

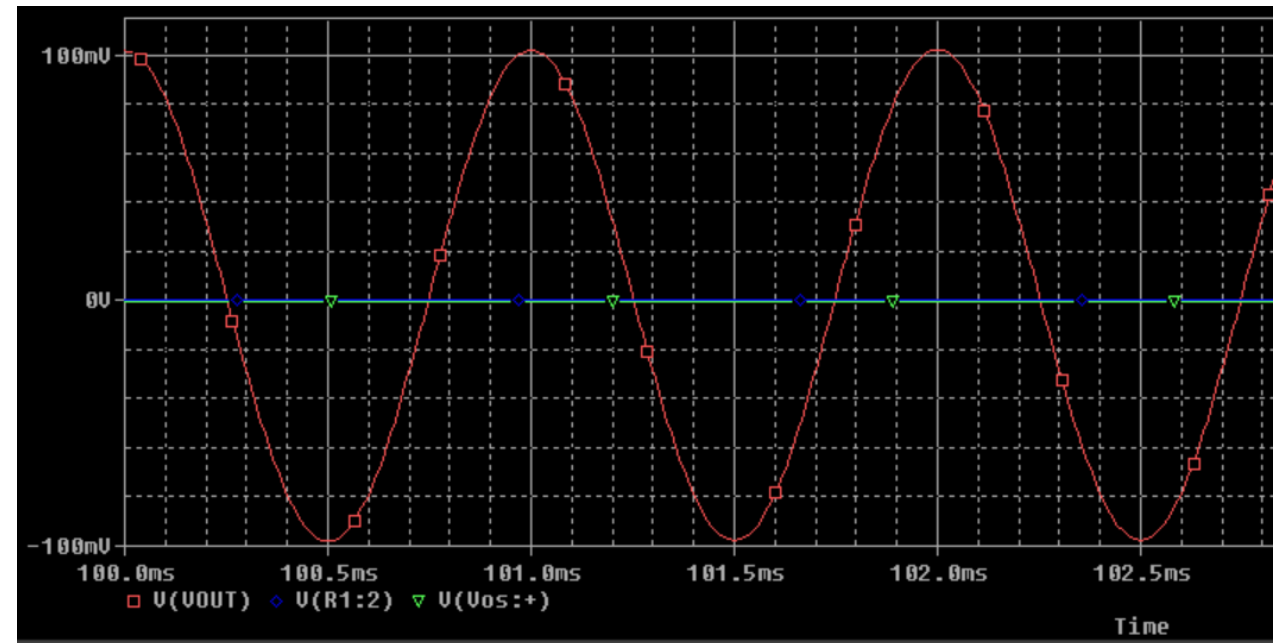
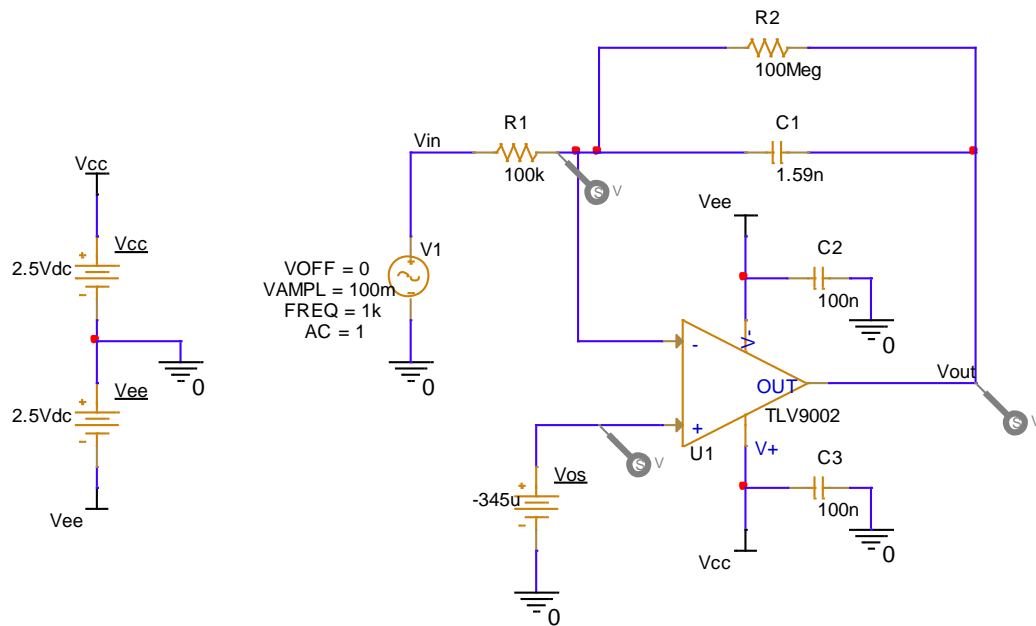
Plots and traces

