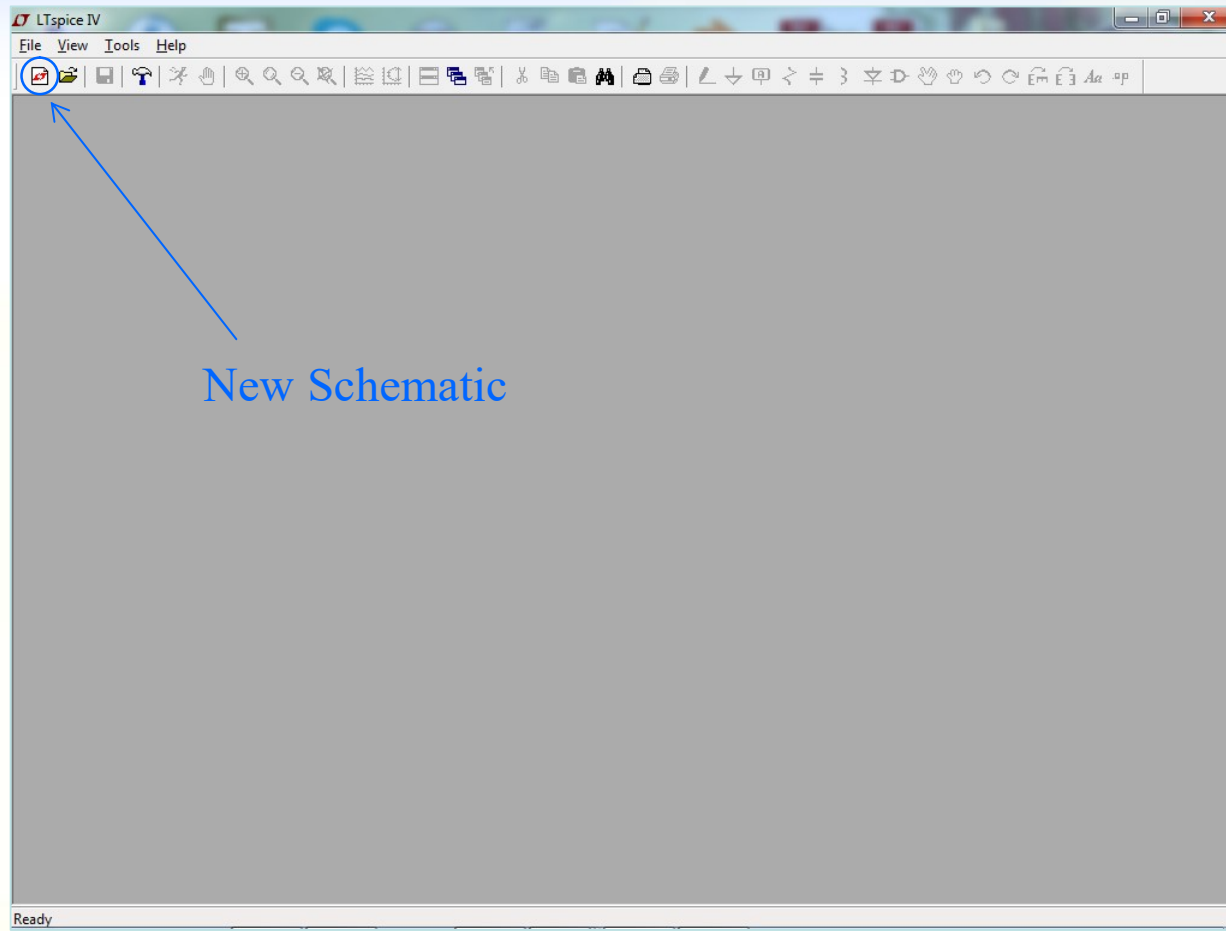
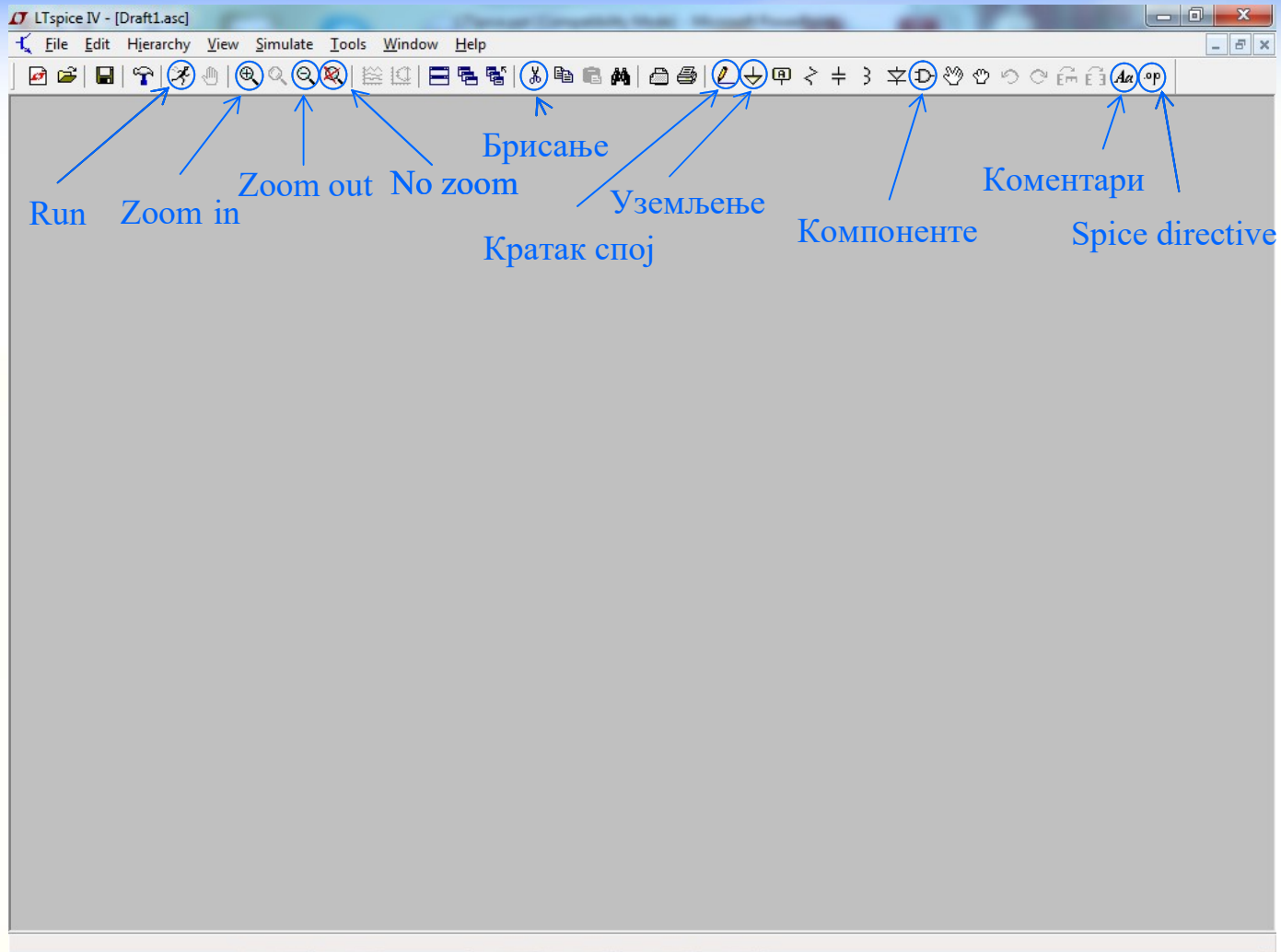


