ENG1027: Instrumentação Eletrônica

3.2) Projeto e Simulação de Circuitos

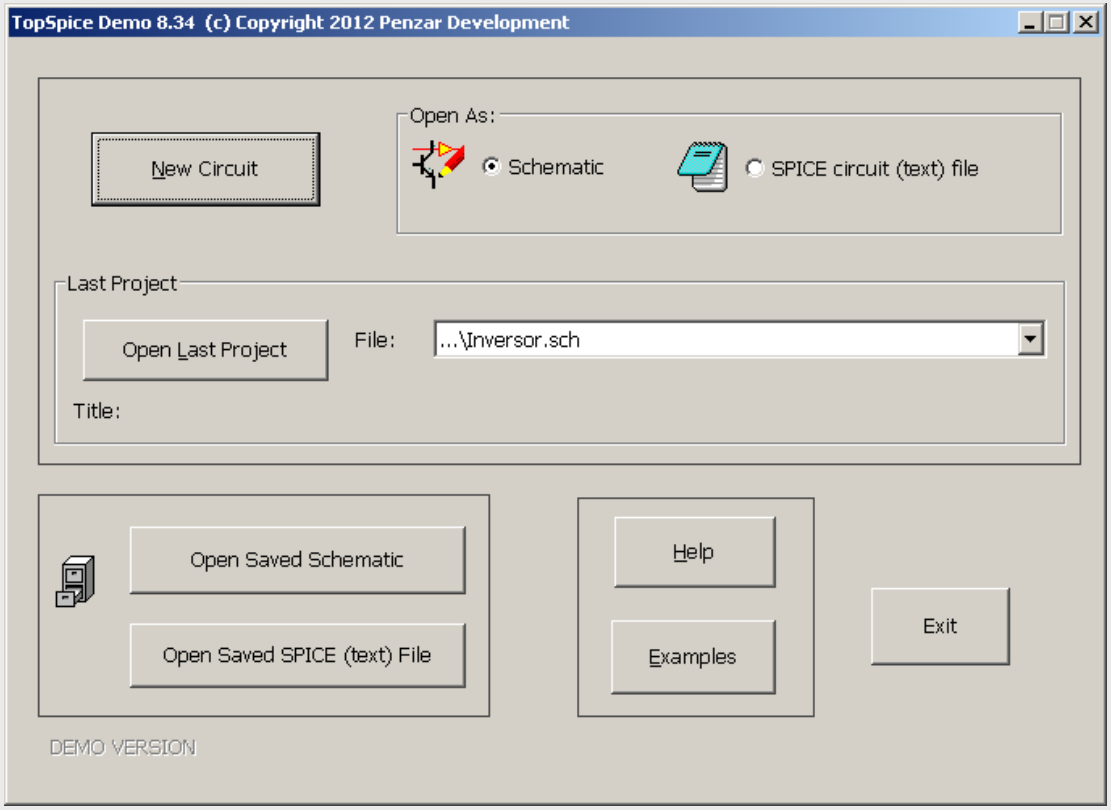
- A imensa maioria dos tipos de software de simulação de circuitos é baseada no SPICE original
  - Desenvolvido em 1975 em Berkeley
  - Inicialmente codificado em FORTRAN, e posteriormente em C (em 1983)
  - Acrônimo de Simulated Program with Integrated Circuits Emphasis (Programa de Simulação com Ênfase em Circuitos Integrados)
  - Poderosa ferramenta usada para testar e antever o comportamento de circuitos contendo circuitos integrados, resistores, transistores, capacitores, diodos e outros componentes elétricos e eletrônicos
  - Dezenas de versões comerciais

3.2) Projeto e Simulação de Circuitos

- Exemplos de SPICEs comerciais:
  - PSpice/OrCAD
  - SPICE OPUS
  - HSpice
  - HSIM
  - MicroCad
  - Dr. Spice
  - T-Spice
  - Intusoft
  - Spice-It!
  - SIMetrix
  - TopSPICE
  - NG-spice
  - MultiSIM
  - SmartSpice
  - TINA
  - Spectre
  - Eldo
  - UltraSim
  - MacSpice
  - NanoSim
  - NSPICE
  - B2SPICE
  - ICAP/4
  - TINA-TI
  - Proteus ISIS

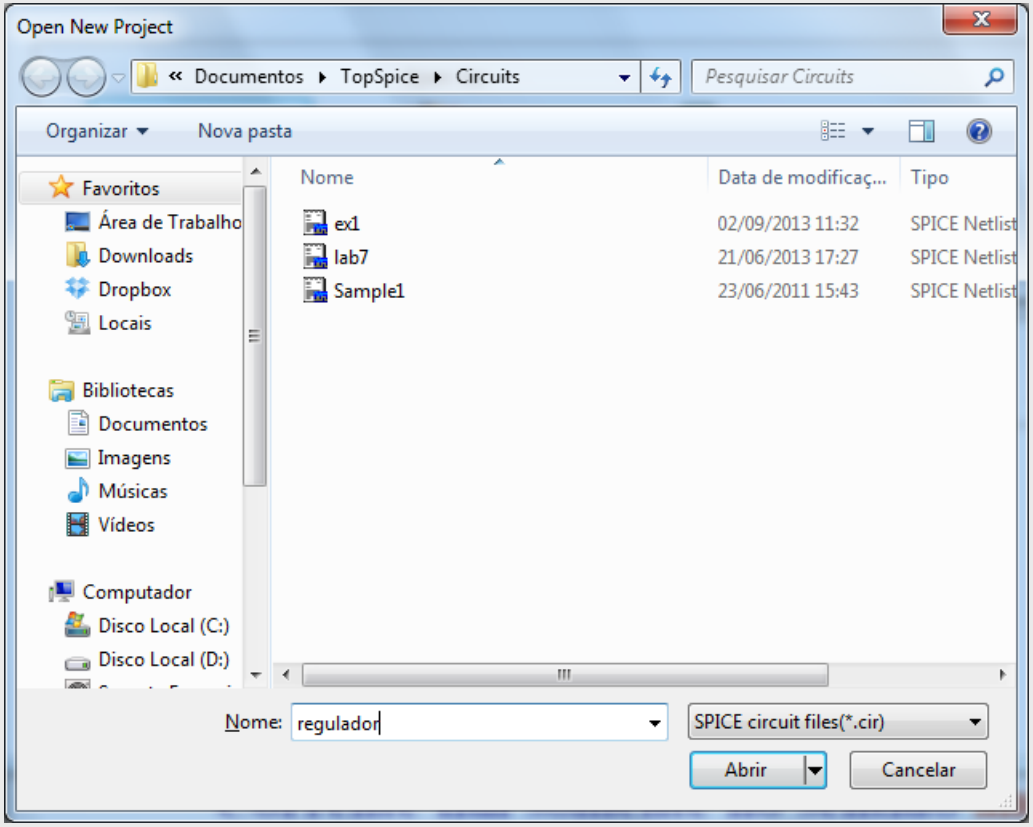
3.2) Projeto e Simulação de Circuitos

- Nesta disciplina: TopSPICE 8 (Demo)



