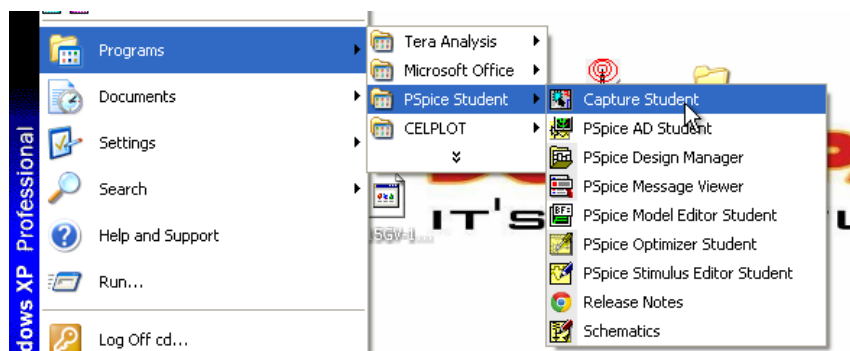
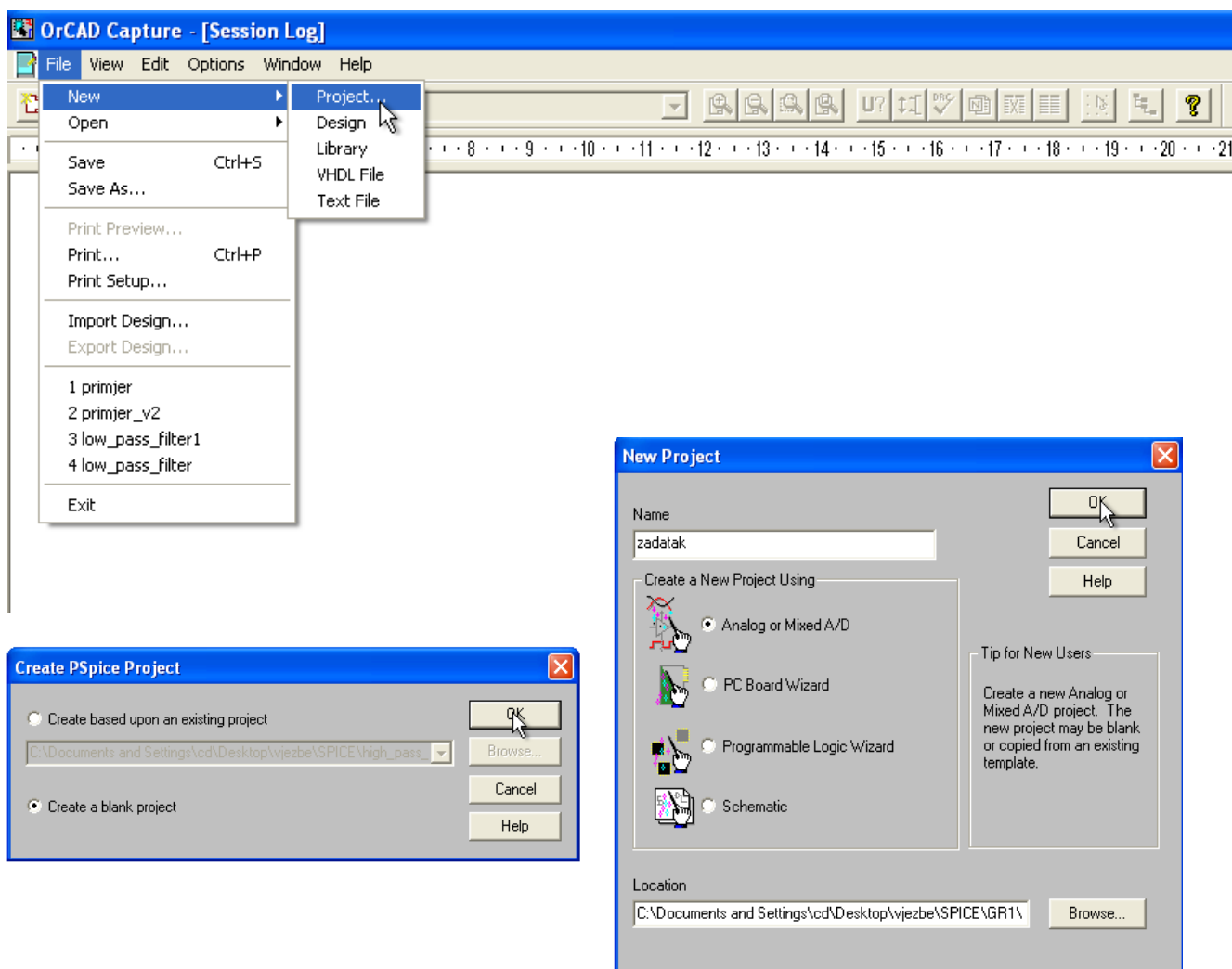


