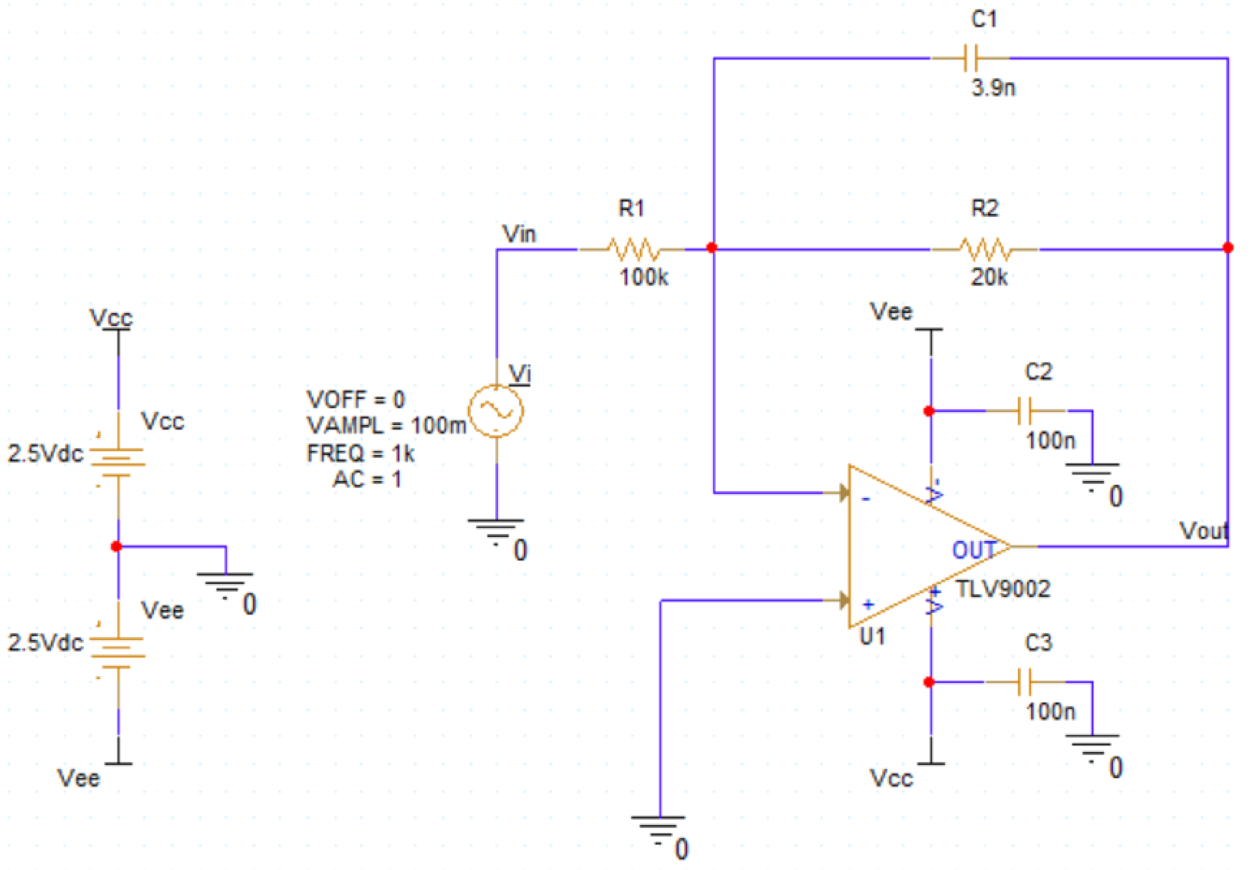
# Transient Analysis

## TI Precision Labs – PSpice® for TI

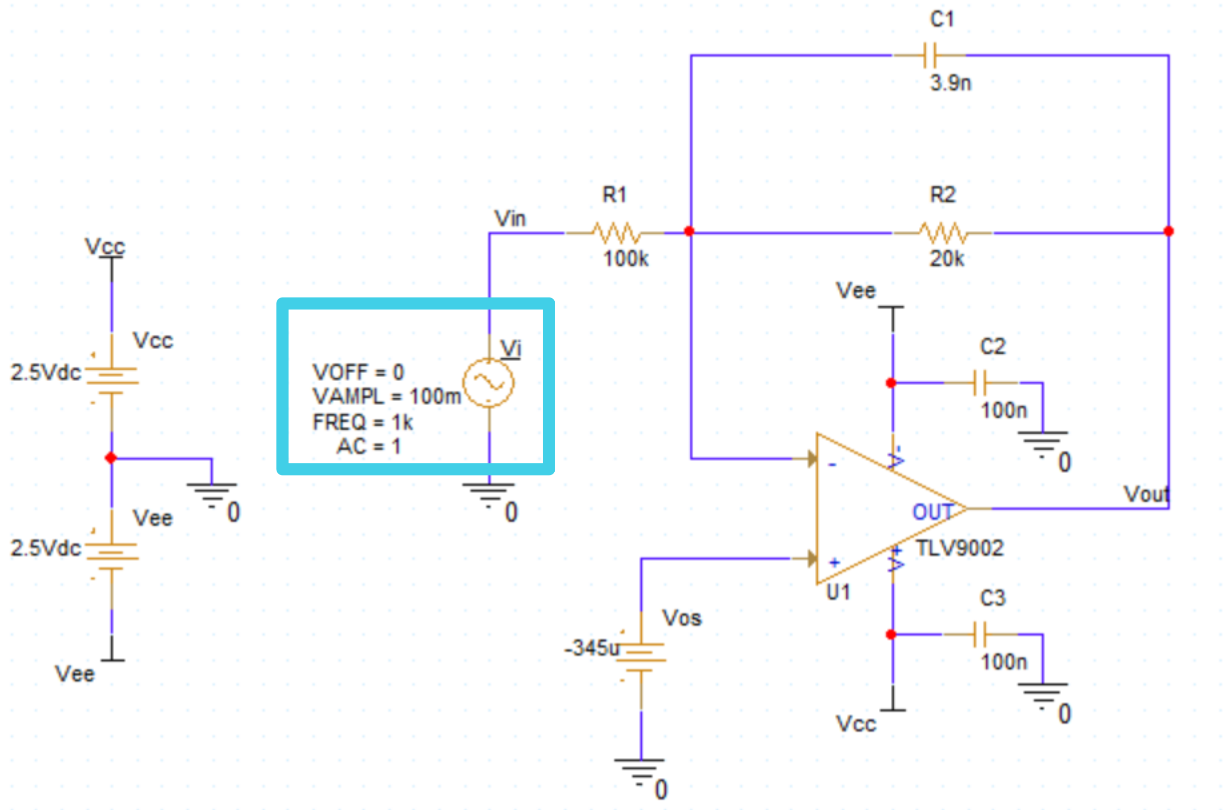
# Example Circuit

- In this example, we will be using the following Integrator circuit, which can be found in the Analog Engineer's Cookbook



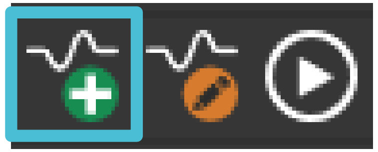
# Time Domain Stimulus

- Time domain stimulus must be present in order to run transient analysis, here are a few examples:
  - Sine wave
  - Pulse
  - DC
  - Exponential
  - Impulse

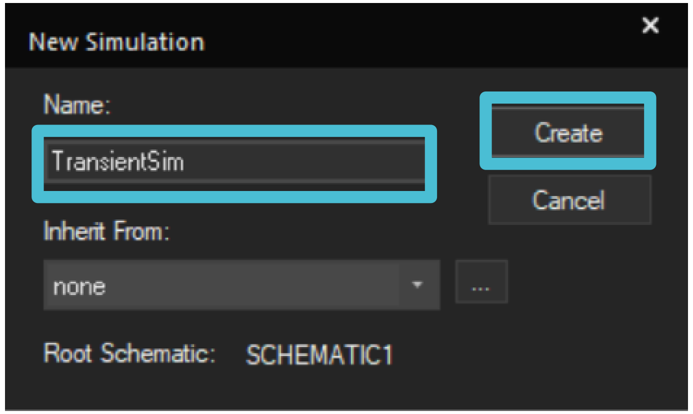


# Create a new simulation

- In your toolbar, select the new simulation button



- Name your simulation, then press “Create”

