

# OPA2991 PSpice

Sim model test

Art Kay

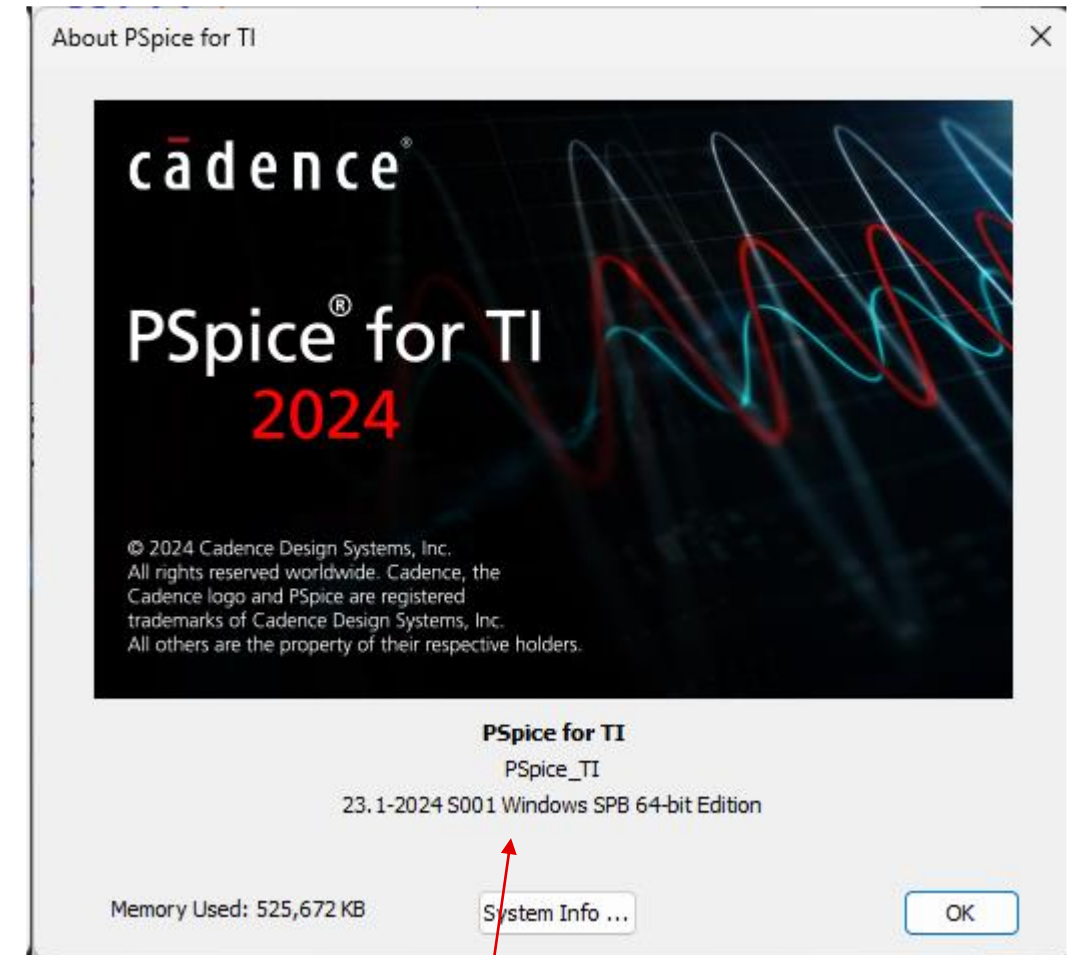
1-28-2025



# Step 1: install / use PSpice for TI

[https://www.ti.com/tool/PSPICE-FOR-TI?keyMatch=pspice%20for%20ti%20download&tisearch=universal\\_search](https://www.ti.com/tool/PSPICE-FOR-TI?keyMatch=pspice%20for%20ti%20download&tisearch=universal_search)