Vježba br. 1 PSPICE – Niskopropusni filter

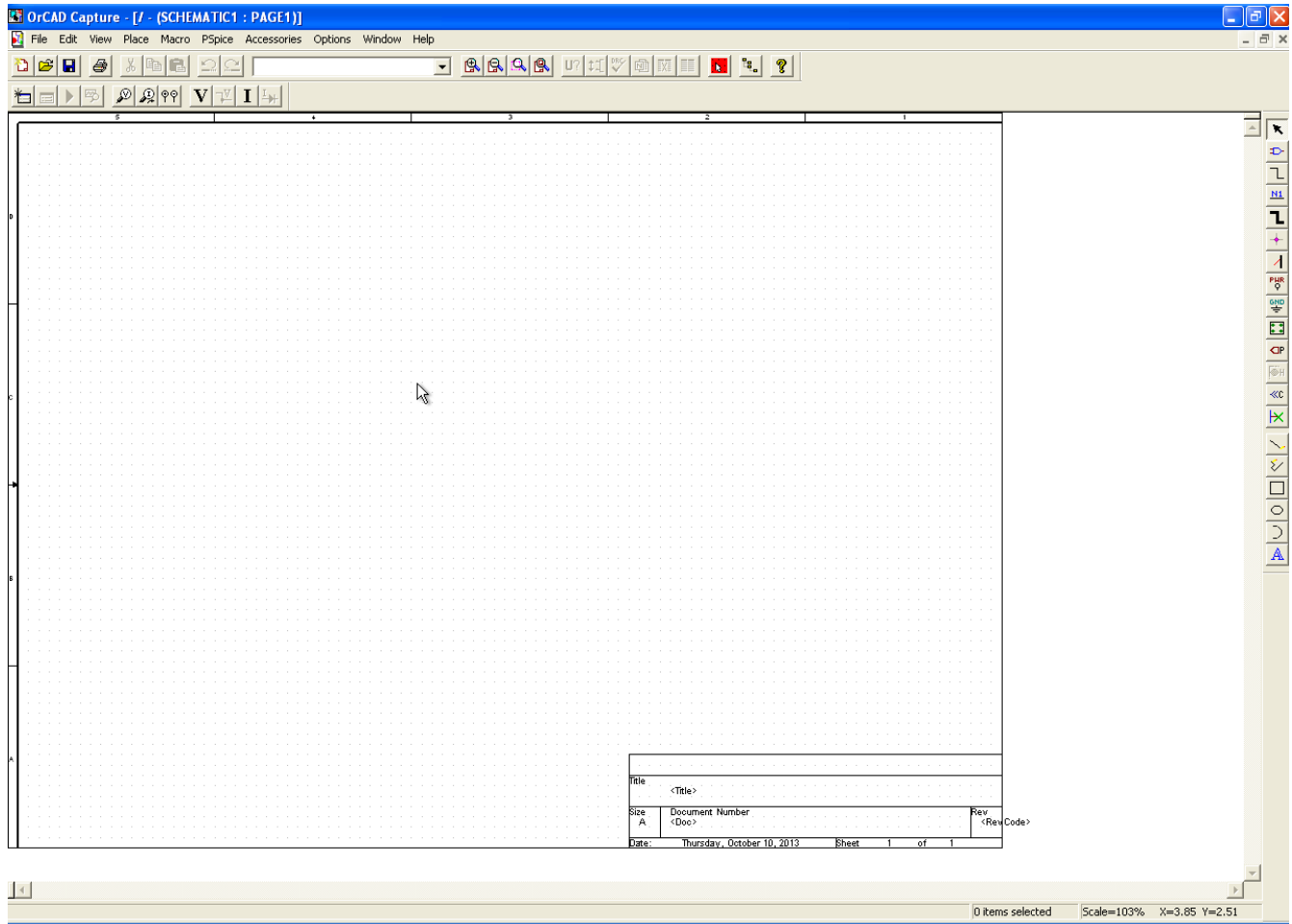
Otvaranje programa: *Start* → *Programs* → *Pspice Student* → *Capture Student*