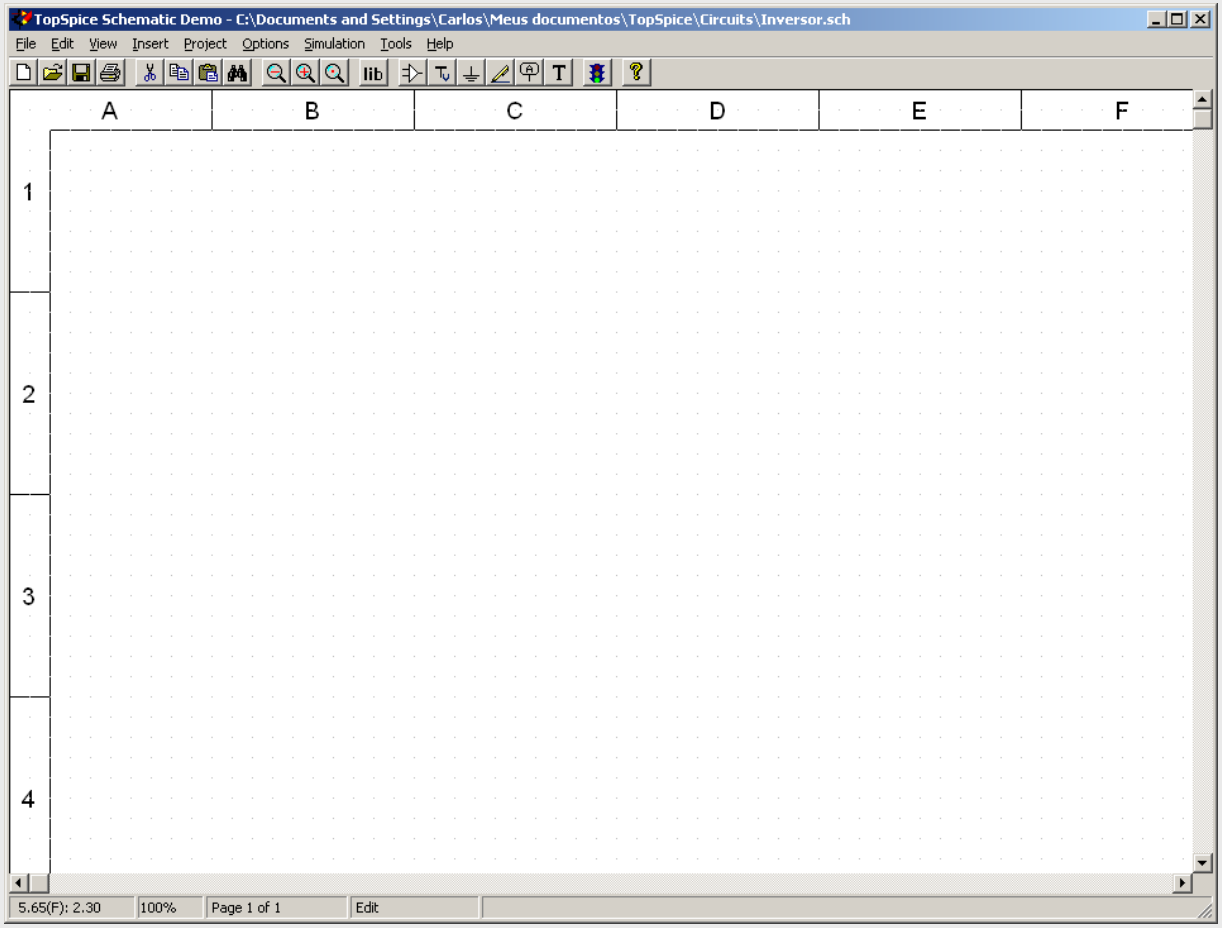
3.2) TopSPICE

- Criando novo projeto



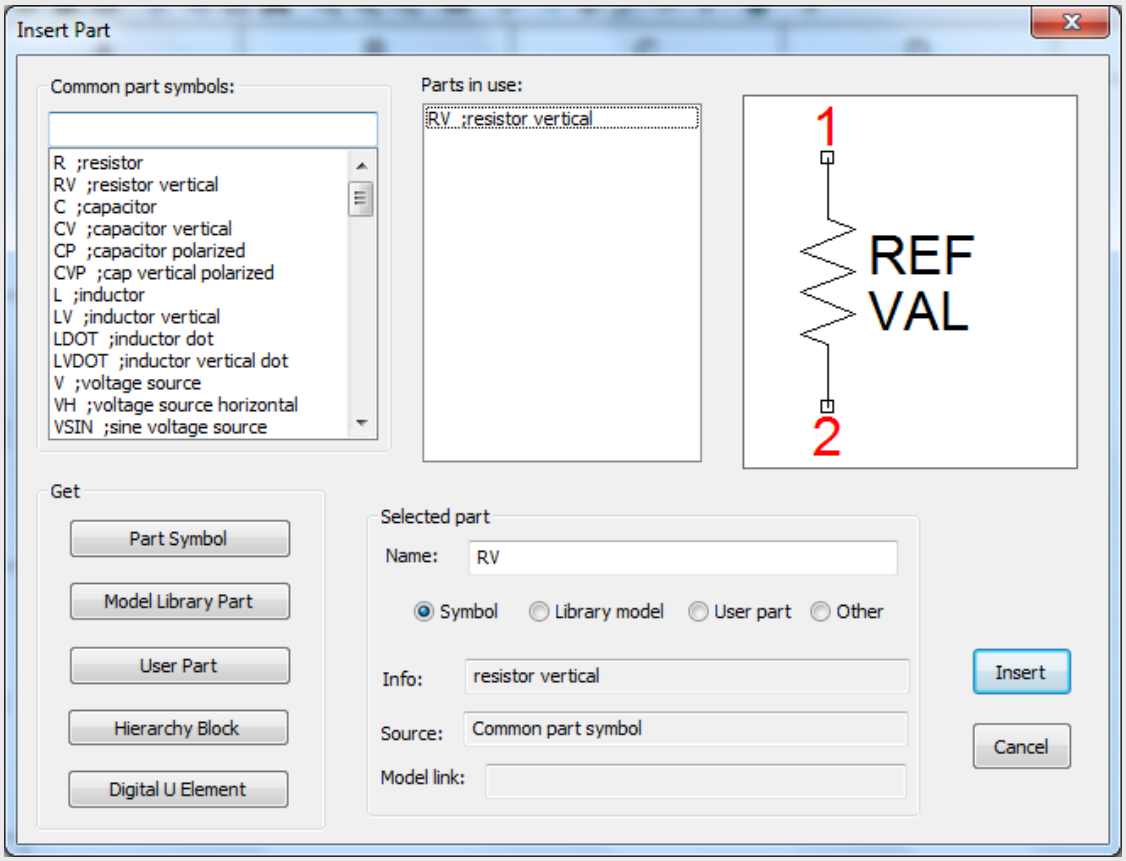
3.2) TopSPICE

- Editor de Esquemático



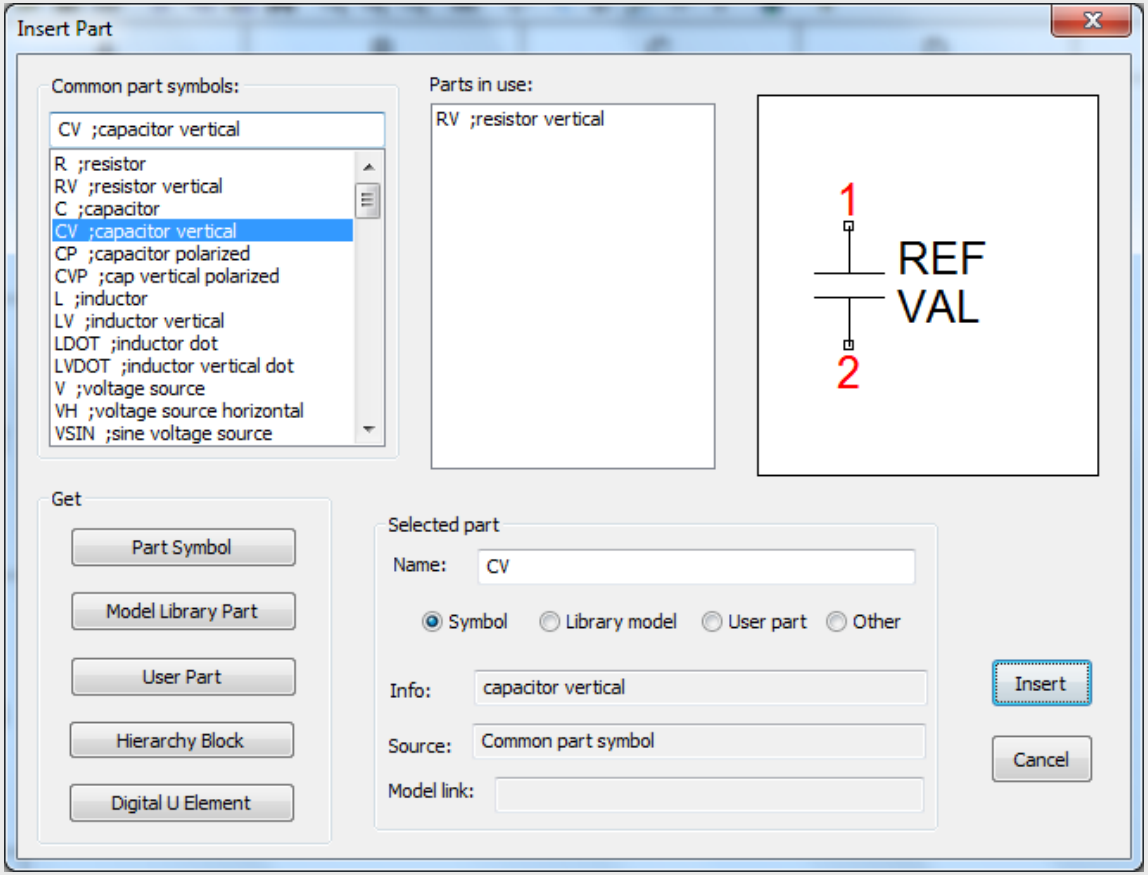
3.2) TopSPICE

- Inserir componente (Insert -> Part): Resistor vertical



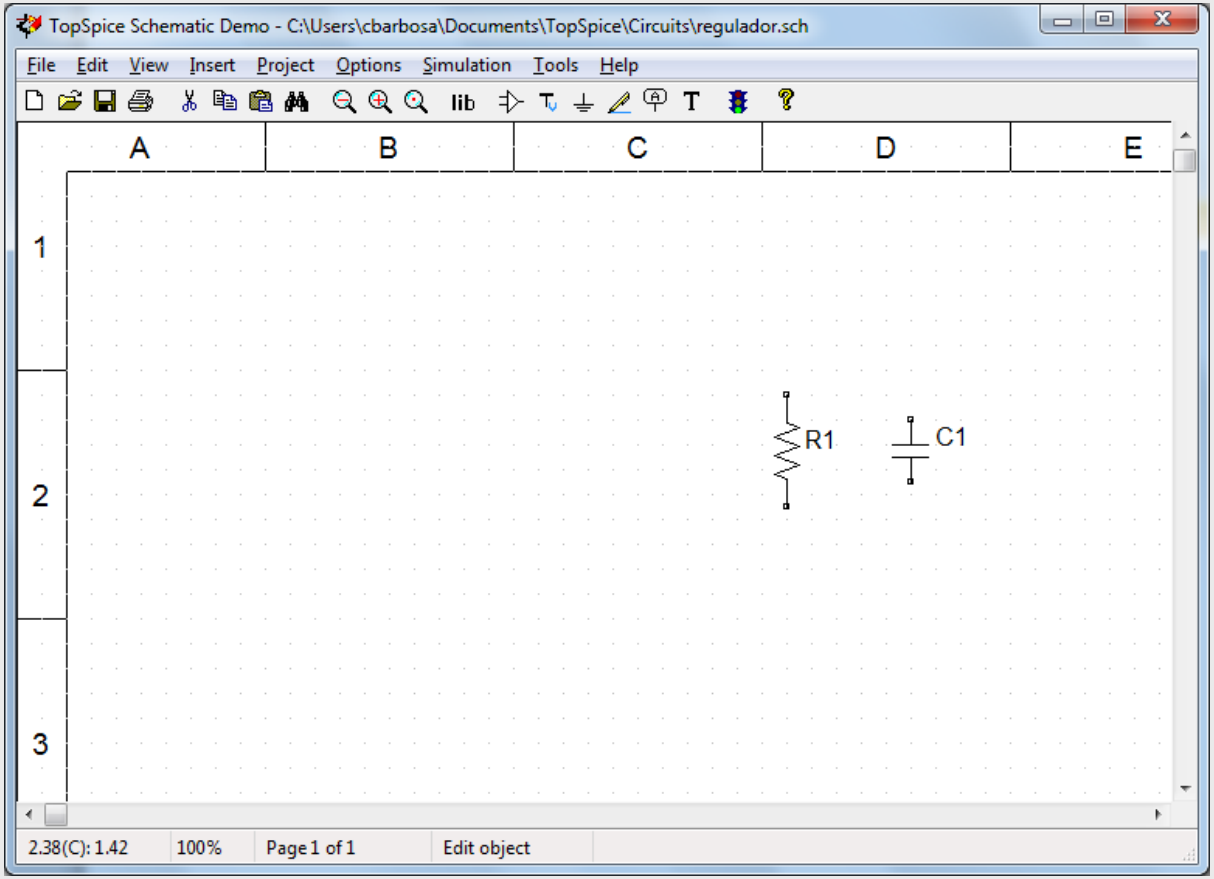
3.2) TopSPICE

- Inserir componente (Insert -> Part): Capacitor vertical



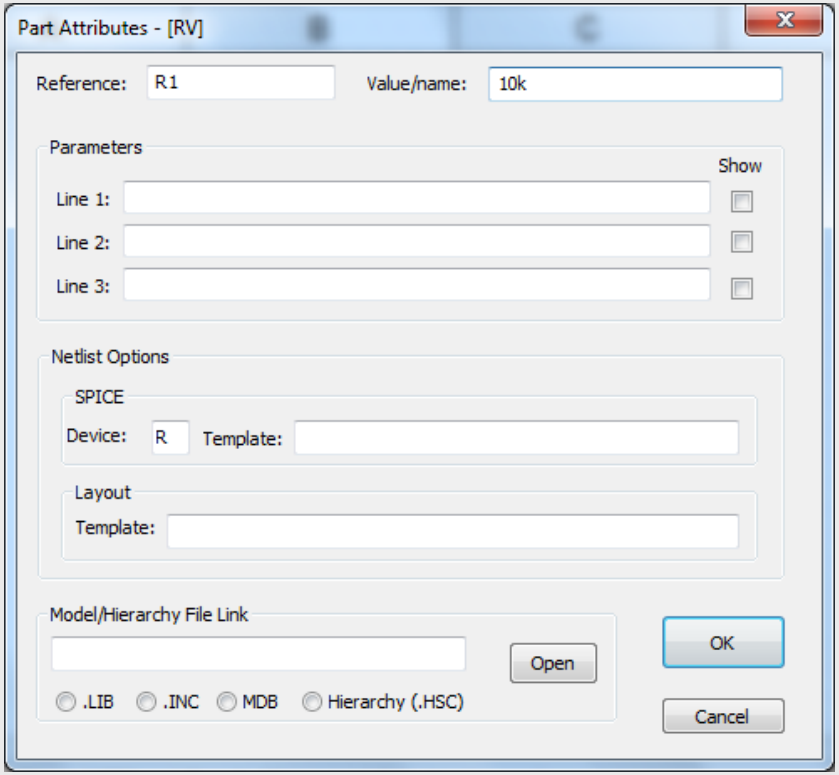
3.2) TopSPICE

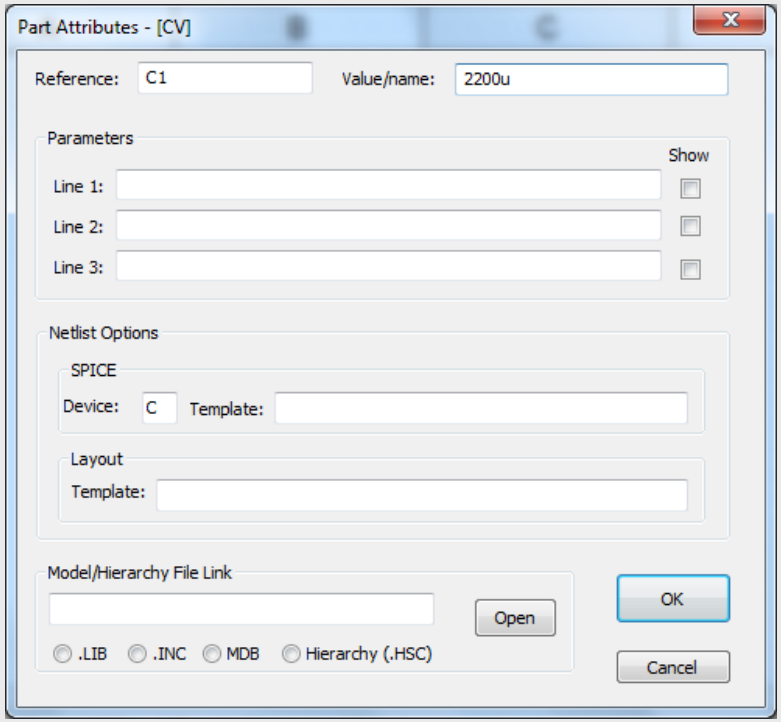
⌘ Circuito Parcial:



3.2) TopSPICE

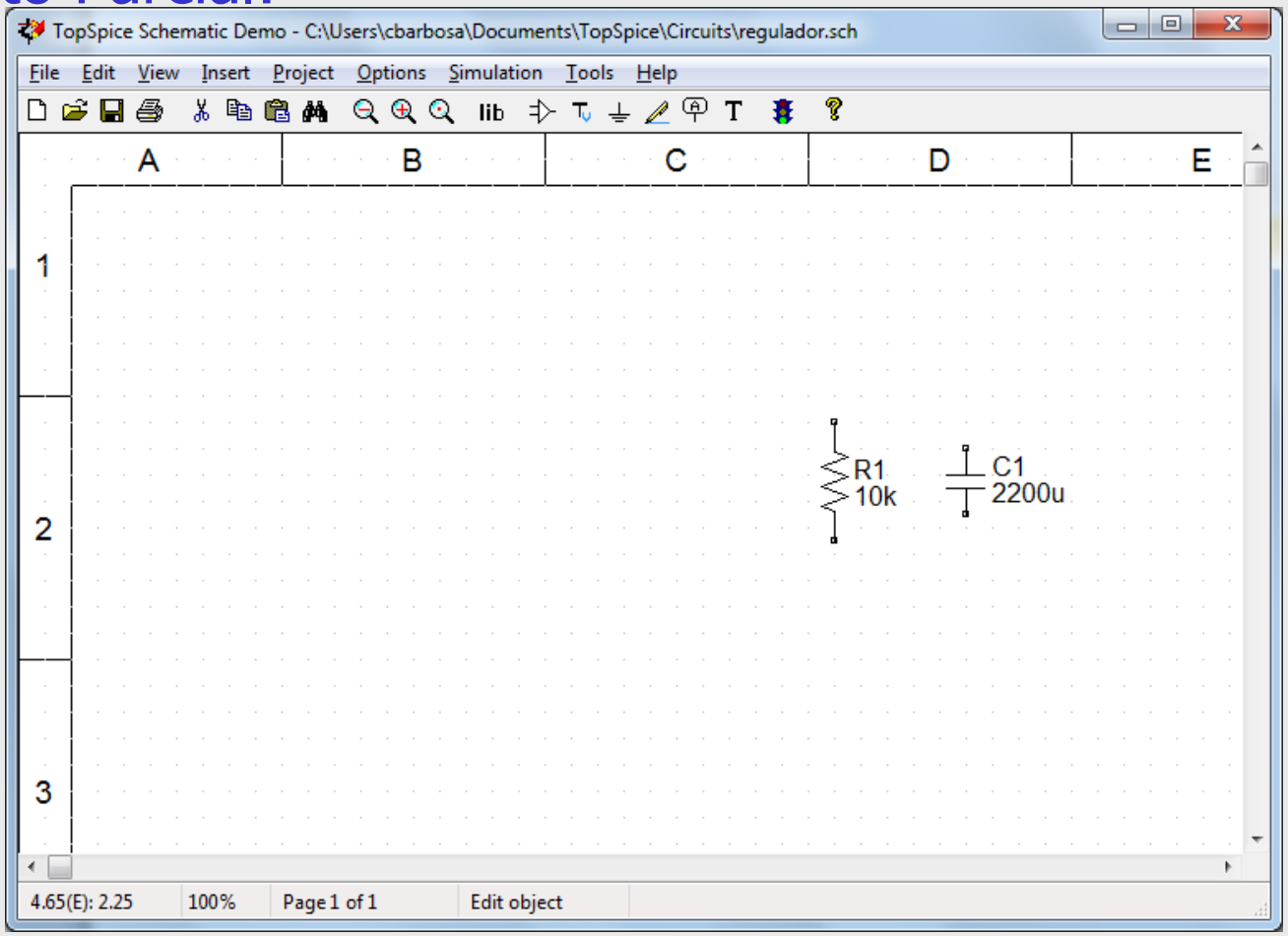
⌘ Parâmetros dos componentes:





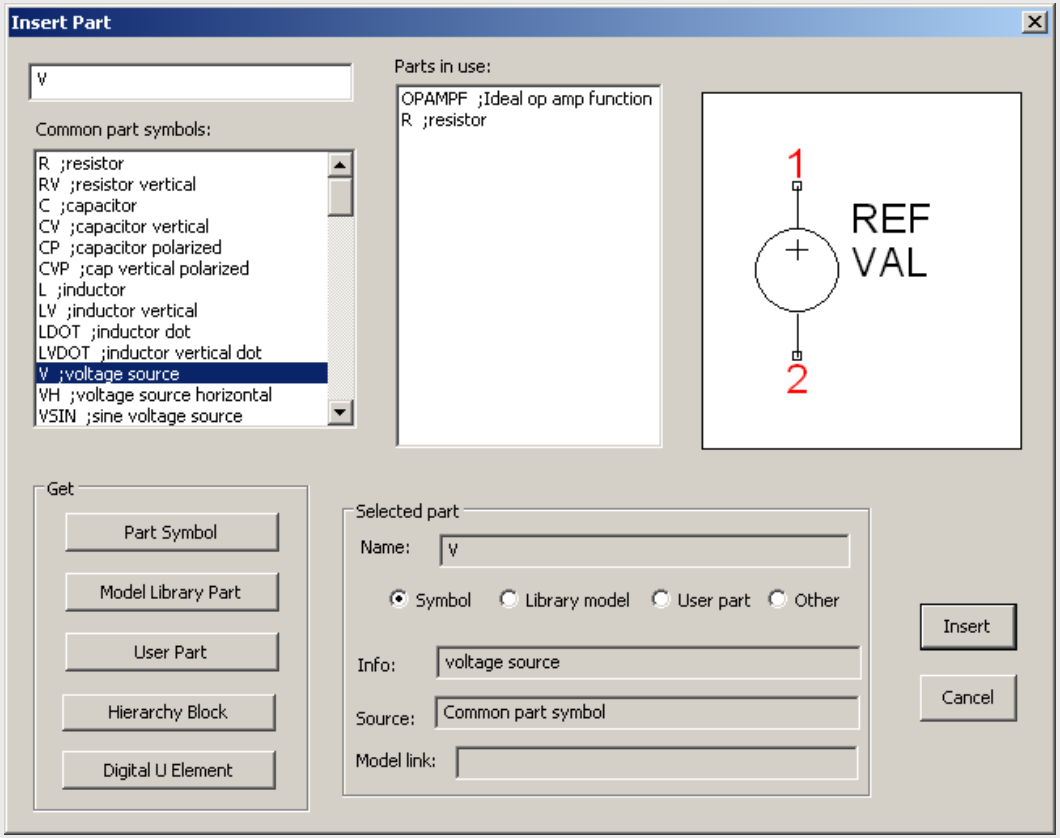
3.2) TopSPICE

- Circuito Parcial:



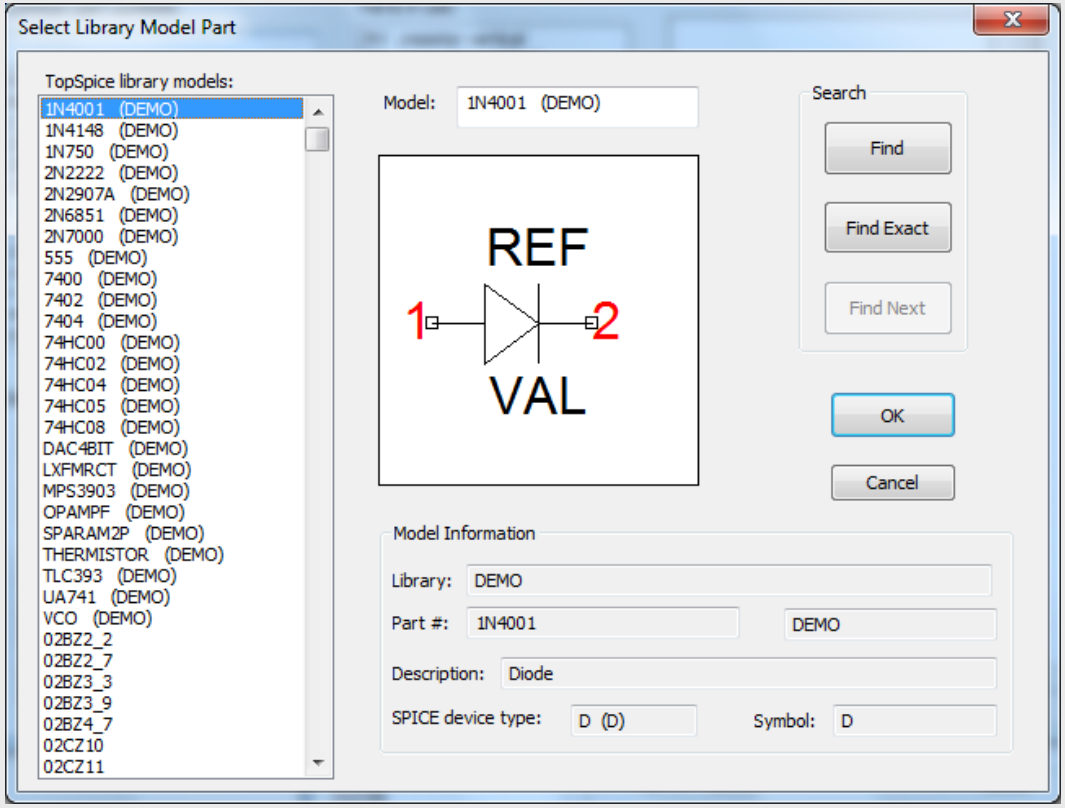
3.2) TopSPICE

- Inserir componente (Insert -> Part): Fonte de Tensão



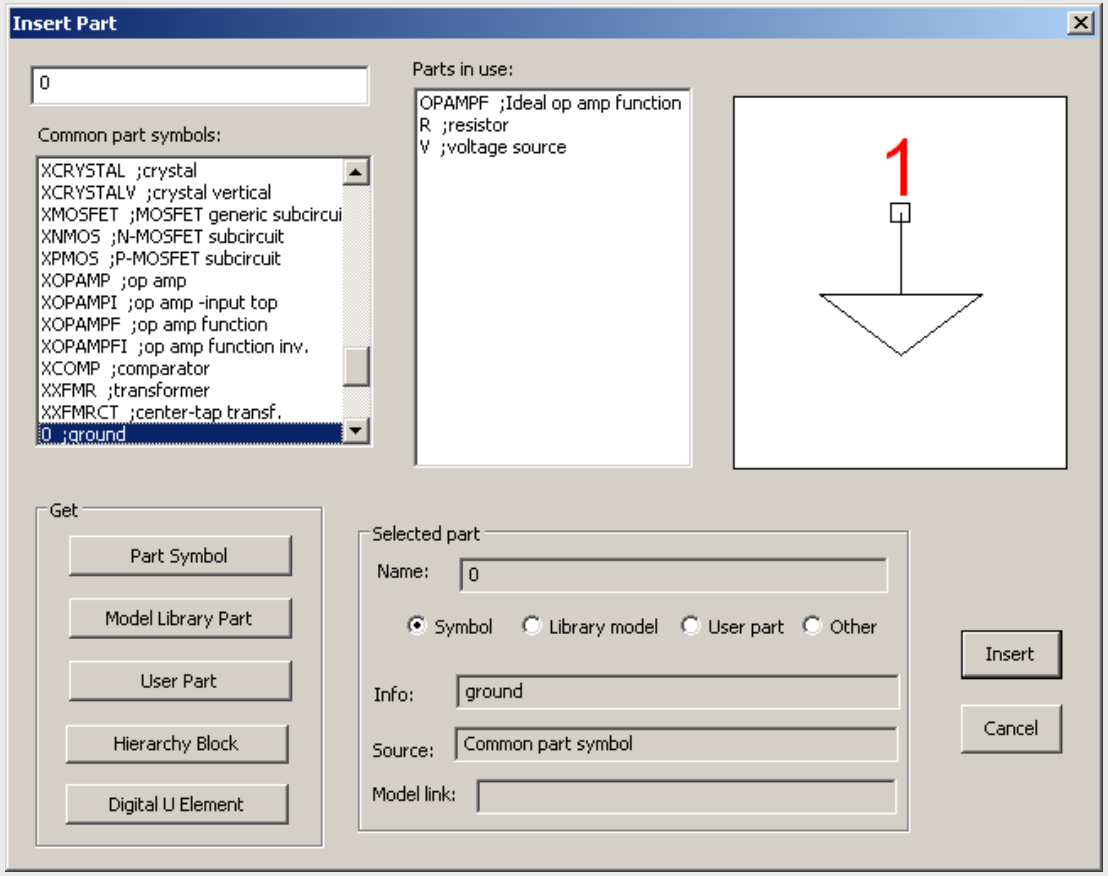
3.2) TopSPICE

- Inserir componente (Insert -> Part): Diodo



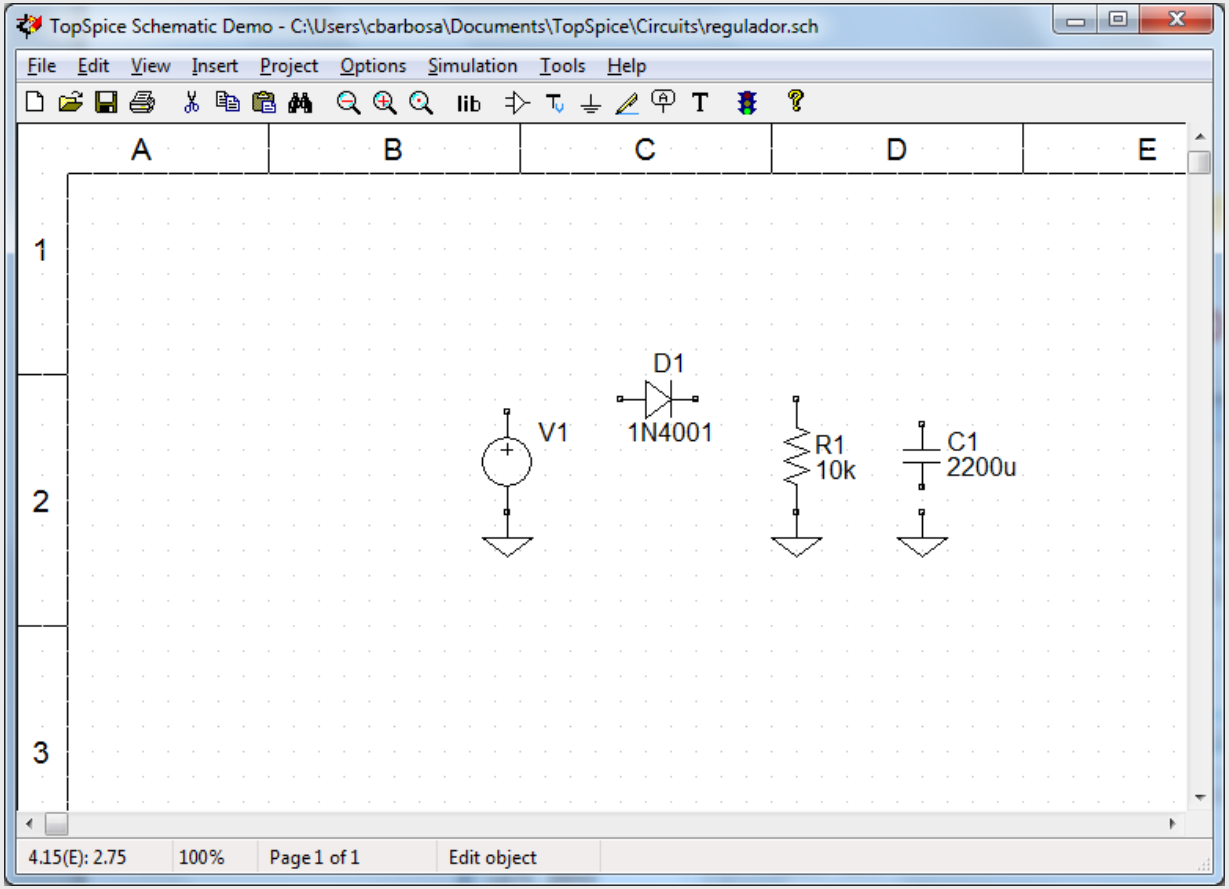
3.2) TopSPICE

- Inserir componente (Insert -> Part): Terras



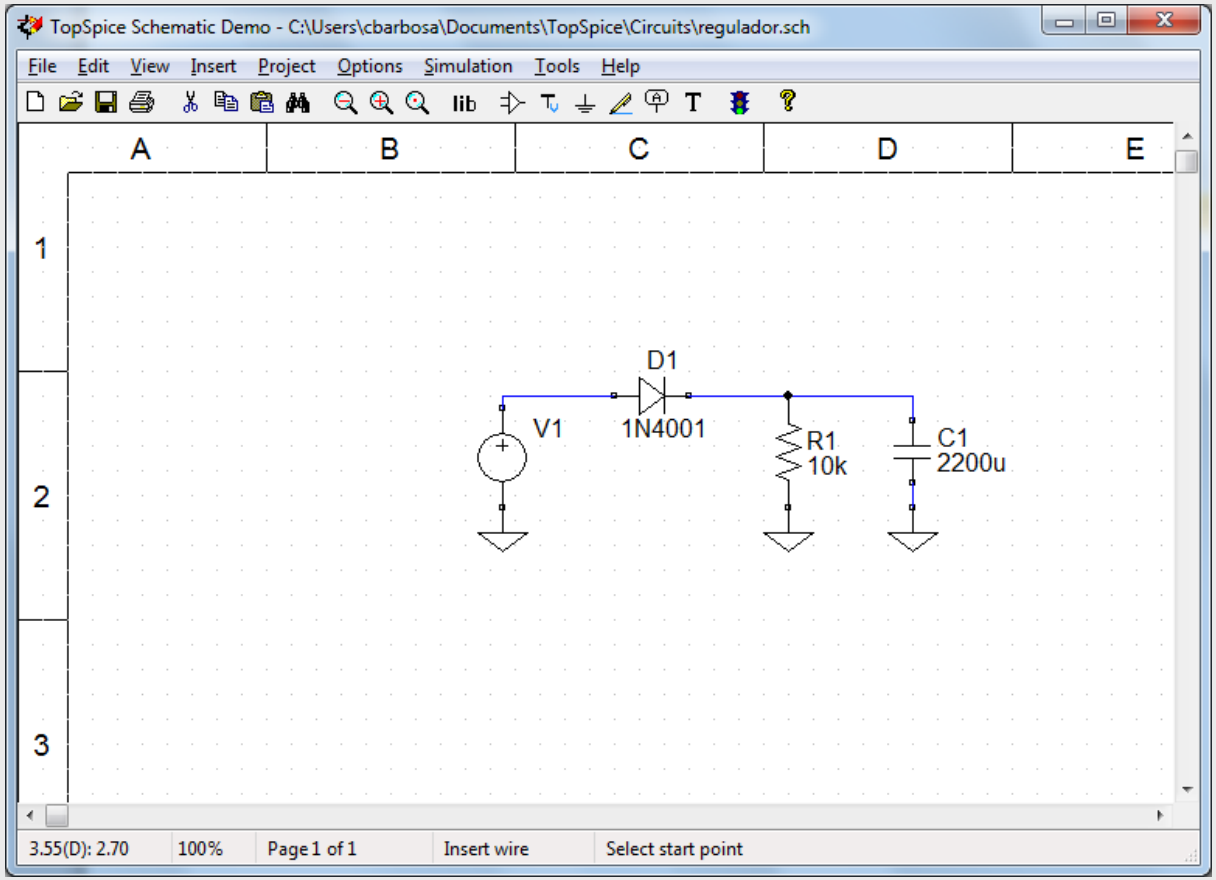
3.2) TopSPICE

- Circuito Parcial:



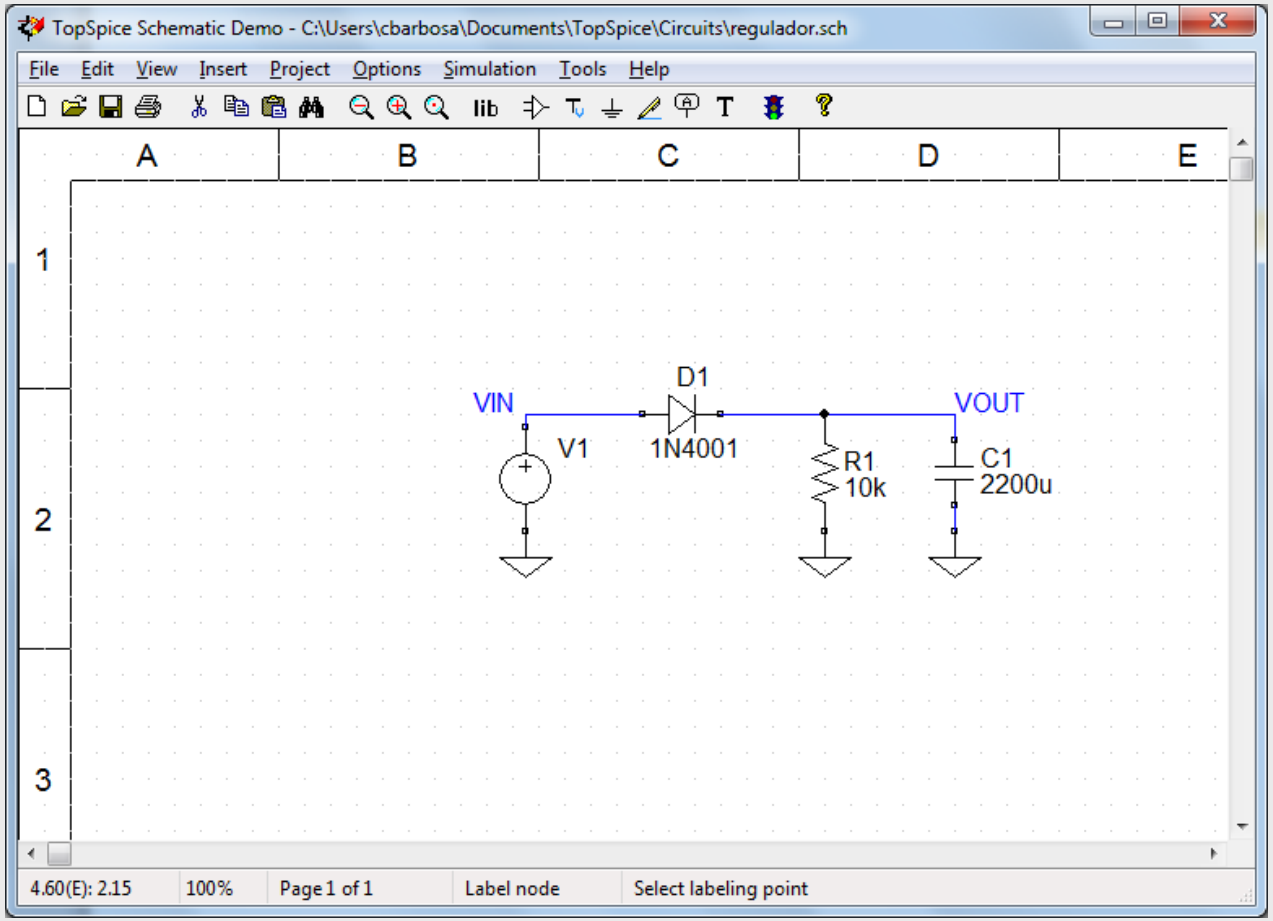
3.2) TopSPICE

- Inserir fiação (Insert -> Wire):



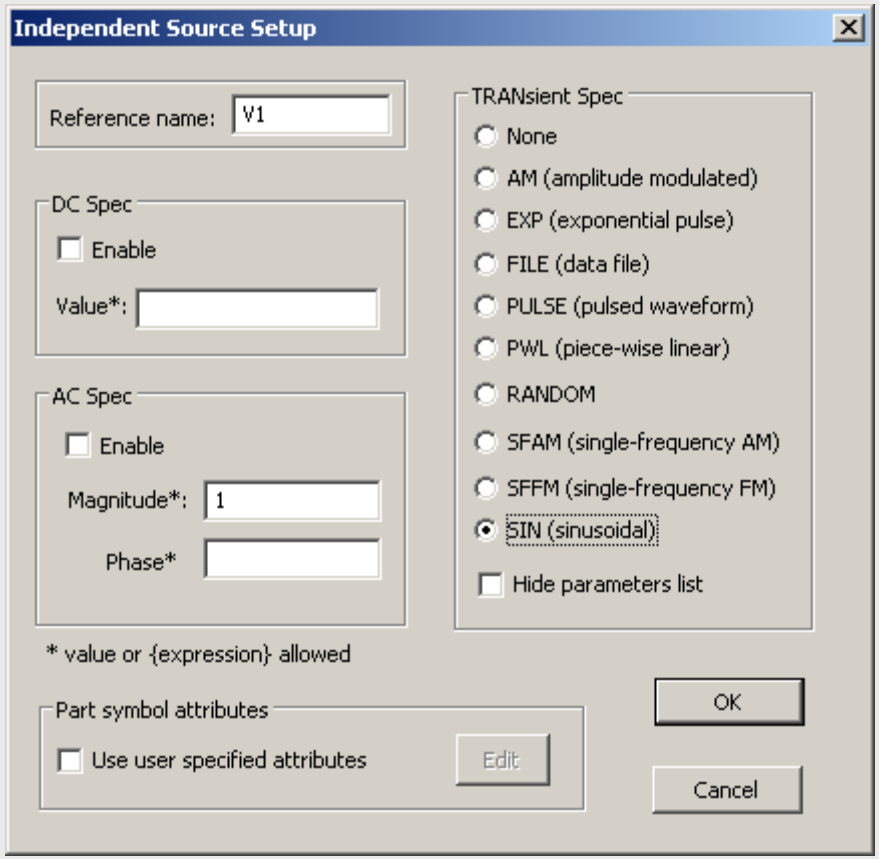
3.2) TopSPICE

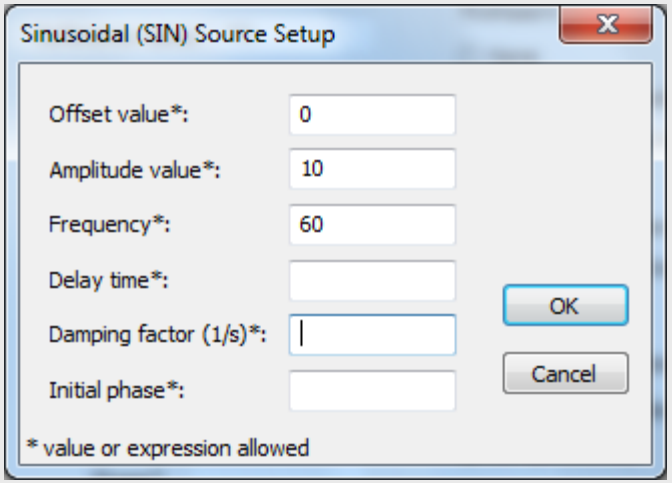
- Definir Rótulos (Insert -> Label Node):



3.2) TopSPICE

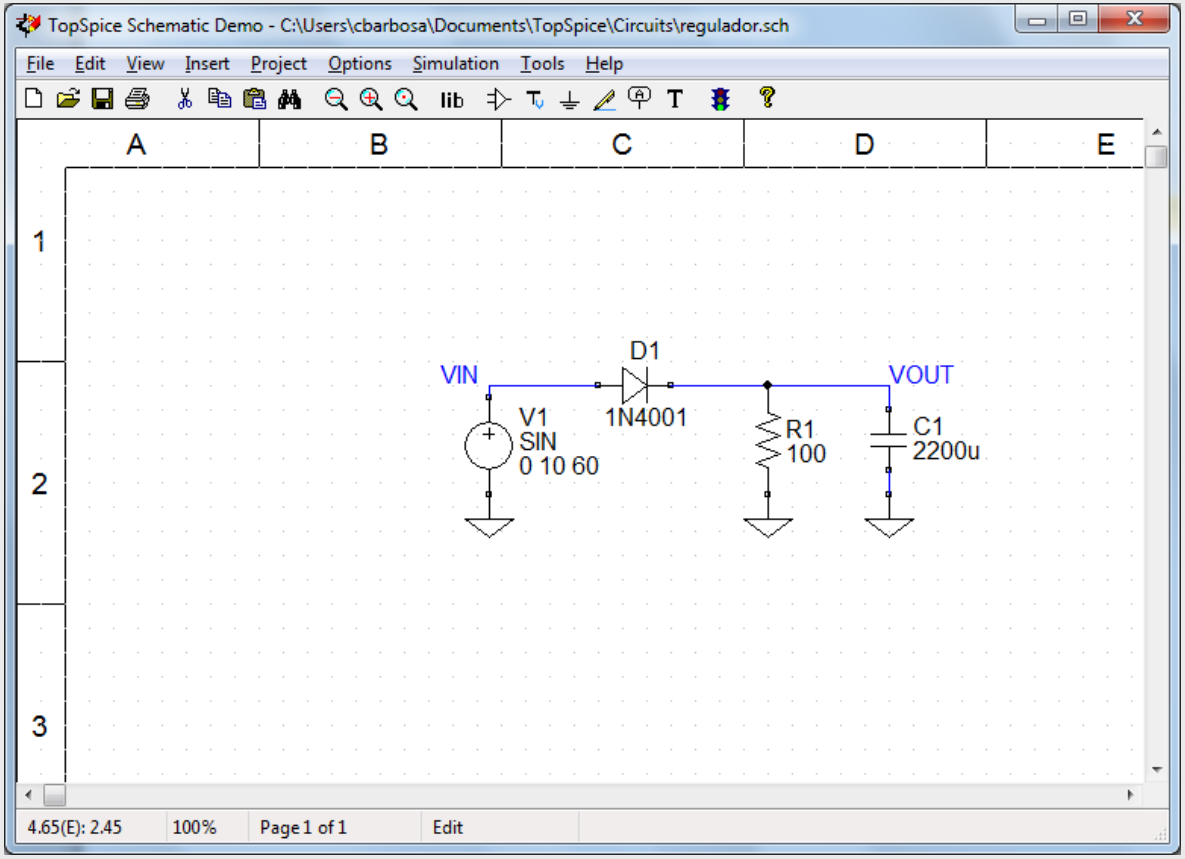
- Definir Fonte de Tensão





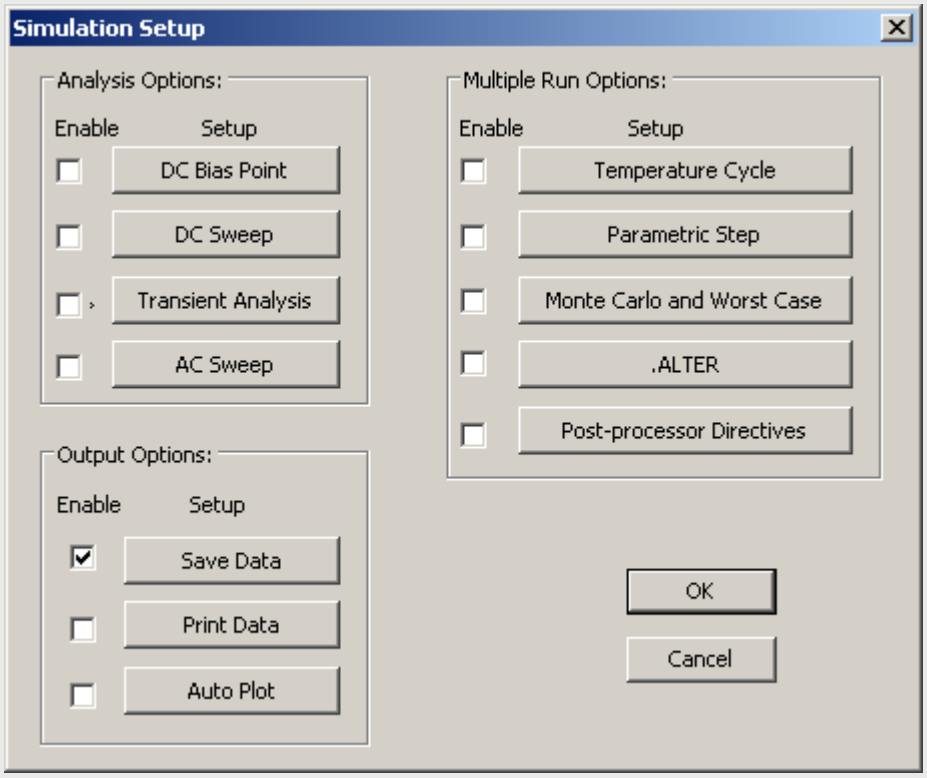
3.2) TopSPICE

- Circuito Parcial:



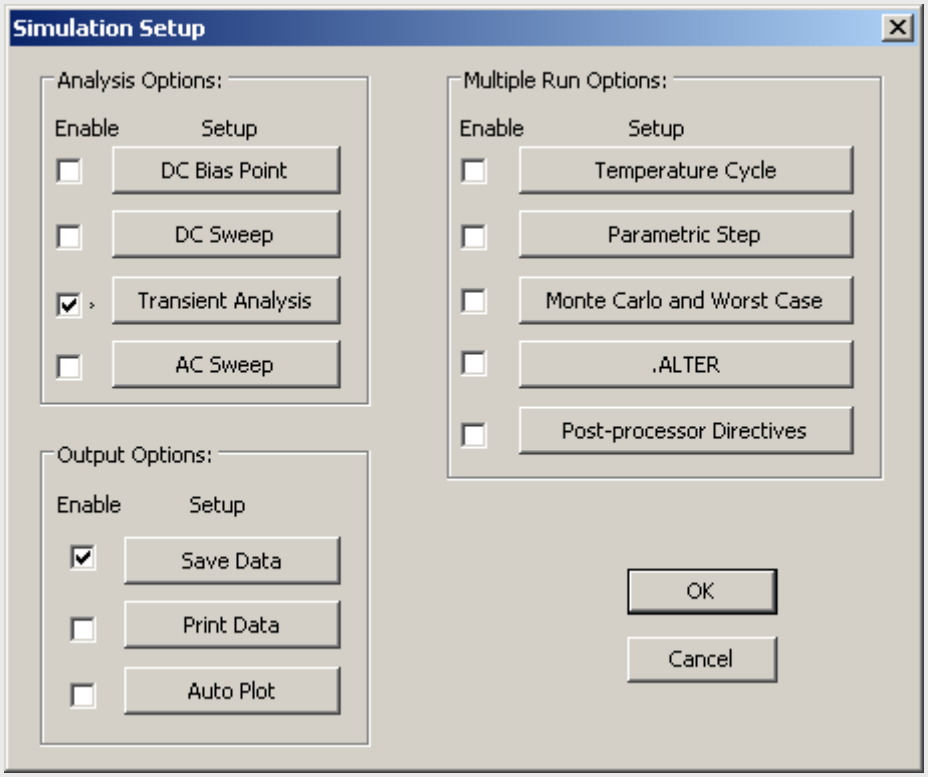
3.2) TopSPICE

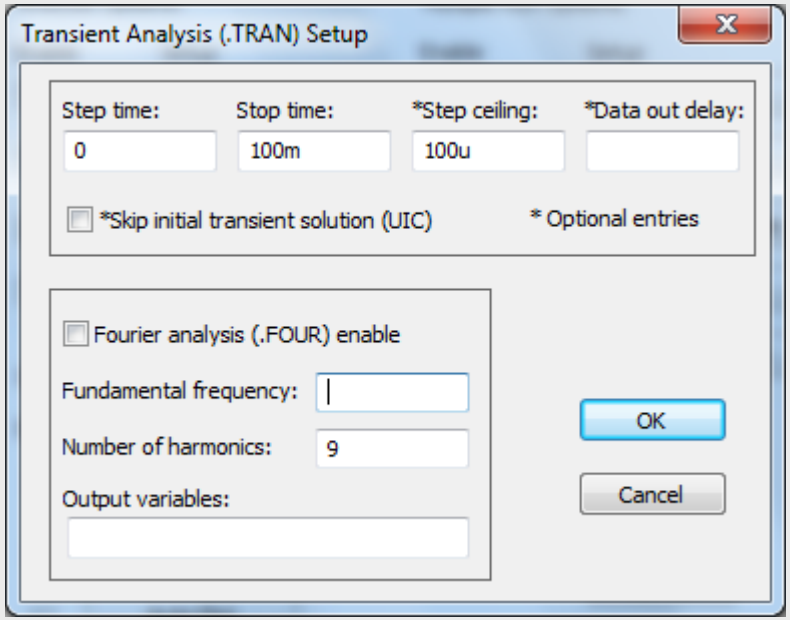
⌘ Configurar Simulação (Simulation -> Setup):



3.2) TopSPICE

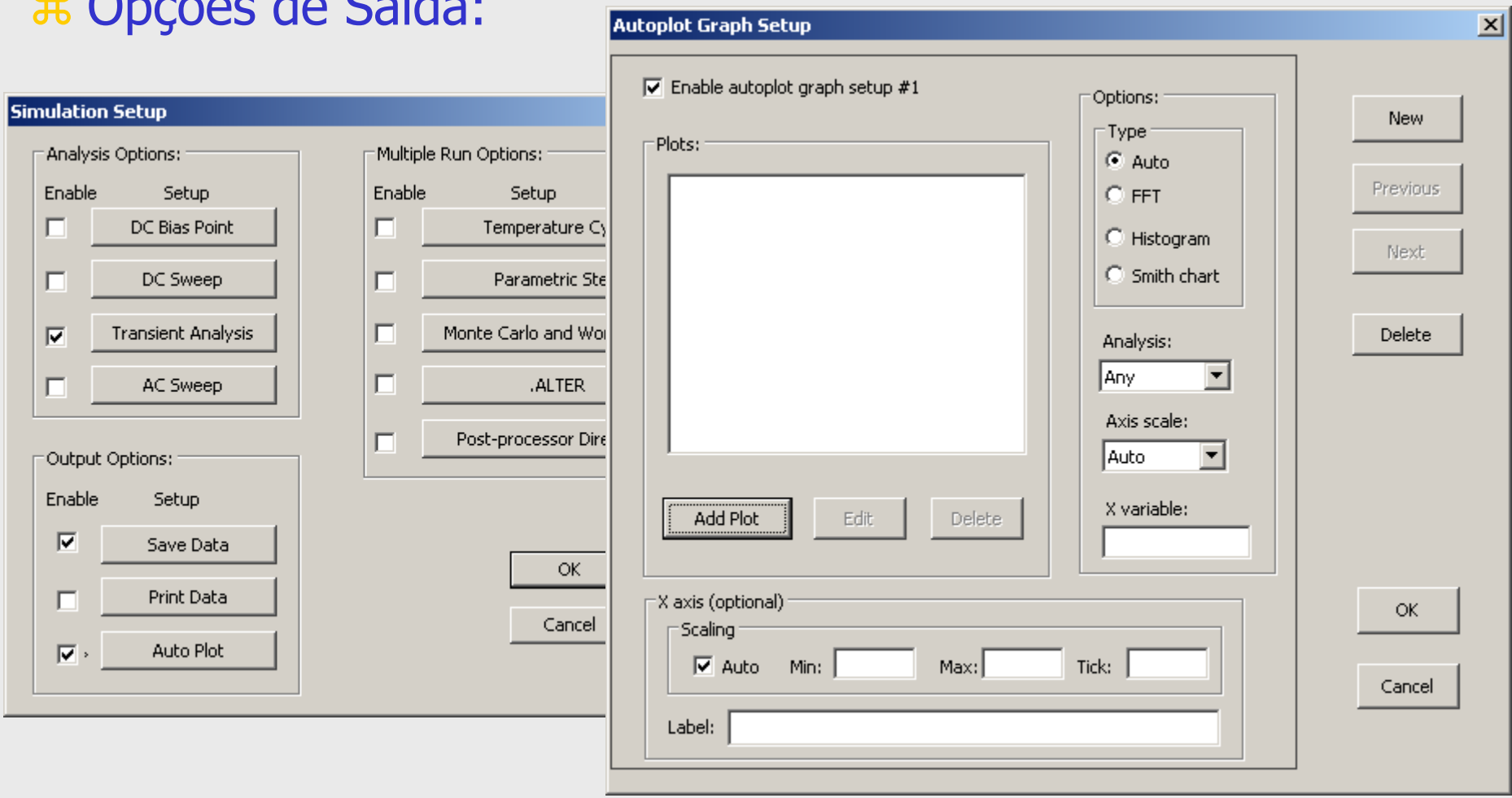
⌘ Análise de Transiente:





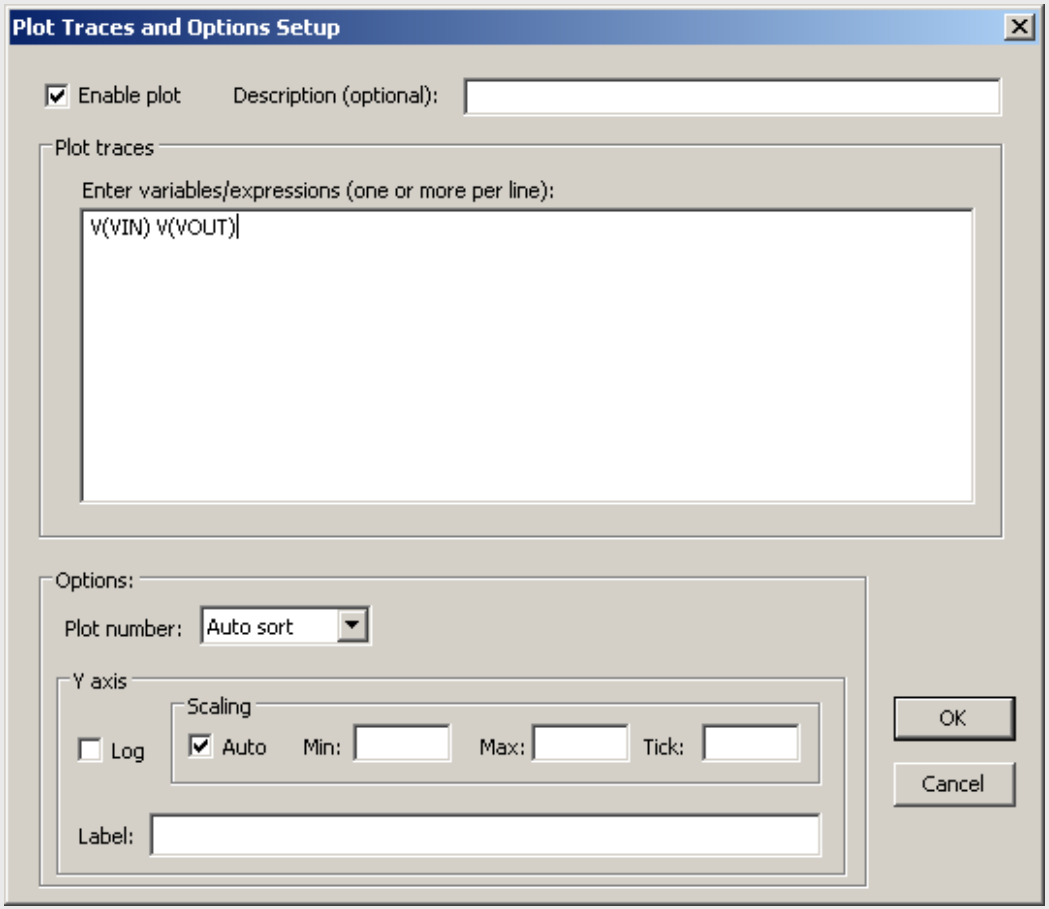
3.2) TopSPICE

⌘ Opções de Saída:



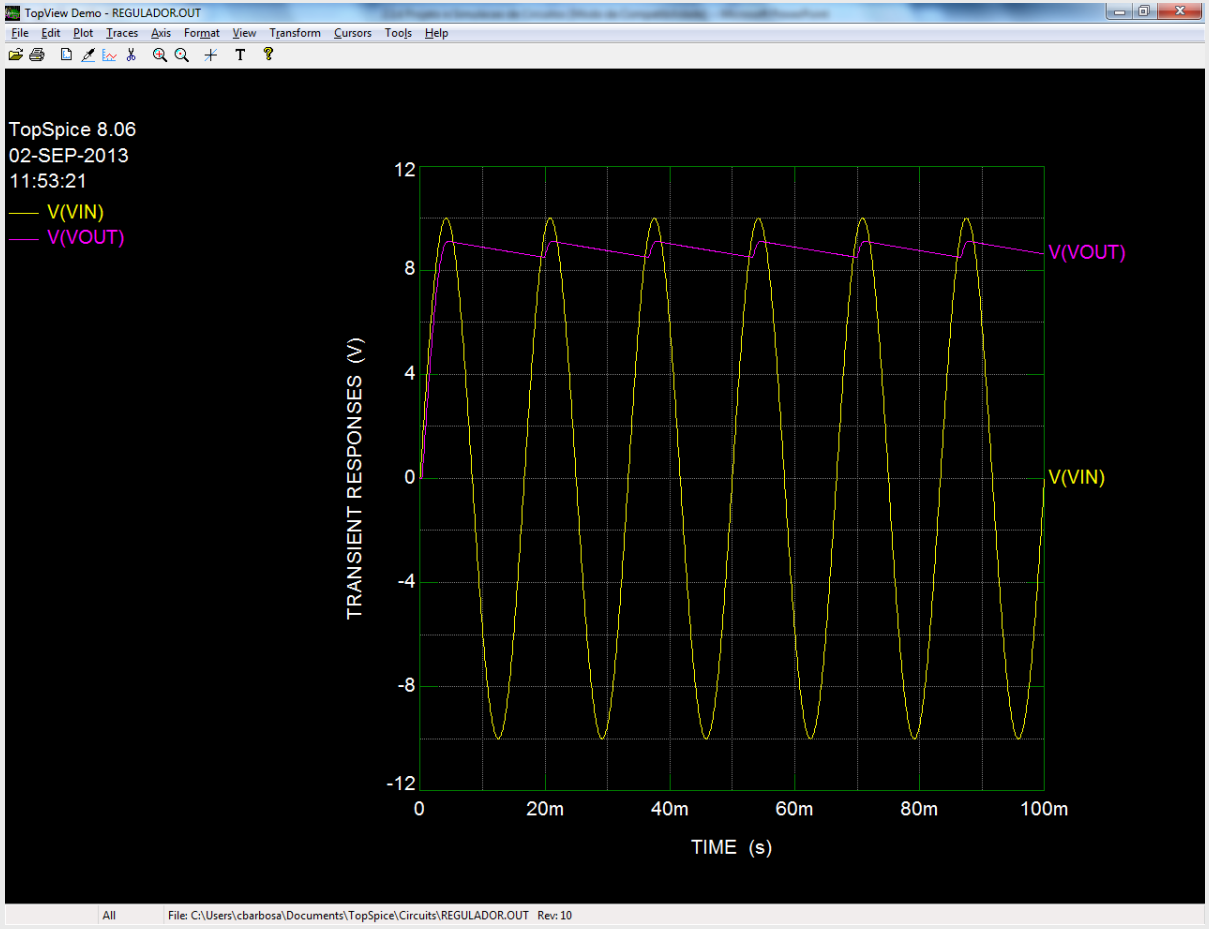
3.2) TopSPICE

⌘ Definição do Gráfico:



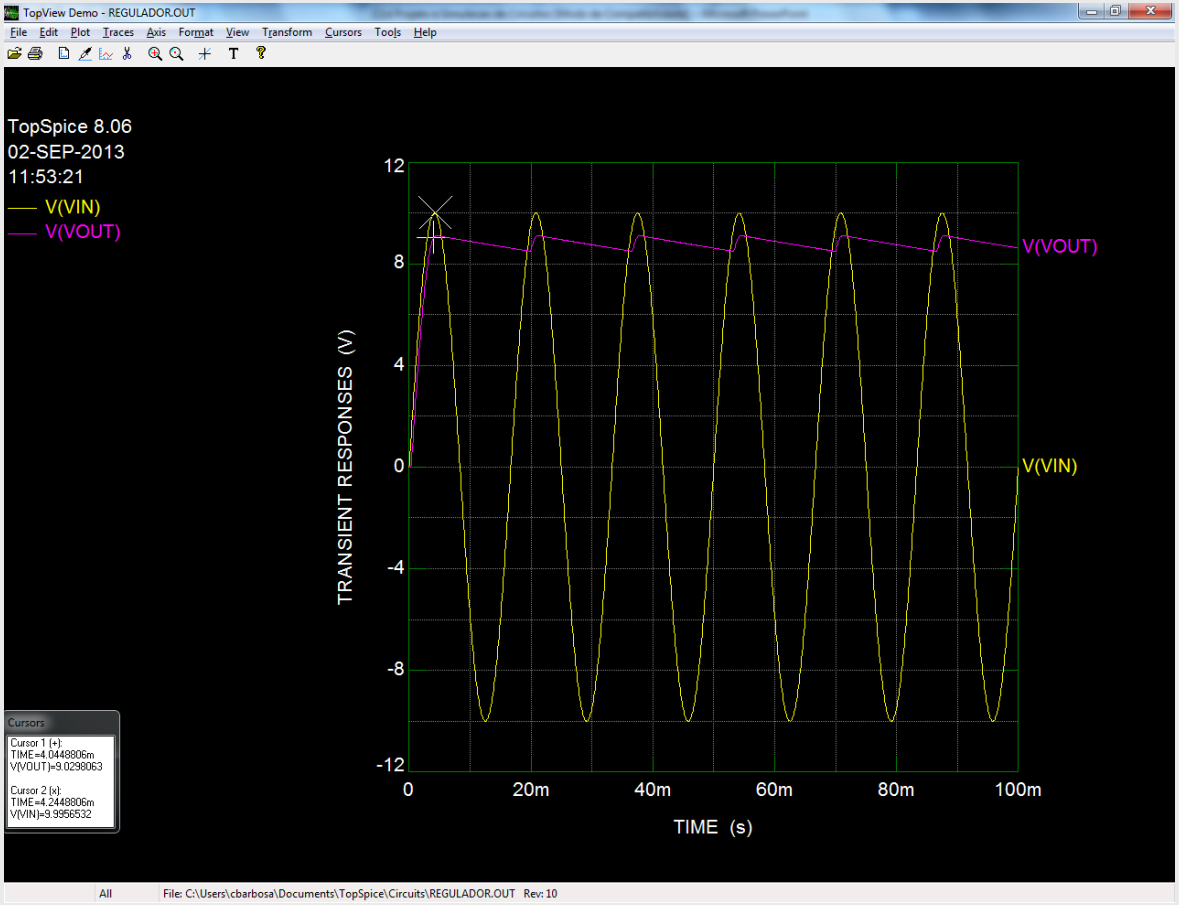
3.2) TopSPICE

- Executar Simulação (F9)