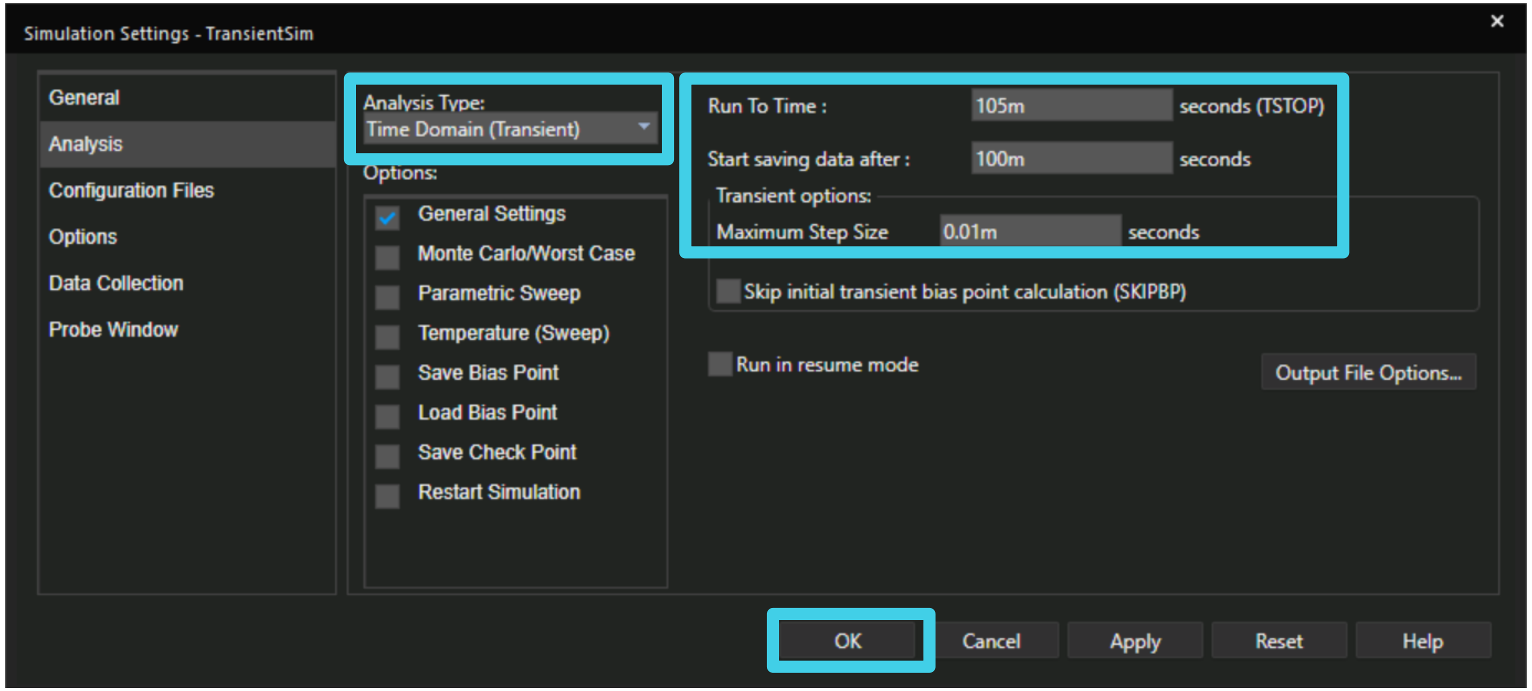


# Initializing your simulation

- Under Analysis Type
  select "Time Domain (Transient)"