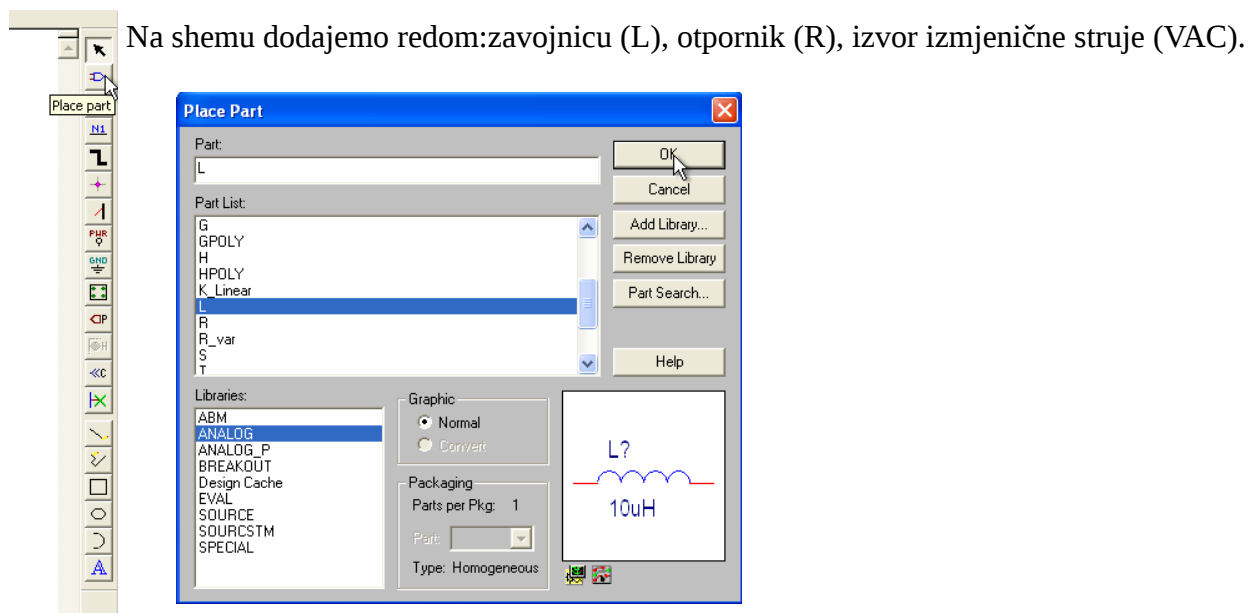
Stvaranje novog projekta: *File* → *New* → *Project* → *Create a blank project* → *Analog or Mixed A/D*