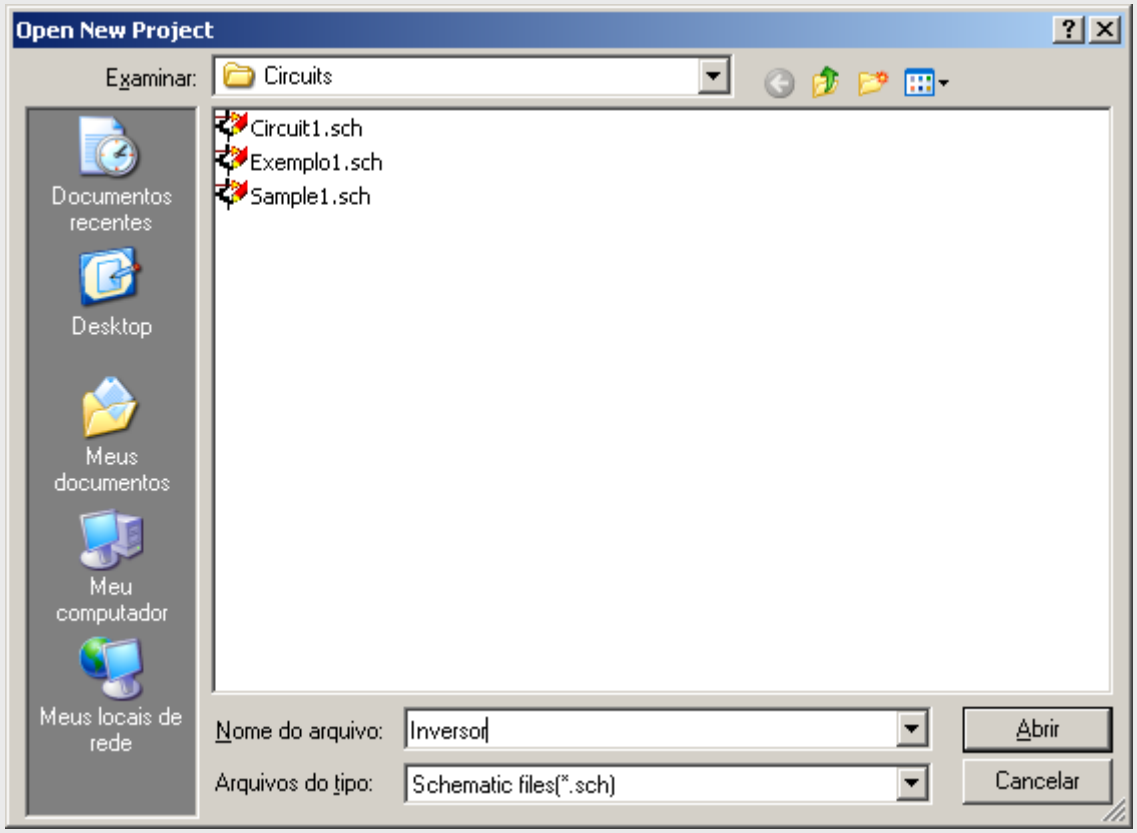
3.2) TopSPICE

- Utilizando Cursores



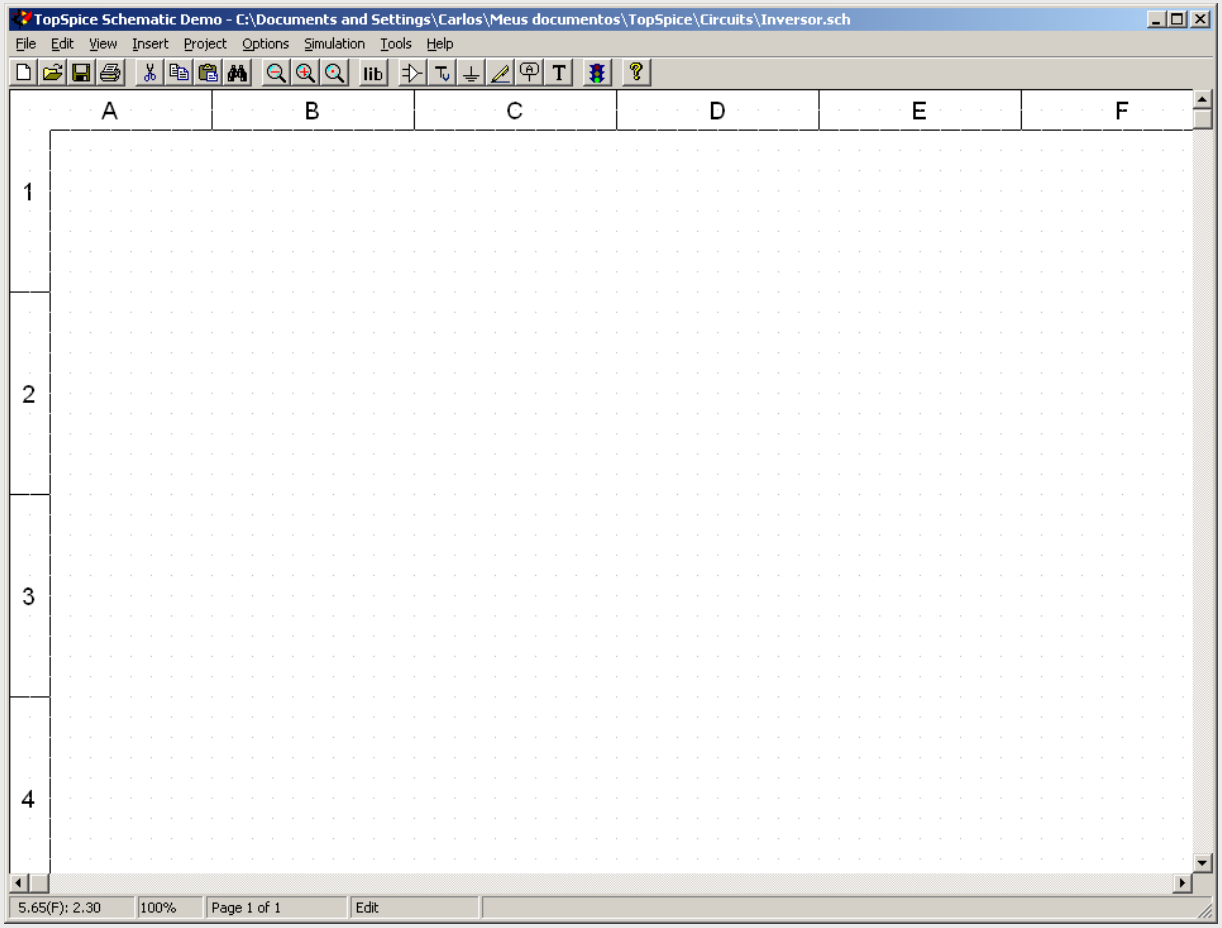
3.2) TopSPICE

- Criando novo projeto



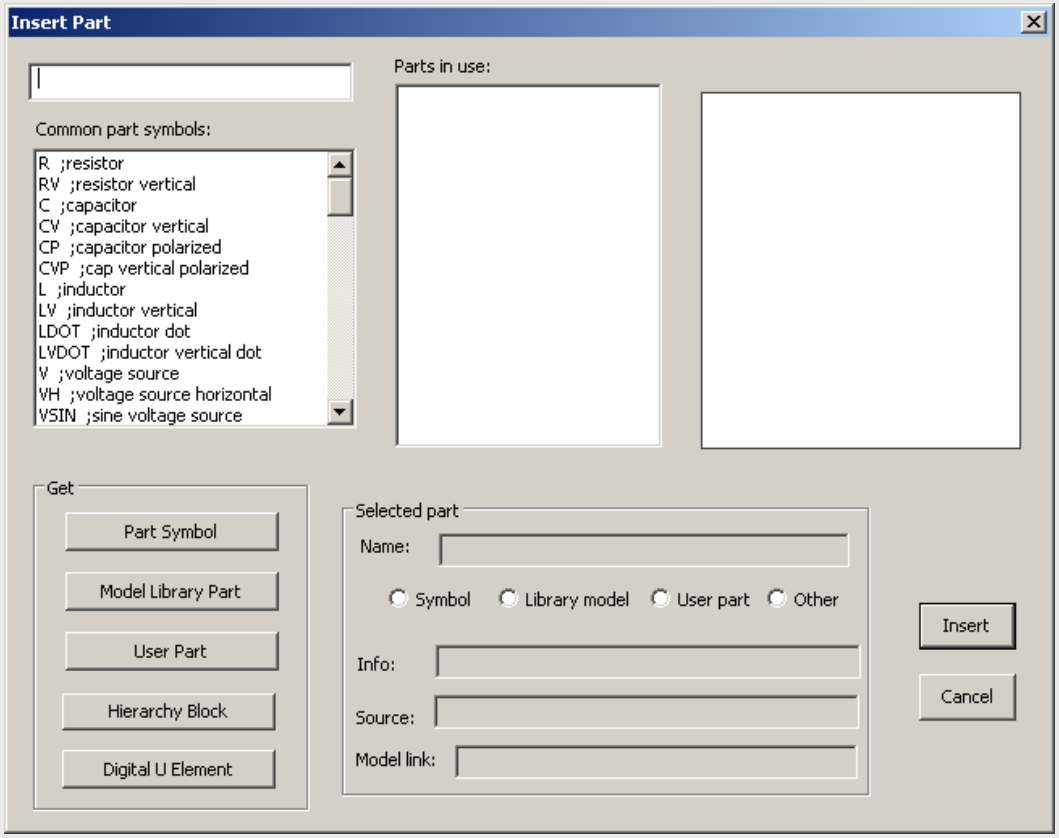
3.2) TopSPICE

- Editor de Esquemático



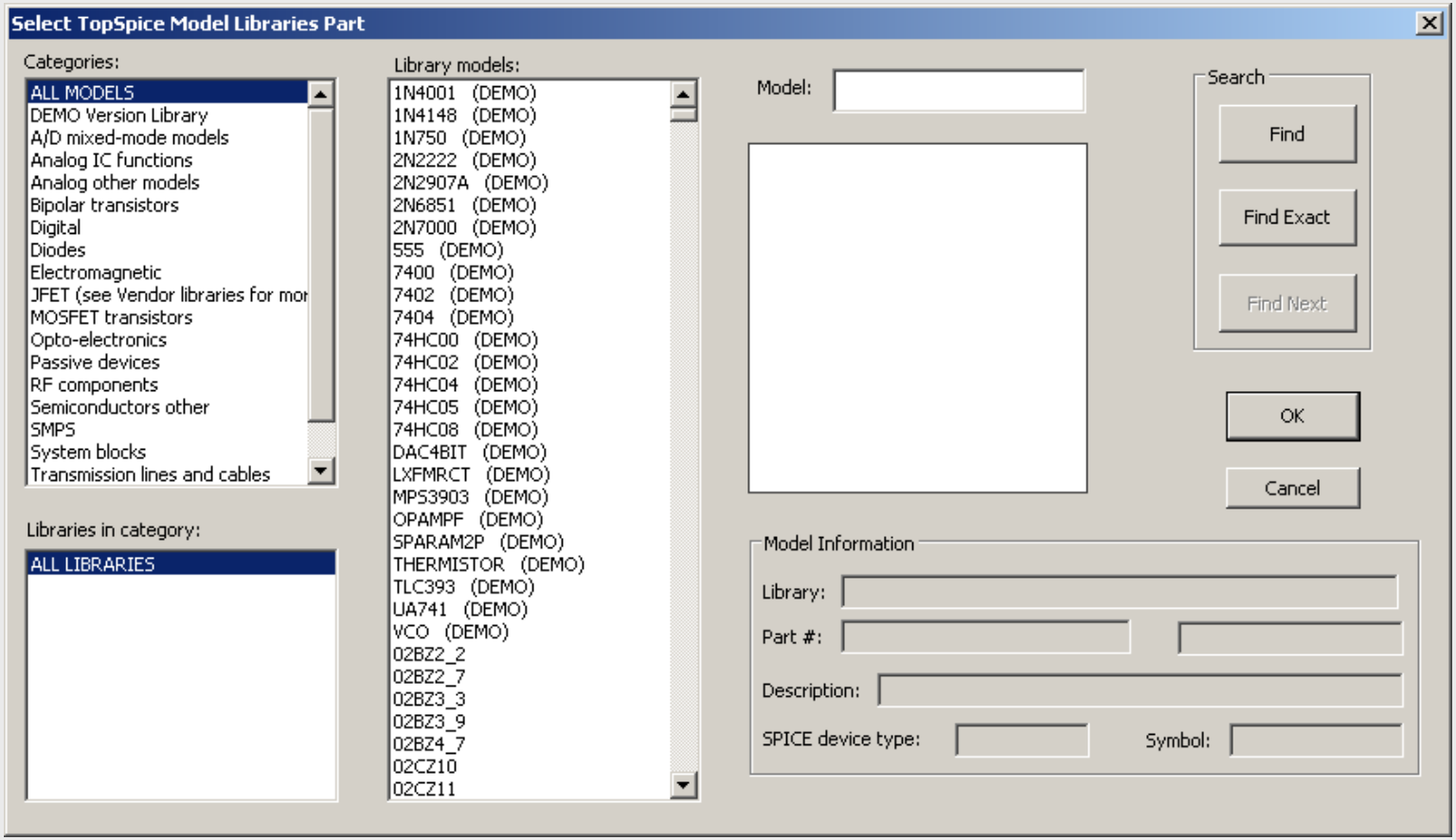
3.2) TopSPICE

- Inserir componente (Insert -> Part): Diodo Zener



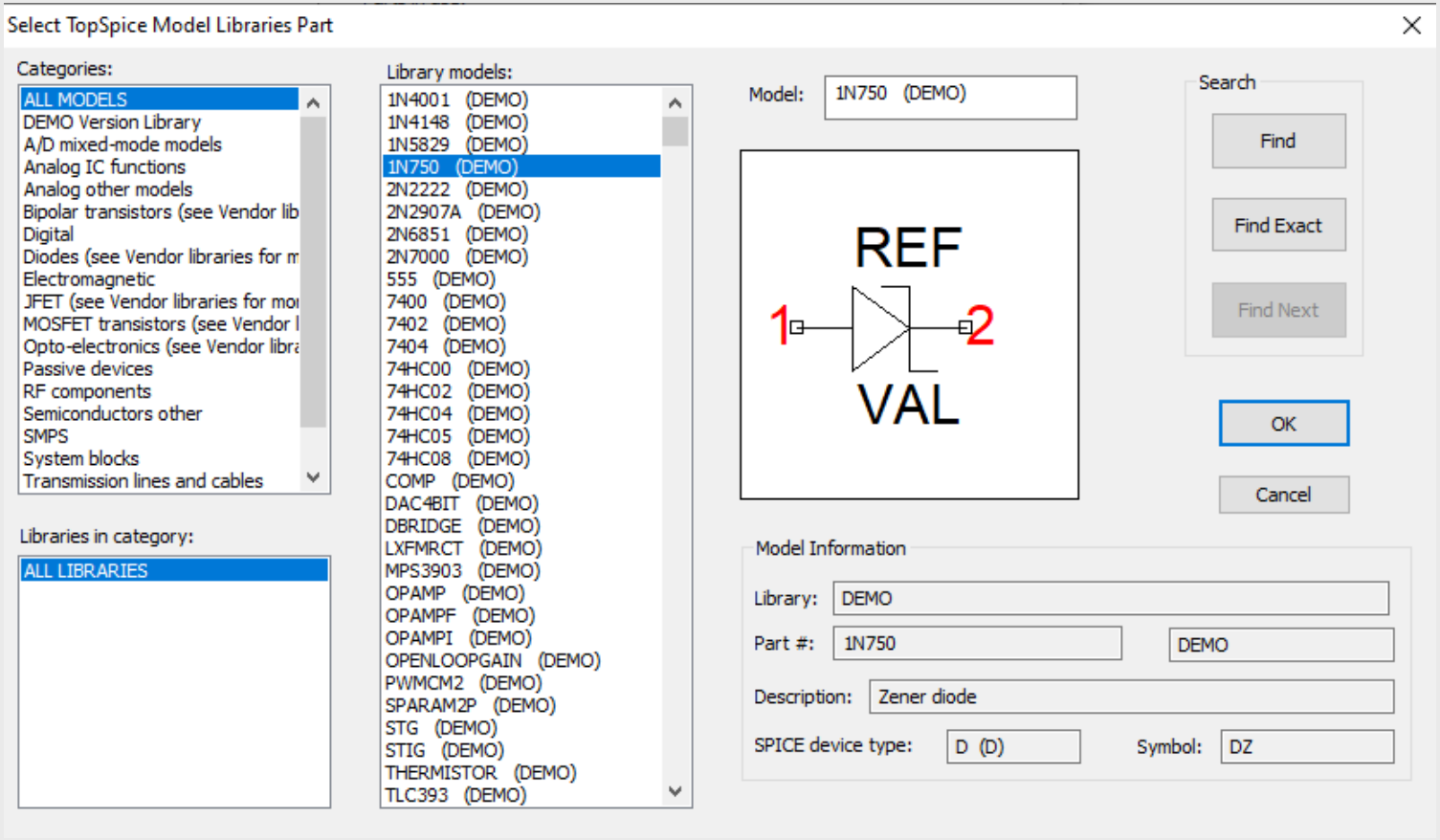
3.2) TopSPICE

- Biblioteca de componentes (Model Library Part)