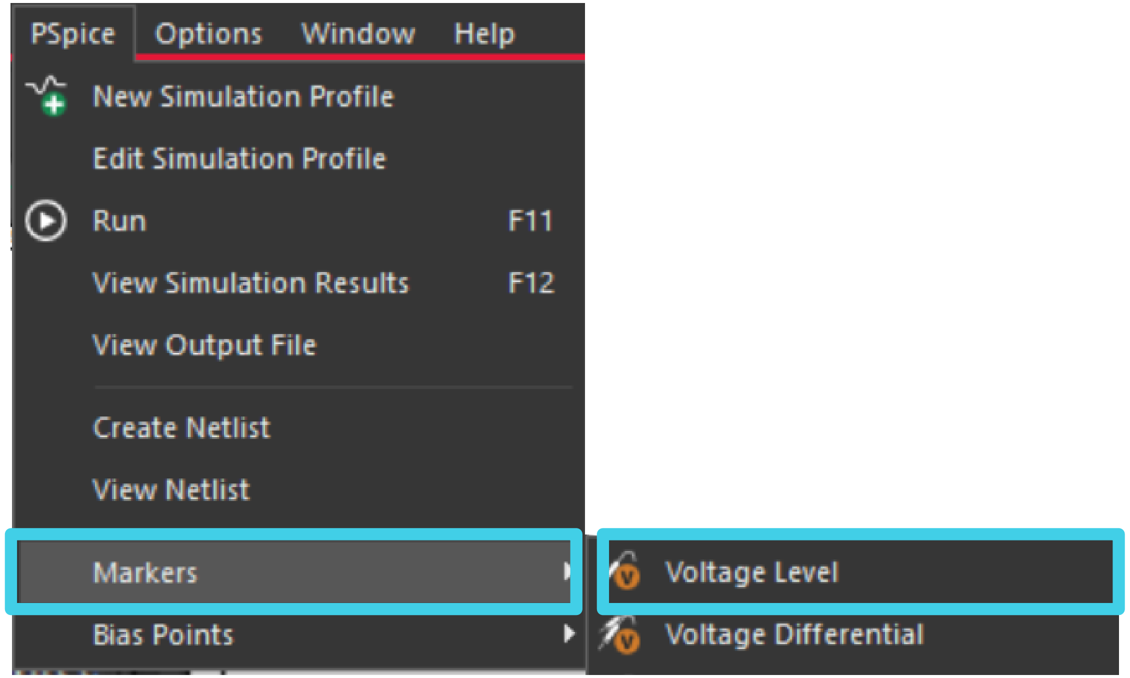
- Under "Run to Time" enter
  the Stop Value as 105ms,
  Start saving data after 100ms,
  and maximum step size 0.01ms

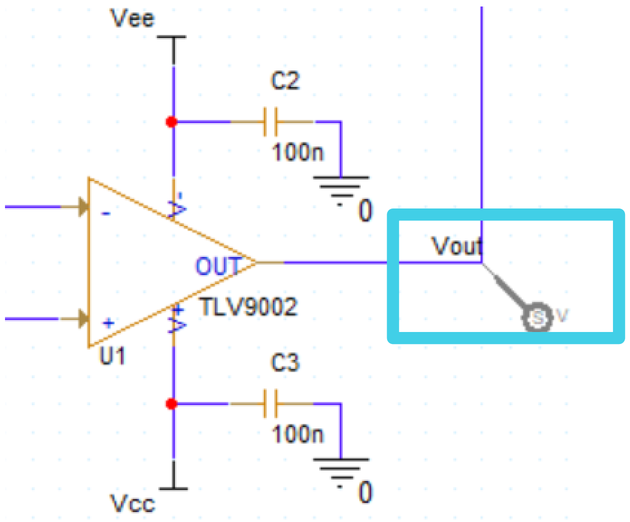
- Click OK



# Initializing your outputs

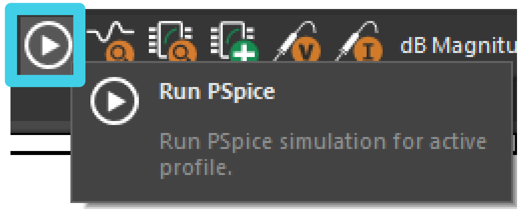
- To measure the output voltage in volts
  navigate to PSpice > Markers > Voltage Level or click voltage icon in taskbar
- Use left-click to place the marker on the wire