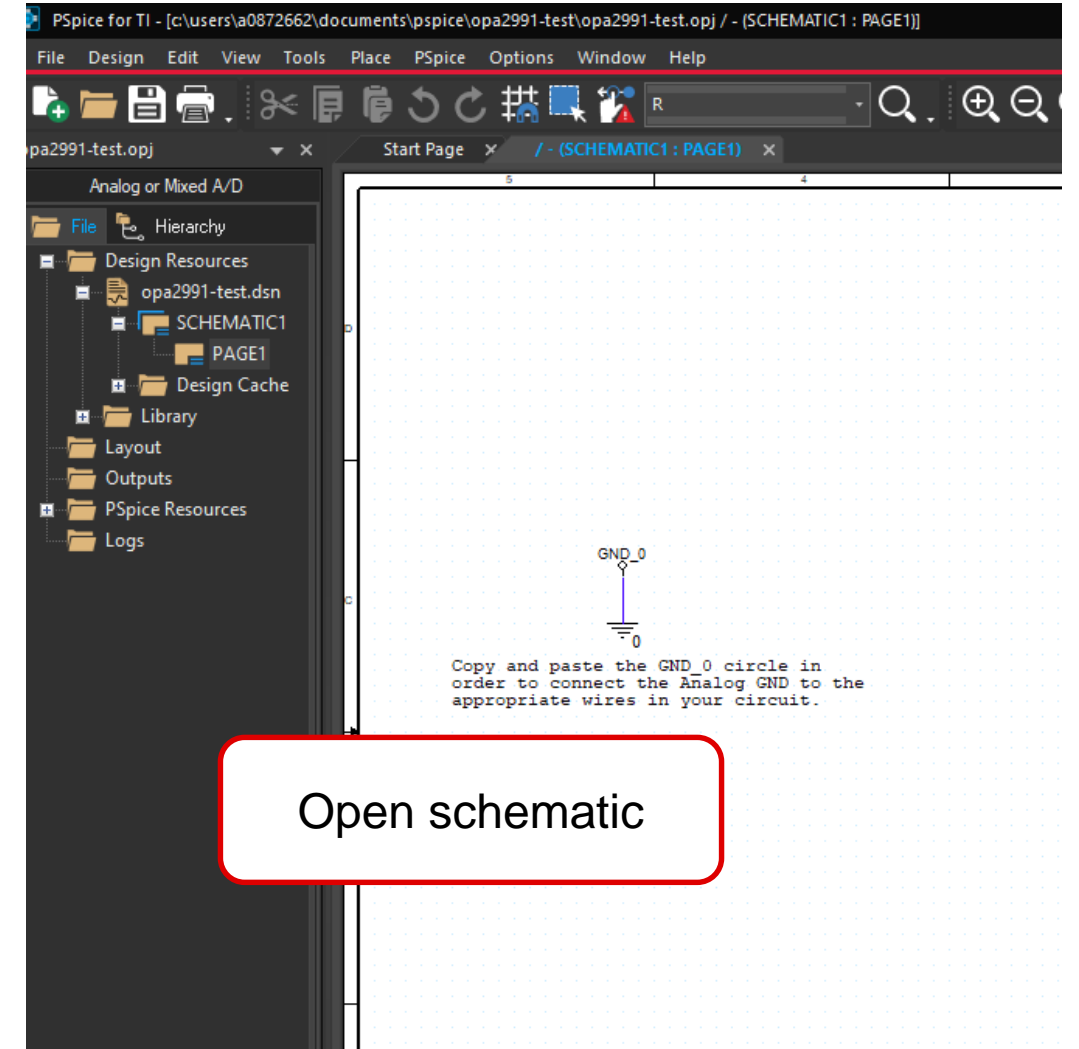
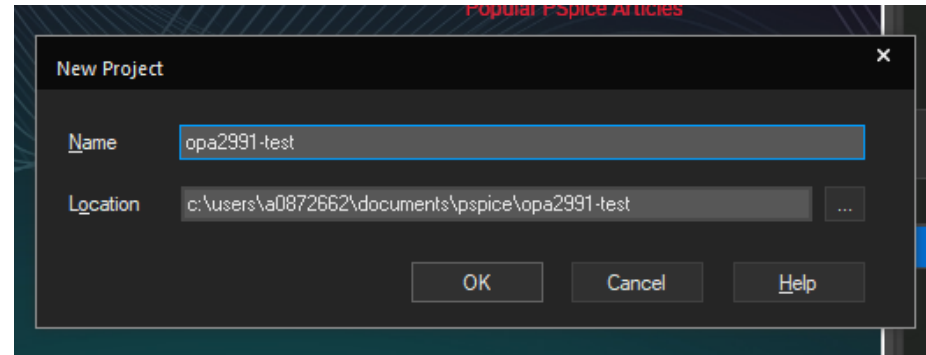