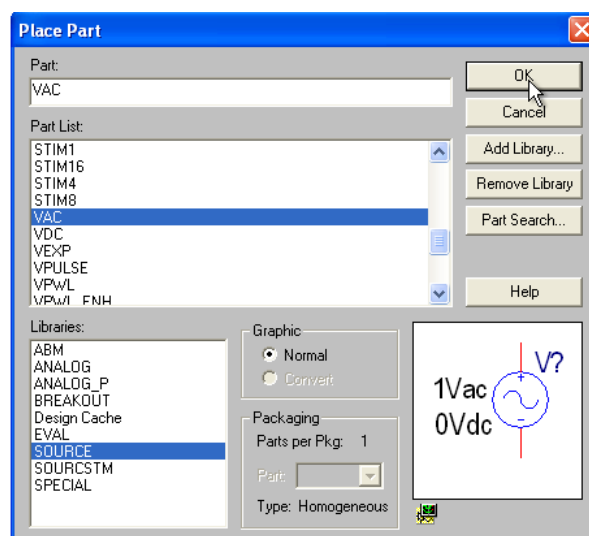
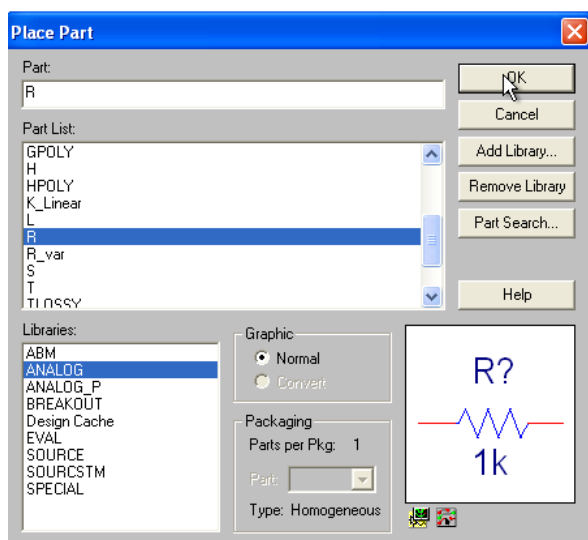


Izgled prozora novog projekta:

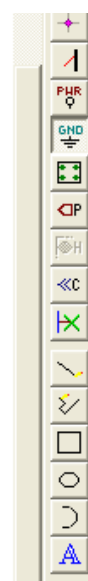
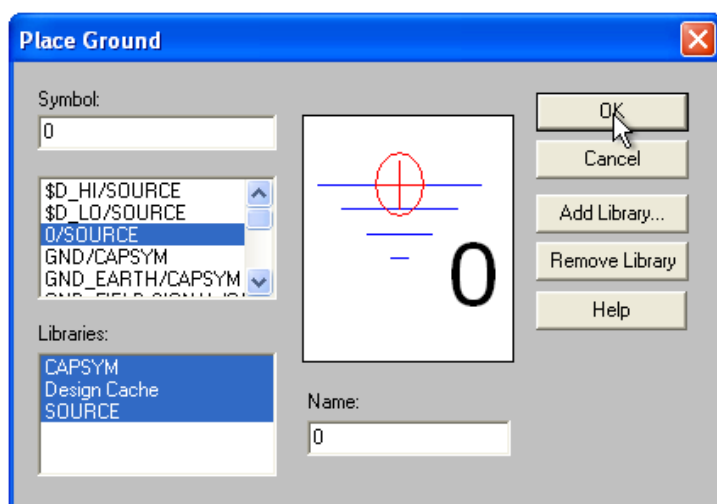


Za crtanje šeme potrebno je iz desnog izbornika odabrati odgovarajući “alat”.
Osnovni elementi za crtanje sheme nalaze se pod ikonom *Place part*.