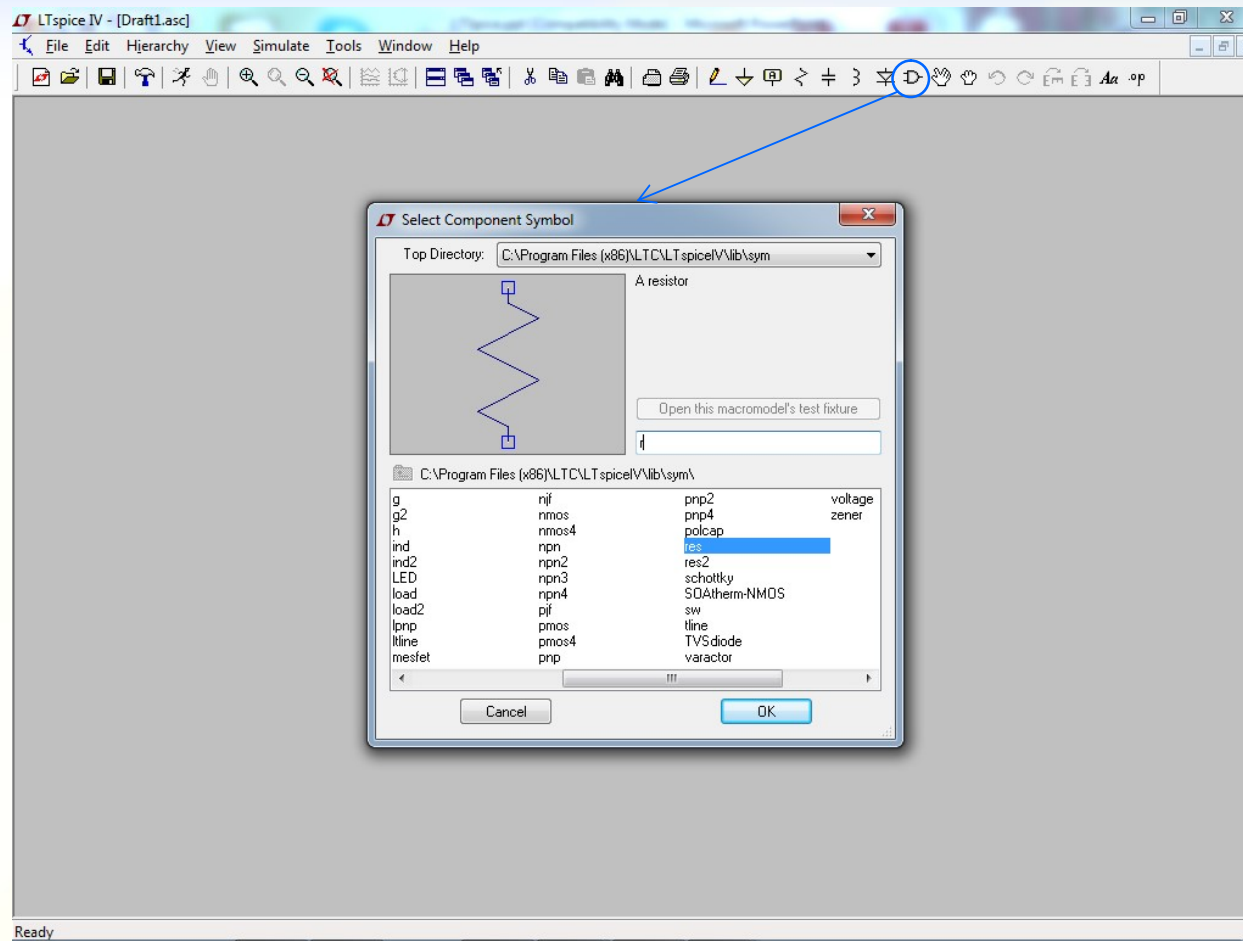
LTspice



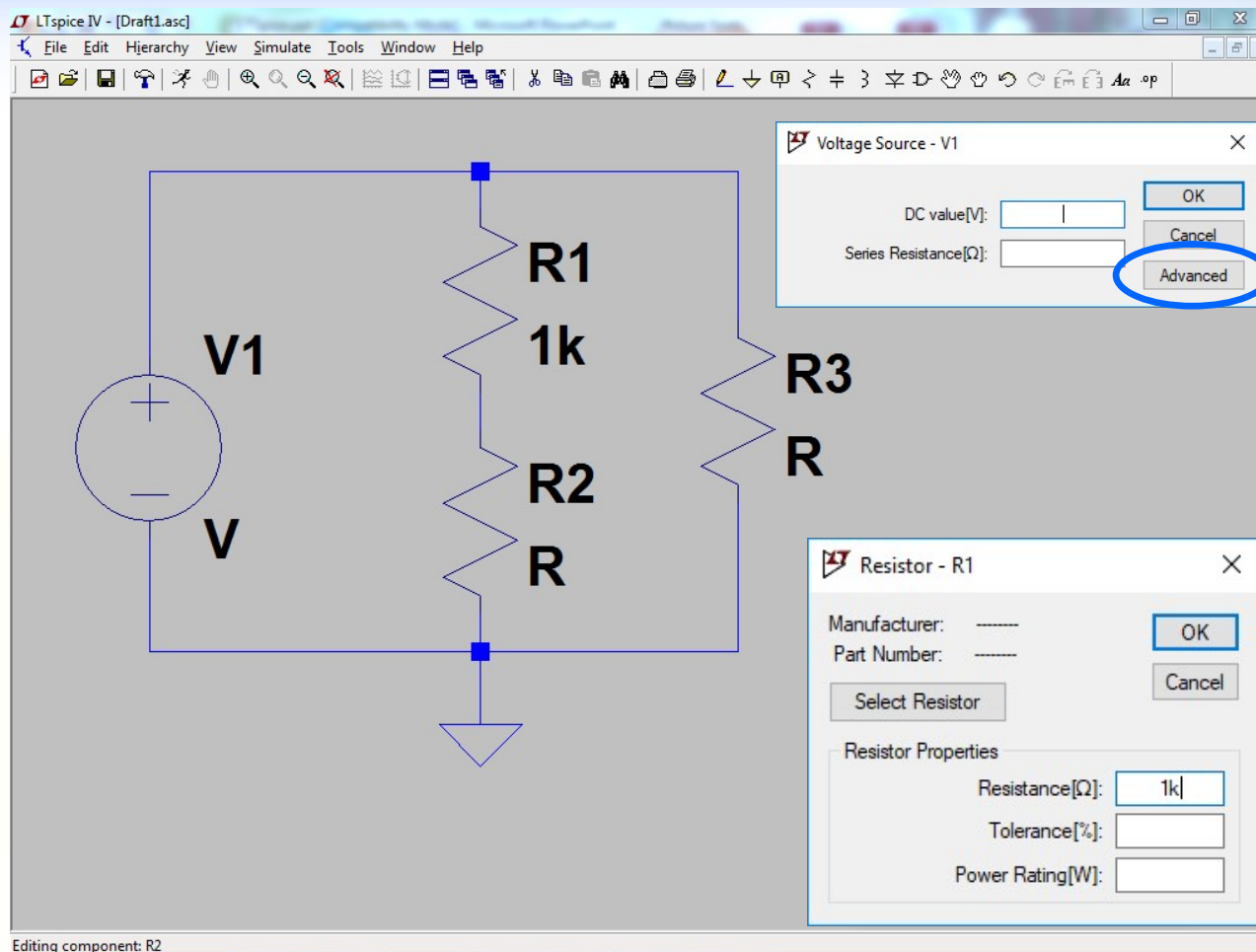
LTspice Schematic



Проналажење елемената

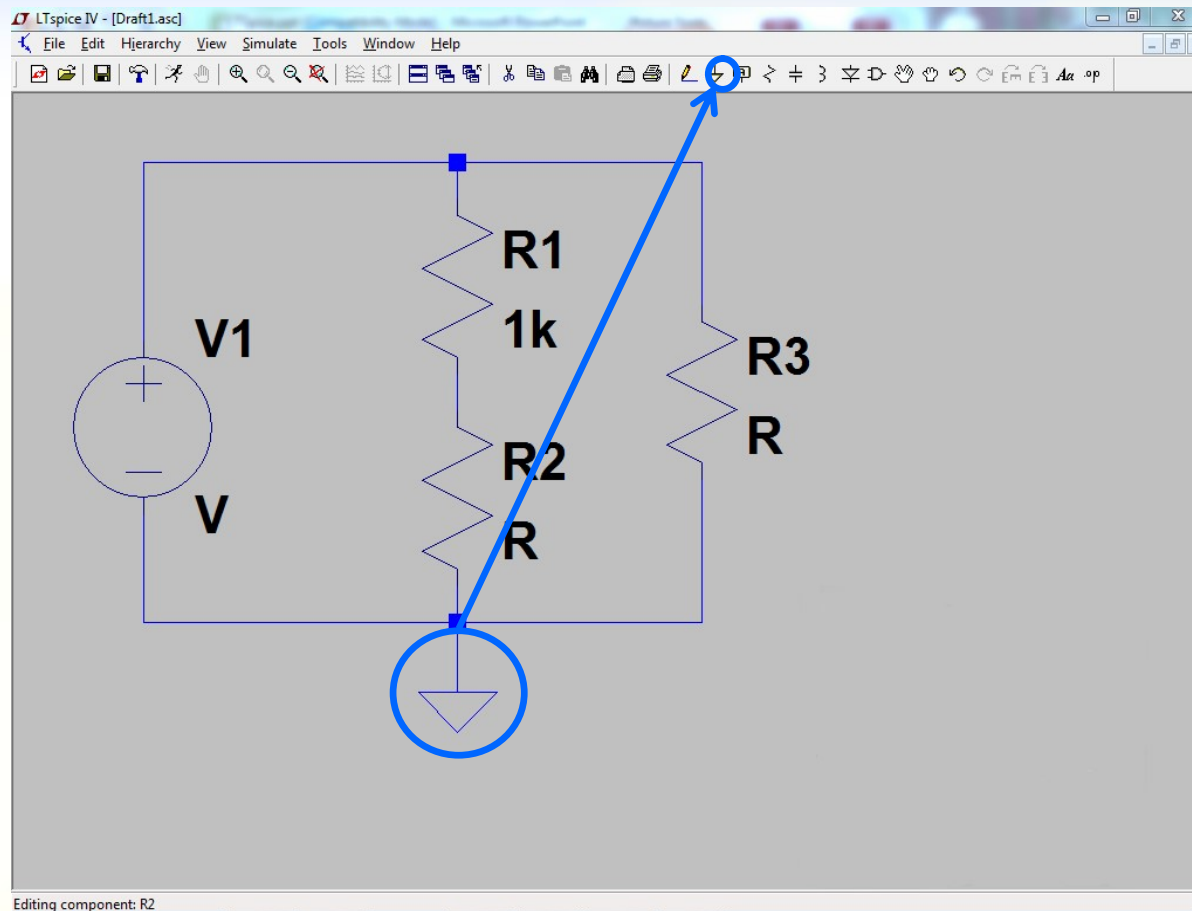


Подешавање параметара елемената



Editing component: R2

Додавање уземљења



Нетлиста (Netlist)

The screenshot displays the LTspice IV interface with a circuit schematic and a pop-up window showing the generated SPICE netlist.

Circuit Schematic: The schematic shows a voltage source V1 connected to a 1k resistor, which is in series with a 2k resistor R3. The circuit is grounded. The components are labeled V1, 1, 1k, R3, and 2k.

SPICE Netlist: The netlist is displayed in a window titled "SPICE Netlist: E:\Software\LTC\LTspiceIV\Draft...". The netlist content is as follows:

```
* E:\Software\LTC\LTspiceIV\Draft1.asc
R1 N001 N002 1k
R2 N002 0 1k
R3 N001 0 2k
V1 N001 0 1
.backanno
.end
```

Display the SPICE netlist for this schematic

Подешавање анализе

The screenshot displays the LTspice IV interface with a circuit diagram and two open dialog boxes for configuring simulations.

Circuit Diagram: A DC voltage source **V1** (labeled **1**) is connected in series with a resistor **R1** (1k). This series combination is connected to a parallel network of two resistors, **R2** (1k) and **R3** (2k), which are then connected to ground.

