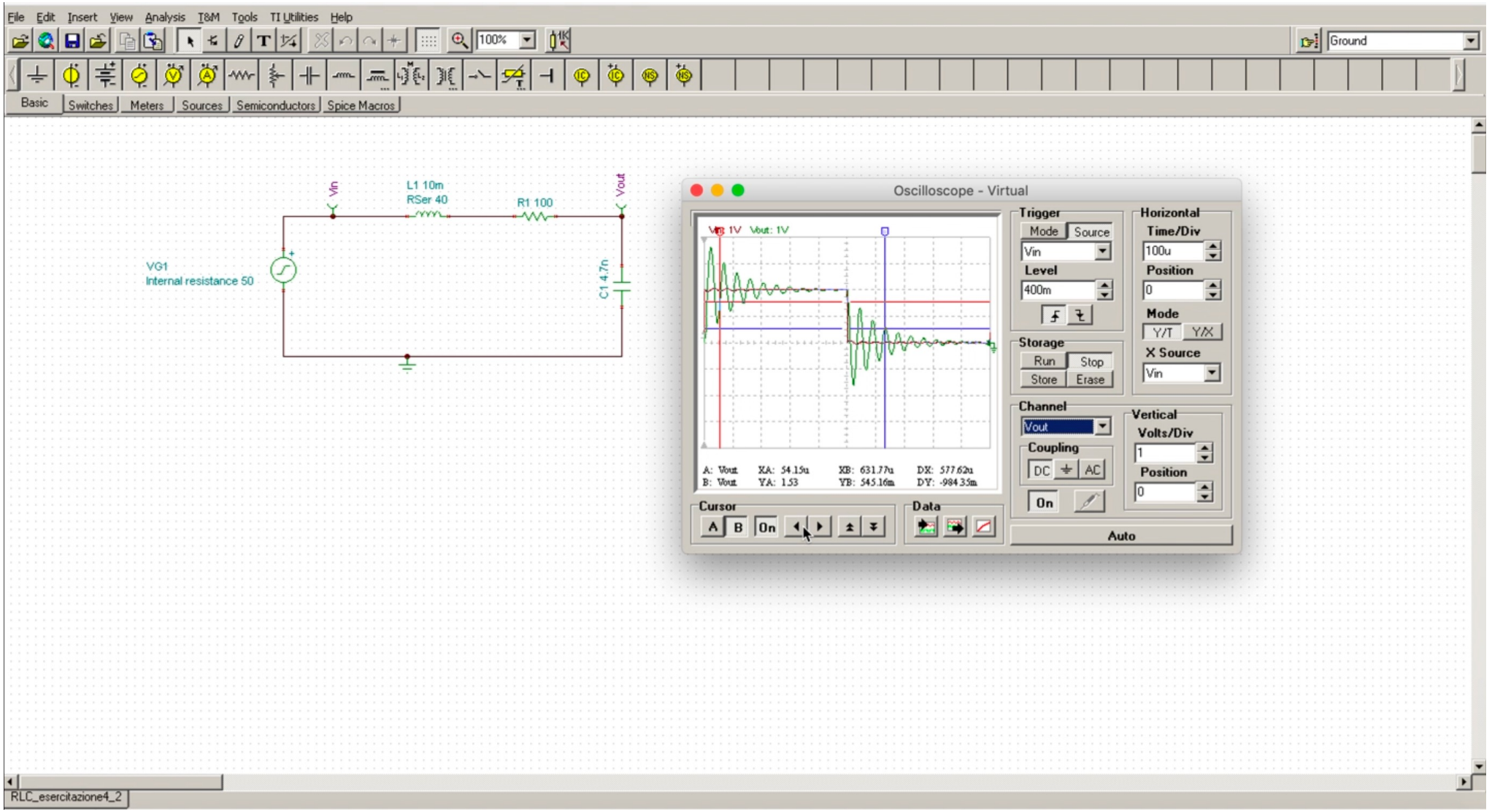
Esempi: TINA-TI