TI Precision Labs – PSpice® for TI



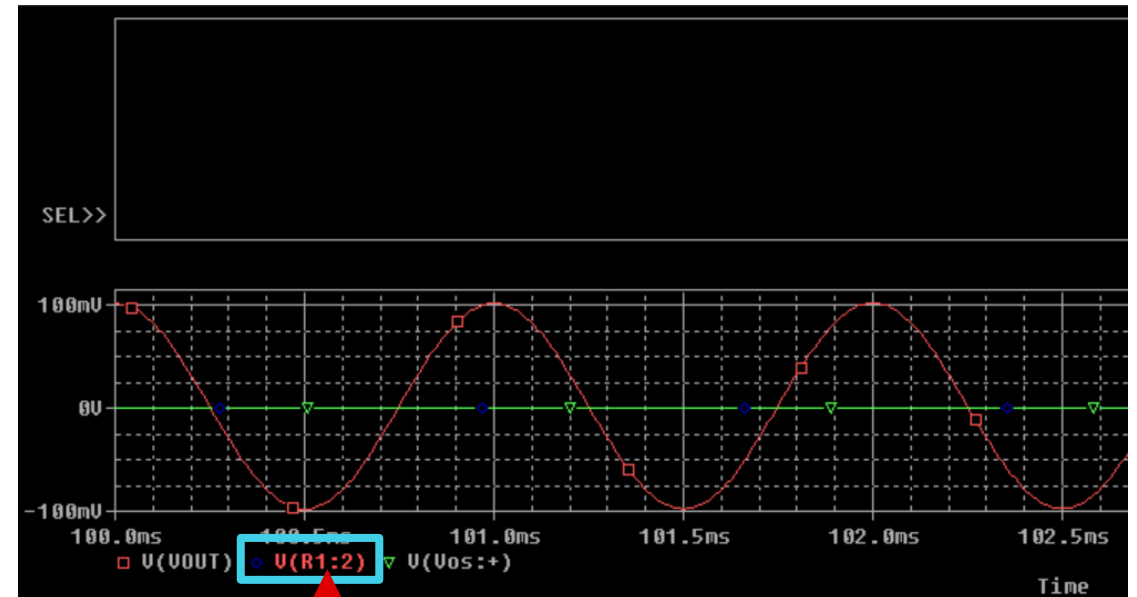
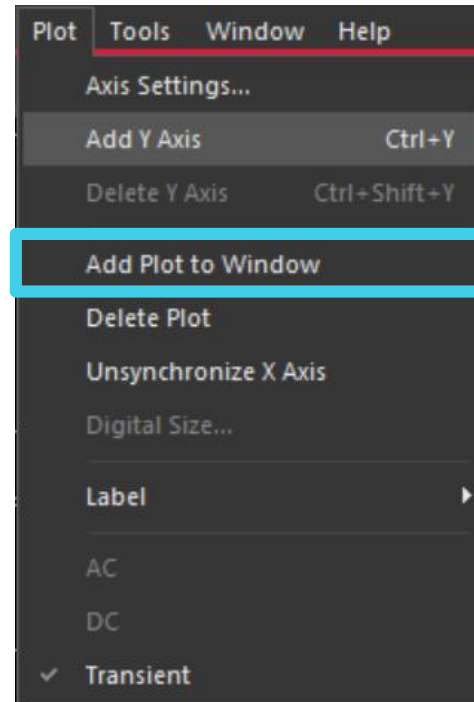
Example circuit

- In this example, we will be using the following Integrator circuit, which can be found in the Analog Engineer's Cookbook
- Run a transient simulation
 - Reference transient simulation video for step-by-step instructions



