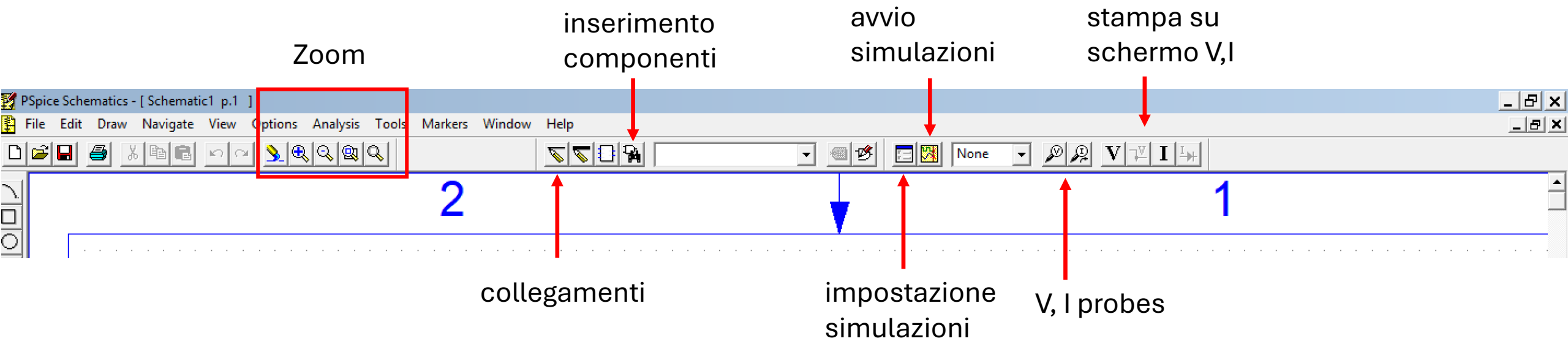


ELETTRONICA IIB: Lab - I

Introduzione all'uso di PSPICE 9.1

- Schematic editor
- Simulazione delle curve statiche di un BJT
- Simulazione di un amplificatore invertente



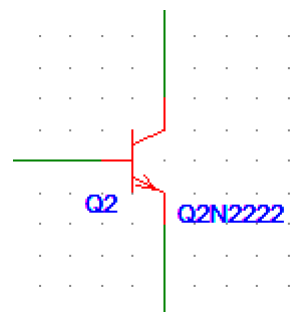
Tasti di scelta rapida.

Selezionare (click) il componente:

ctrl+c -> copia

ctrl+x -> taglia

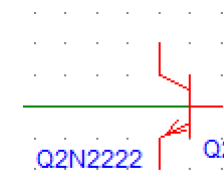
ctrl+v -> incolla



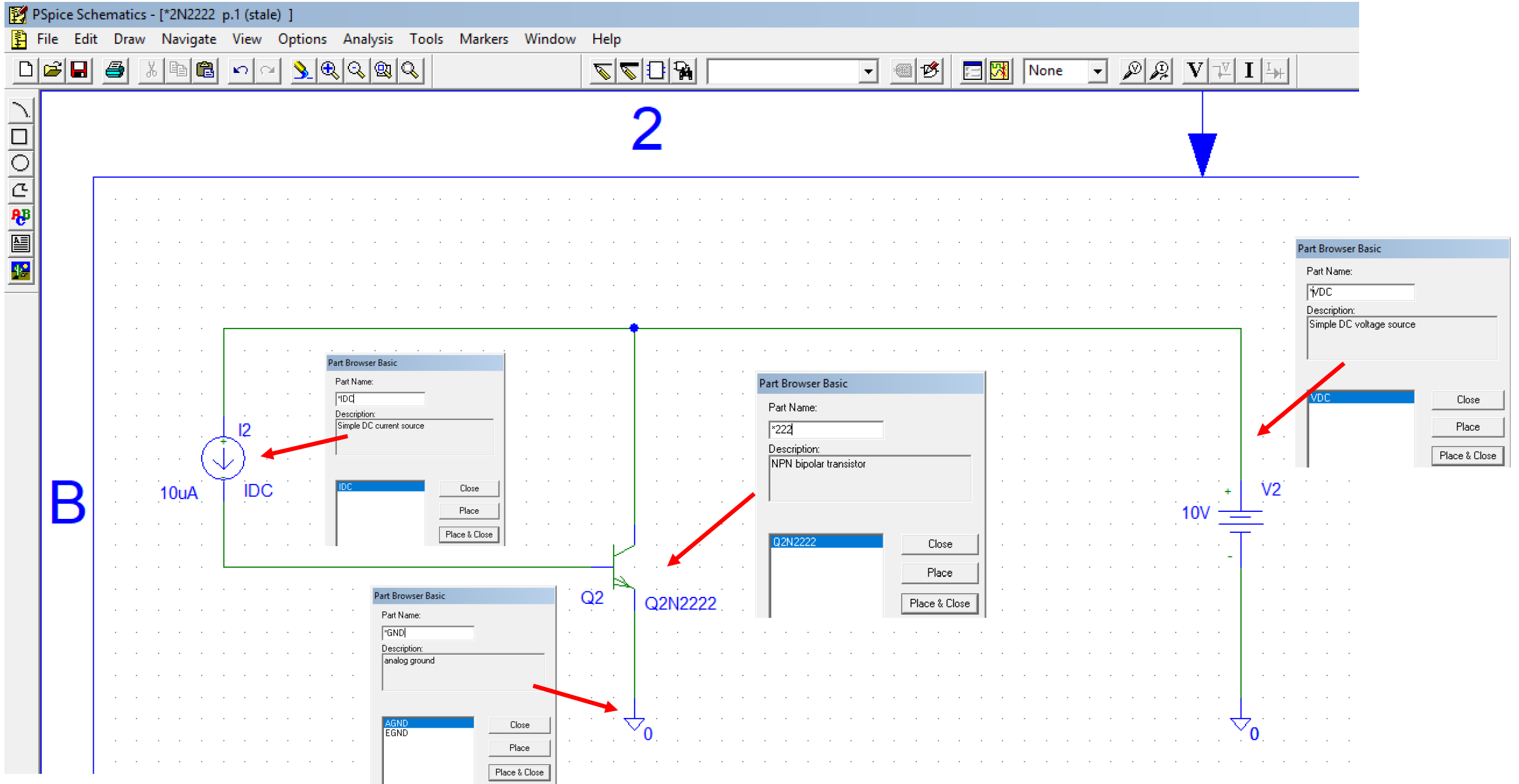
ctrl+r -> ruota



ctrl+f -> flip (capovolgi)