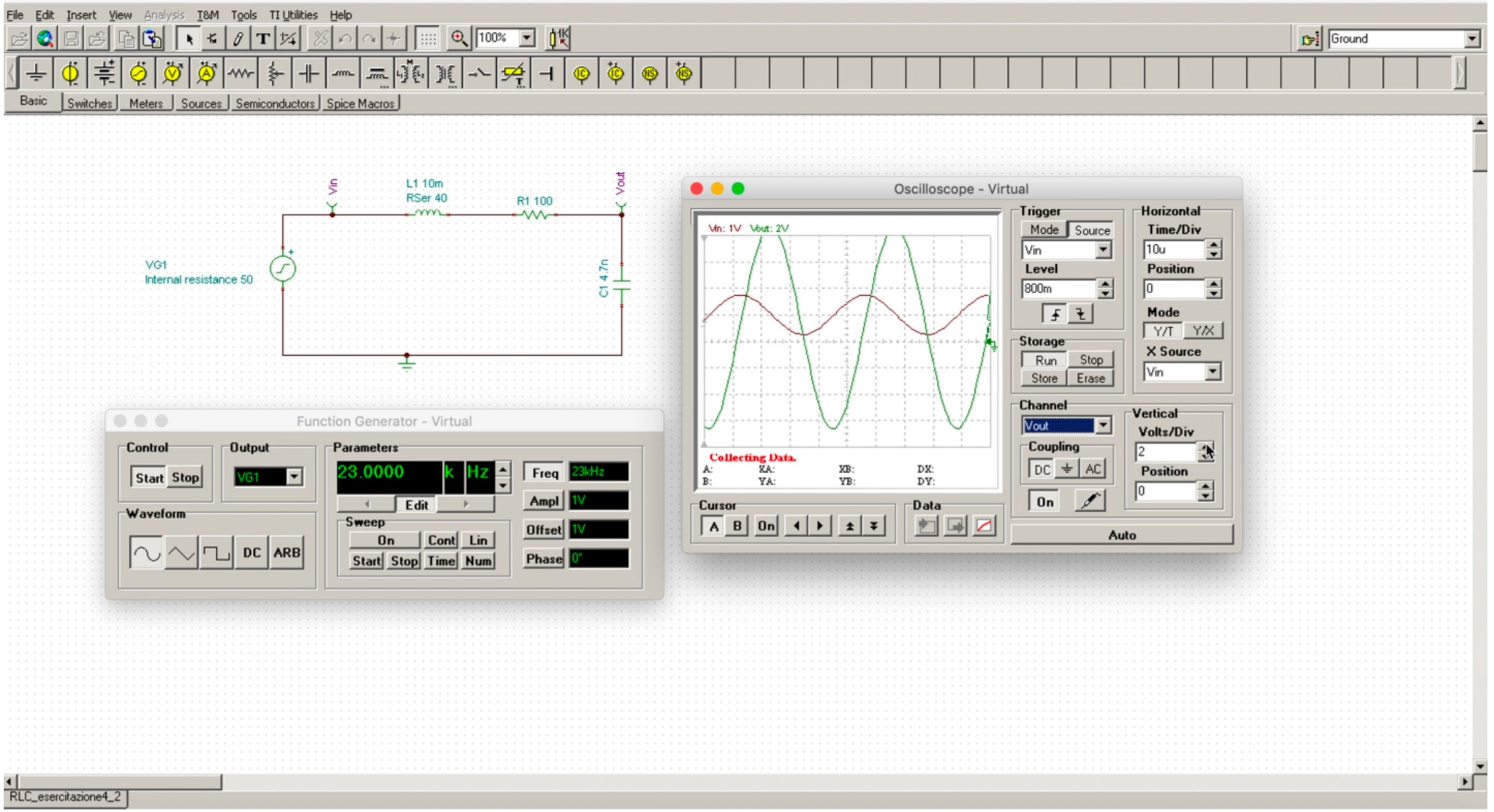
Esempi: TINA-TI



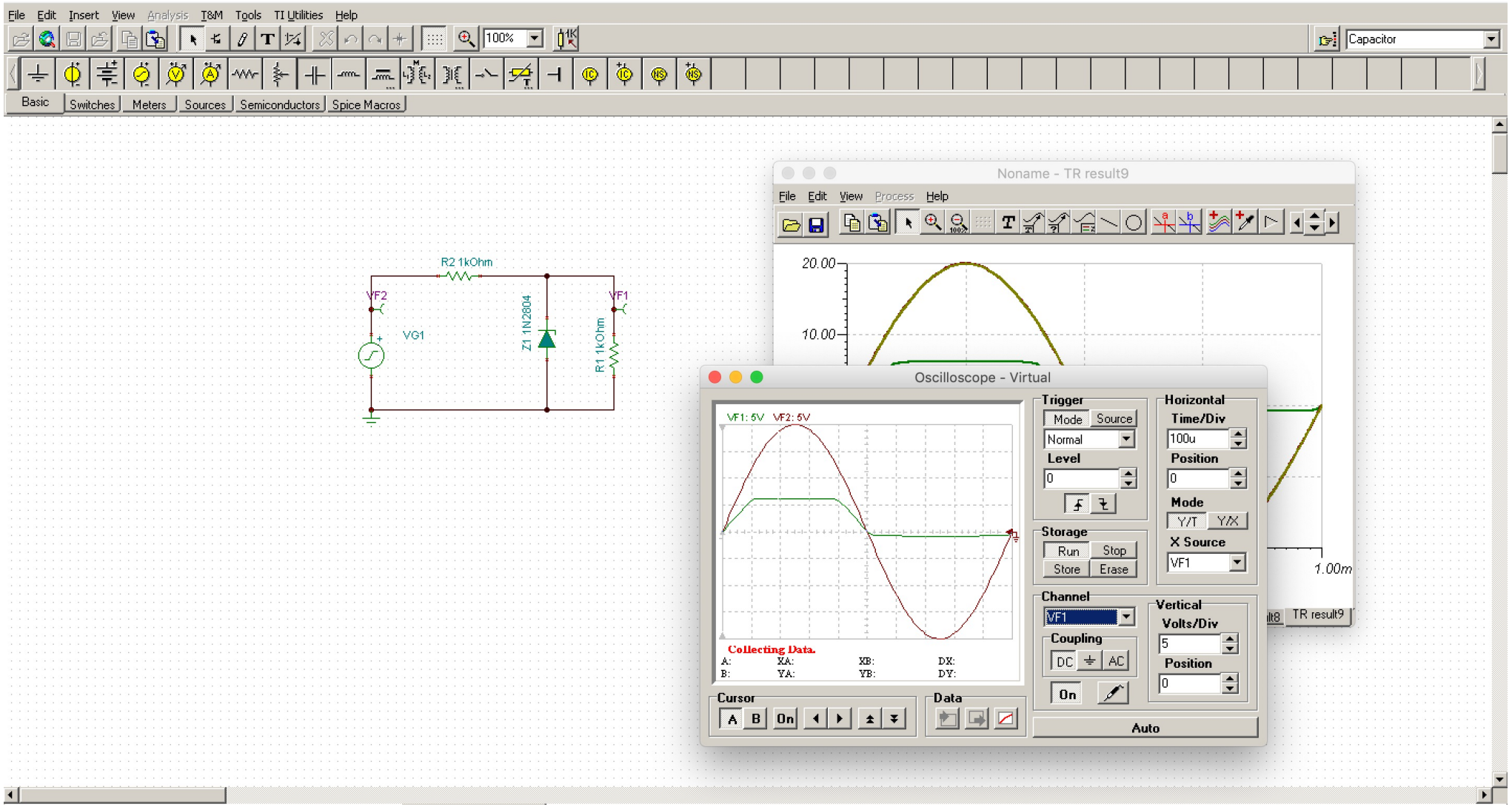