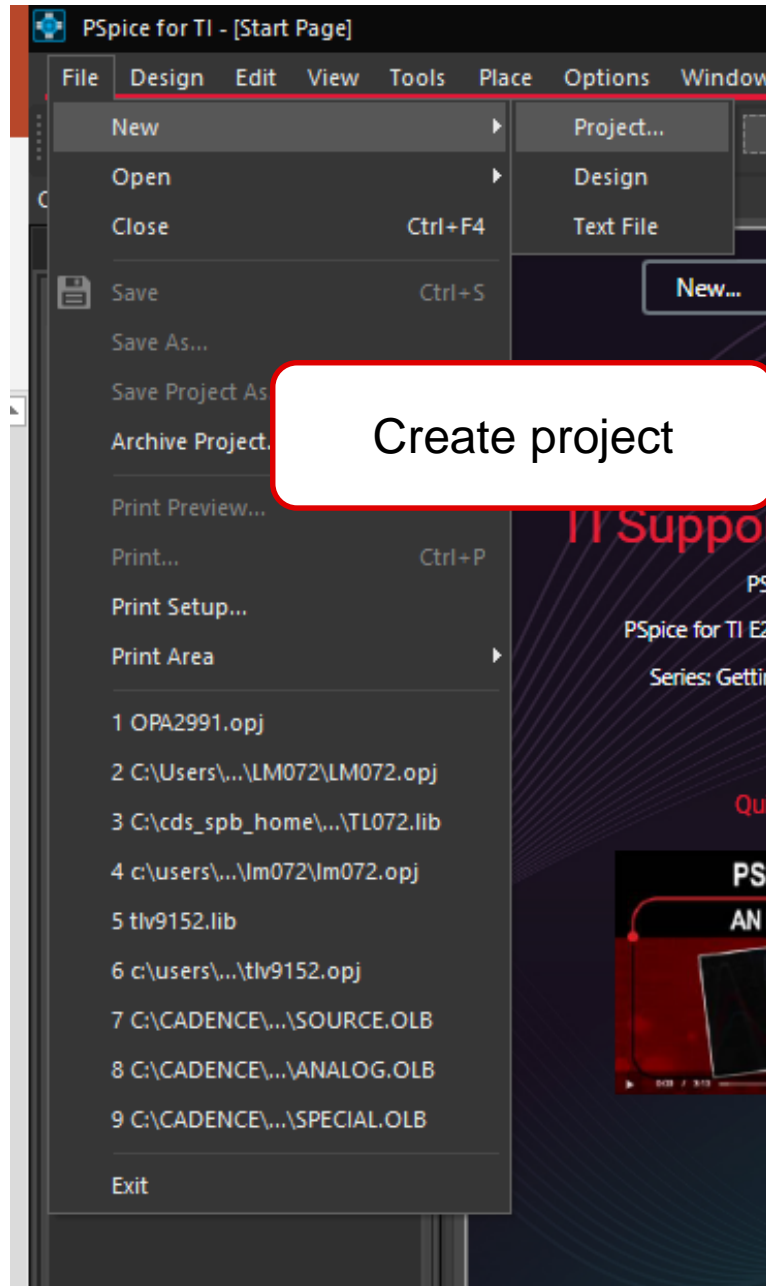
Search for PSpice for TI to download and install this software