Simulazione punto operativo di un BJT: (1) DISEGNO SCHEMATICO



Simulazione punto operativo di un BJT: (2) impostazione valori componenti

Doppio click sul componente da modificare

The image shows the PSpice Schematics interface with two component property dialog boxes open. The background schematic shows a BJT circuit with a current source IDC (10uA) and a voltage source V2 (10V) connected to a Q2N2222 transistor. A large blue number '2' is overlaid on the schematic, indicating the second step of the process. A blue arrow points from the 'V2' dialog box to the voltage source symbol in the schematic. A red arrow points from the 'IDC' dialog box to the current source symbol. Another red arrow points from the 'V2' dialog box to the 'DC=10V' line in its list.

Dialog Box 1: I2 PartName: IDC

Name	Value
DC	= 10uA

* REFDES=I2
* TEMPLATE=I^@REFDES %+ %- ?DCIDC @DCI ?ACIAC @ACI
DC=10uA
* SIMULATIONONLY=
* PART=IDC
* MODEL=
* PKGREF=I2

Include Non-changeable Attributes
 Include System-defined Attributes

Buttons: Save Attr, Change Display, Delete, OK, Cancel

Dialog Box 2: V2 PartName: VDC

Name	Value
DC	= 10V

* REFDES=V2
* TEMPLATE=V^@REFDES %+ %- @DC
DC=10V
* SIMULATIONONLY=
* PART=VDC
* MODEL=
* PKGREF=V2

Include Non-changeable Attributes
 Include System-defined Attributes

Buttons: Save Attr, Change Display, Delete, OK, Cancel

Simulazione punto operativo di un BJT: (3) impostazione simulazione

The image shows the PSpice Schematics interface for a BJT circuit simulation. The circuit includes a 10uA current source labeled I2 (IDC) connected to the base of a BJT transistor labeled Q2 (Q2N2222). The emitter is connected to ground (0), and the collector is connected to a 10V DC voltage source labeled V2. The Analysis Setup dialog box is open, showing the following settings:

Enabled	Enabled
<input type="checkbox"/>	AC Sweep...
<input type="checkbox"/>	Load Bias Point...
<input type="checkbox"/>	Save Bias Point...
<input type="checkbox"/>	DC Sweep...
<input type="checkbox"/>	Monte Carlo/Worst Case...
<input checked="" type="checkbox"/>	Bias Point Detail
<input type="checkbox"/>	Digital Setup...
<input type="checkbox"/>	Options...
<input type="checkbox"/>	Parametric...
<input type="checkbox"/>	Sensitivity...
<input type="checkbox"/>	Temperature...
<input type="checkbox"/>	Transfer Function...
<input type="checkbox"/>	Transient...

A red arrow points from the Analysis Setup dialog box to the Analysis toolbar in the top menu bar. A blue arrow points from the toolbar to the circuit schematic. A large blue number '2' is centered above the dialog box, and a large blue letter 'B' is on the left side of the schematic area.

Simulazione punto operativo di un BJT: (4) esecuzione sim. e stampa risultato

2

avvio simulazione

stampa su schermo V,I

B

10.00uA
10uA
IDC

10.00V
10.00V

657.14mV

Q2

1.731mA

10V
V2

-1.741mA

0V

0

0

Simulazione punto operativo di un BJT: (4) curve Ic-Vce al variare di Ib

PSpice Schematics - [*2N2222 p.1 (stale)]

File Edit Draw Navigate View Options Analysis Tools Markers Window Help

(1)

2

(4)

DC Sweep

Swept Var. Type

- Voltage Source
- Temperature
- Current Source
- Model Parameter
- Global Parameter

Name: V2

Model Type:

Model Name:

Param. Name:

Sweep Type

- Linear
- Octave
- Decade
- Value List

Start Value: 0

End Value: 10

Increment: 0.01

Values:

Nested Sweep... OK Cancel

Analysis Setup

Enabled

- AC Sweep...
- Load Bias Point...
- Save Bias Point...
- (2) DC Sweep...
- Monte Carlo/Worst Case...
- Bias Point Detail
- Digital Setup...

Enabled

- (3) Parametric...
- Sensitivity...
- Temperature...
- Transfer Function...
- Transient...