Esempi: TINA-TI



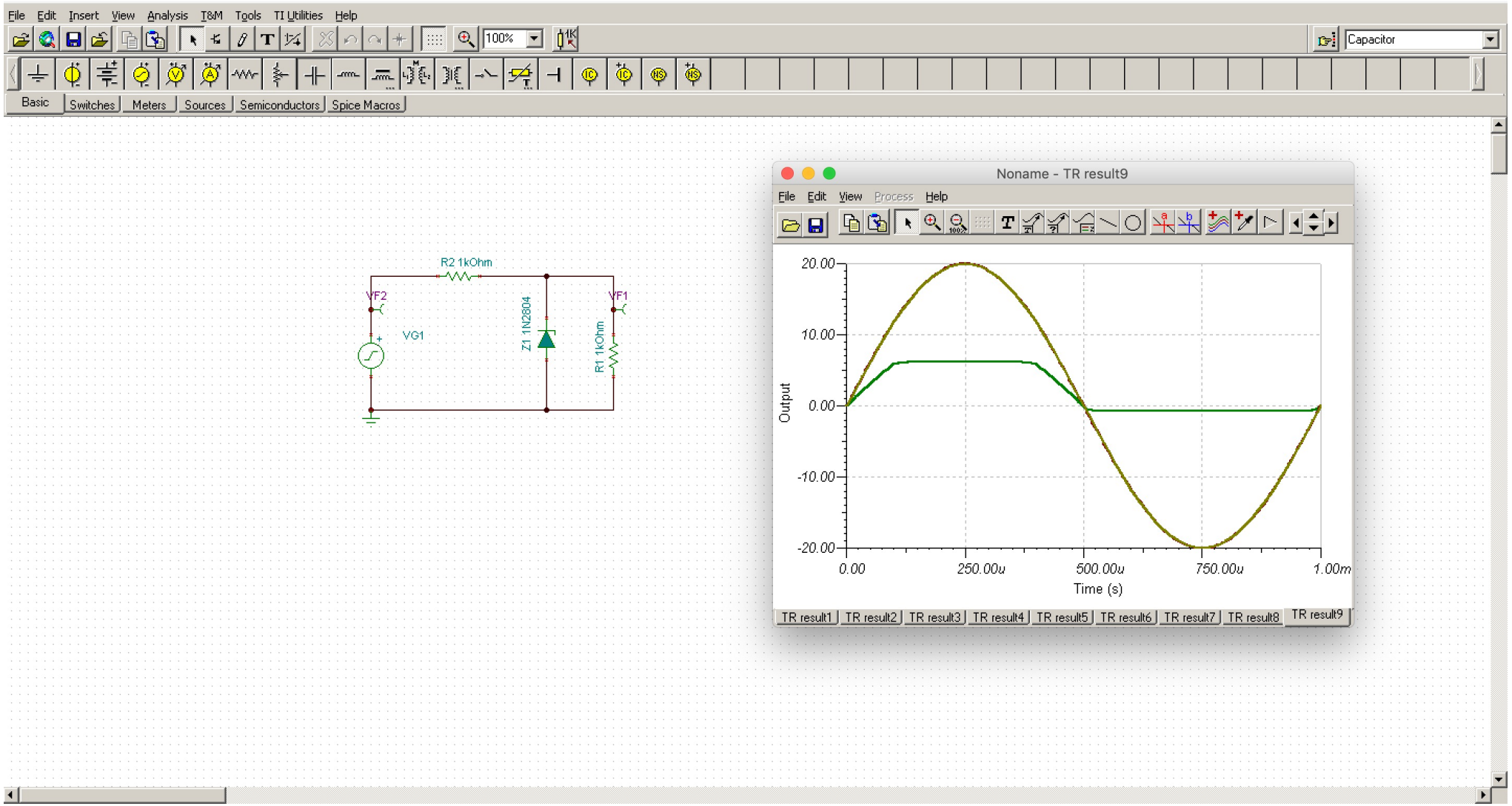
