

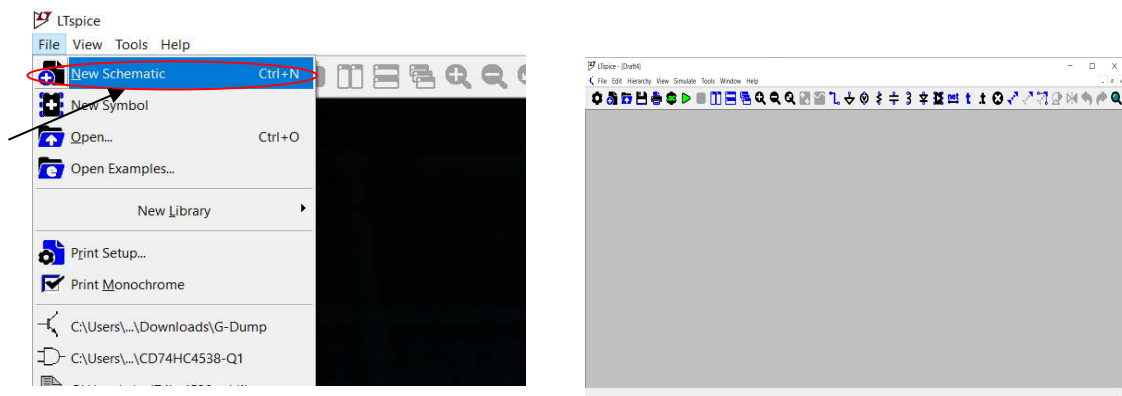
LTspice Simulation

- Download and install LTspice from the following website:

<https://www.analog.com/en/resources/design-tools-and-calculators/ltspice-simulator.html>

- Create a new schematic:-

Open LTspice >Go to “File”> select “New Schematic”



- Design the schematic diagram by using the tools available in the top of the window.


Also by adding the required components.

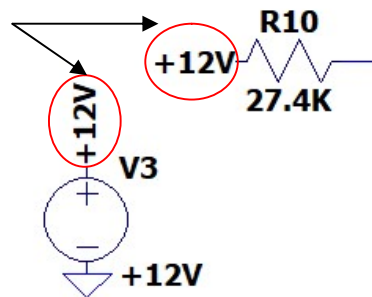


Use Shortcuts, if required:

Draw Wire:	W
Place Ground:	G
Place Voltage Source:	V
Place Resistor:	R
Place Capacitor:	C
Place Inductor:	L
Place Diode:	D
Place Component:	P

Move Mode:	M
Stretch Mode:	S
Rotate:	Ctrl+R
Mirror:	Ctrl+E
Delete Mode:	Backspace or Del
Duplicate Mode:	Ctrl+C
Undo:	Ctrl+Z
Redo:	Ctrl+Shift+Z

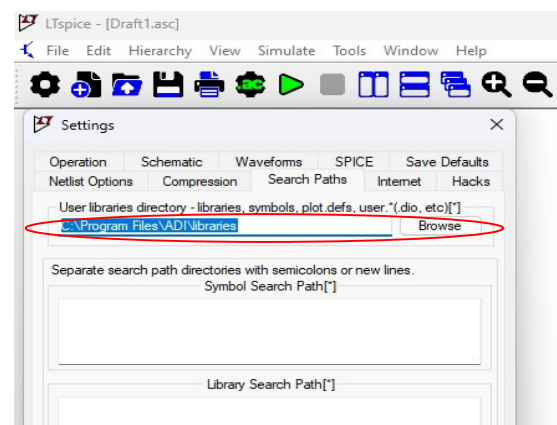
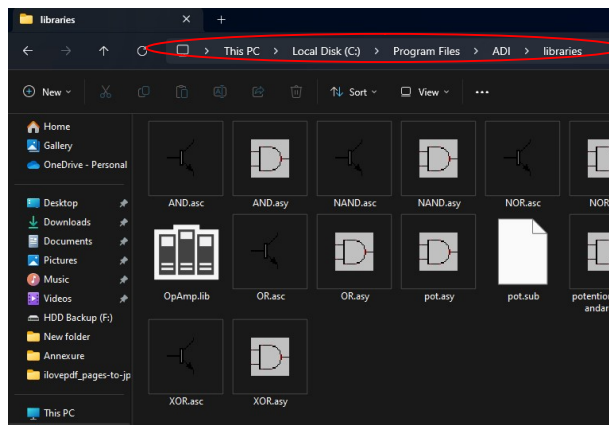
Use 'nets' , to give common connections such as power source, without connecting wires.