My version



# Create project



# Get model from integrated library

Click on IC symbol with magnifying glass to launch PSpice part search (integrated library)

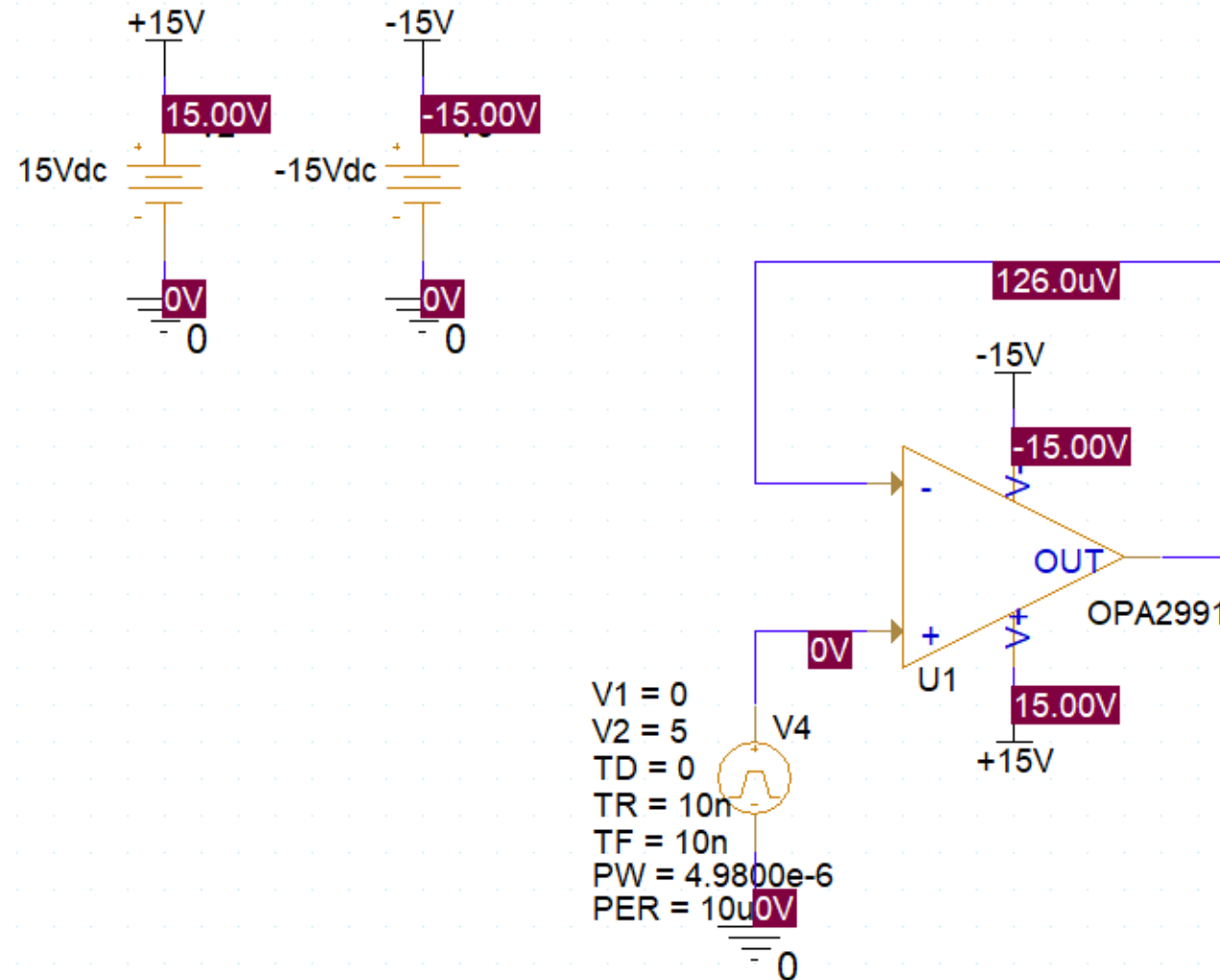
The screenshot shows the PSpice Part Search dialog box overlaid on a schematic diagram. The schematic includes an operational amplifier component labeled U1 (OPA2991) and a ground connection labeled GND\_0. The PSpice Part Search window has a search bar containing 'OPA2991' and a search icon. Below the search bar is a table with the following data:

PART NAME	DESCRIPTION
OPA2991	Dual, 40-V, 4.5-MHz, low-power oper...

A red arrow points from the magnifying glass icon in the schematic toolbar to the search bar in the PSpice Part Search dialog. Another red arrow points from the search results table to a callout box.

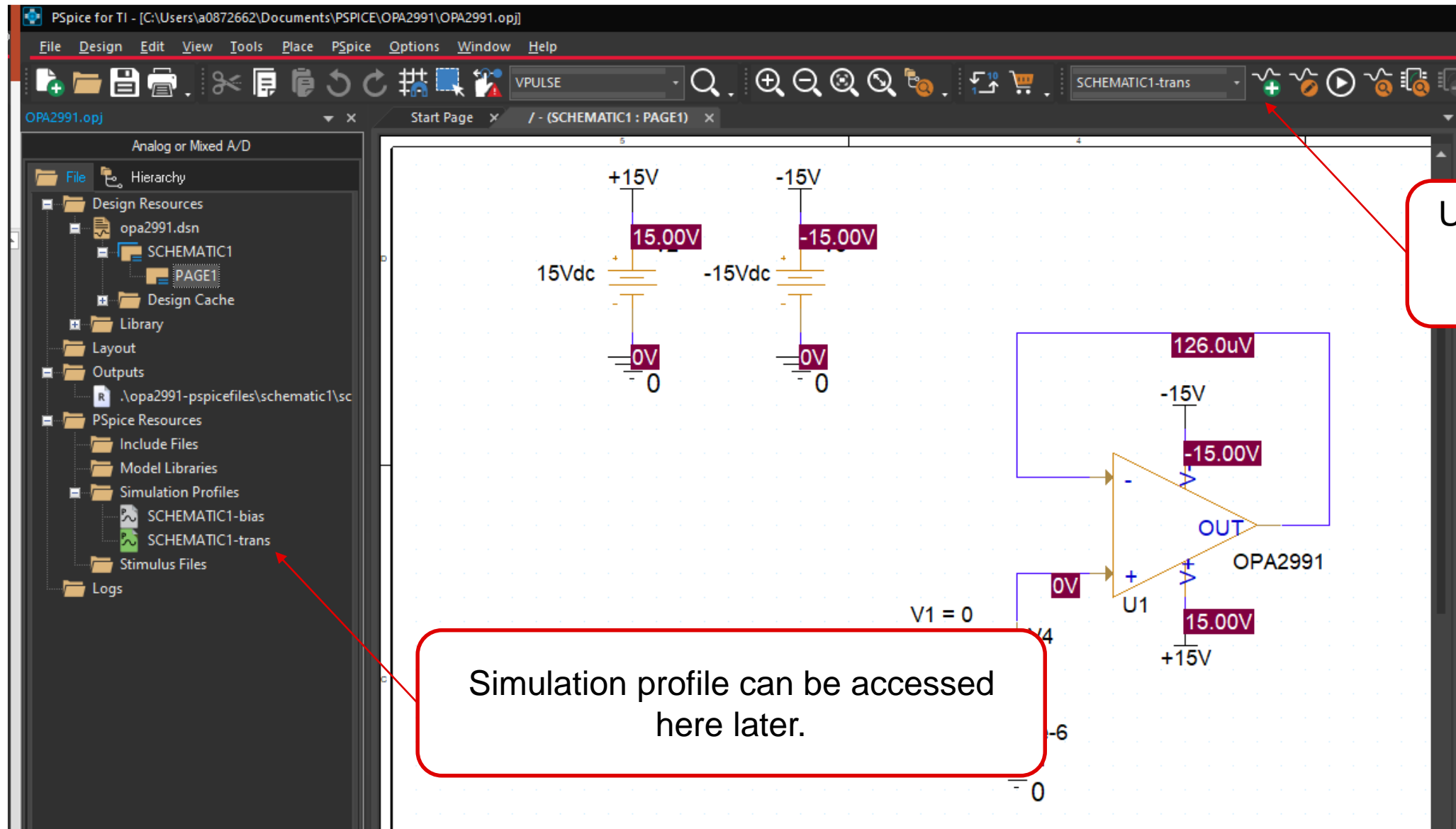
Search for the desired device. Double click and drag to schematic.