Options... Close


10uA IDC

V2

0

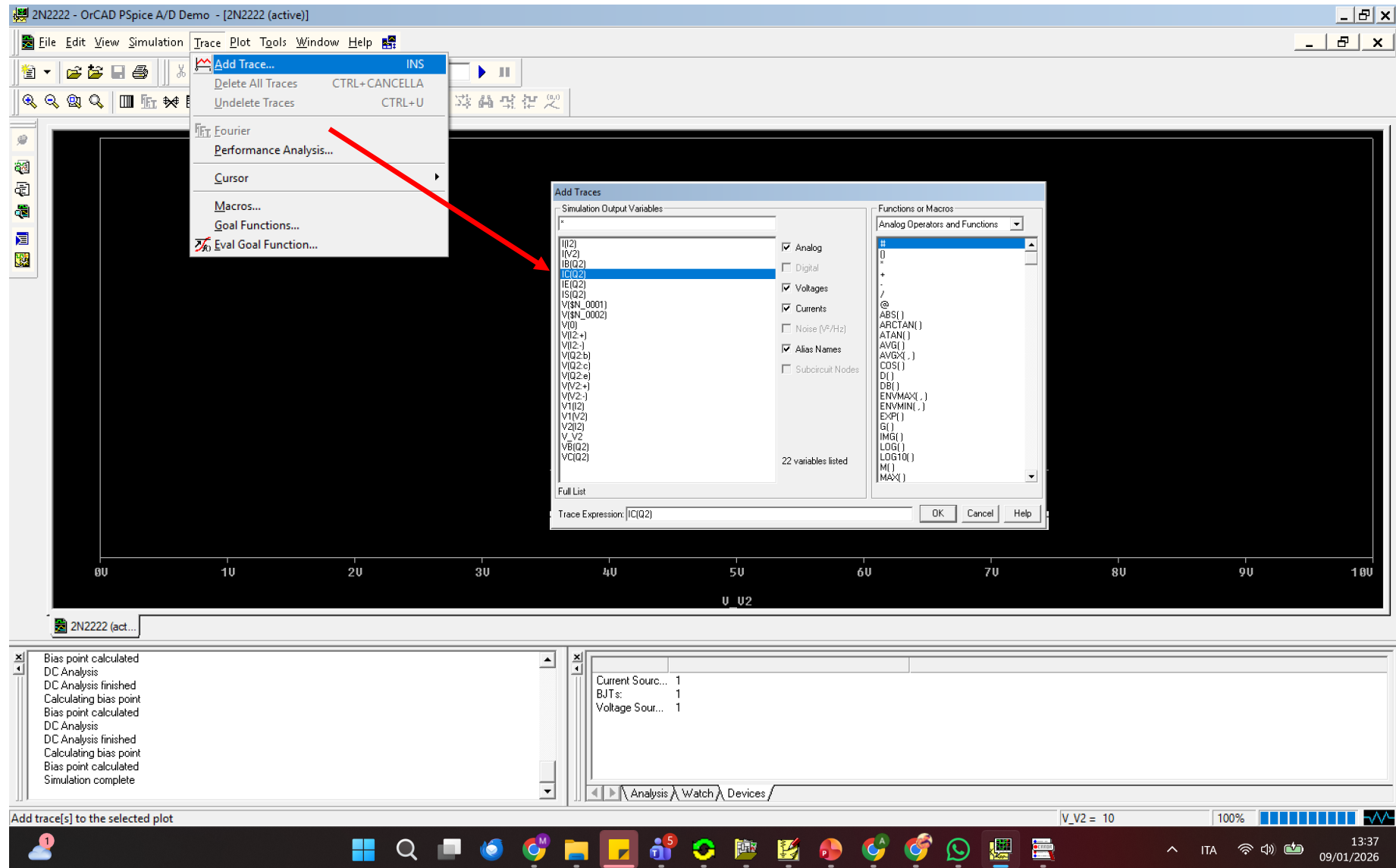
B

Simulazione punto operativo di un BJT: (4) curve Ic-Vce al variare di Ib : plot risultati

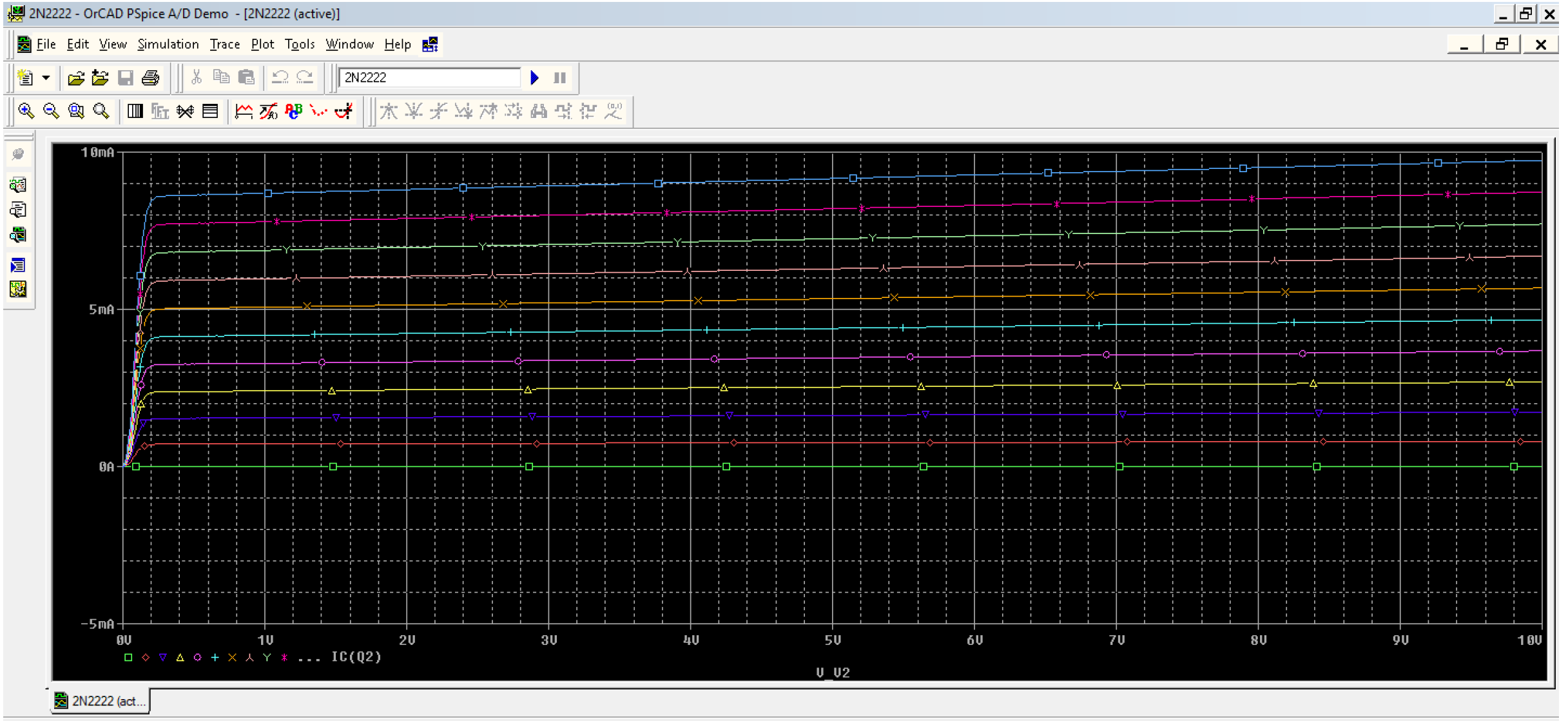
Avviata la simulazione con il bottone  in *schematic*, si apre la finestra di visualizzazione dei risultati