Esempi: TINA-TI



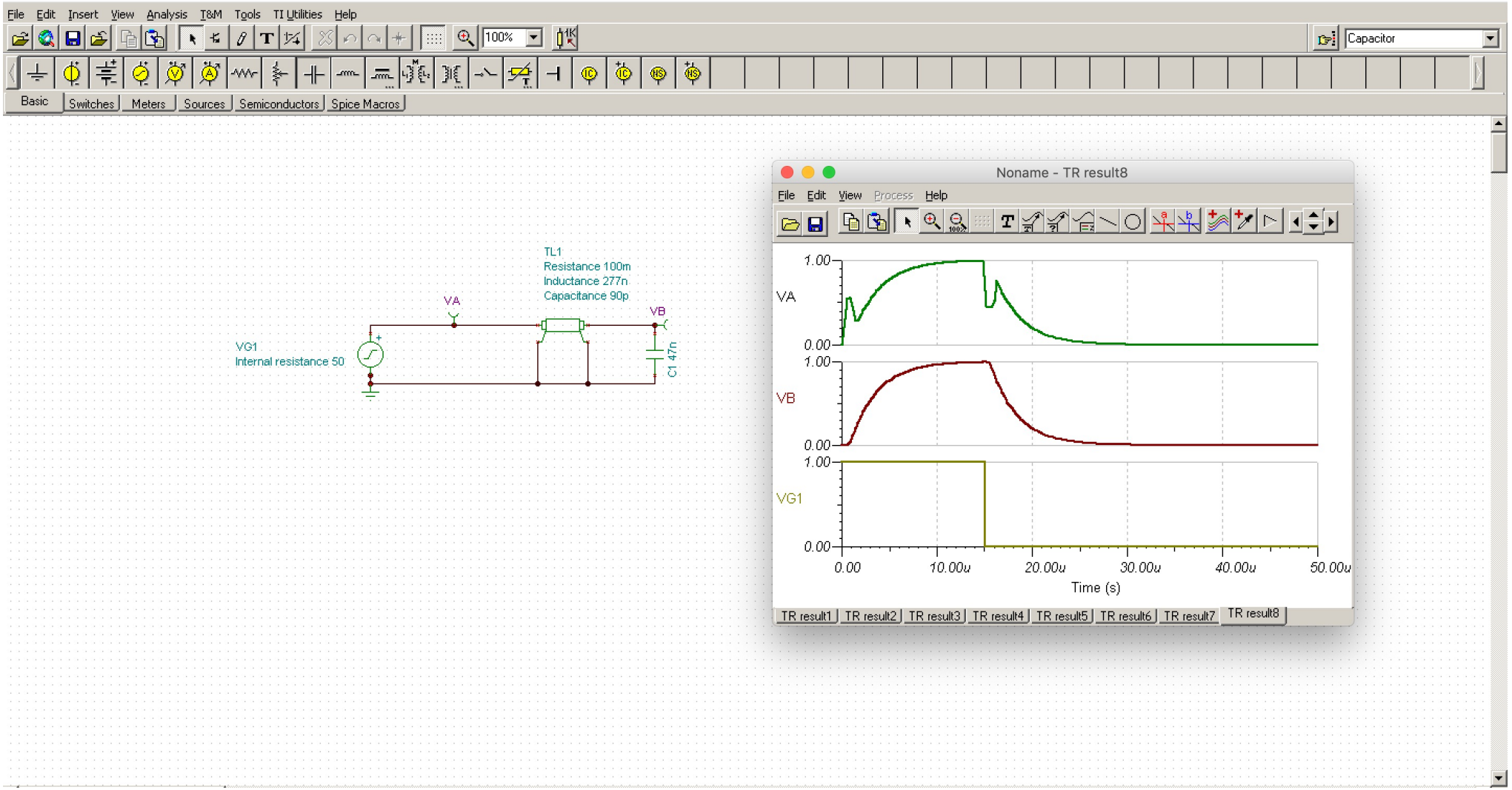