Separate plots

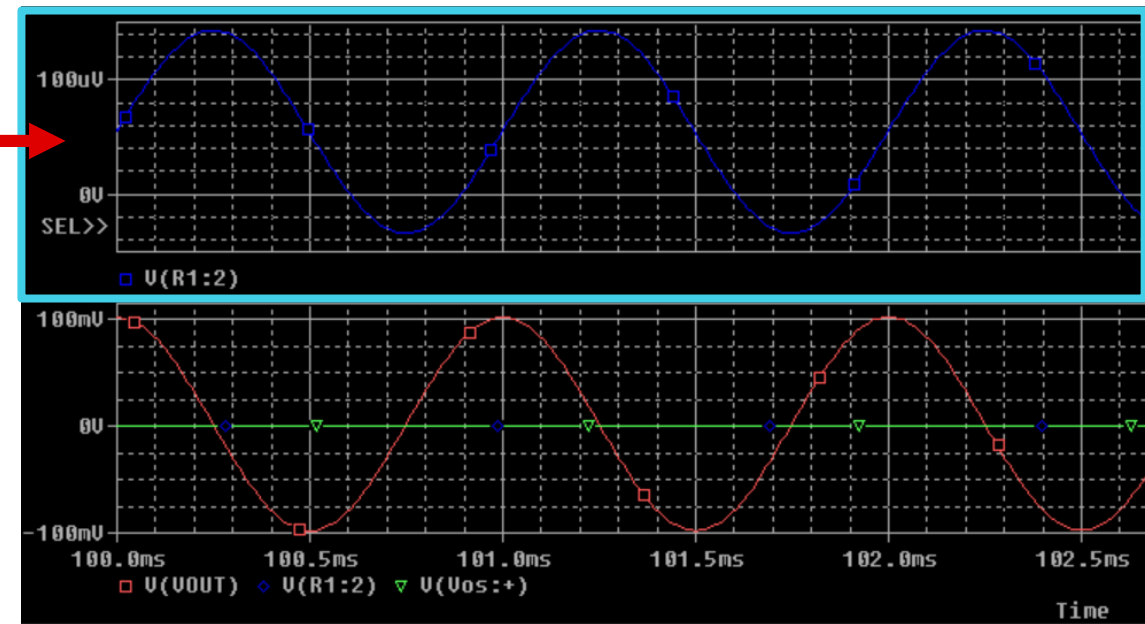
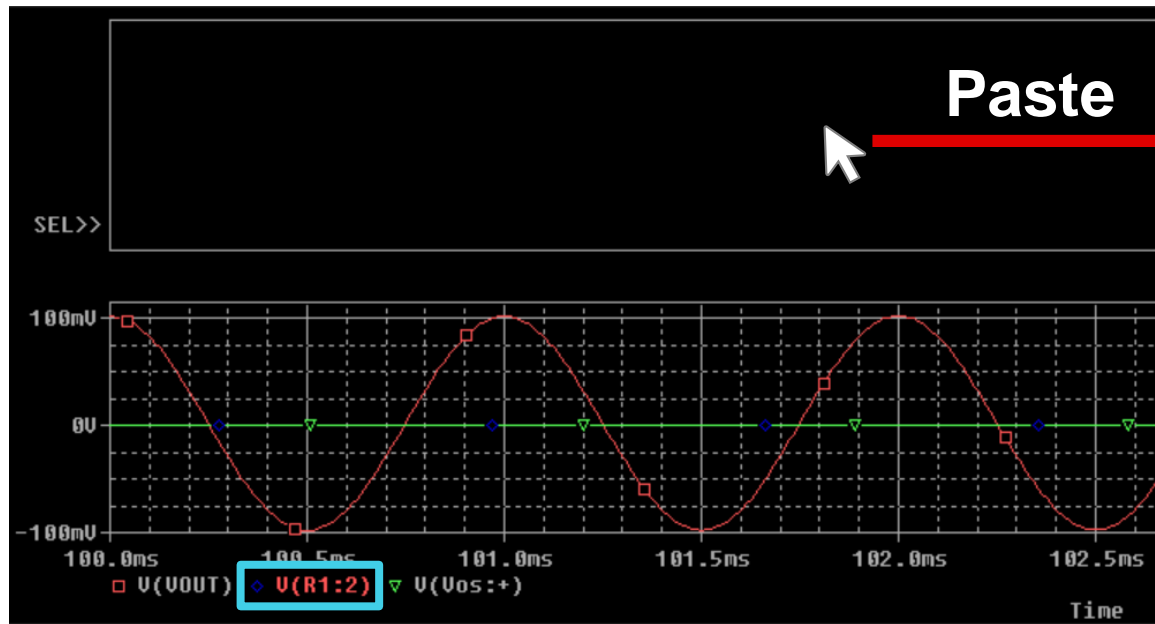
- Separate plots to gain better resolution of traces
- Select Plot > Add Plot to Window
- Left click on trace name you want to move in legend
 - Will be highlighted
- Copy trace (Ctrl C)