Dal menu Trace è possibile scegliere quale parametro visualizzare

Scelgere la corrente di collettore del transistor



Simulazione punto operativo di un BJT: (4) curve Ic-Vce al variare di Ib : plot risultati



Doppio click sugli assi per modificare la scala o etichette
Bottoni in alto per marker e misure

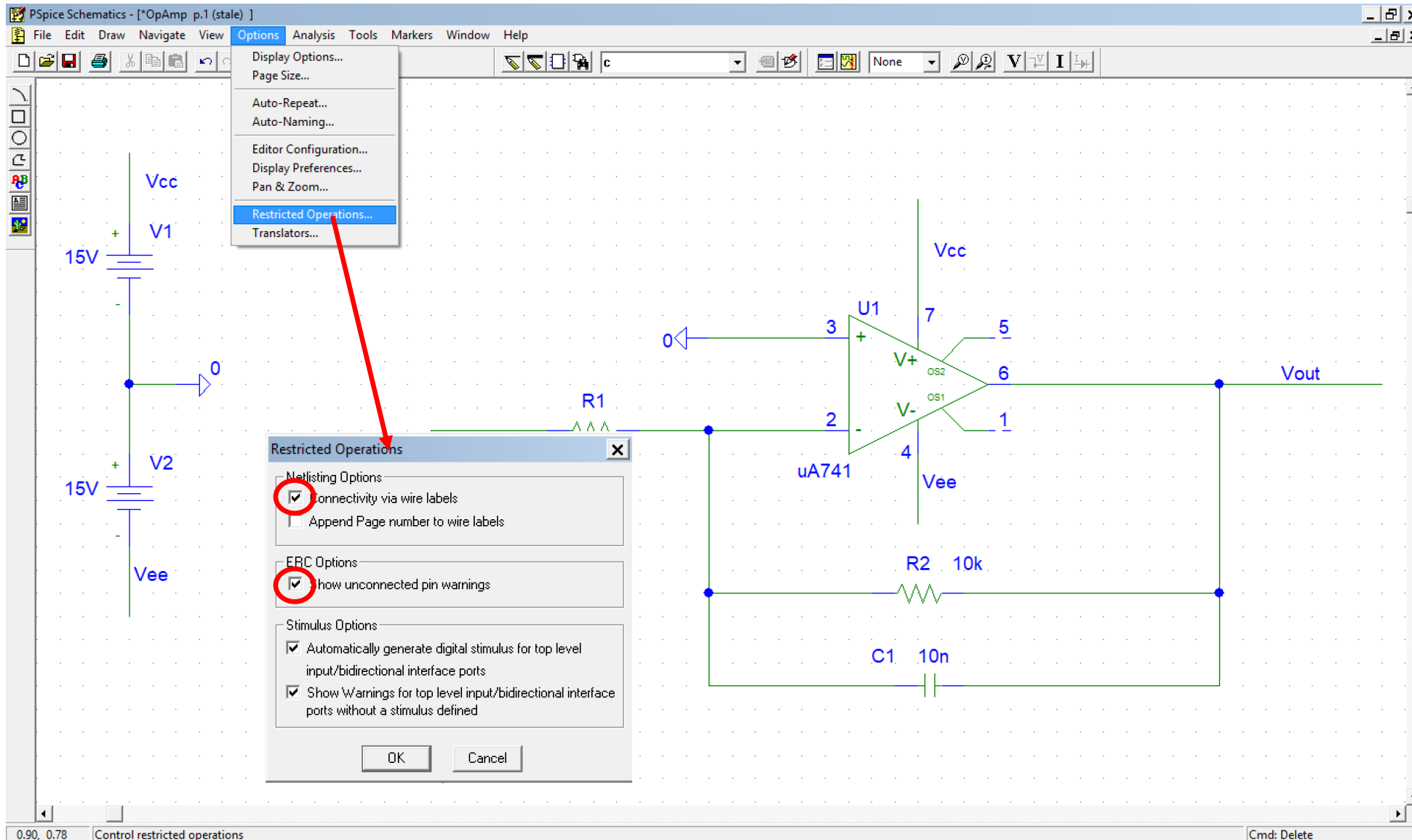
PARTE 2:

Connessione con labels

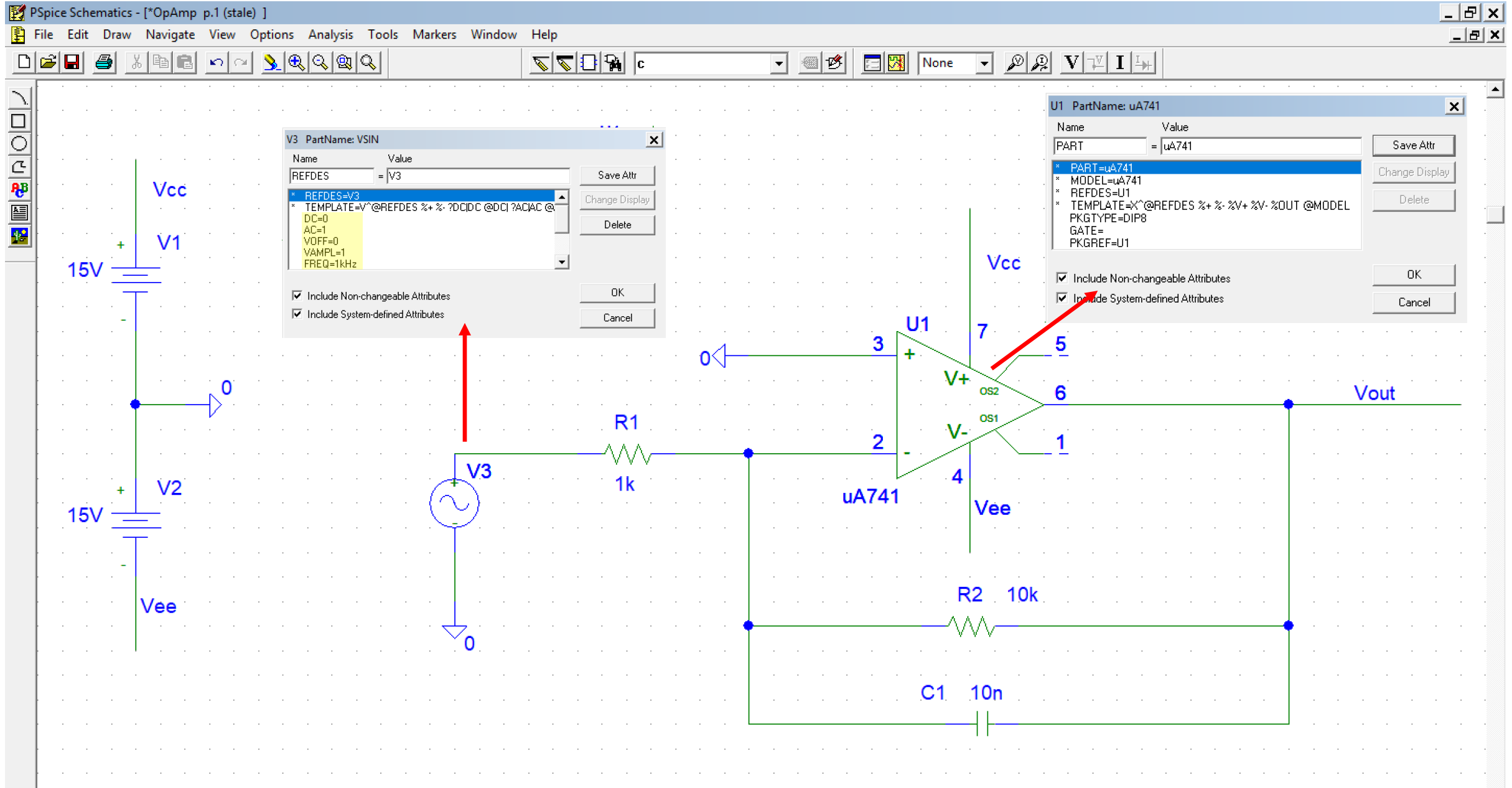
Simulazioni AC e TRANSIENT

Abilitare connessioni con labels sui fili

Con doppio click sui fili è possibile inserire labels. Stesse labels sono automaticamente collegate



Schema: Op-Amp conf. invertente passa-basso



Risposta in frequenza: simulazione AC

The image shows a screenshot of the PSpice Schematics software interface. The main window displays a circuit diagram for an AC simulation. The circuit includes a uA741 operational amplifier. The non-inverting input is connected to a 15V DC source (V1) through a 1k resistor (R1). The inverting input is connected to a 15V DC source (V2) through a 10k resistor (R2). A 10nF capacitor (C1) is connected between the inverting input and the output. The output is labeled Vout. The op-amp is powered by two 15V DC sources (V1 and V2) labeled Vcc and Vee. An AC voltage source (V3) is connected to the non-inverting input through a 1k resistor (R1). The output is labeled Vout.

Two dialog boxes are open over the circuit:

- AC Sweep and Noise Analysis:**
 - AC Sweep Type: Decade
 - Sweep Parameters: Pts/Decade: 101, Start Freq.: 10, End Freq.: 100K
 - Noise Analysis: Noise Enabled (unchecked)
- Analysis Setup:**
 - Enabled: AC Sweep... (checked), Bias Point Detail (checked), Digital Setup... (checked)
 - Options... (unchecked), Parametric... (unchecked), Sensitivity... (unchecked), Temperature... (unchecked), Transfer Function... (unchecked), Transient... (unchecked)

The software interface includes a menu bar (File, Edit, Draw, Navigate, View, Options, Analysis, Tools, Markers, Window, Help) and a toolbar with various simulation and editing tools. The status bar at the bottom shows the version (2.41, 0.78) and the current task: "Set up the simulation analyses for the active schematic".

Risposta in frequenza: simulazione AC, Risultato

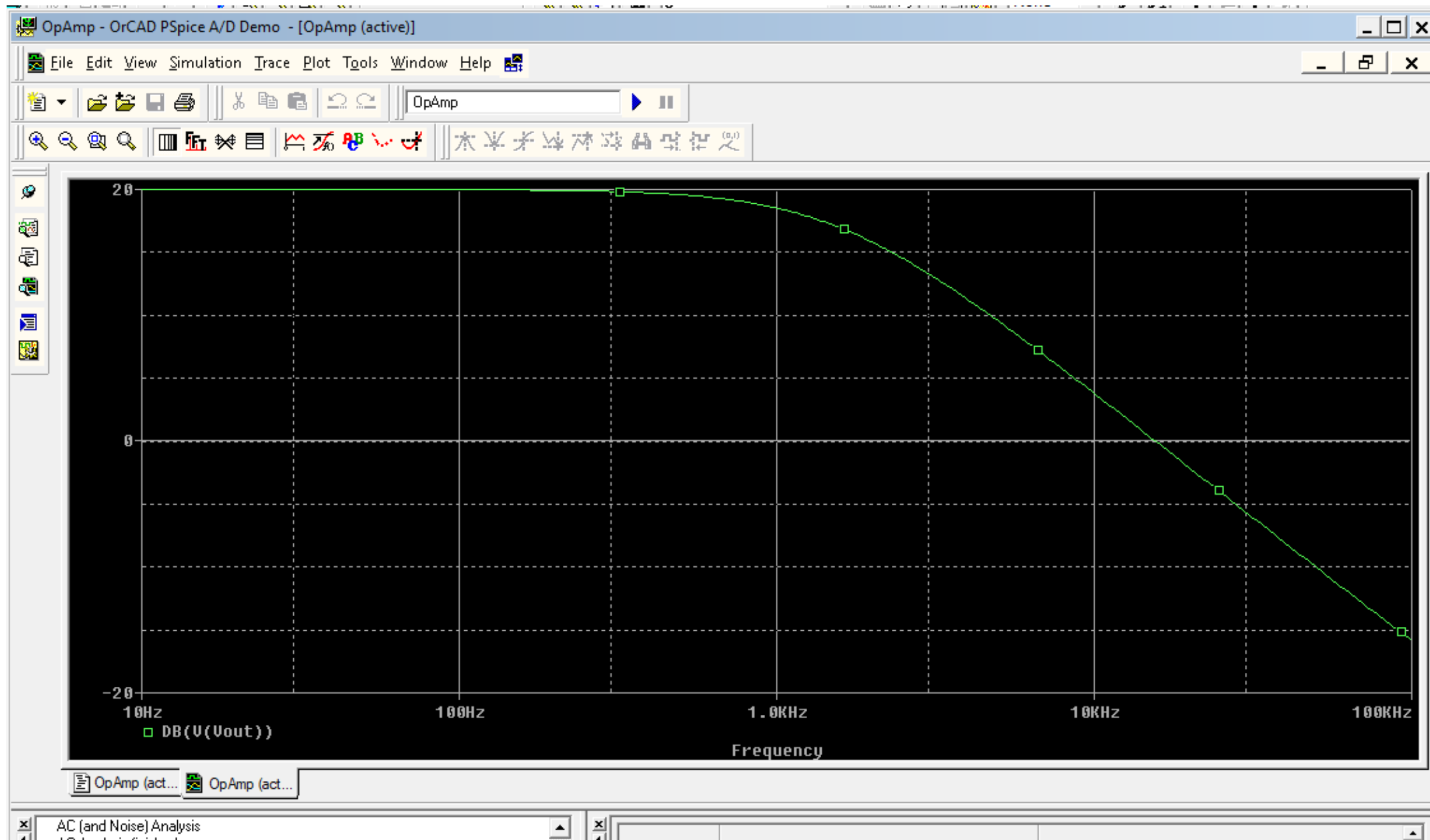
The screenshot displays the OrCAD PSpice A/D Demo software interface. The main window shows a frequency plot with a logarithmic x-axis ranging from 10Hz to 100kHz. The plot area is currently empty, with the x-axis labeled 'Frequency' and tick marks at 10Hz, 30Hz, 100Hz, 300Hz, 1.0kHz, 3.0kHz, 10kHz, 30kHz, and 100kHz. The y-axis is not visible.