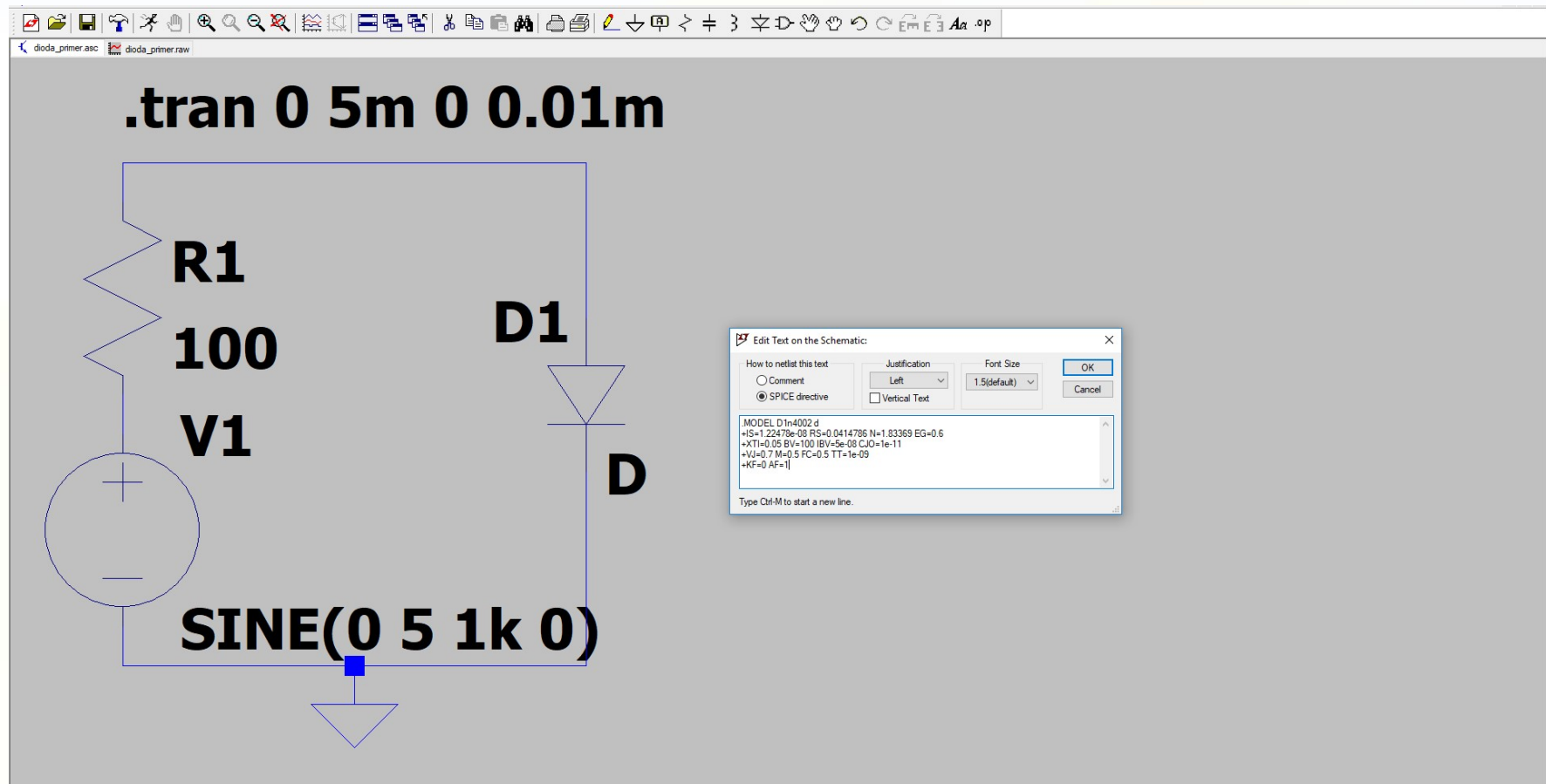
Simulation Command Dialogs:

- Top Dialog (AC Analysis):** Titled "Edit Simulation Command", it shows the "AC Analysis" tab selected. The description reads: "Compute the small signal AC behavior of the circuit linearized about its DC operating point." Fields for "Type of sweep:", "Number of points:", "Start frequency:", and "Stop frequency:" are present. The syntax field contains ".ac".
- Bottom Dialog (Transient):** Also titled "Edit Simulation Command", it shows the "Transient" tab selected. The description reads: "Perform a non-linear, time-domain simulation." Fields for "Stop time:" (200u), "Time to start saving data:" (0), and "Maximum Timestep:" (0.01u) are visible. The syntax field contains ".tran 0 200u 0 0.01u".

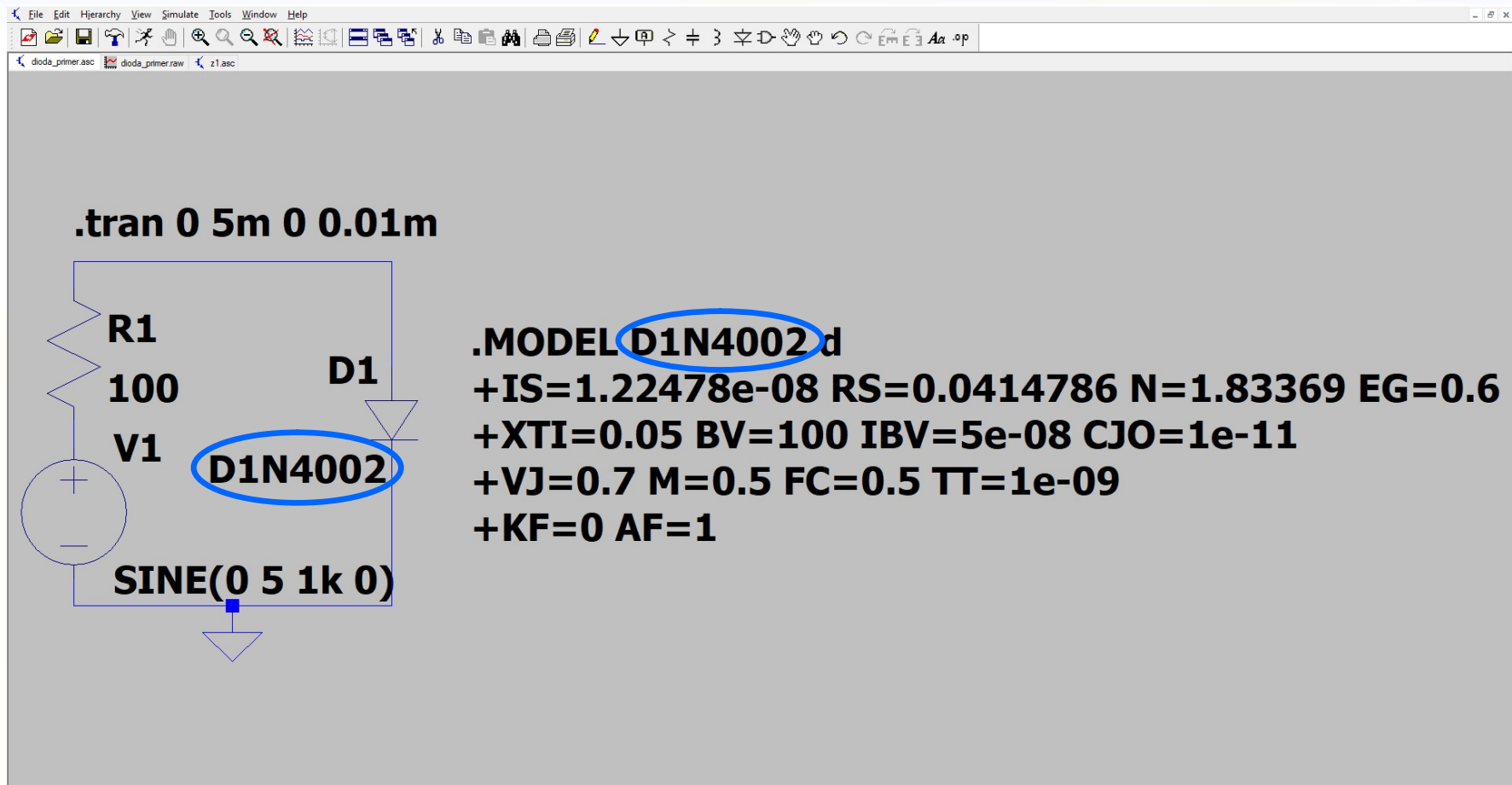
A blue oval highlights the command **.tran 0 200u 0 0.01u** in the bottom dialog, with a blue arrow pointing to it from the right.