Esempi: TINA-TI



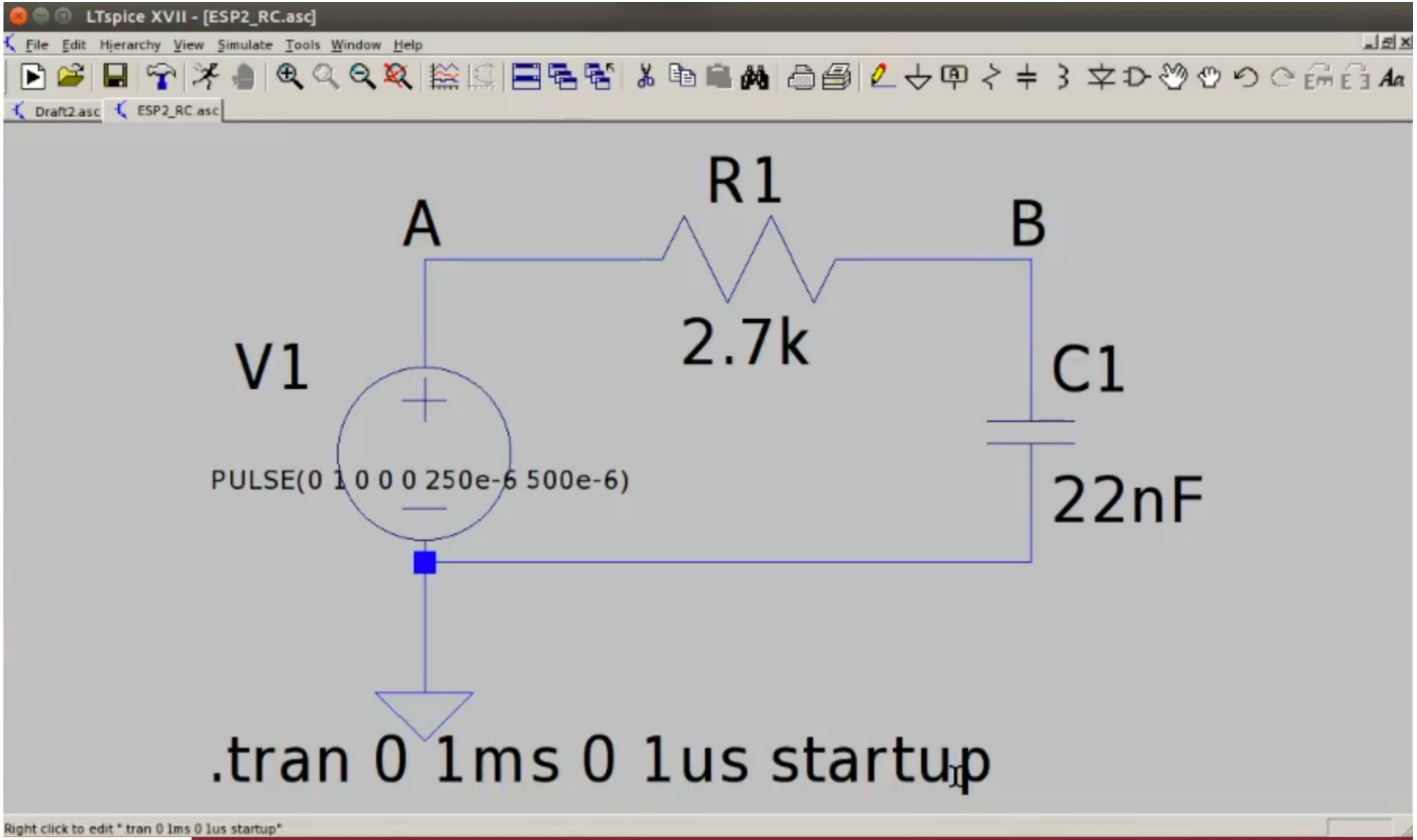
Esempi: TINA-TI