# Finish wiring schematic



Use IC symbol with green plus sign for "modeling application". This has the transient source.

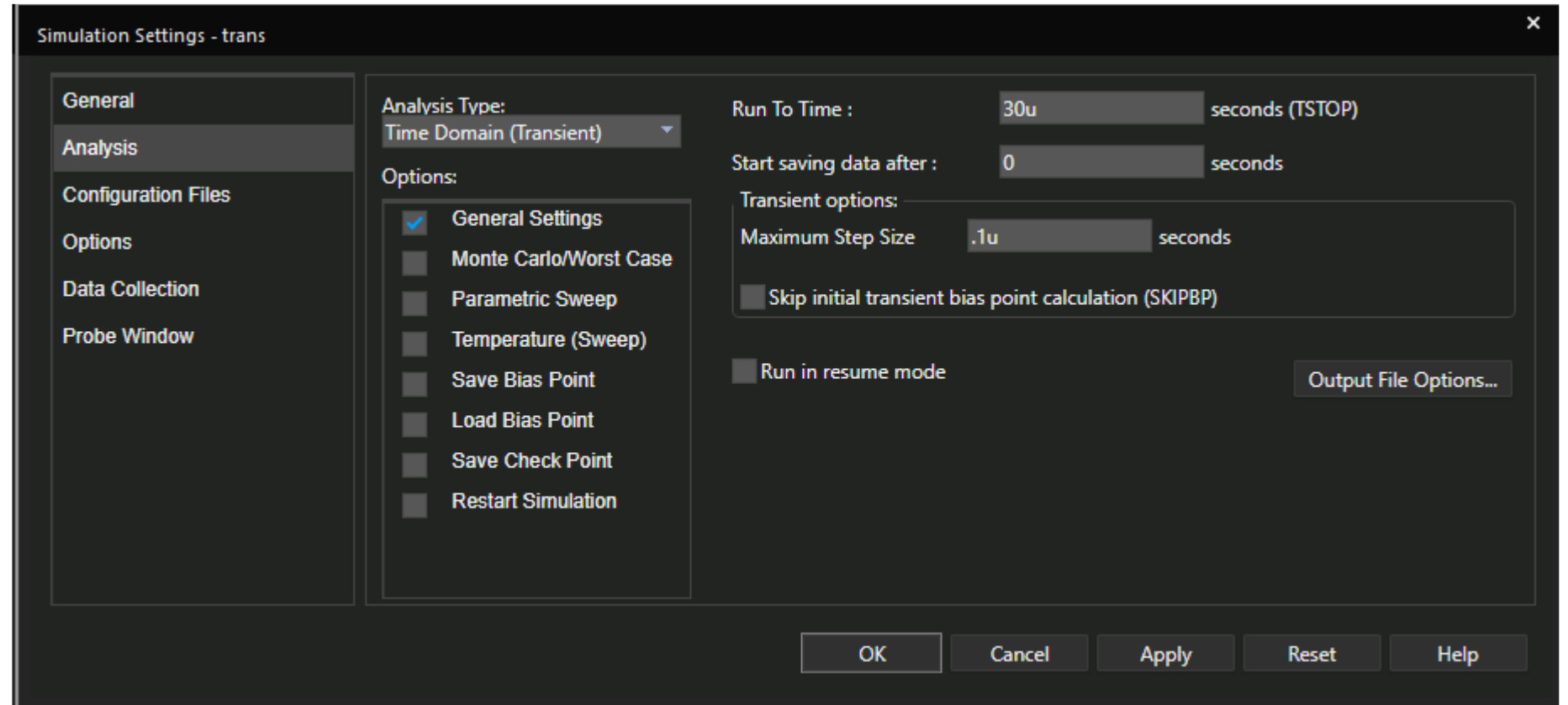
# Create simulation profile



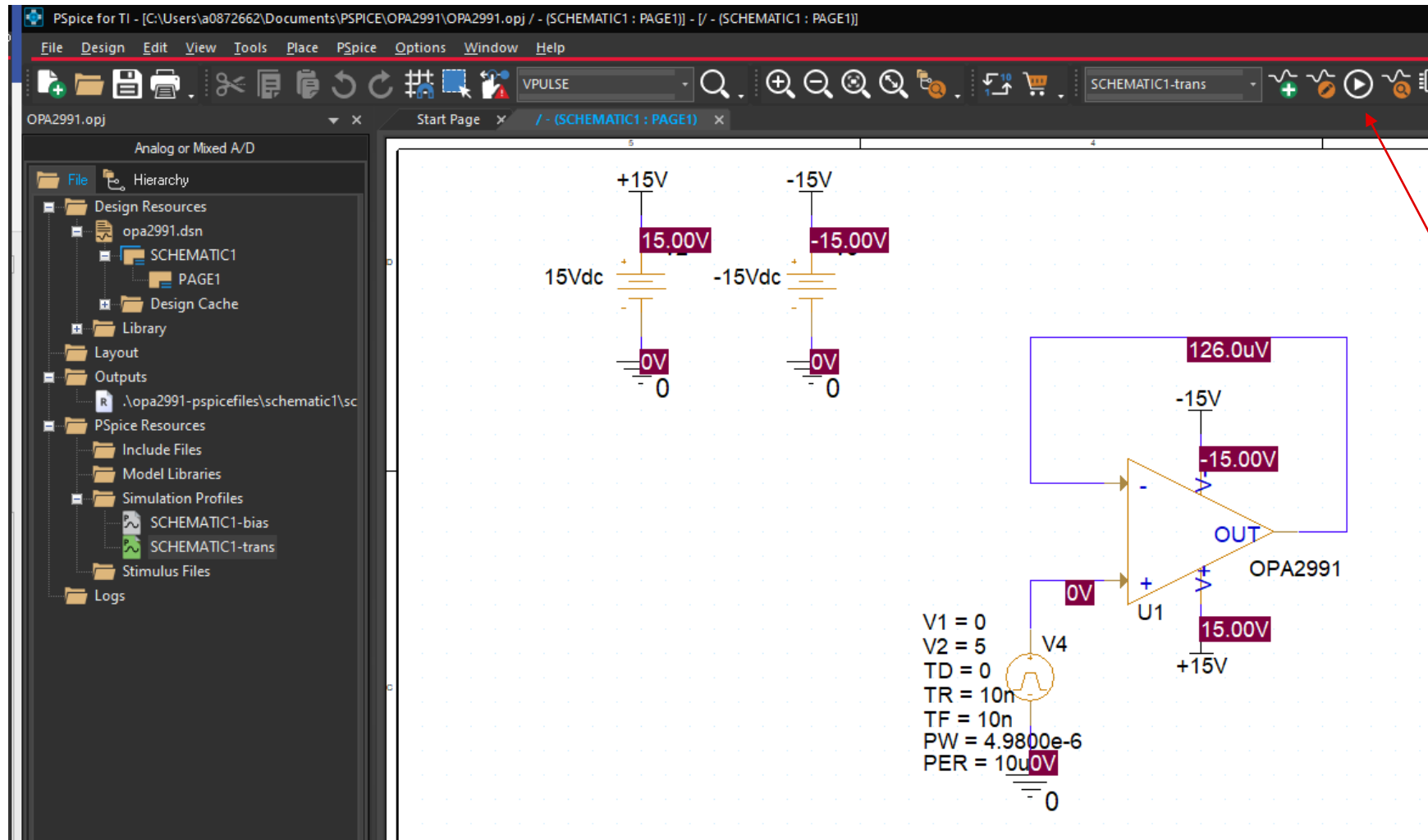
Use waveform with green plus sign to create a new simulation profile. Transient is use in this example.

Simulation profile can be accessed here later.