At the bottom of the window, the text "Enter, Edit, or Select the Simulation Command" is visible.

Модел елемента – SPICE директива



Модел елемента



Подешавање побуде и анализе

The screenshot displays a circuit simulation software interface. The main workspace shows a circuit diagram with a voltage source **V1** (a circle with a plus sign), a resistor **R1** (a zigzag line) labeled **100**, and a diode **D1** (a triangle with a line). The circuit is connected to ground. Two blue ovals highlight specific components: the top oval contains the text **.tran 0 5m 0 0.01m**, and the bottom oval contains the text **SINE(0 5 1k 0)**. Two blue arrows point from these ovals to their respective configuration dialog boxes.

The **Edit Simulation Command** dialog box is open, showing the **Transient** tab. It contains the following settings:

- Stop time: **5m**
- Time to start saving data: **0**
- Maximum Timestep: **0.01m**
- Start external DC supply voltages at 0V: ☐
- Stop simulating if steady state is detected: ☐
- Don't reset T=0 when steady state is detected: ☐
- Step the load current source: ☐
- Skip initial operating point solution: ☐

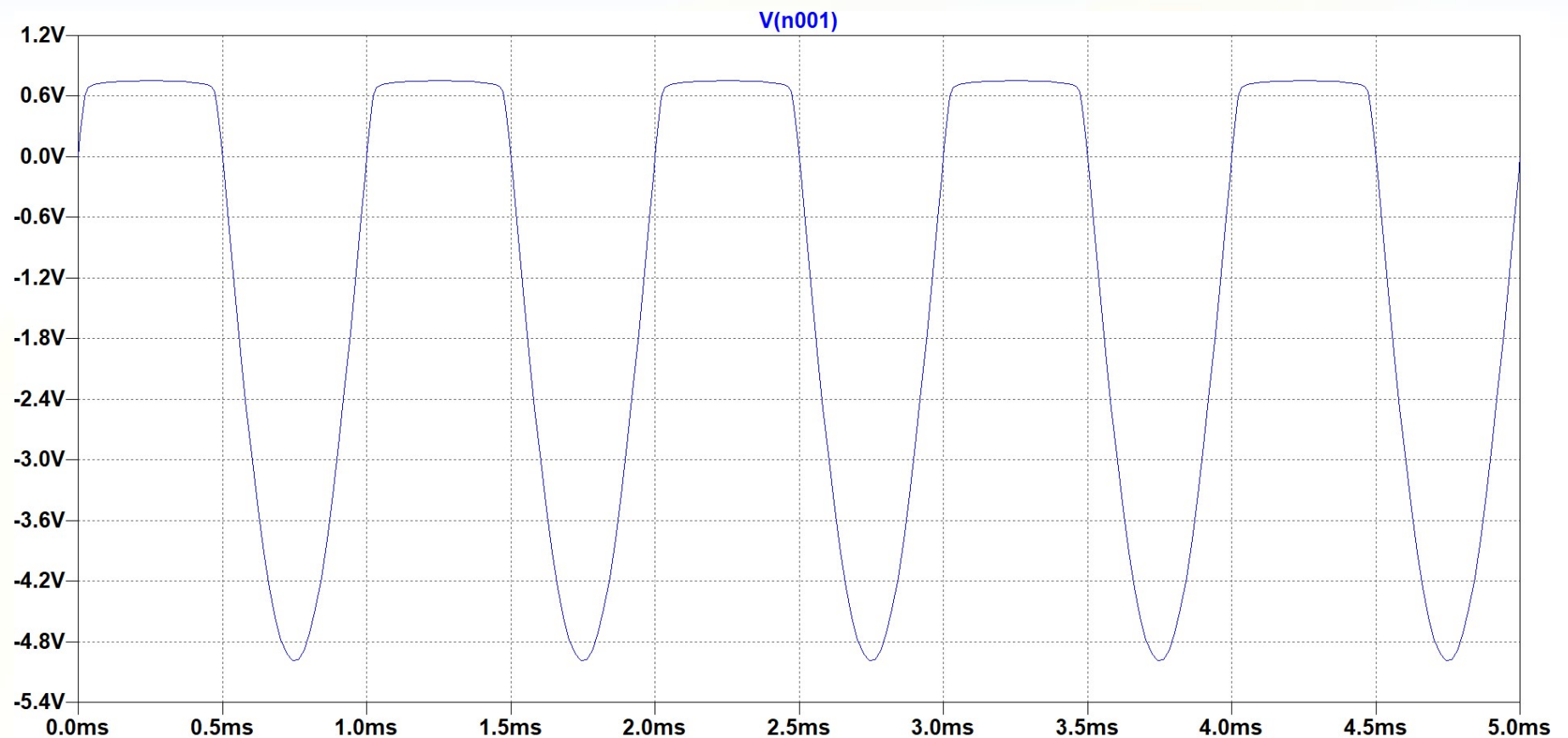
The syntax field shows: `.tran <Tprint> <Tstop> [<Tstart> [<Tmaxstep>]] [<option> [<option>] ...]`. The command field contains: `.tran 0 5m 0 0.01m`.

The **Independent Voltage Source - V1** dialog box is open, showing the **Functions** tab. It contains the following settings:

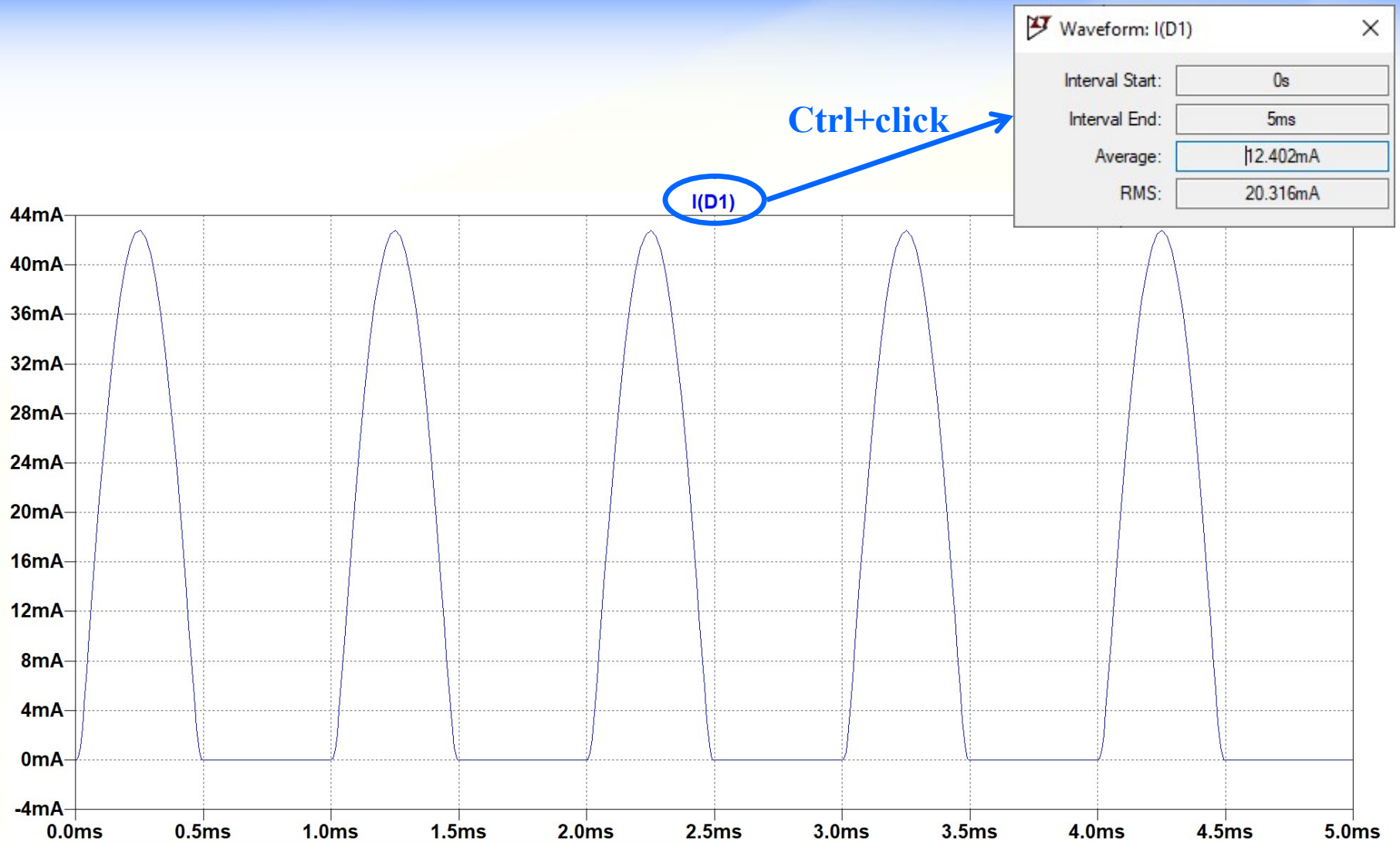
- Functions: **SINE(Voffset Vamp Freq Td Theta Phi Ncycles)** (selected)
- DC Value: **DC value:** (empty)
- Small signal AC analysis (AC): ☐
- AC Amplitude: (empty)
- AC Phase: (empty)
- Parasitic Properties: ☐
- Series Resistance[R]: (empty)
- Parallel Capacitance[F]: (empty)

The command field contains: `SINE(0 5 1k 0)`.

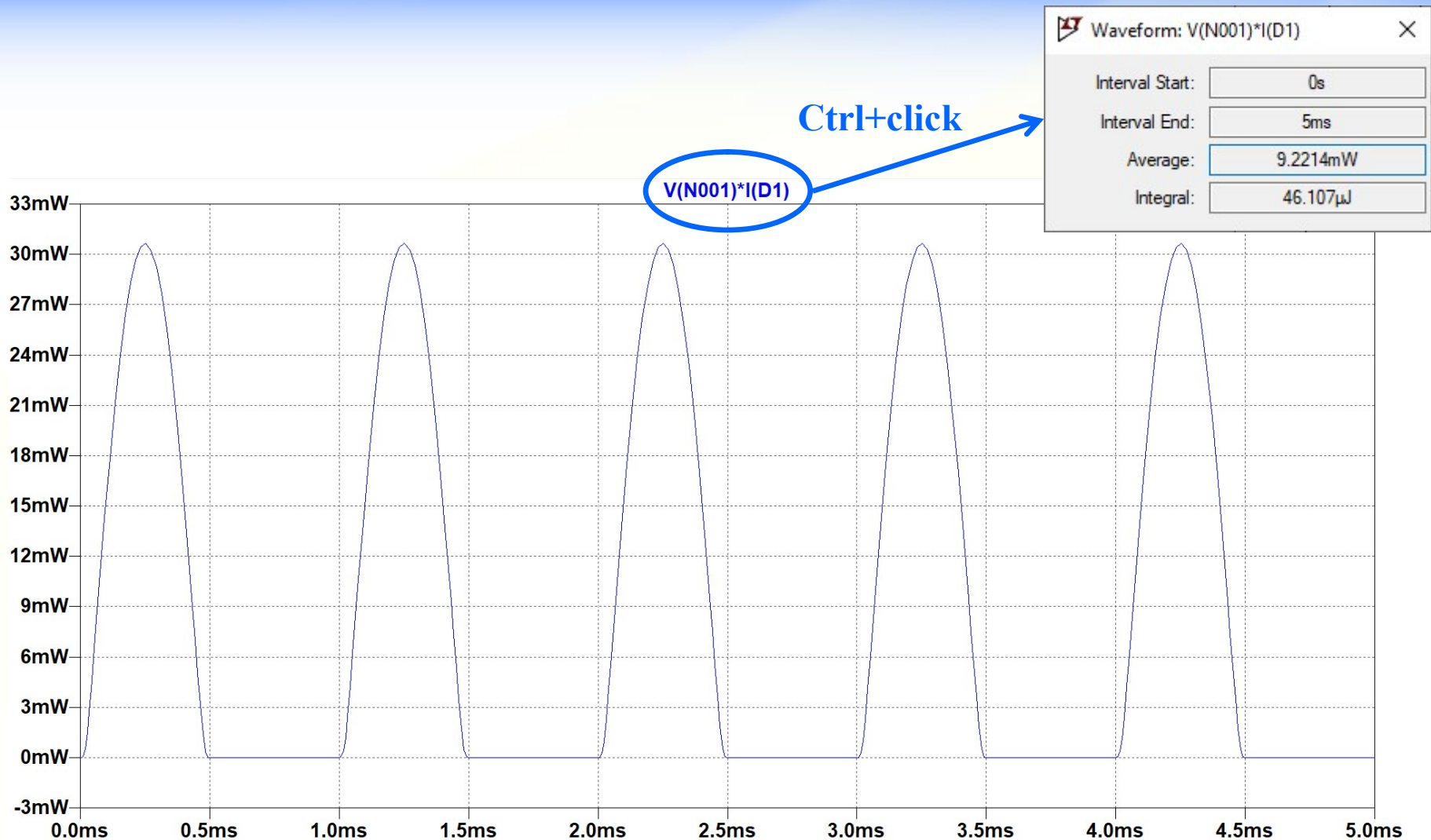
Резултати симулације - напон (Transient analysis)