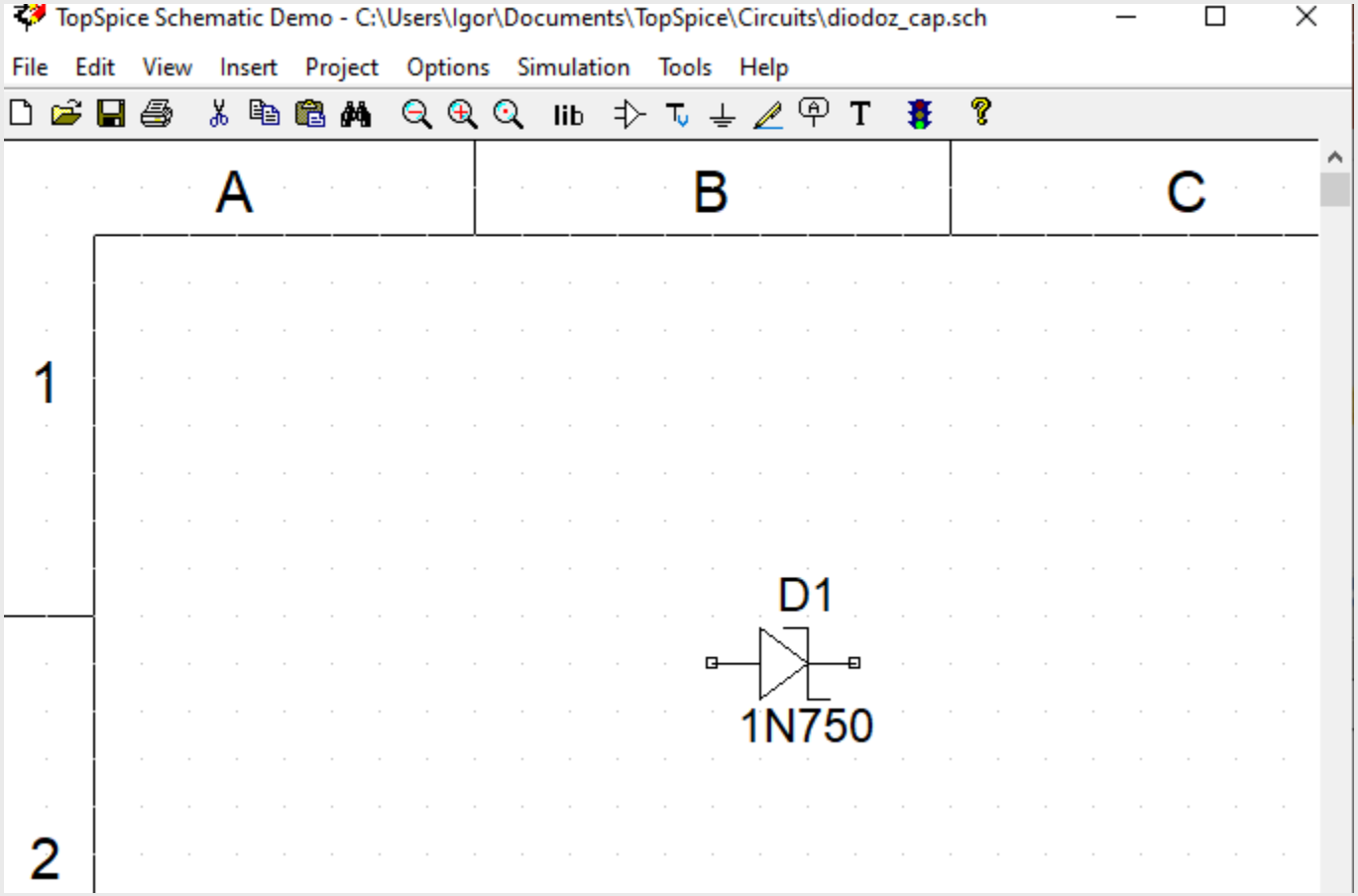
3.2) TopSPICE

- Diodo Zener



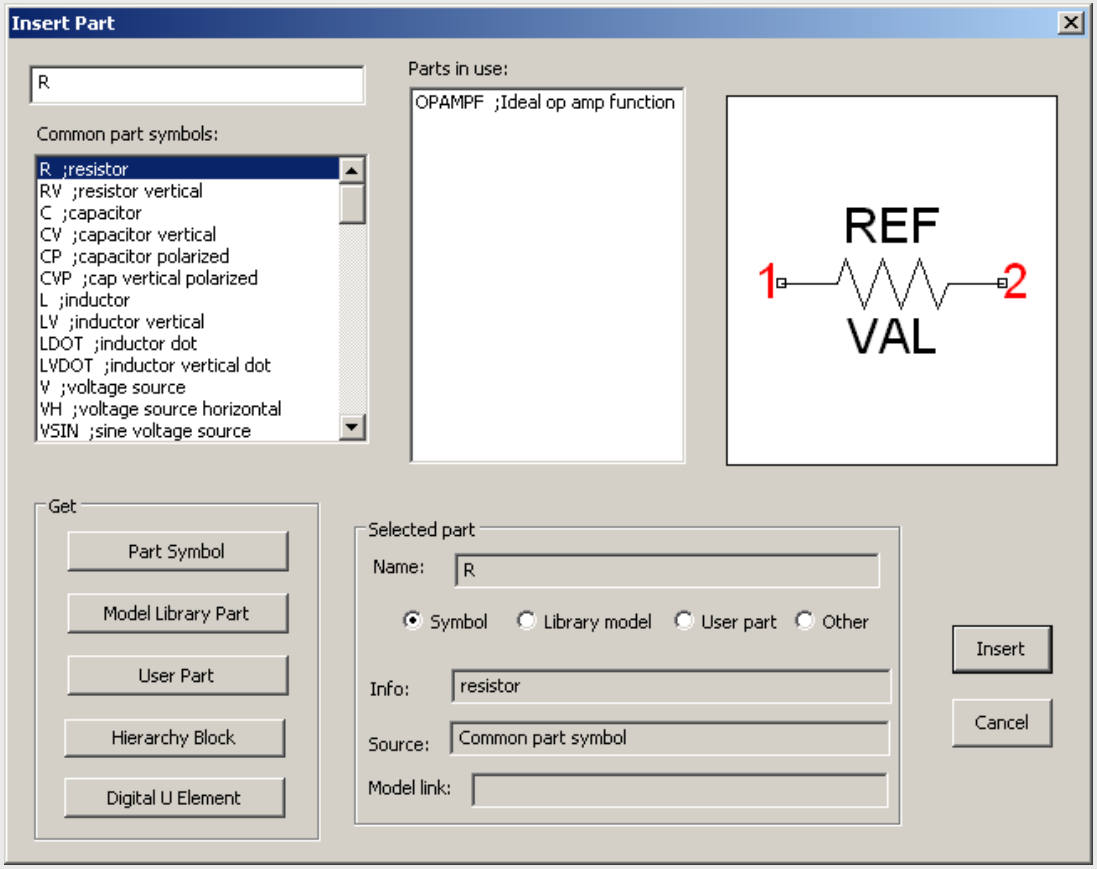
3.2) TopSPICE

- Circuito Parcial:



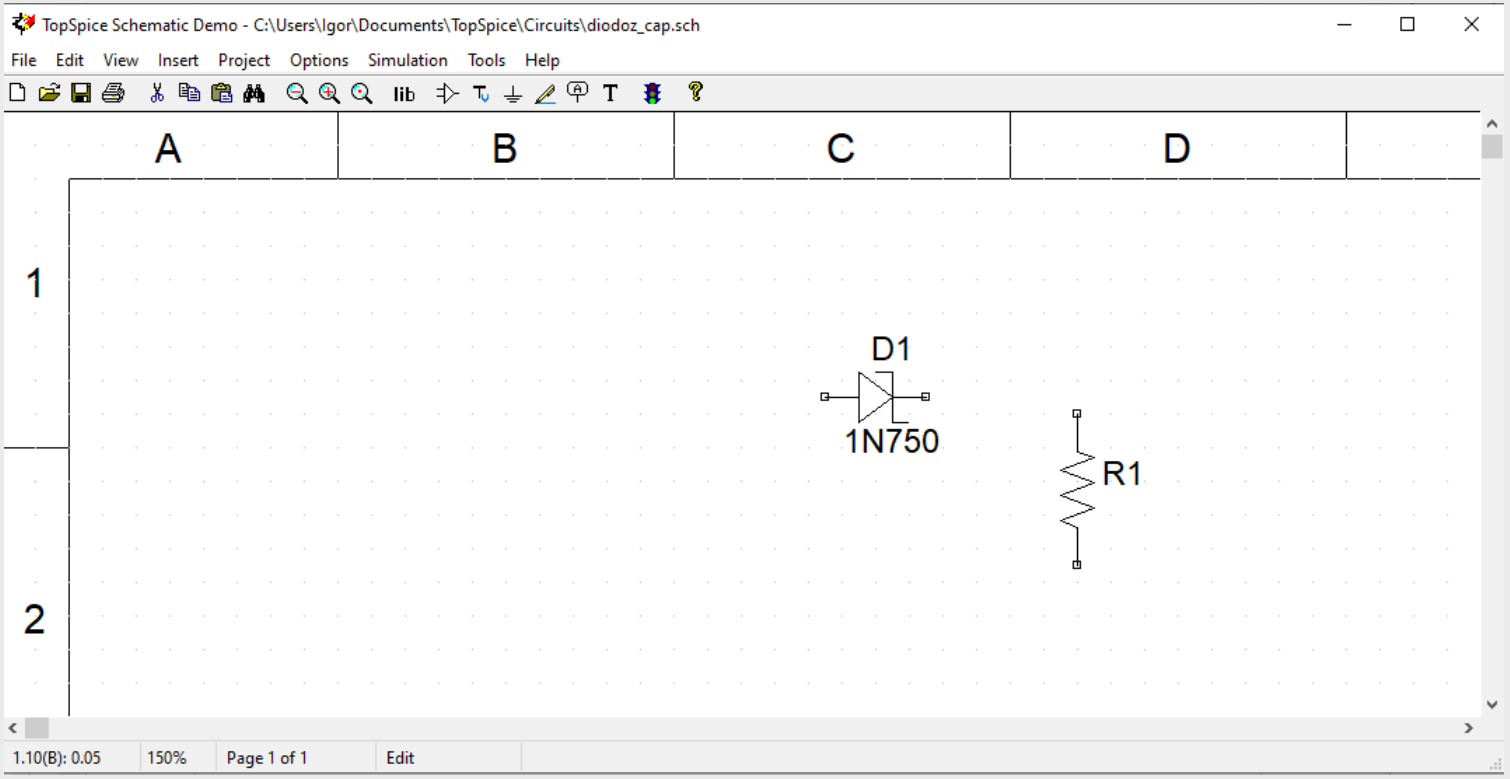
3.2) TopSPICE

- Inserir componente (Insert -> Part): Resistores e fonte



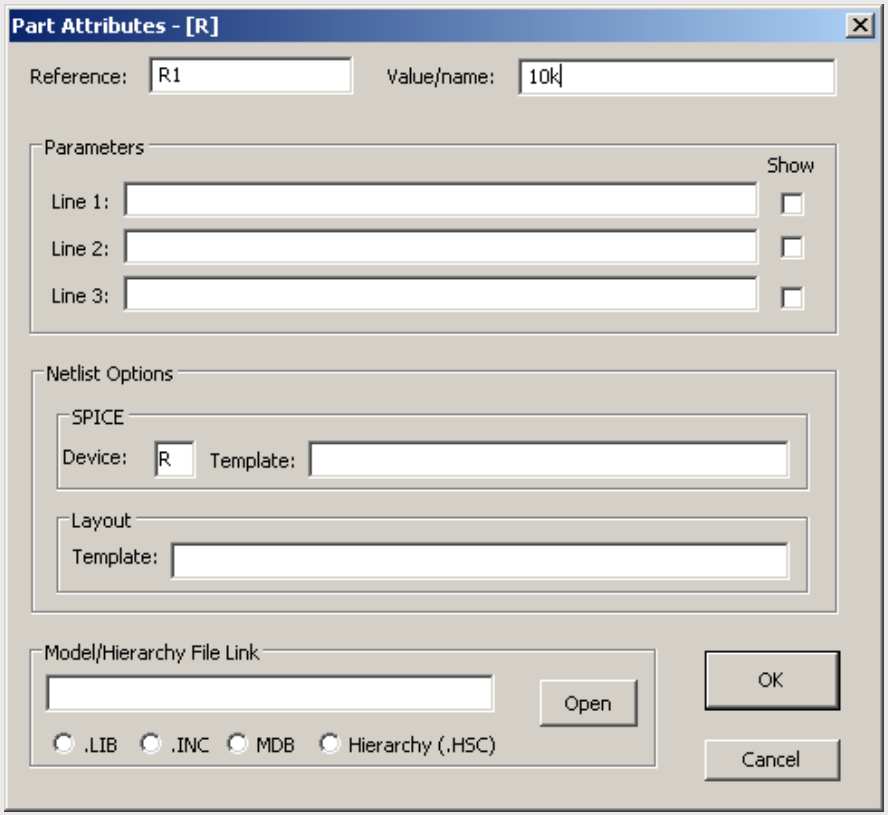
3.2) TopSPICE

⌘ Circuito Parcial:



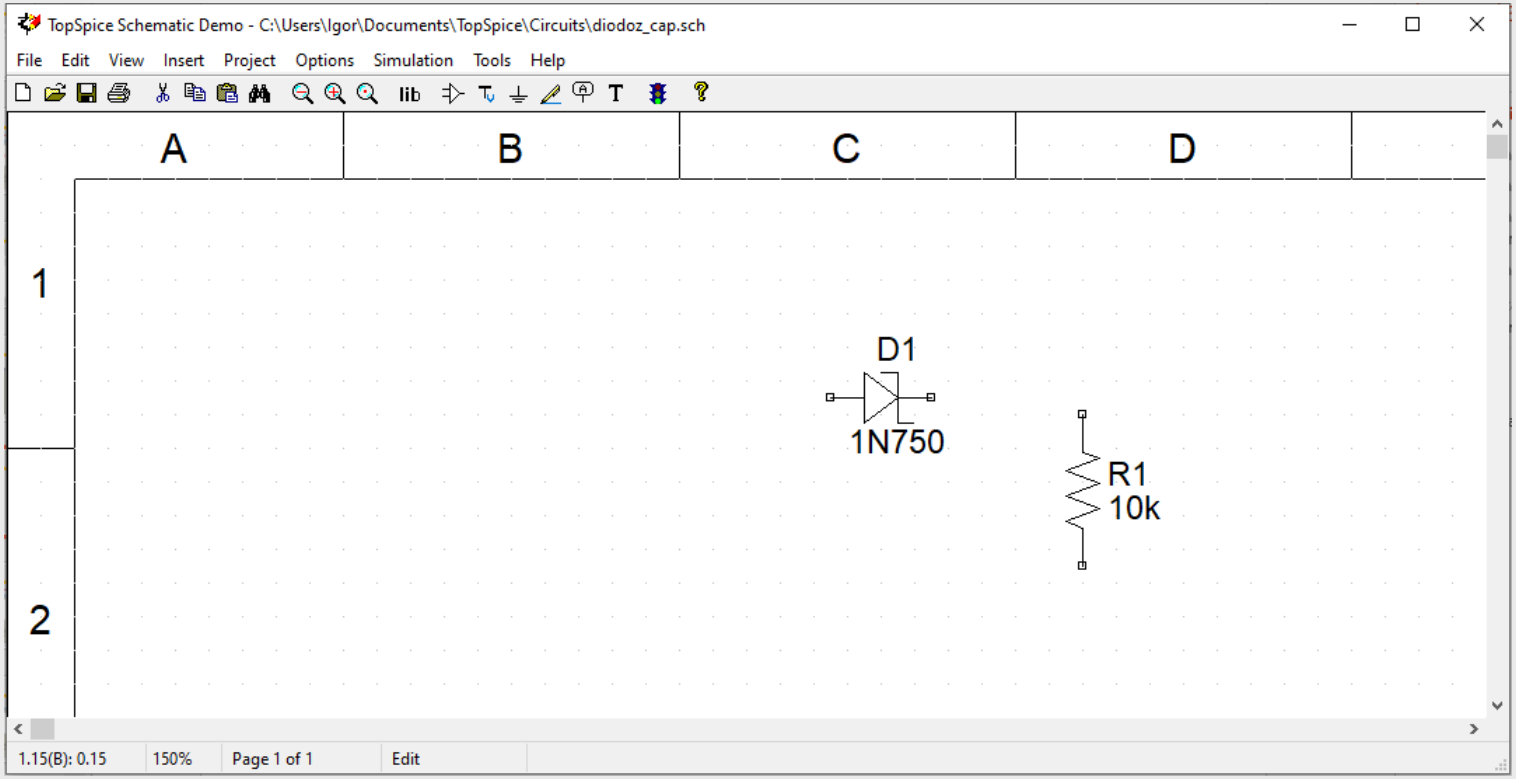
3.2) TopSPICE

⌘ Parâmetros dos componentes:



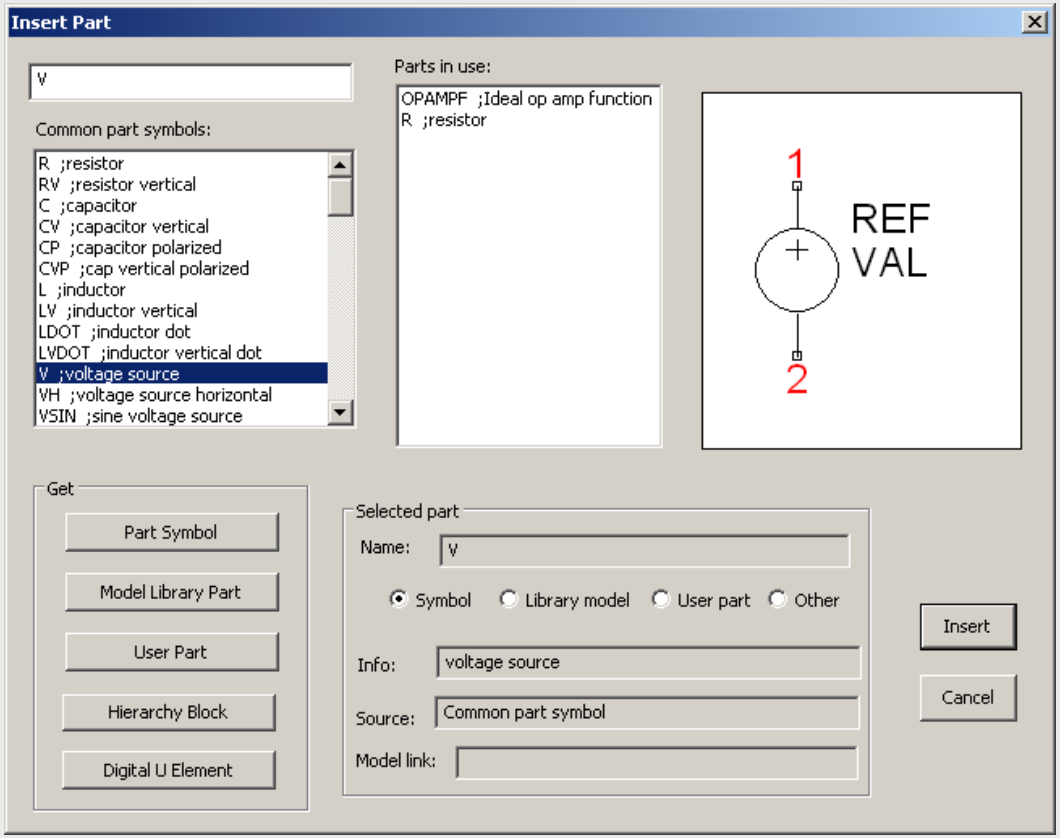
3.2) TopSPICE

⌘ Circuito Parcial:



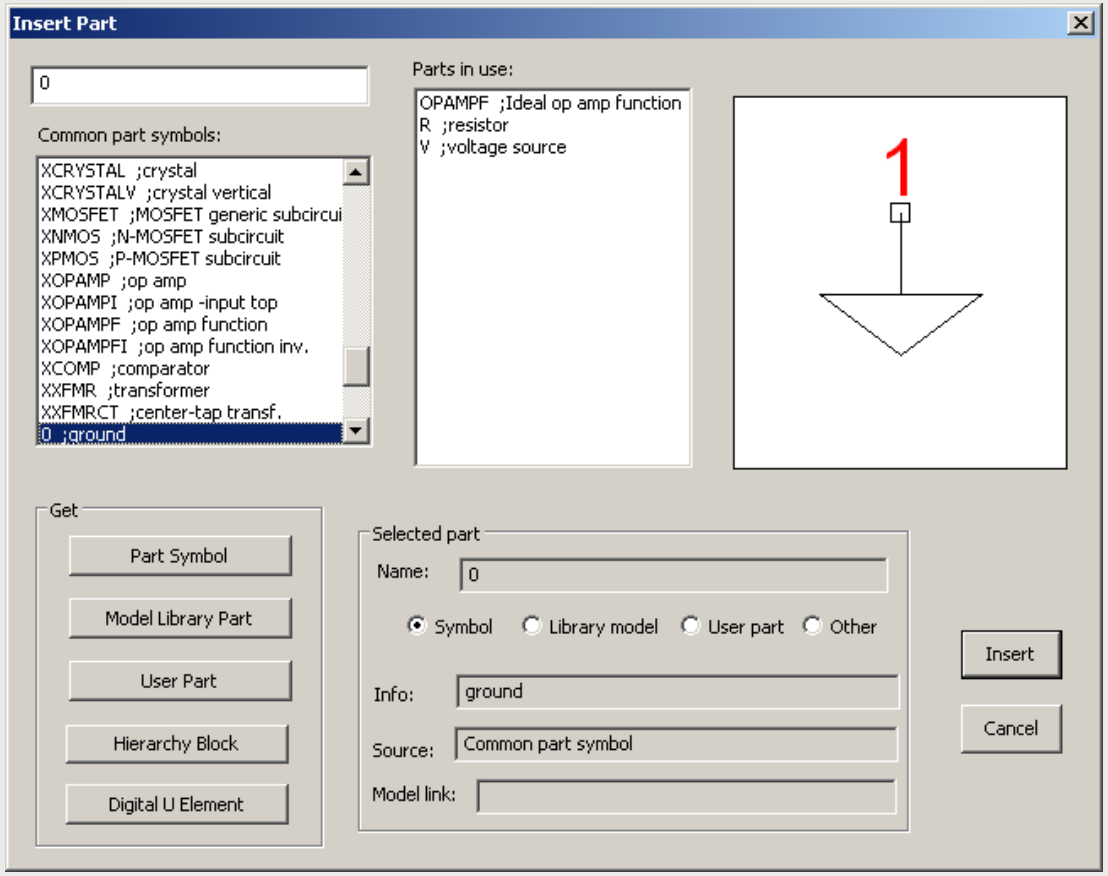
3.2) TopSPICE

- Inserir componente (Insert -> Part): Fonte de Tensão



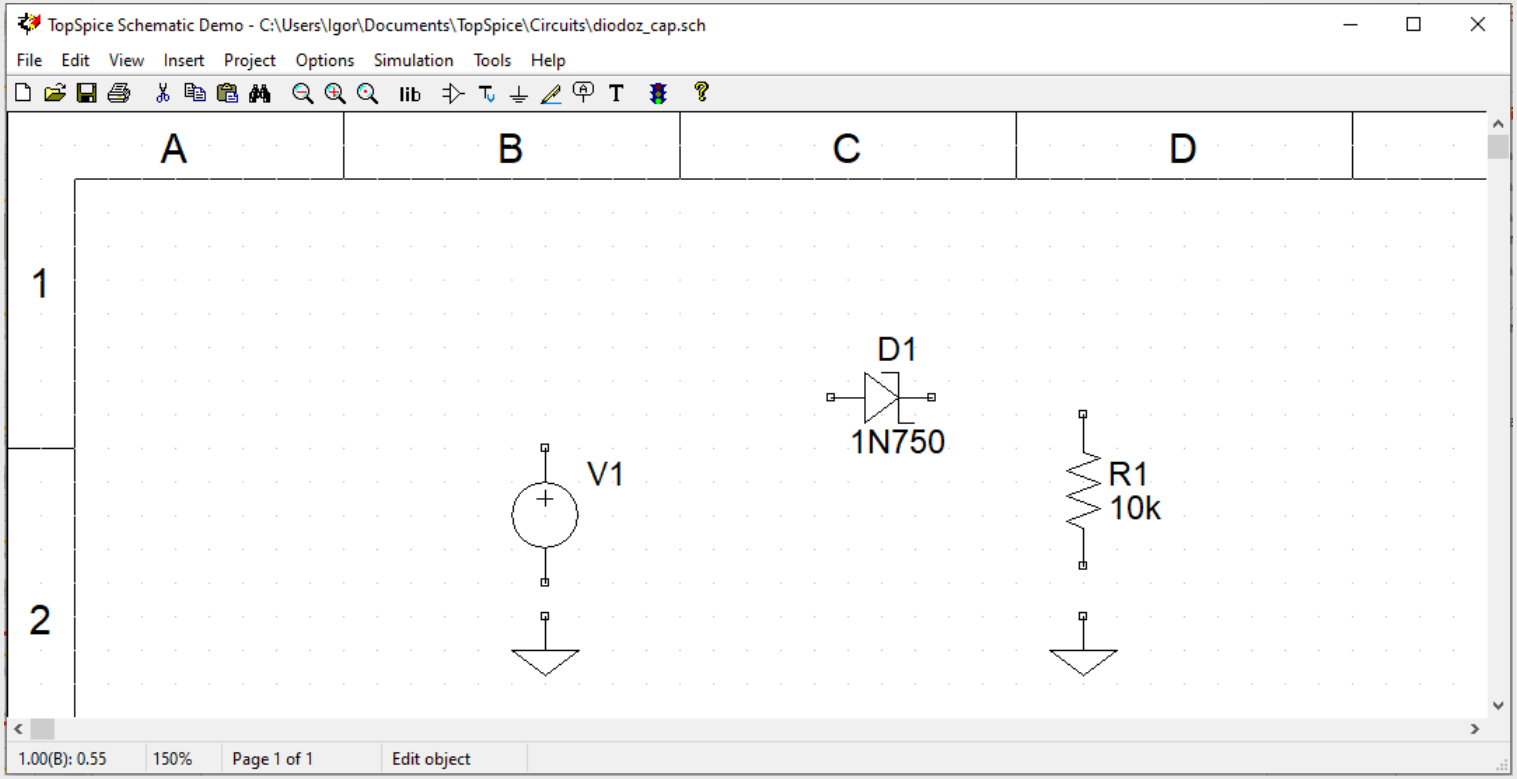
3.2) TopSPICE

- Inserir componente (Insert -> Part): Terras



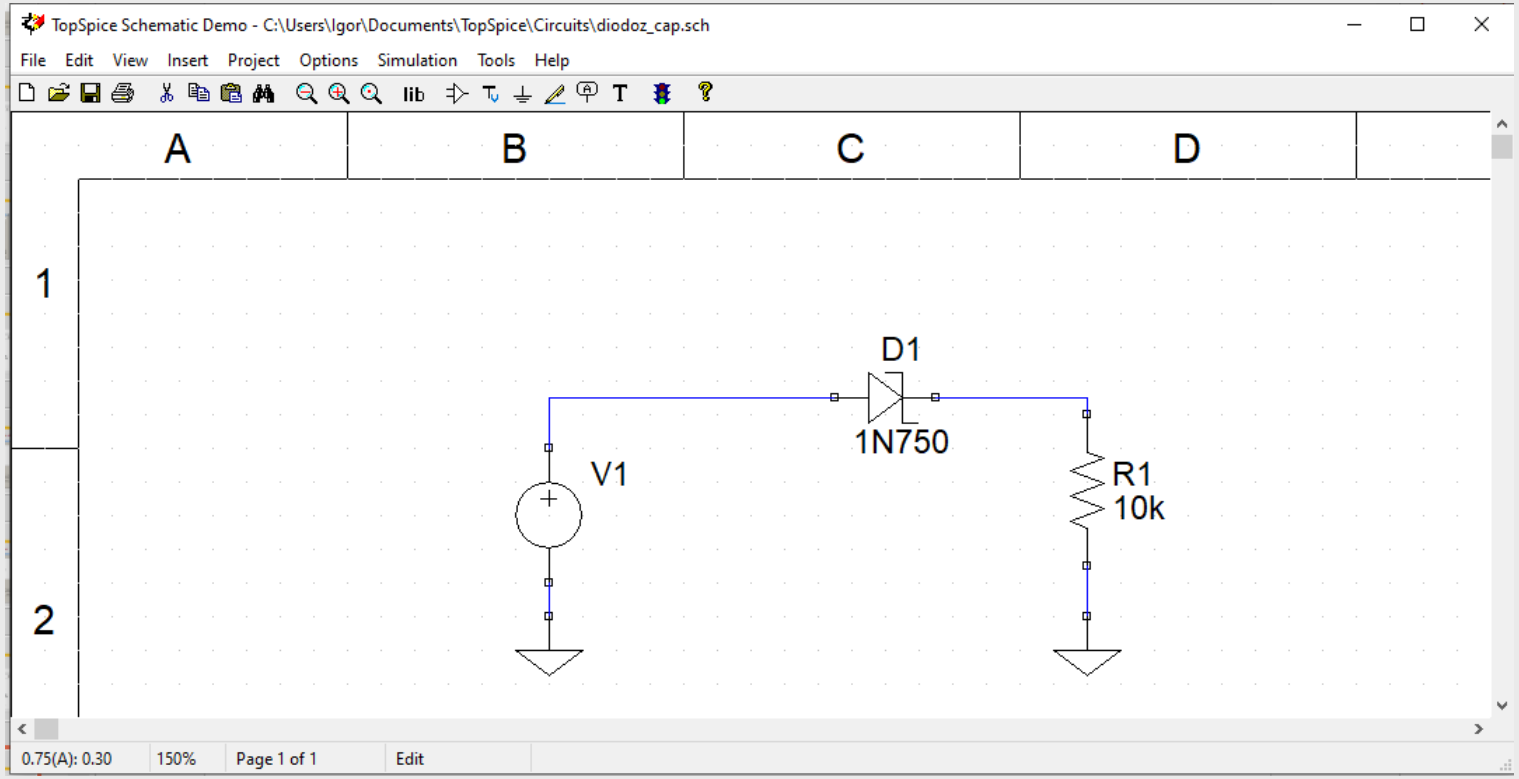
3.2) TopSPICE

⌘ Circuito Parcial:



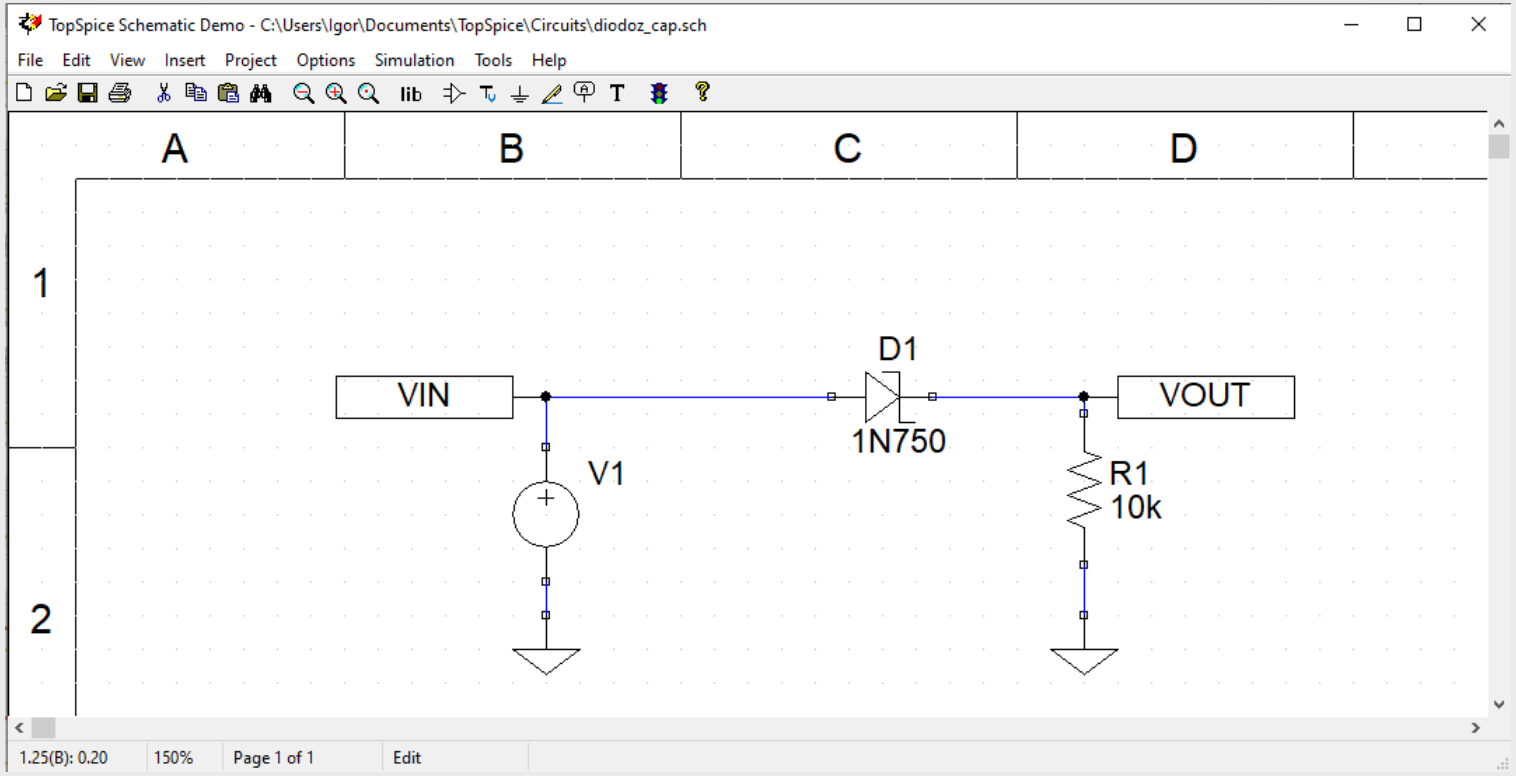
3.2) TopSPICE

- Inserir fiação (Insert -> Wire):