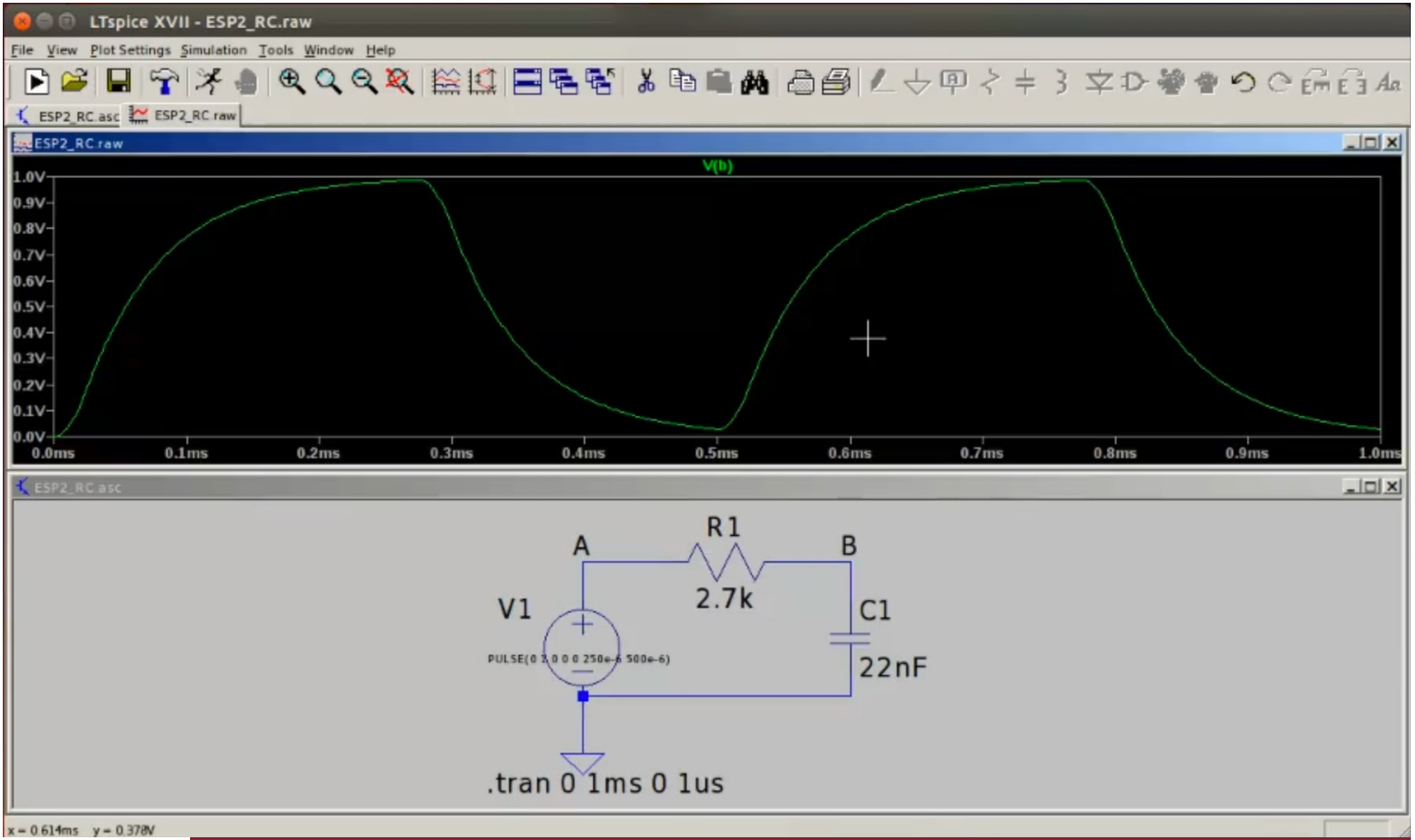
Esempi: TINA-TI