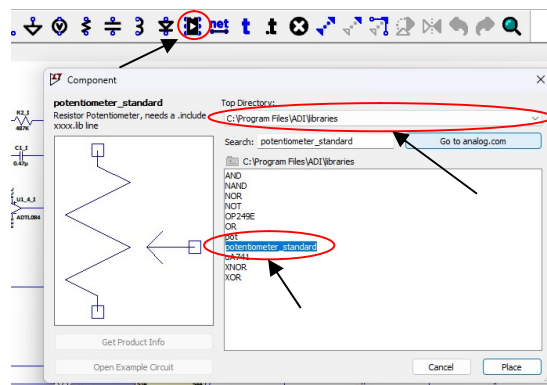
For custom components/IC's/libraries/symbols, downloaded from the internet.

→To add them to LTspice, Add the respective path of the library directory in the **Search Paths**.

LTspice→Settings→Search Paths



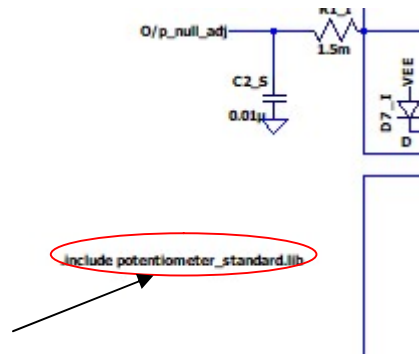
→Then choose the components from the component menu.





For custom libraries:-

→ You must also add the **SPICE Directive** by clicking this button  in the schematic.

→ It must be in the format - **.include xxxxxxx.lib**



Analysis:

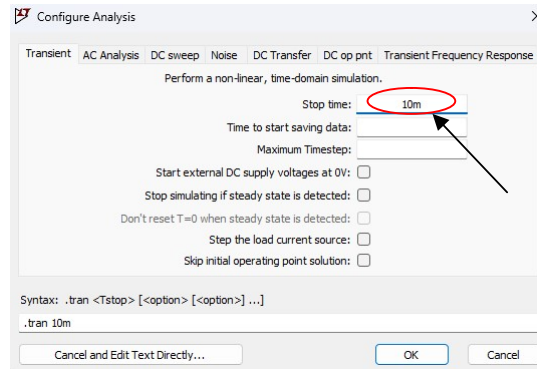
- To configure the analysis, Click this icon 
 - Select the type of analysis and give the required parameters.
 - Click the play button , to perform the analysis.
 - Click on a node, using the probe to analyse the signal at the respective node such as input and output.
 - The waveform will be displayed as in the figure below.
 - Right click the graph → Place cursor on active trace.
- Toggle the nodes to analyse the values at a point at different nodes, using the cursor.

1) Transient Analysis:

This is used to analyse the change in output with respect to time.

Steps to perform analysis:

- Configure the analysis, by specifying the time. (say, 10m)
- Configure the analysis parameters, as per the image below:

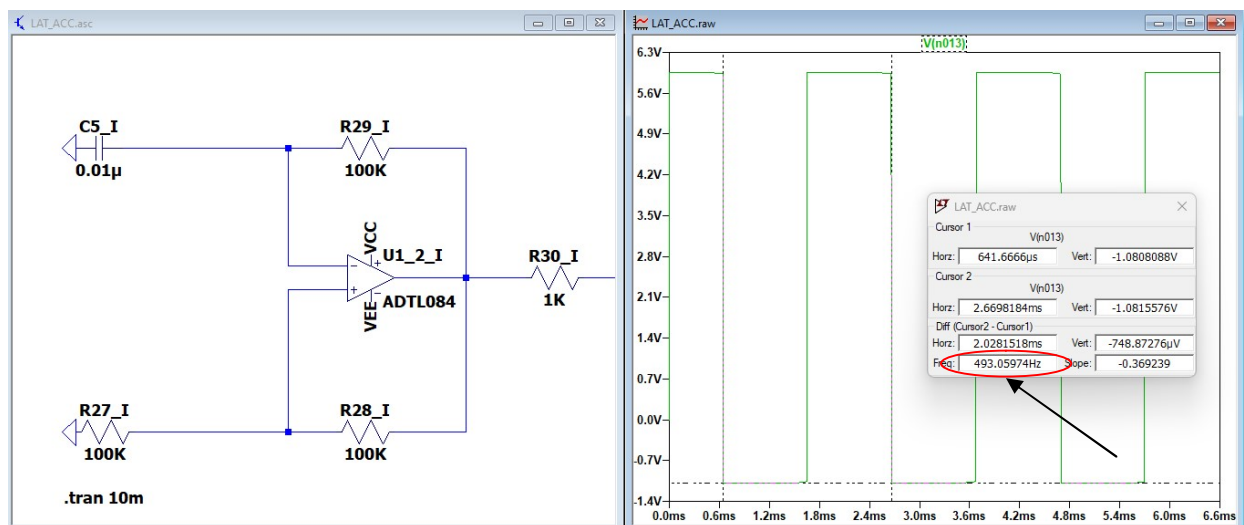


(Vary the start and stop time as per the circuit)

Click OK and place the 'SPICE Directive' text, by clicking on the schematic.