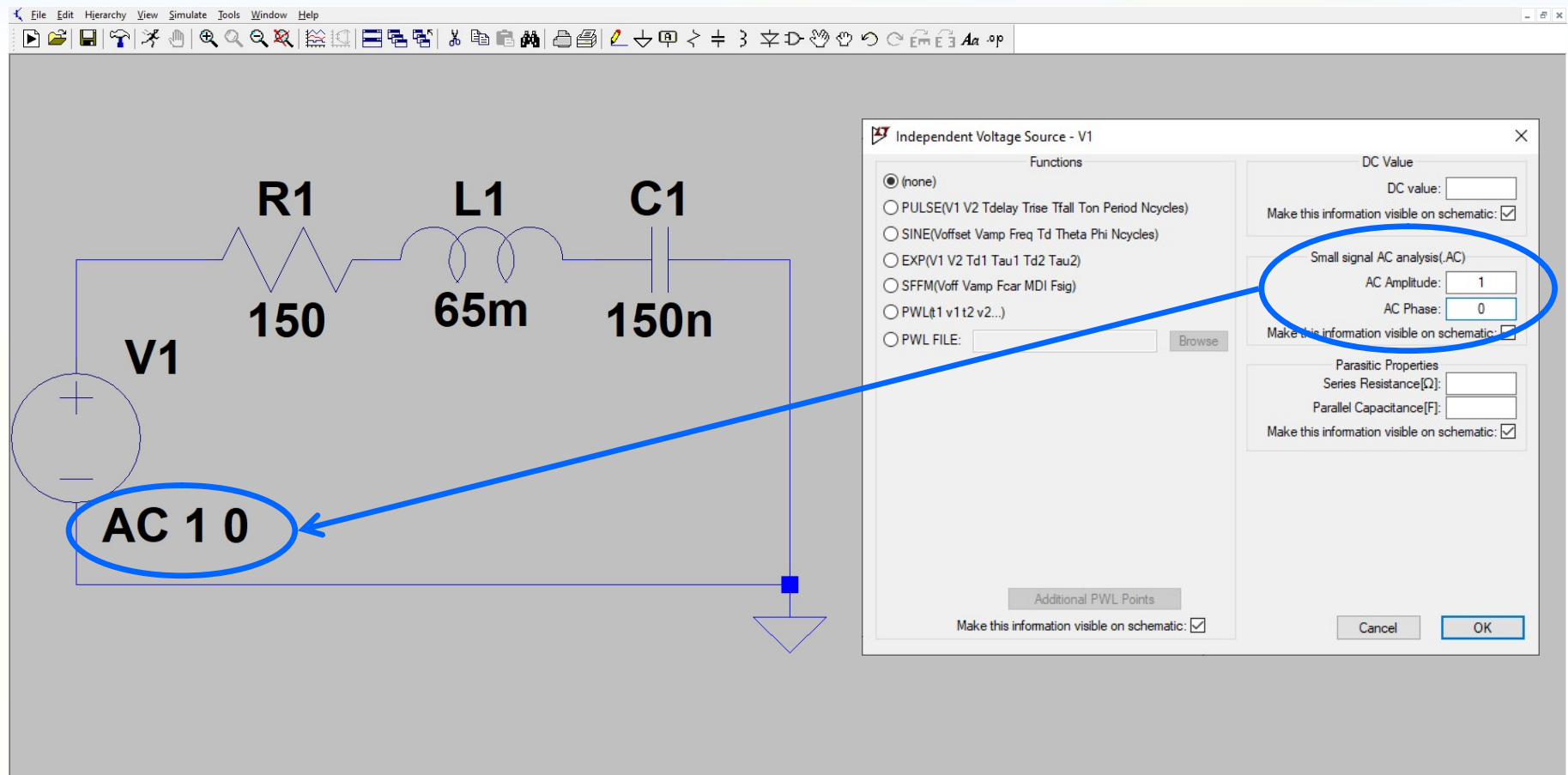
Резултати симулације - струја (Transient analysis)



Резултати симулације - дисипација (Transient analysis)



AC analysis Модел



AC analysis

Подешавање анализе

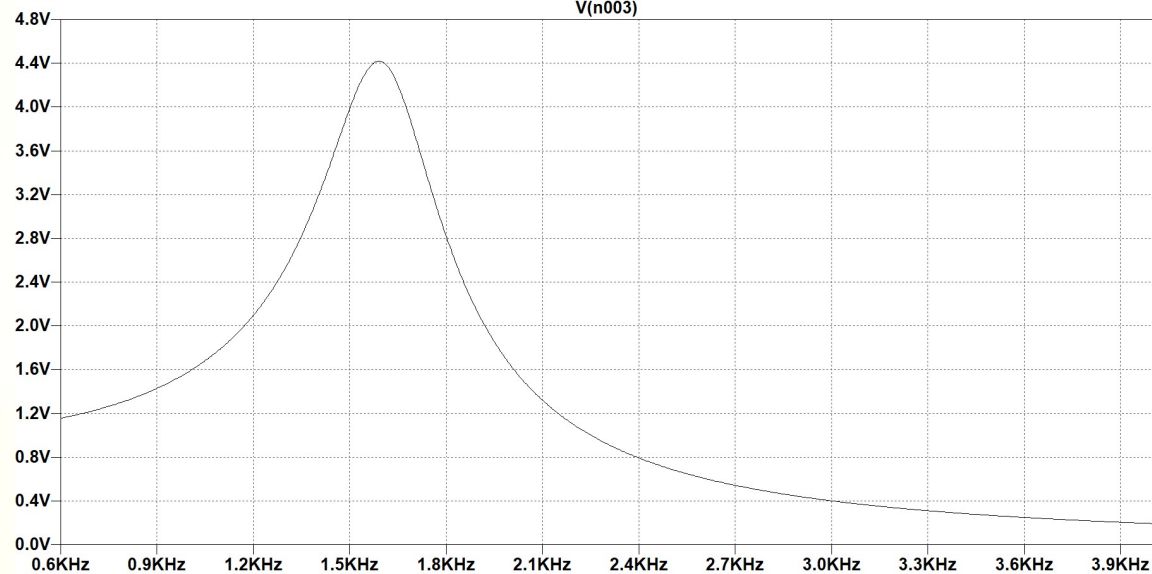
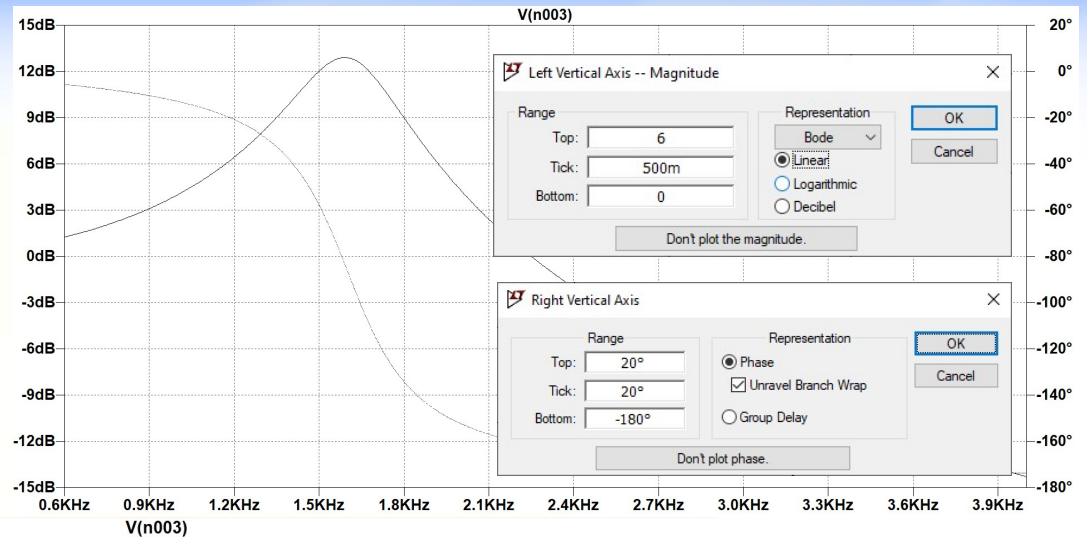
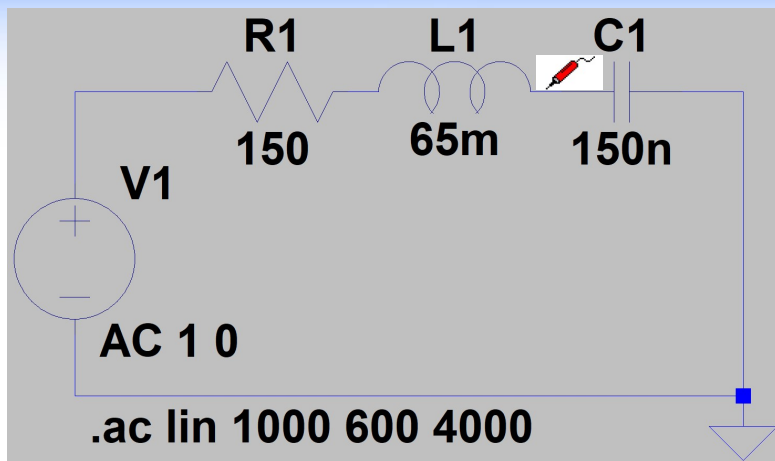
The image shows a circuit diagram and a simulation command dialog box. The circuit consists of an AC voltage source V1 connected in series with a resistor R1 (150), an inductor L1 (65m), and a capacitor C1 (150n). The circuit is grounded. The simulation command `.ac lin 1000 600 4000` is entered in the command window, which is highlighted with a blue oval. A blue arrow points from this command to the 'Edit Simulation Command' dialog box.