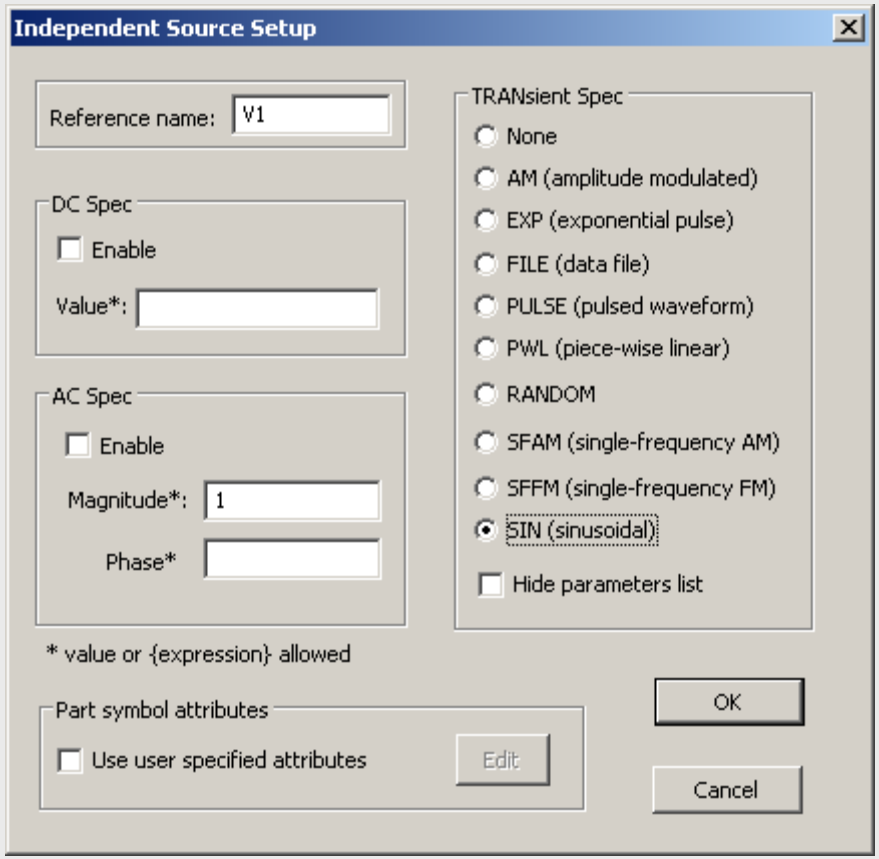
3.2) TopSPICE

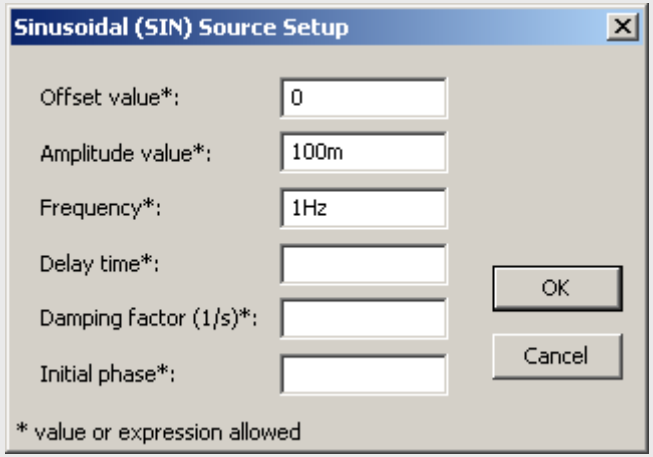
- Definir Rótulos (Insert -> Label Node):



3.2) TopSPICE

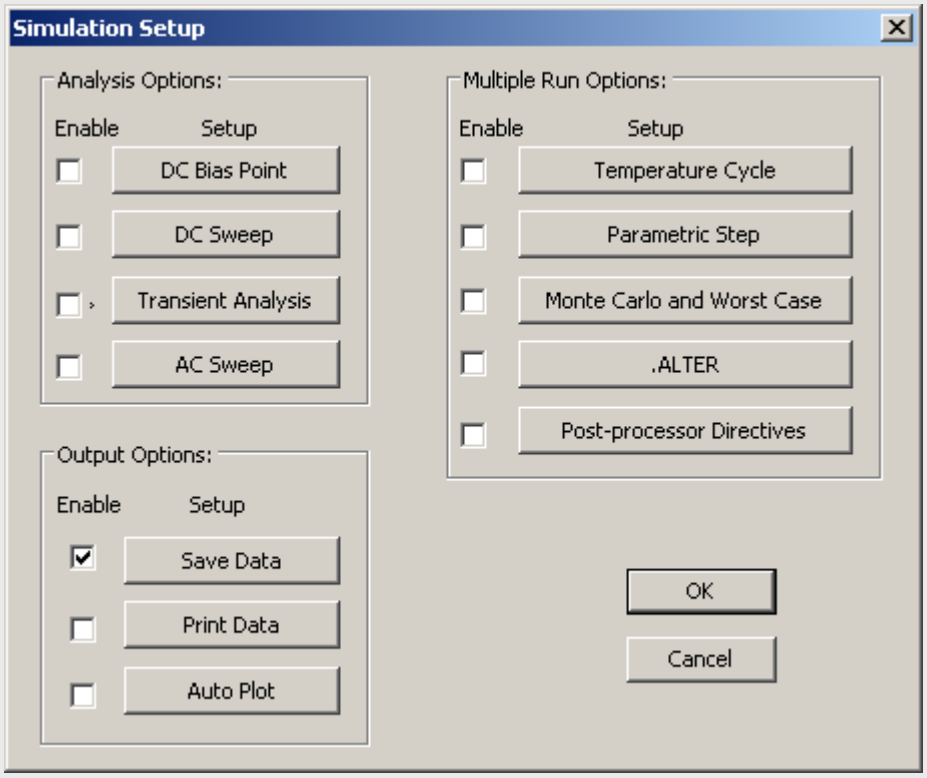
- Definir Fonte de Tensão





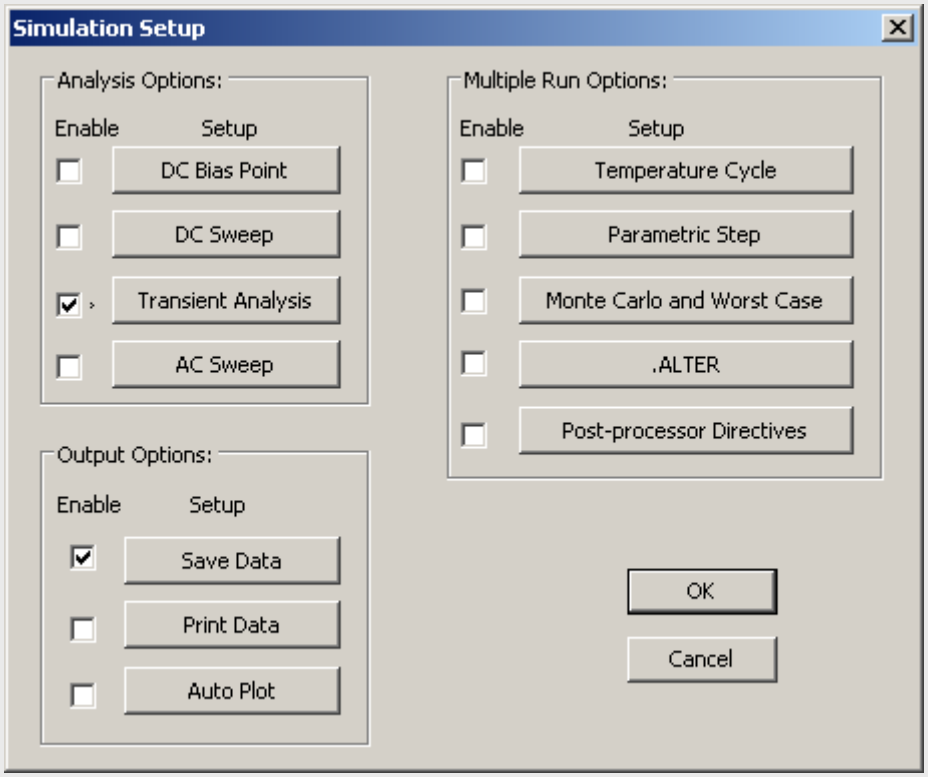
3.2) TopSPICE

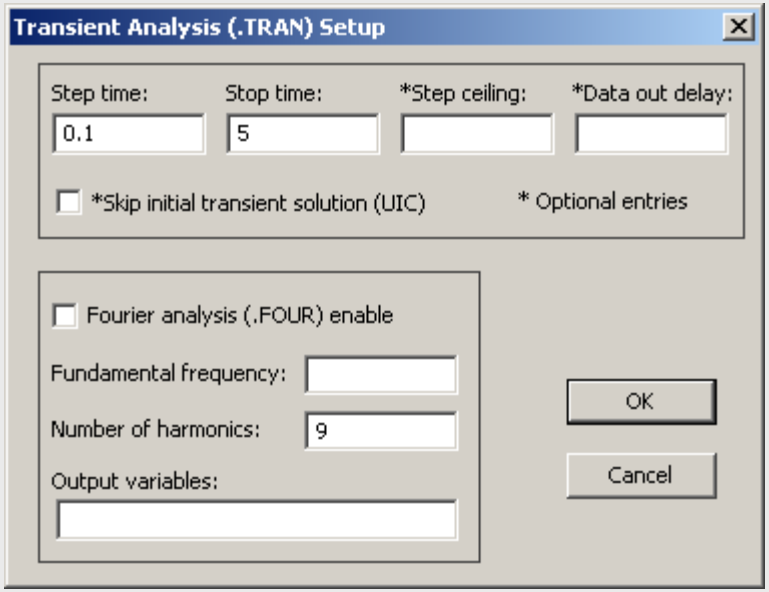
⌘ Configurar Simulação (Simulation -> Setup):



3.2) TopSPICE

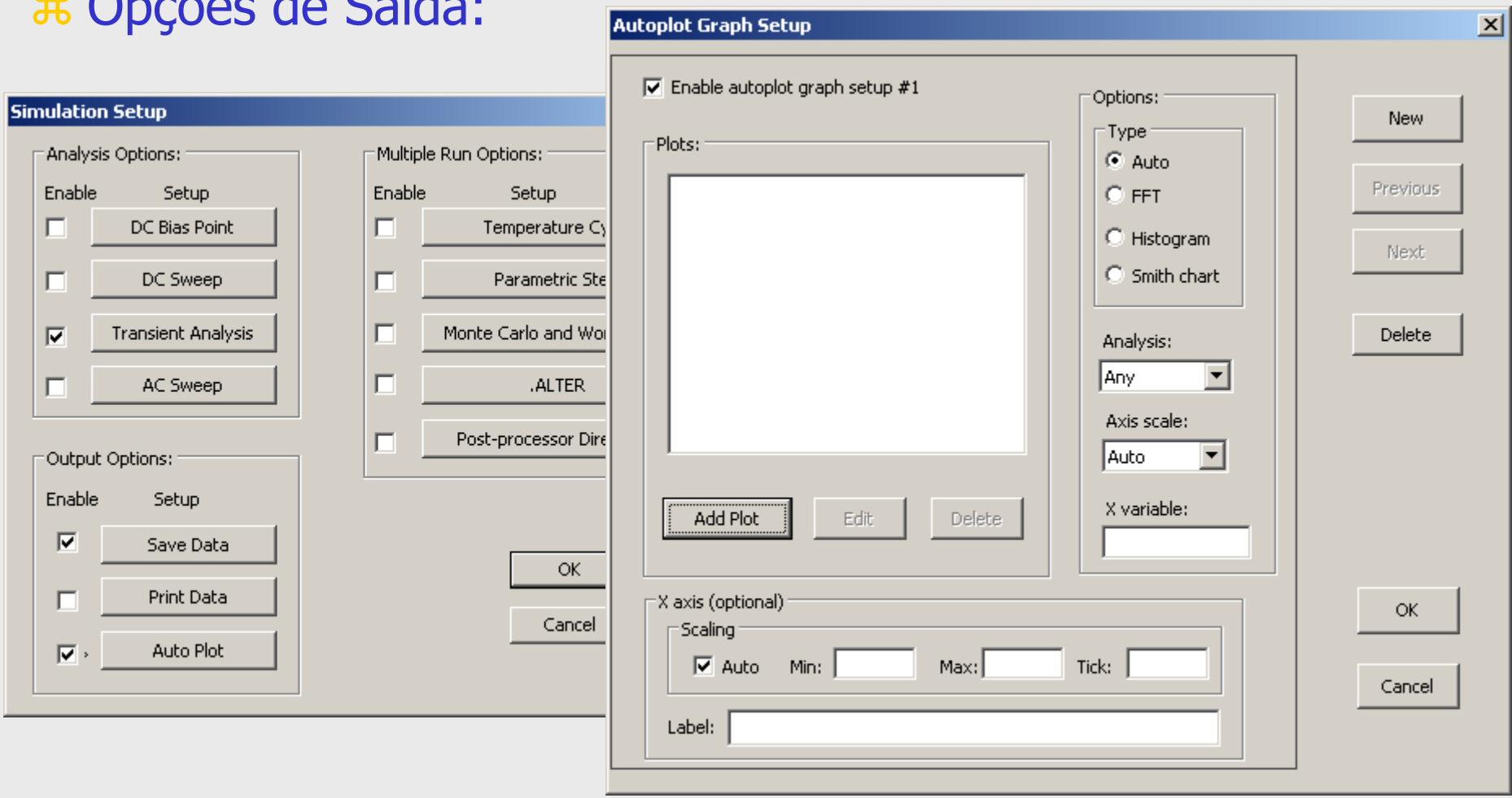
⌘ Análise de Transiente:





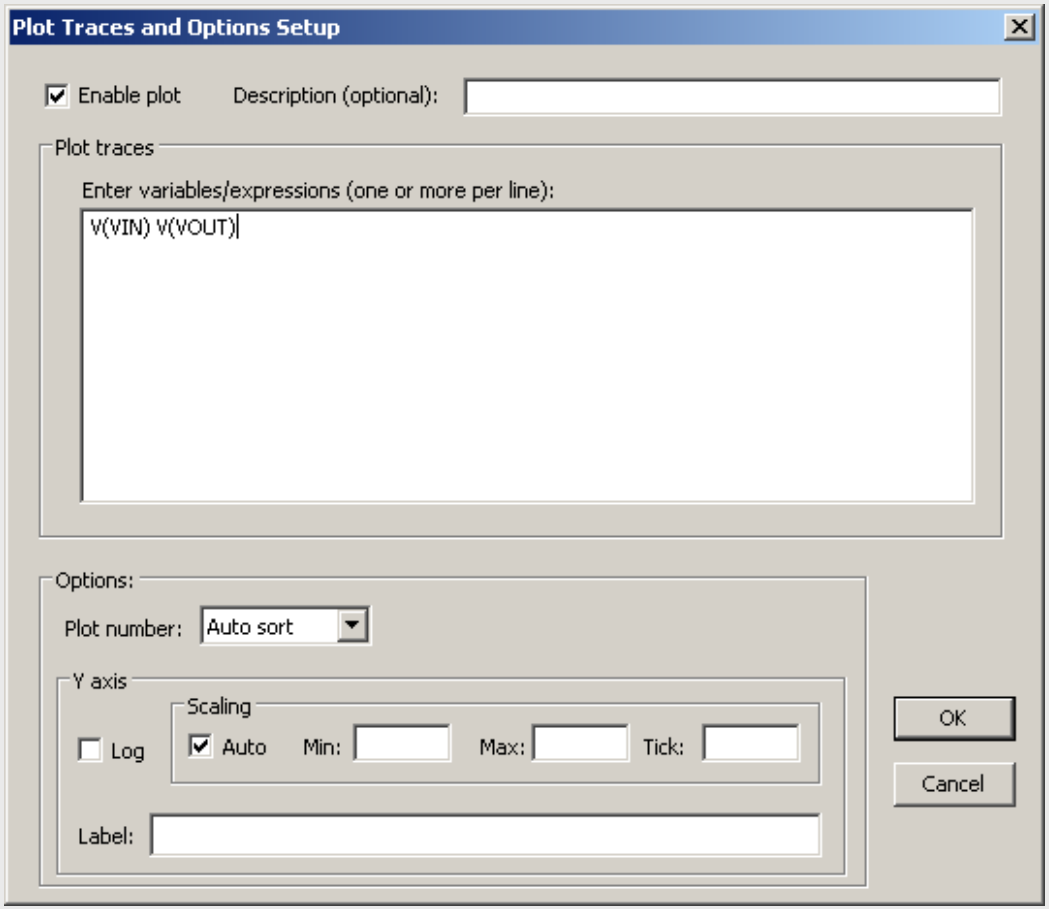
3.2) TopSPICE

⌘ Opções de Saída:



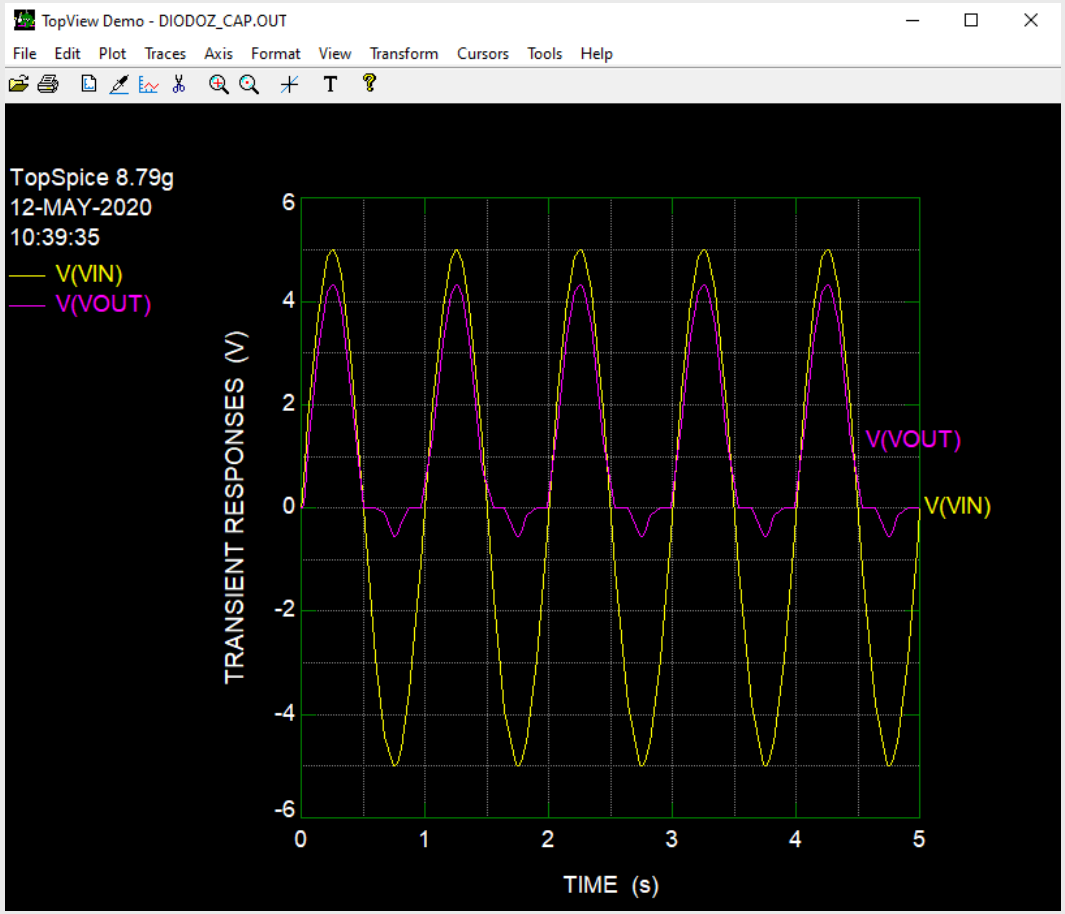
3.2) TopSPICE

⌘ Definição do Gráfico:



3.2) TopSPICE

- Executar Simulação (F9)



3.2) TopSPICE

⌘ Utilizando Cursores