.tran 10m

- Perform analysis
- Click the nodes at input and output to view the graph.
- Use cursor to analyse the output for a particular input.



(This is a square wave oscillator circuit; Using transient analysis we can measure that, it is oscillating at a frequency of 493 Hz)

2) AC Analysis:

This is used to analyze the response of circuits to AC signals at different frequencies, essentially examining the output as the frequency changes.

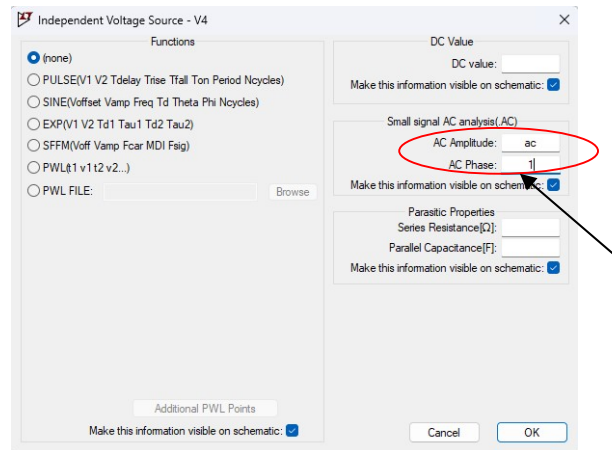
Ex: Analysing the behaviour of a low pass filter circuit.

Steps to perform analysis:

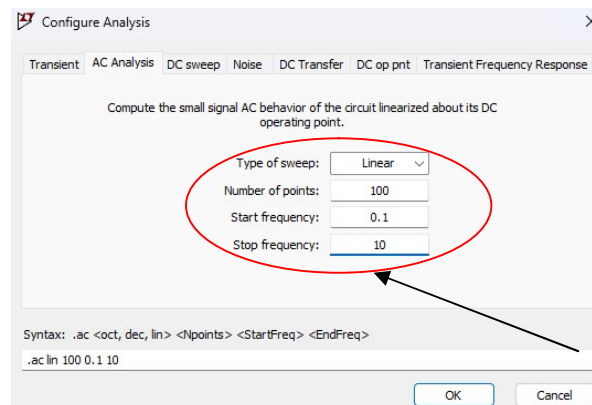
- Configure 'signal source' for analysis,
Create a Input signal source → Open its properties → Click 'Advanced' →

Ac amplitude -- ac

Ac phase -- 1



- Configure the analysis parameters, as per the image below:

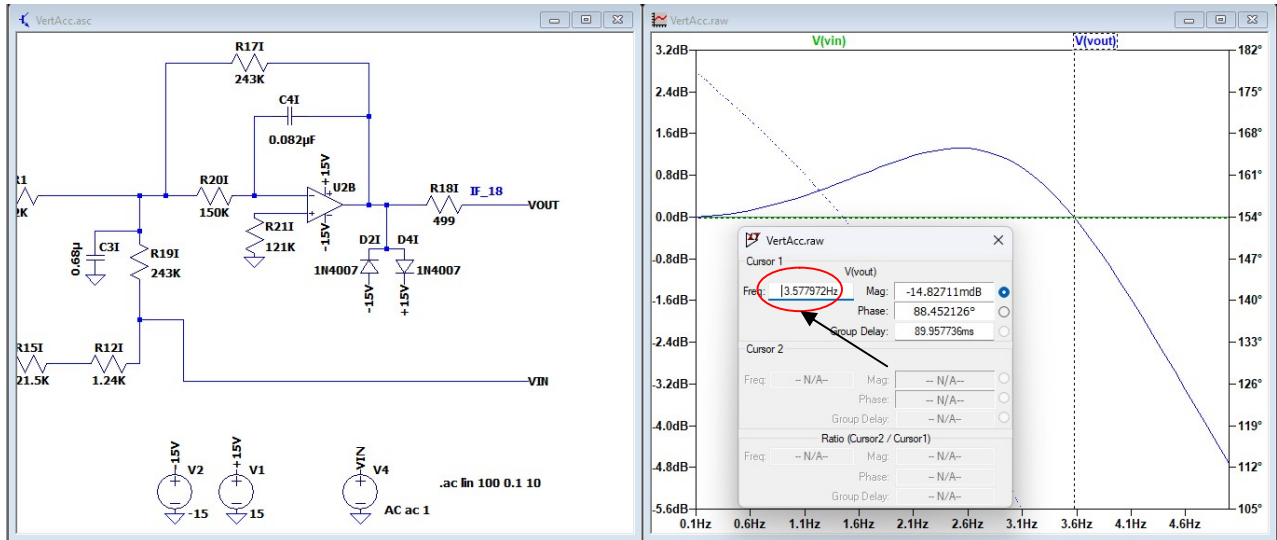


(Vary the start and stop frequency as per the circuit)

Click OK and place the 'SPICE Directive' text, by clicking on the schematic.