Copy trace

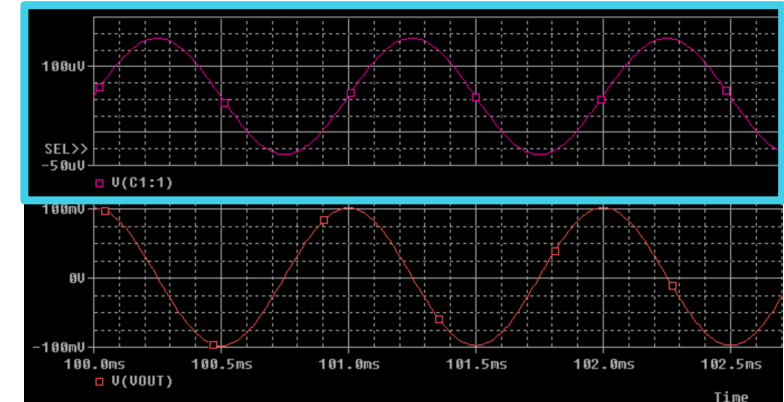
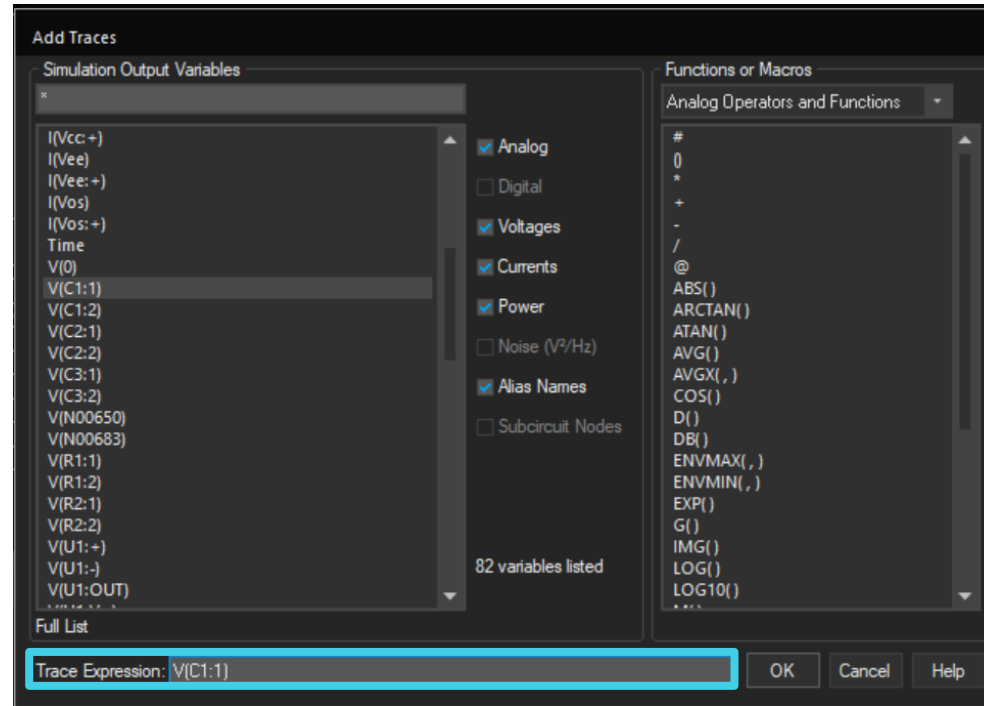
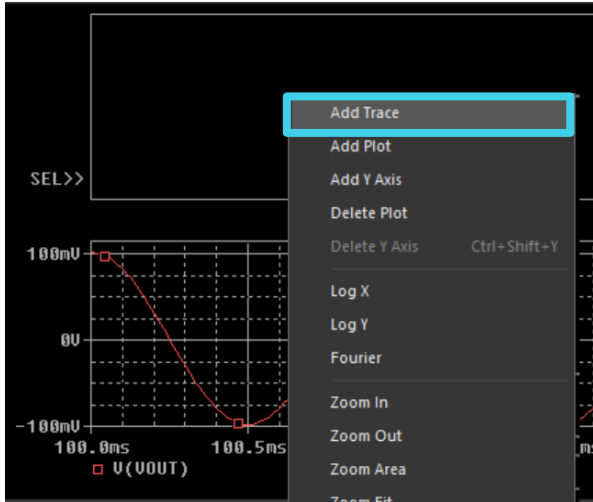
Paste trace into new plot