The 'Edit Simulation Command' dialog box is open, showing the 'AC Analysis' tab. The settings are as follows:

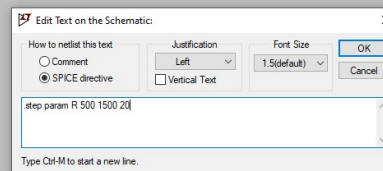
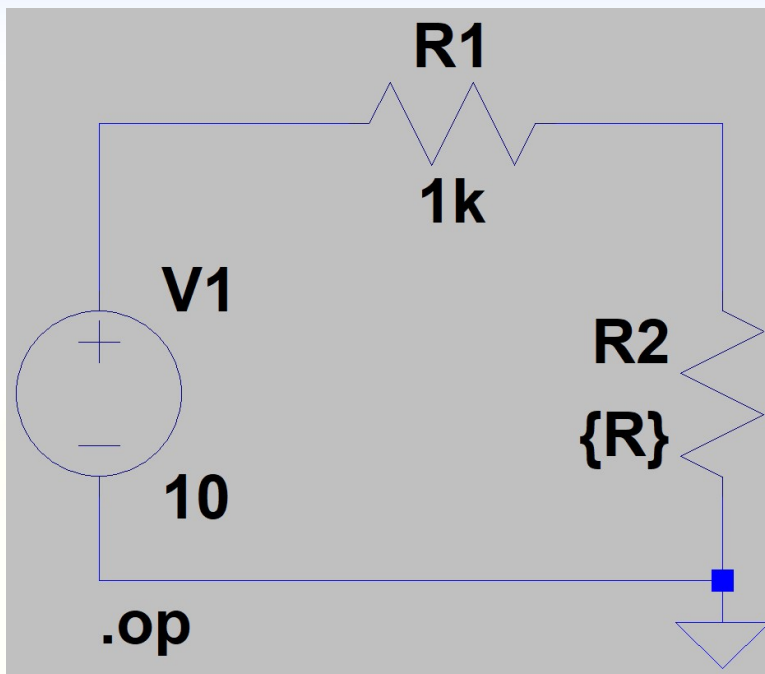
- Type of sweep: Linear
- Number of points: 1000
- Start frequency: 600
- Stop frequency: 4000

The syntax for the command is shown as: `.ac <oct, dec, lin> <Npoints> <StartFreq> <EndFreq>`. The command `.ac lin 1000 600 4000` is entered in the command field.

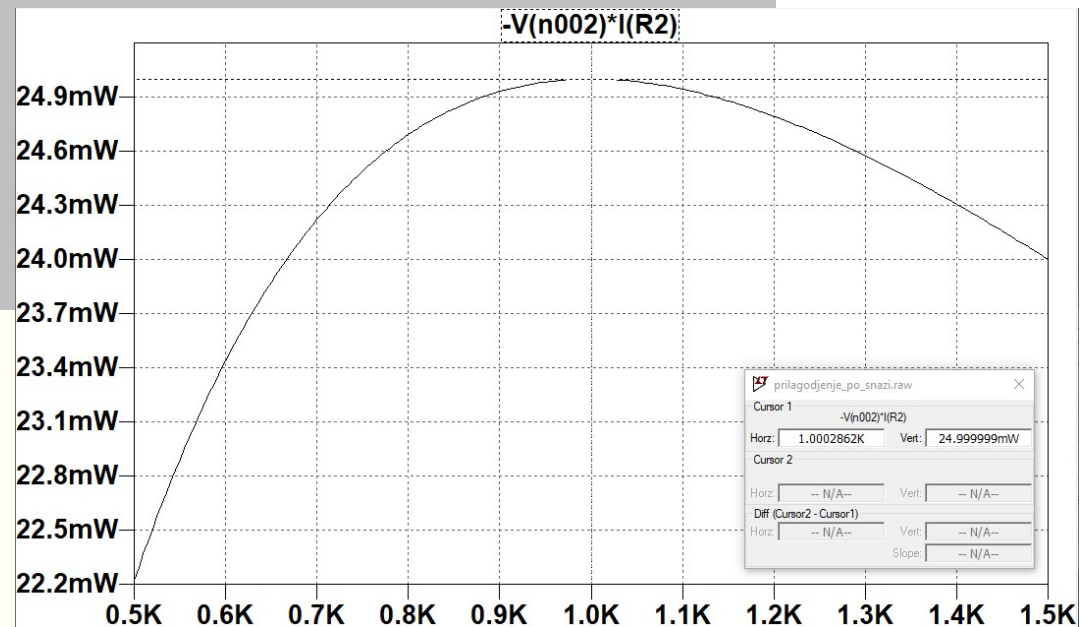
Резултати симулације (AC analysis)



“Sweep” по параметру



.step param R 500 1500 20



Спрегнуті калемови