# Example transient simulation profile



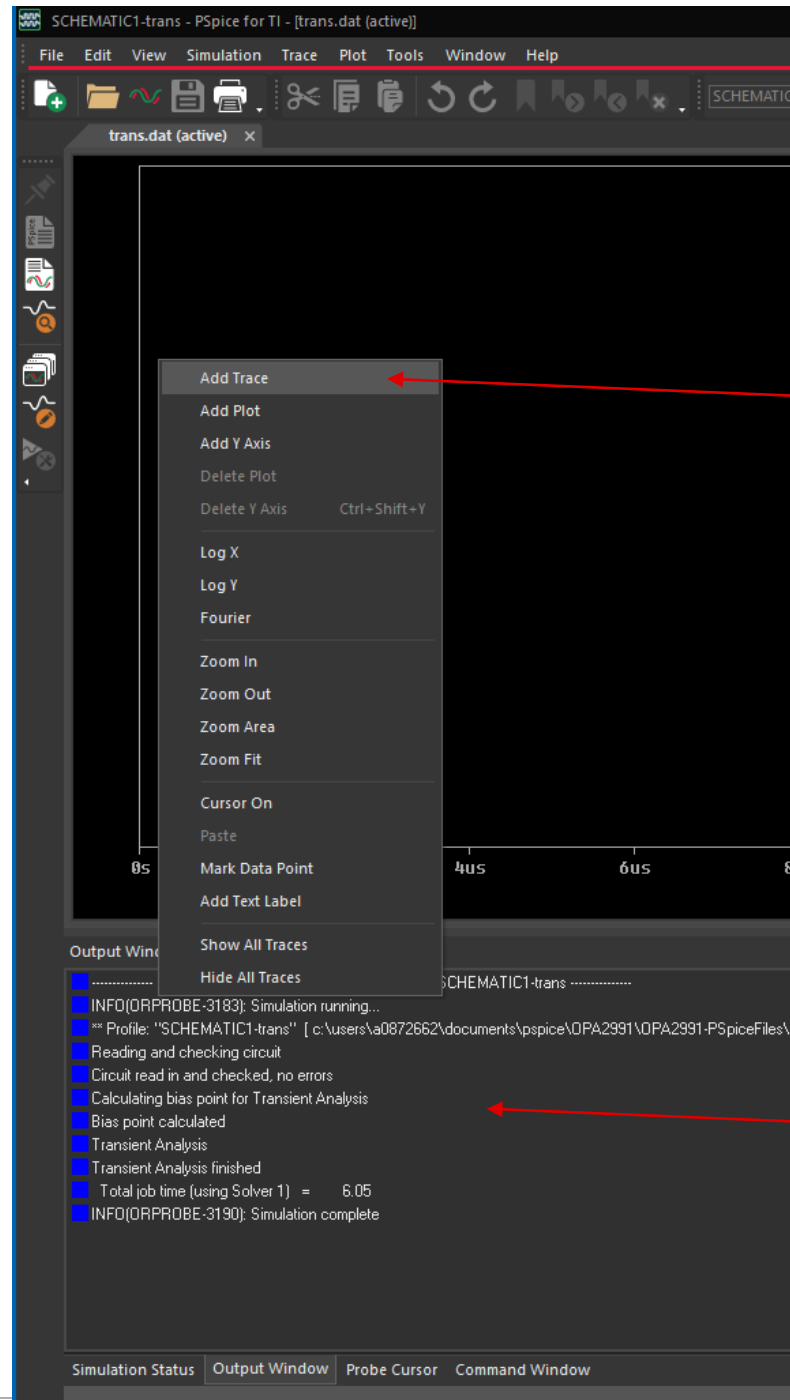
# Run simulation



Press arrow (play symbol) to run currently selected simulation profile



# Simulation results



3. Click on graph and select add trace

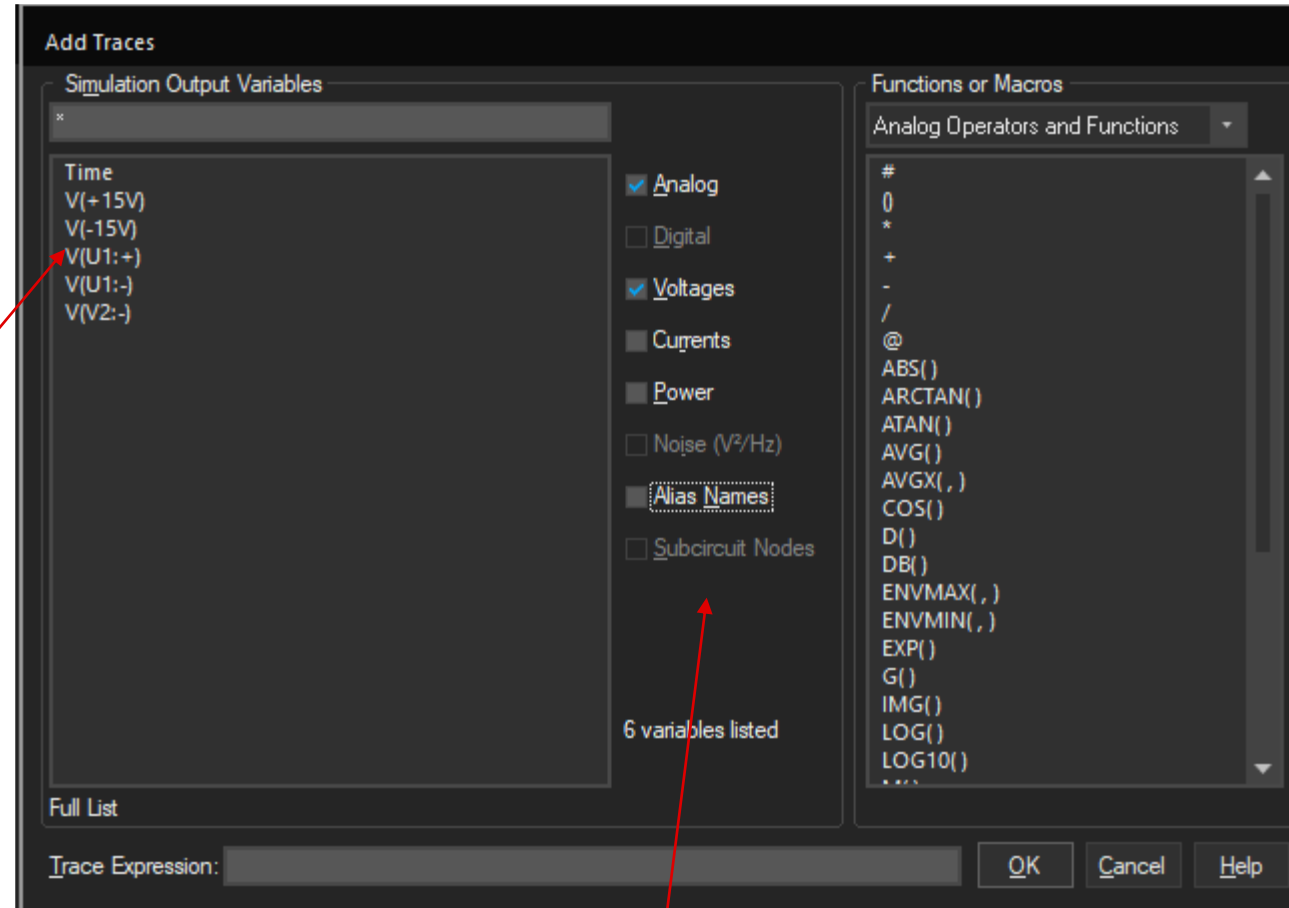
2. Progress of simulation will show here.