- Left click blank plot
- Paste trace into blank plot (Ctrl V)
- Alternatively, a trace can also be cut and pasted and would only appear on the new selected plot



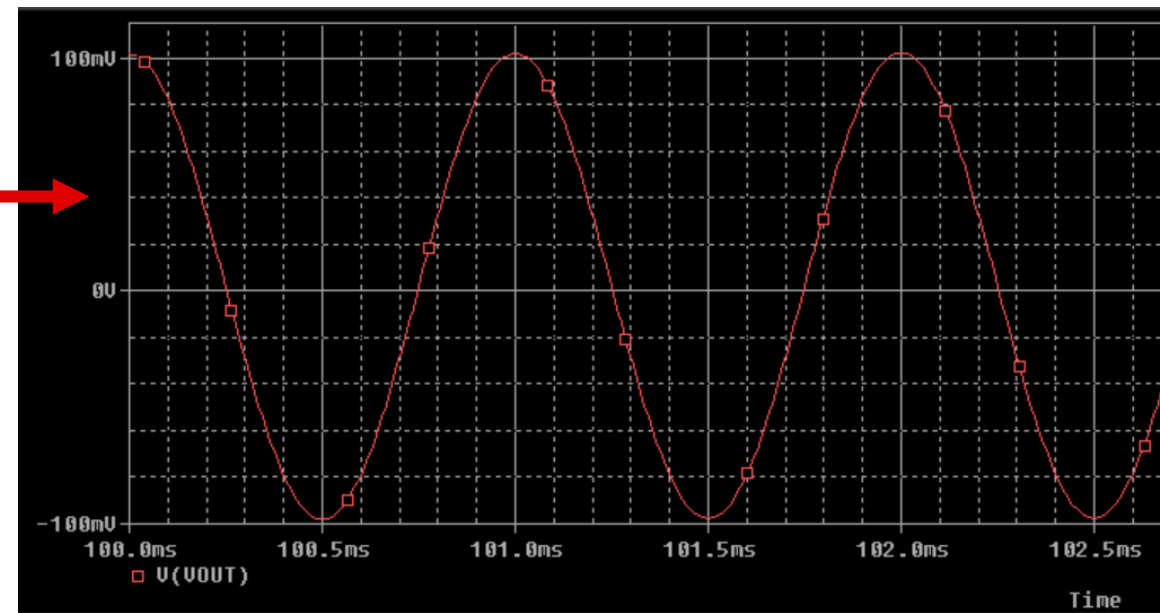
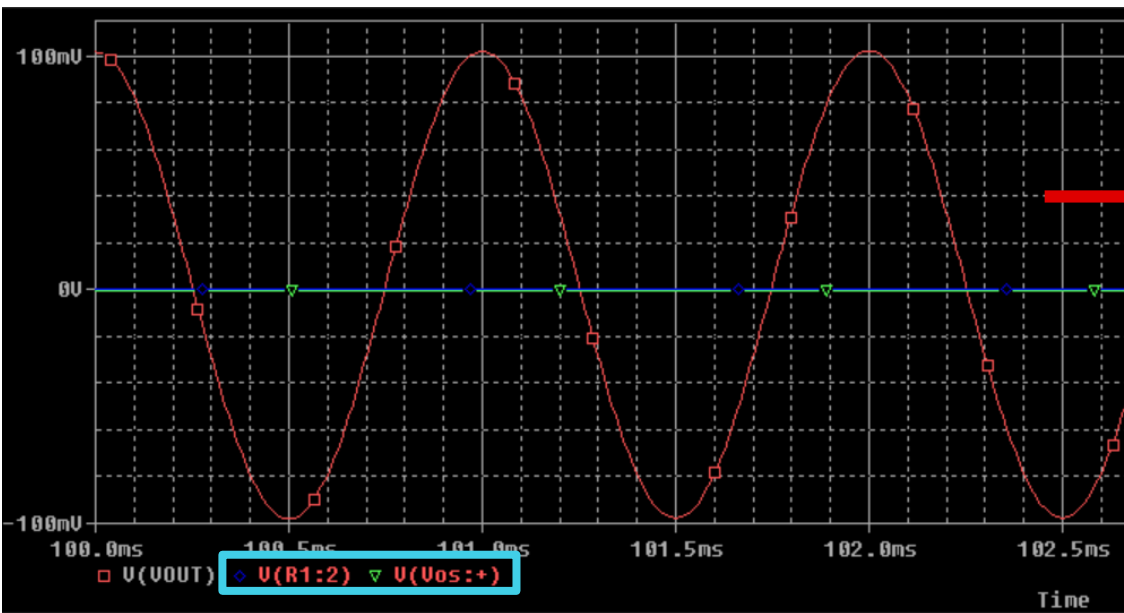
Add trace into new plot