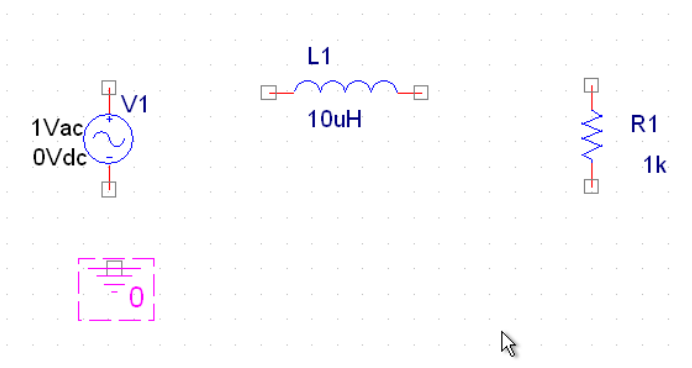




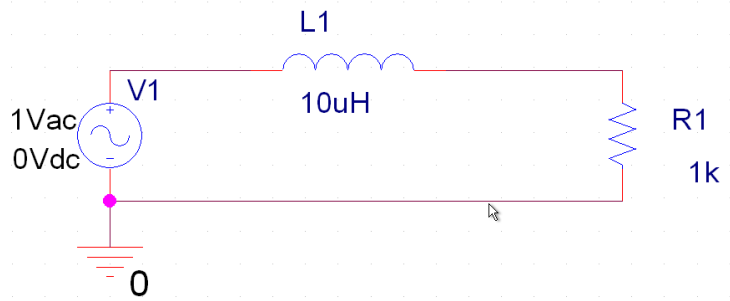
Naposljetku treba dodati i uzemljenje (0/SOURCE) koje se nalazi pud ikonom GND (u desnom izborniku):



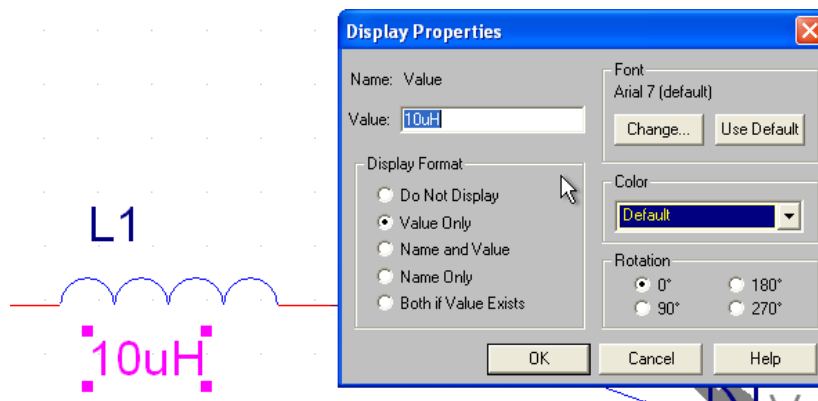
Sve elemente sheme je potrebno povezati žicom koja se odabire iz desnog izbornika (*Place wire*).



Nakon povezivanja dobivamo konačnu shemu:

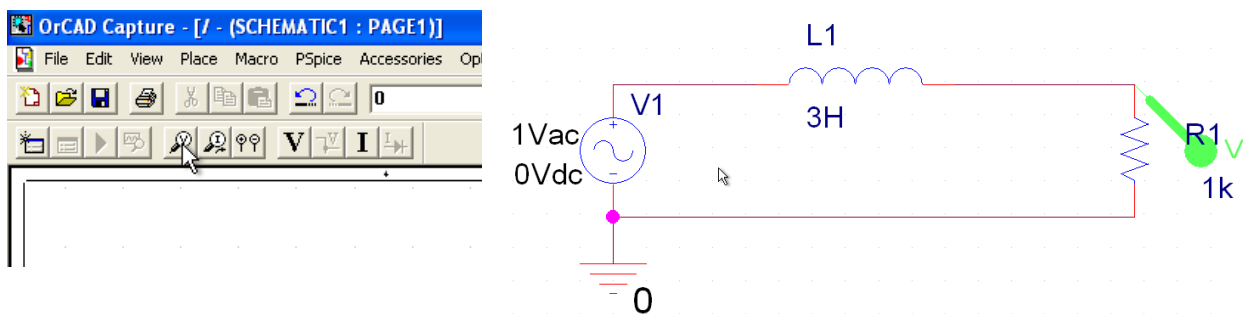


Po potrebi je moguće mjenjati nazive elemenata i/ili im mjenjati iznos (dvostruki klik na naziv ili iznos):

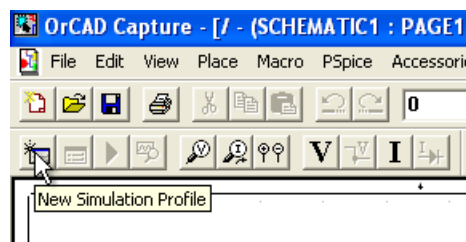


Time završava crtanje sheme. Sada je potrebno postaviti parametre simulacije.