1. It takes some time for the first simulation result to show up. Watch for the separate simulation window to pop up.

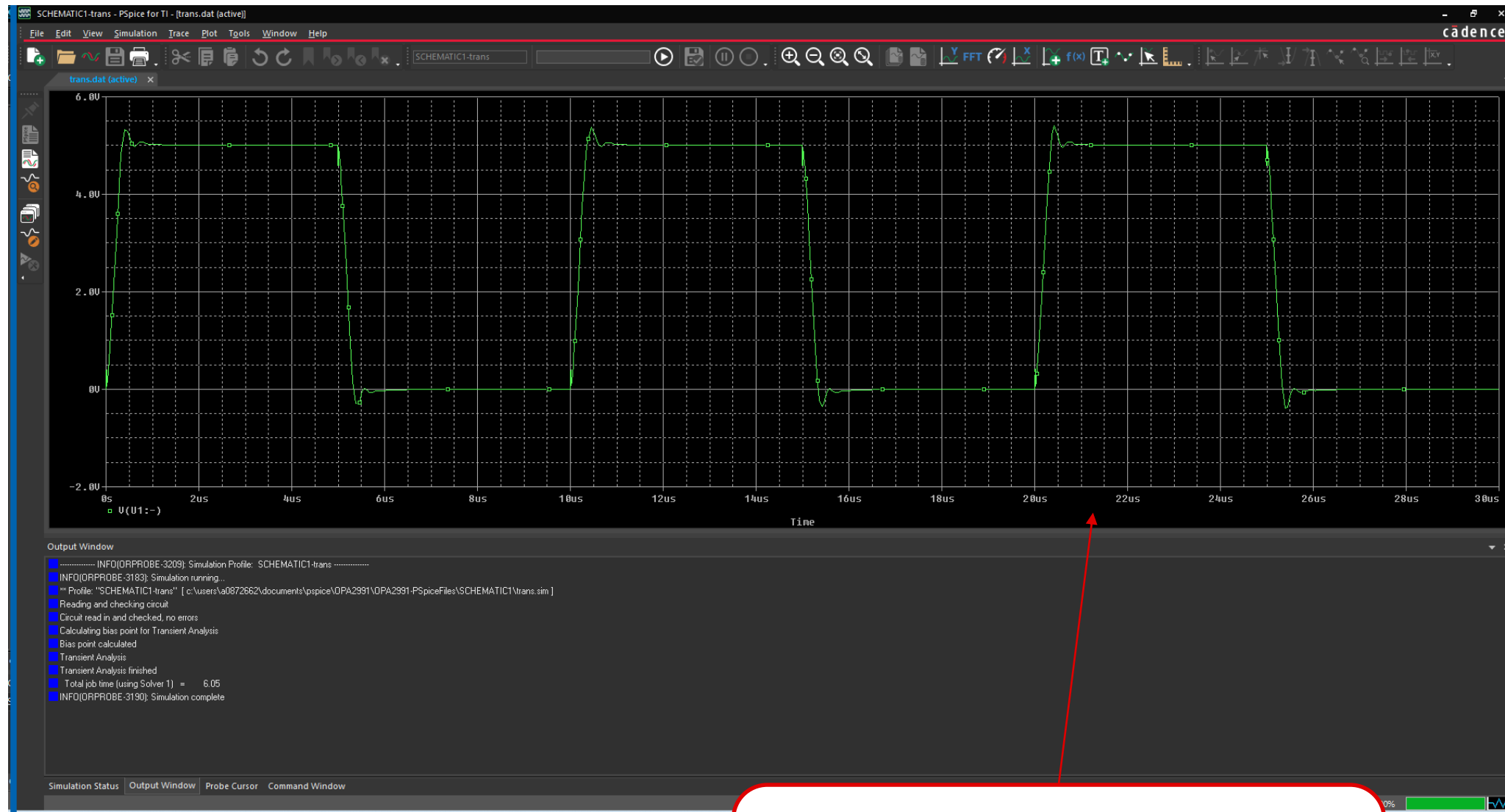
# Simulation results

2. For this example choose V(U1:-) The inverting node.



1. There are many nodes you can display. Narrow down the list by selecting analog voltages only.

# Simulation results



Here is the result.