The 'Add Traces' dialog box is open, showing a list of simulation output variables. The variable 'V(Vout)' is selected. The 'Trace Expression' field contains 'DB[V(Vout)]'. The dialog also shows a list of functions or macros, with 'DB()' selected. The 'Trace Expression' field is set to 'DB[V(Vout)]'. The 'Functions or Macros' list includes: #, (), *, +, -, /, @, ABS(), ARCTAN(), ATAN(), AVG(), AVGX(), COS(), D(), DB(), ENVMAX(), ENVMIN(), EXP(), G(), IMG(), LOG(), LOG10(), M(), and MAX().

The 'Simulation Output Variables' list includes: V(V3-), V(Vcc), V(Vee), V(Vout), V(X_U1.10), V(X_U1.11), V(X_U1.12), V(X_U1.13), V(X_U1.14), V(X_U1.53), V(X_U1.54), V(X_U1.6), V(X_U1.7), V(X_U1.8), V(X_U1.9), V(X_U1.90), V(X_U1.91), V(X_U1.92), V(X_U1.99), V1(C1), V1(R1), V1(R2), V1(V1), and V1(V3). The 'Trace Expression' field is set to 'DB[V(Vout)]'. The 'Functions or Macros' list includes: #, (), *, +, -, /, @, ABS(), ARCTAN(), ATAN(), AVG(), AVGX(), COS(), D(), DB(), ENVMAX(), ENVMIN(), EXP(), G(), IMG(), LOG(), LOG10(), M(), and MAX().

The status bar at the bottom shows 'Simulation running...' and 'No recognized product configuration selected.' The system tray at the bottom right shows the date and time: 09/01/2026, 18:15.

Risposta in frequenza: **simulazione AC, Risultato**



Trans-caratteristica (Vout_vs_Vin) : sweep DC

The image shows a PSpice Schematics window titled "OpAmp p.1 (simulating)". The circuit diagram includes two 15V DC voltage sources, V1 and V2, connected to the non-inverting input of an operational amplifier. The output of the op-amp is labeled Vout. A feedback network is connected between the output and the inverting input, consisting of a resistor R2 (10k) and a capacitor C1 (10n) in parallel. Two dialog boxes are open over the circuit:

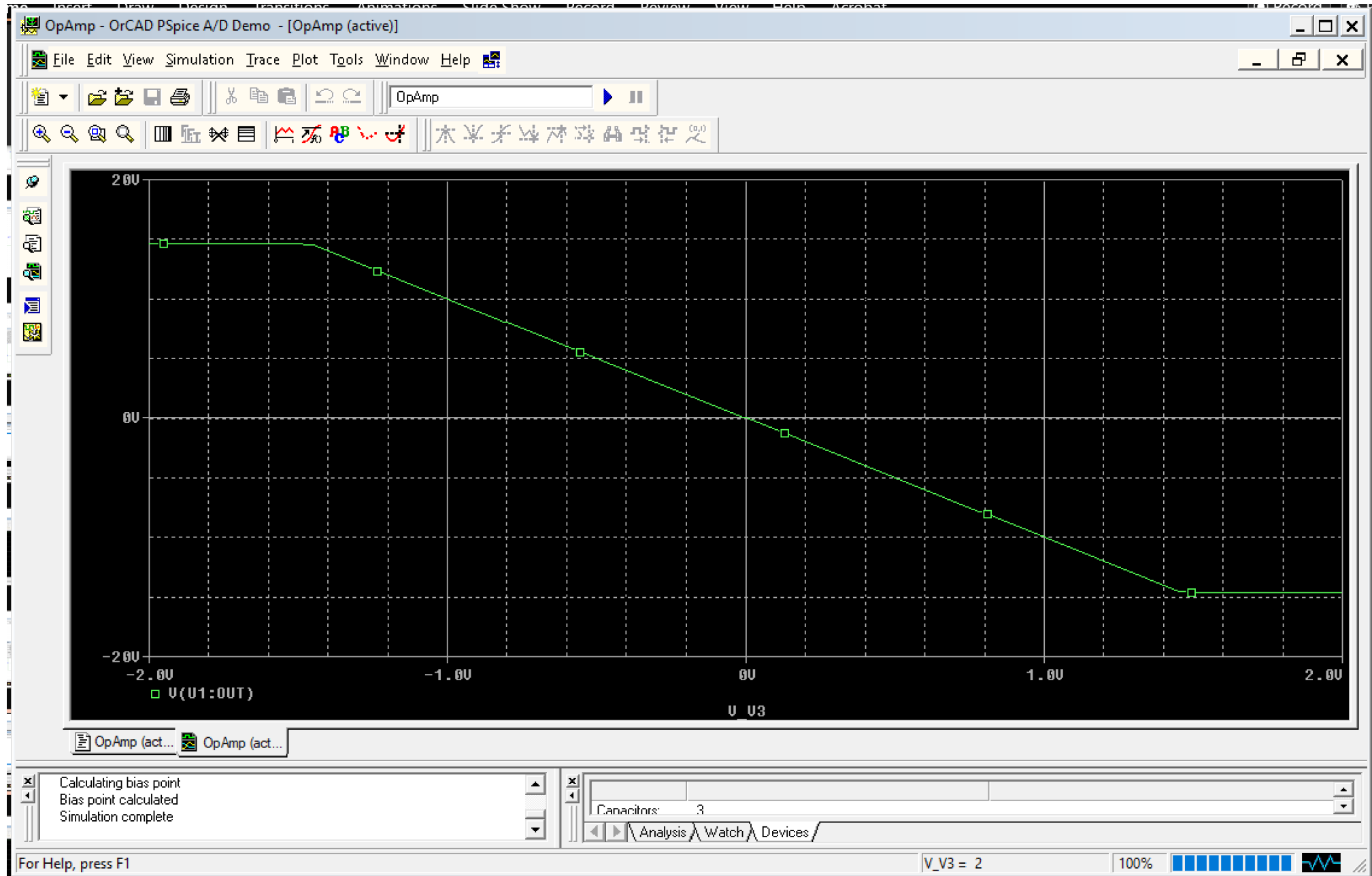
- DC Sweep:** This dialog is used to configure a DC sweep analysis. It shows "Swept Var. Type" set to "Voltage Source" with "Name: V3". Under "Sweep Type", "Linear" is selected. The "Start Value" is -2, "End Value" is 2, and "Increment" is 0.01.
- Analysis Setup:** This dialog shows the configuration for the simulation analysis. Under the "Enabled" section, "DC Sweep..." is checked, indicating that a DC sweep analysis is being performed.

The status bar at the bottom of the window displays "4.44, 0.78" and "Set up the simulation analyses for the active schematic". The Windows taskbar at the very bottom shows the date and time as "09/01/2026 19:00".

Trans-caratteristica (Vout_vs_Vin) : Risultato

The screenshot displays the PSpice software interface during a simulation. The main window shows a plot of the transfer characteristic, with the x-axis labeled 'Frequency' and markers at 10Hz, 30Hz, and 100Hz. The 'Add Traces' dialog box is open, showing a list of simulation output variables. The variable 'V(U1:OUT)' is selected and highlighted in blue. The 'Trace Expression' field at the bottom of the dialog contains 'V(U1:OUT)'. The dialog also includes checkboxes for 'Analog', 'Digital', 'Voltages', 'Currents', 'Noise (V²/Hz)', 'Alias Names', and 'Subcircuit Nodes'. The 'Functions or Macros' section is visible on the right, showing a list of mathematical functions like ABS(), ARCTAN(), ATAN(), etc. The status bar at the bottom indicates 'Freq = 100.0E+03' and '100%'. The Windows taskbar is visible at the bottom of the screen, showing the date and time as 09/01/2026 18:15.

Trans-caratteristica (Vout_vs_Vin), Risultato



Simulazioni transient

PSpice Schematics - [OpAmp p.1 (simulating)]

File Edit Draw Navigate View Options Analysis Tools Markers Window Help

None

Transient

Transient Analysis

Print Step: 0ns

Final Time: 10ms

No-Print Delay:

Step Ceiling: 0.01m

Detailed Bias Pt.

Skip initial transient solution

Fourier Analysis

Enable Fourier

Center Frequency: 1kHz

Number of harmonics: 10

Output Vars.: V(Vout)

OK Cancel

Analysis Setup

Enabled

AC Sweep...

Load Bias Point...

Save Bias Point...

DC Sweep...

Monte Carlo/Worst Case...

Bias Point Detail

Digital Setup...

Enabled

Options...

Parametric...

Sensitivity...

Temperature...

Transfer Function...

Transient...