Prvi korak je dodavanje tvz. markera za ono što želimo promatrati na mjesto koje želimo promatrati. Postoje strujni i naponski markeri, što znači da u određenom strujnom krugu na različitim mjestima možemo promatrati stuju i/ili napon. U konkretno slučaju potrebno je promatrati napon na otporniku pa je stoga naponski marker postavljen na + otpornika.



Sljedeći korak je odabir vrste simulacije:



The image displays two side-by-side screenshots of the 'Simulation Settings - freq_sweep' dialog box in a software application.

Left Screenshot:

- Title Bar:** Simulation Settings - freq_sweep
- Tabs:** General, Analysis, Include Files, Libraries, Stimulus, Options, Data Collection, Probe Window.
- Analysis type:** A dropdown menu is open, showing options: Time Domain (Transient), DC Sweep, **AC Sweep/Noise** (highlighted with a mouse cursor), and Bias Point.
- Run to time:** 1000ns seconds (TSTOP)
- Start saving data after:** 0 seconds
- Transient options:**
 - Maximum step size: [] seconds
 - ☐ Skip the initial transient bias point calculation (SKIPBP)
- Buttons:** OK, Cancel, Apply, Help.

Right Screenshot:

- Title Bar:** Simulation Settings - freq_sweep
- Tabs:** General, Analysis, Include Files, Libraries, Stimulus, Options, Data Collection, Probe Window.
- Analysis type:** AC Sweep/Noise
- AC Sweep Type:**
 - ☐ Linear
 - ☒ Logarithmic
 - Decade []
 - Start Frequency: 1
 - End Frequency: 1e5
 - Points/Decade: 10
- Noise Analysis:**
 - ☐ Enabled
 - Output Voltage: []
 - I/V Source: []
 - Interval: []
- Output File Options:**
 - ☐ Include detailed bias point information for nonlinear controlled sources and semiconductors (.OP)
- Buttons:** OK, Cancel, Apply, Help.

The screenshot shows the top portion of the OrCAD Capture software interface. The title bar reads "OrCAD Capture - [I] - (SCHEMATIC1 : PAGE1)". Below it is a standard Windows-style menu bar with options: File, Edit, View, Place, Macro, PSpice, and Accessories. A toolbar follows, containing icons for file operations (Open, Save), editing (Undo, Redo), and simulation (Run). The "Run" icon, which depicts a computer monitor, is highlighted by a mouse cursor. A tooltip box labeled "Run PSpice" appears directly beneath the cursor. The main workspace below the toolbar is mostly blank, showing some faint grid lines.

The screenshot displays the OrCAD PSpice A/D Demo interface. The main window shows an AC analysis plot for a circuit named 'zadatak-SC...'. The plot shows the voltage magnitude (V(R1:2)) versus frequency (Hz) on a logarithmic scale. The curve starts at 1.0V at 1.0Hz and decreases, crossing the 0.707V (-3dB) point at 100Hz. A probe cursor is positioned at 100Hz, showing the following data:

Probe Cursor	A1	A2	dif
1	133.114	371.155m	
2	1.0000	0.9998	
dif	132.114	-628.667m	

The status bar at the bottom indicates the current frequency is 100.00E+03 Hz. The left pane shows the simulation log, and the bottom pane shows the analysis and watch tabs.

ZADATAK:

1. U programskom paketu PSpice simulirati prethodno opisani induktivni niskopropusni filter.
2. U programskom paketu PSpice izraditi model kapacitivnog niskopropusnog filtera te simulirati ponašanje.
3. Komentirati rezultate.

Vježba br 2.

1. U programskom paketu PSpice izraditi model kapacitivnog visokopropusnog filtera te simulirati ponašanje.
2. U programskom paketu PSpice izraditi model induktivnog visokopropusnog filtera te simulirati ponašanje.
3. Komentirati rezultate.

Vježba br 3.

1. U programskom paketu PSpice izraditi model kapacitivnog pojasnopropusnog filtera te simulirati ponašanje.
2. U programskom paketu PSpice izraditi model induktivnog pojasnopropusnog filtera te simulirati ponašanje.
3. U programskom paketu PSpice izraditi model pojasnebrane te simulirati ponašanje.
4. Komentirati rezultate.