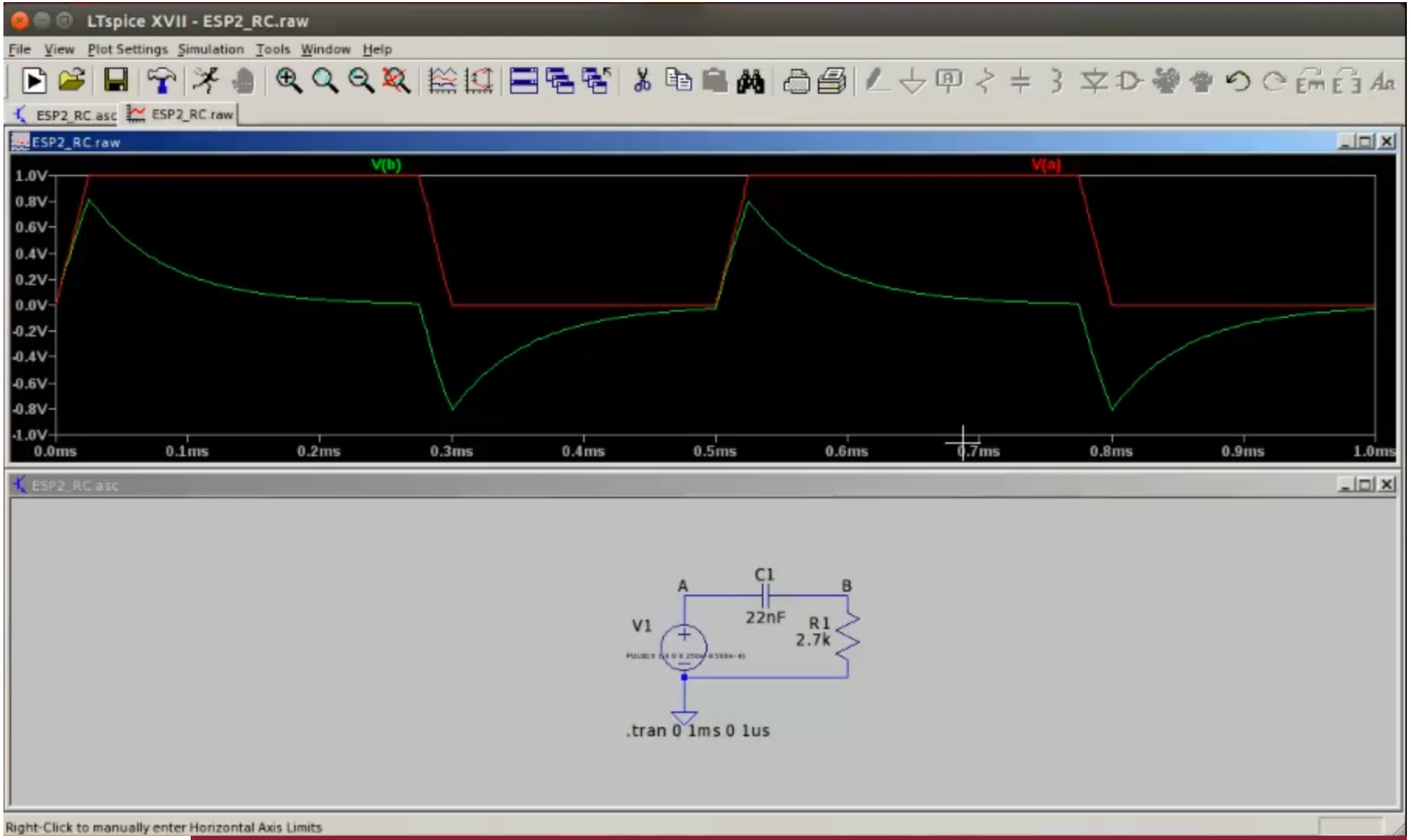
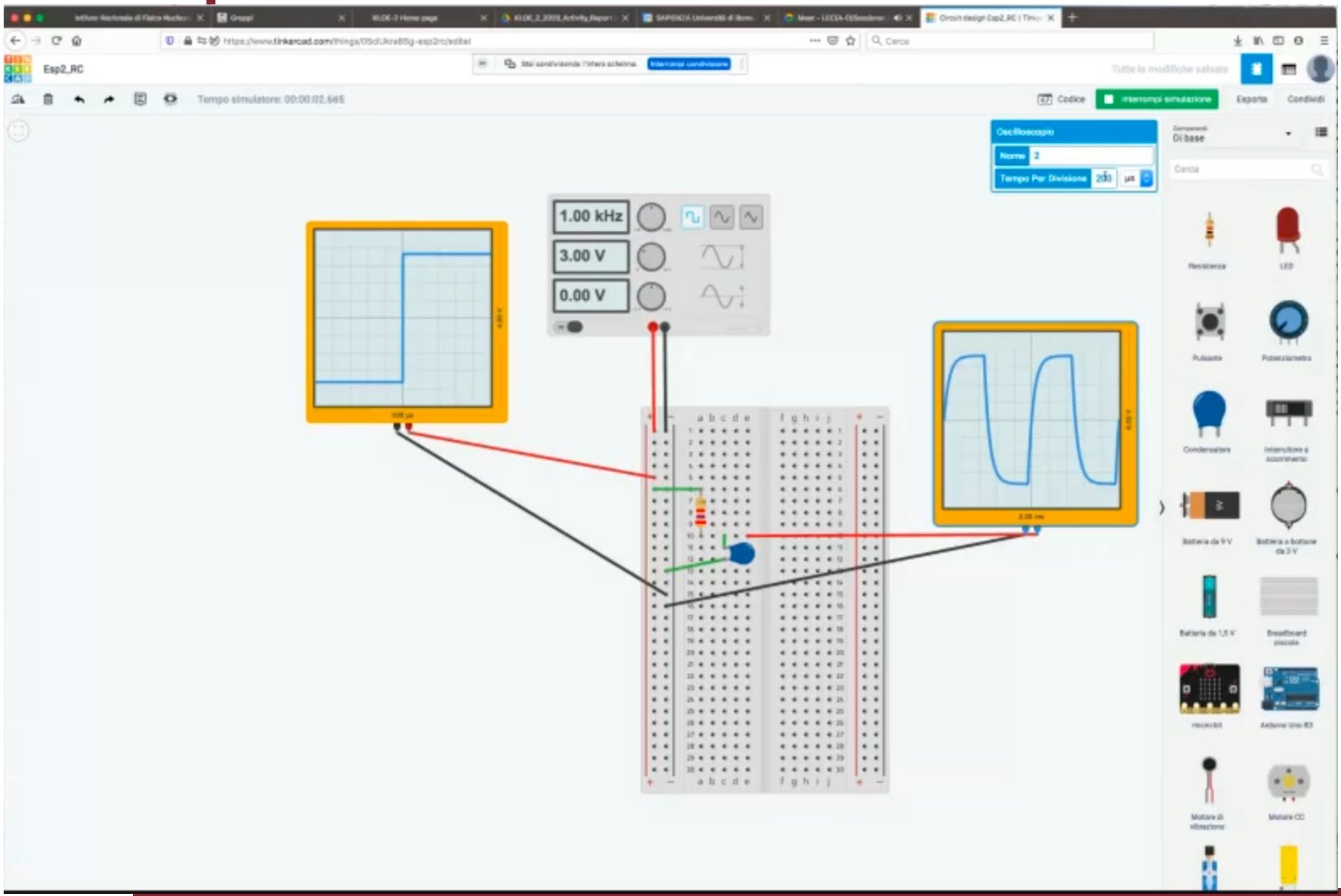
Esempi: LTSpice



Esempi: LTSpice



Esempi: LTSpice



Esempi: TinkerCAD