.ac lin 100 0.1 10

- Perform AC analysis
- Click the nodes at input and output to view the graph.
- Use cursor to analyse the output for a particular input.



(From the analysis, we can see that the circuit allows only upto 3.5Hz and acts as a low pass filter)

3) DC Sweep:

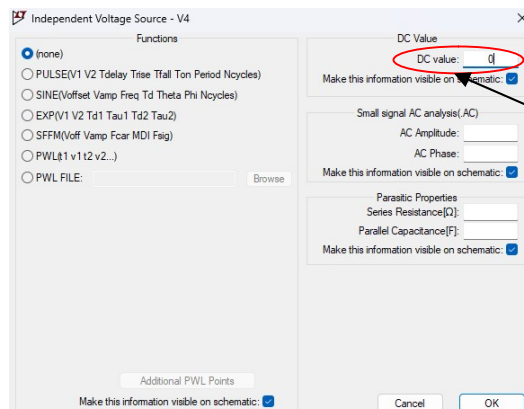
This is used to analyse the change in output with respect to change in DC Voltage.

Ex: Analysing the behaviour of an inverter circuit/ amplifier circuit/ comparator circuit/...

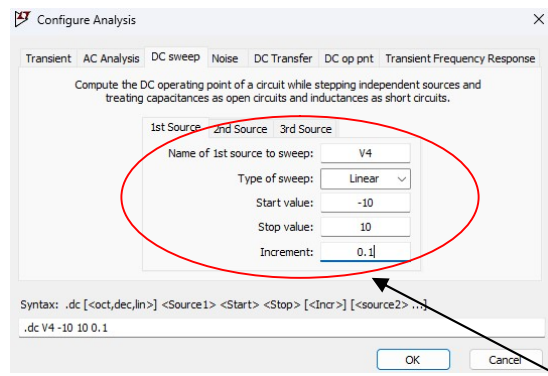
Steps to perform analysis:

- Configure 'signal source' for analysis,
Create a Input signal source→Open its properties→Click 'Advanced'→

DC Value -- 0



- Configure the analysis parameters, as per the image below:

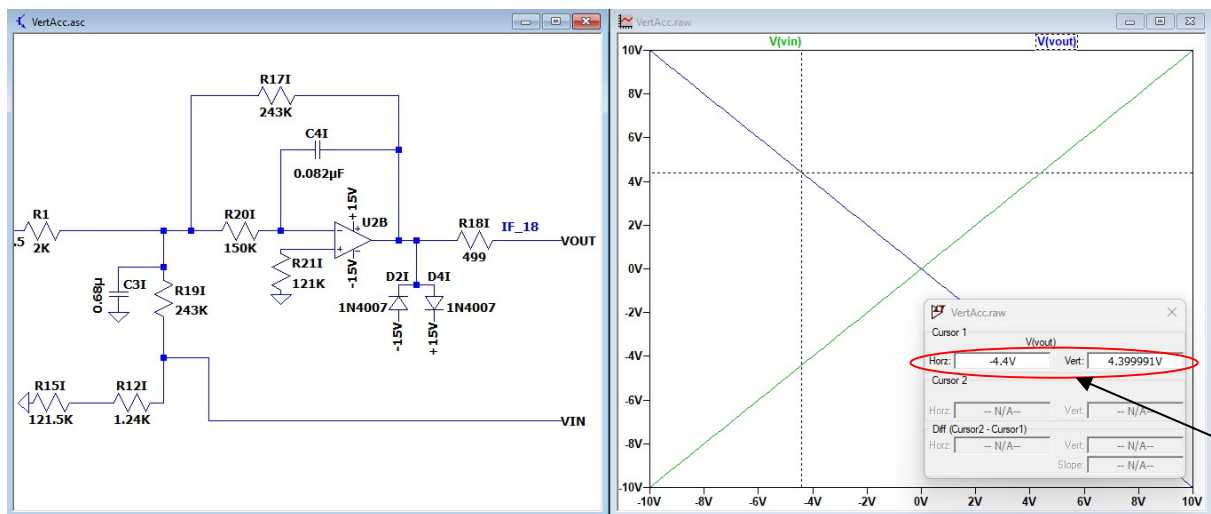


(Vary the start and stop time as per the circuit)

Click OK and place the 'SPICE Directive' text, by clicking on the schematic.

.dc V4 -10 10 0.1

- Perform analysis
- Click the nodes at input and output to view the graph.
- Use cursor to analyse the output for a particular input.



(By the analysis results- The circuit acts as an **Inverter**)