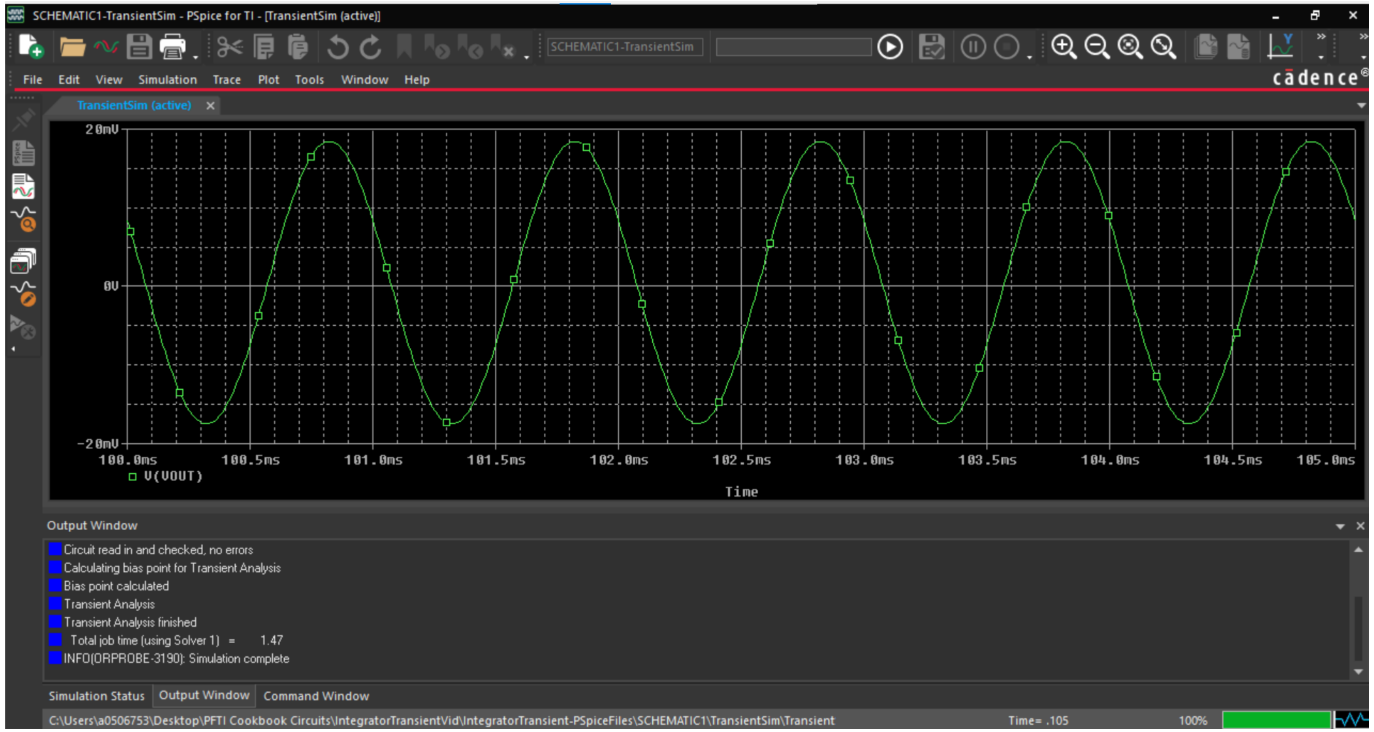



# Running your simulation

- Run your simulation by pressing the play button or by pressing F11
- A new window will open with your simulation results



# Thanks for your time!

To find more **PSpice® for TI** technical resources and search products, visit **ti.com/tool/PSPICE-FOR-TI**.