Close

Vcc

V1

15V

V2

15V

Vee

U1

5

6

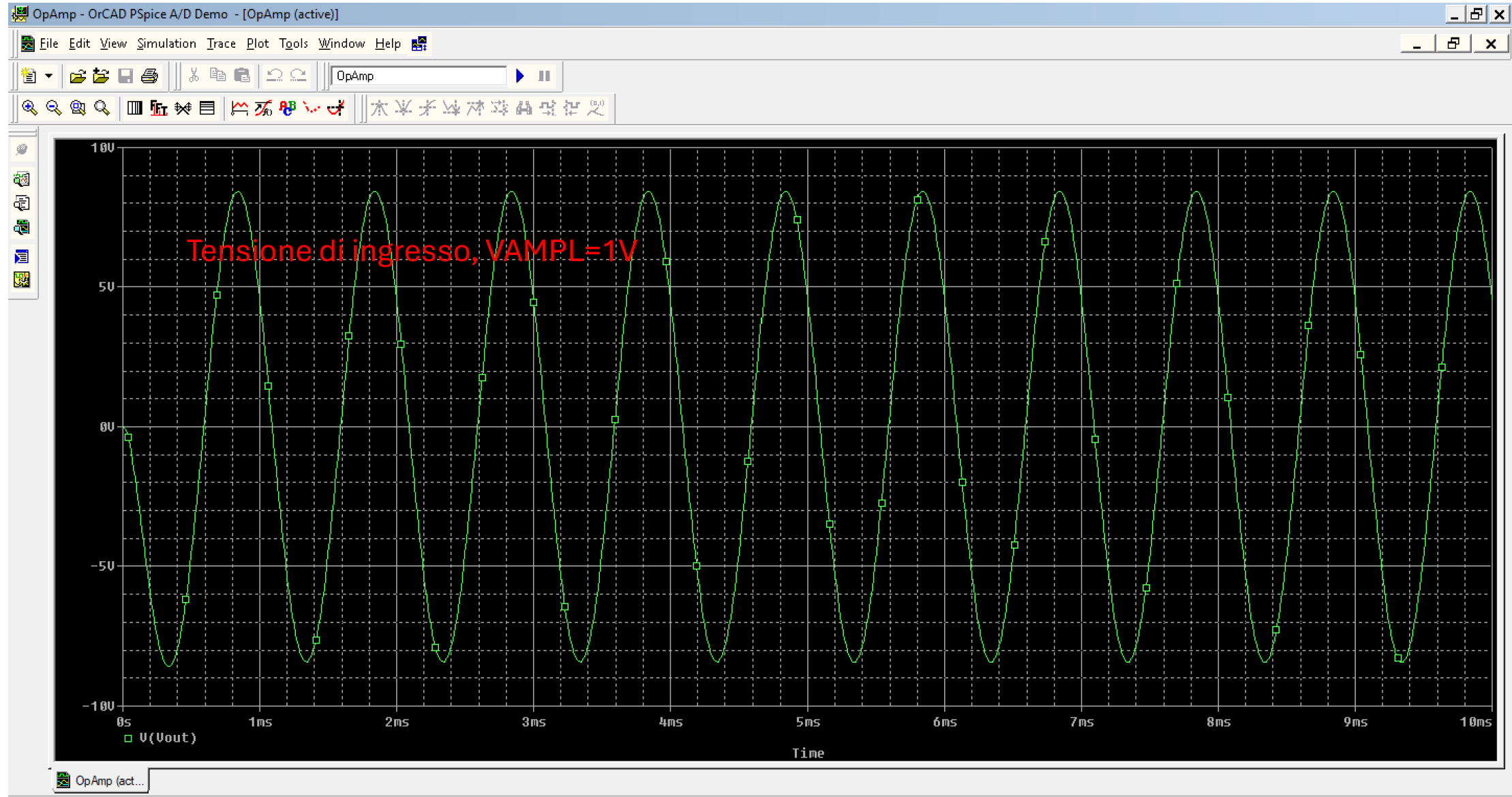
1

Vout

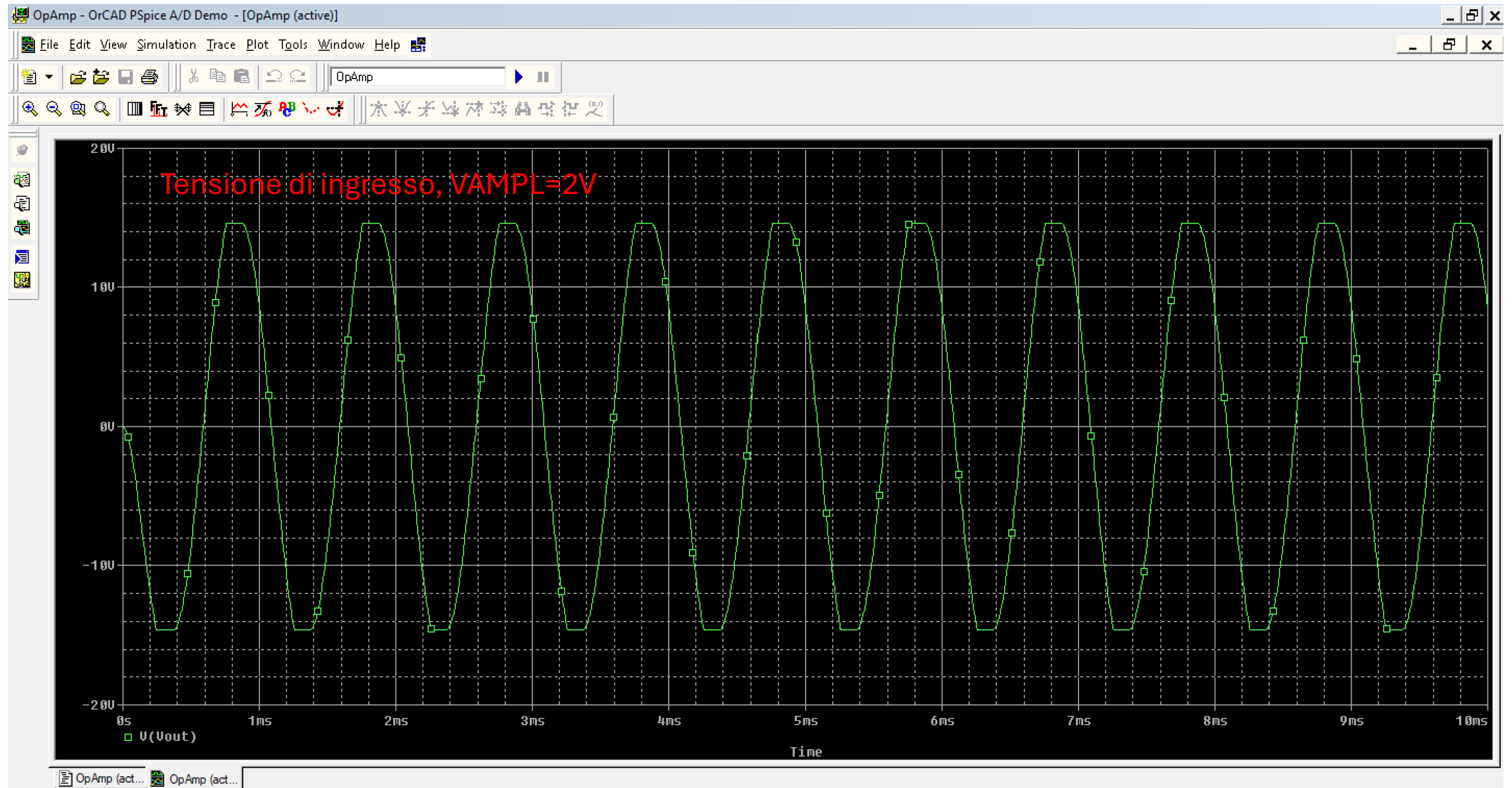
R2 10k

C1 10n

Simulazioni transient : Risultato



Simulazioni transient : Risultato



Simulazioni transient : FFT