- Right click blank plot > Select “Add Trace” > Select desired trace > OK



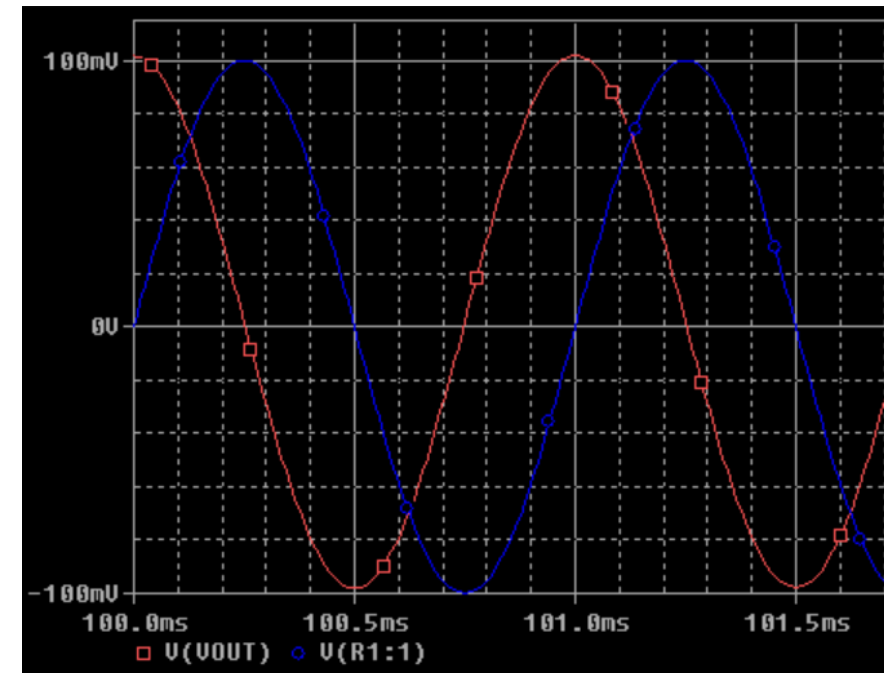
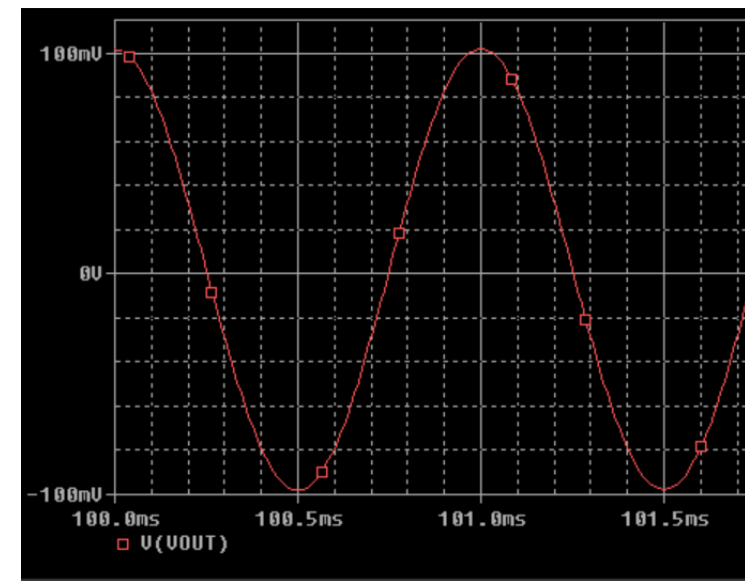
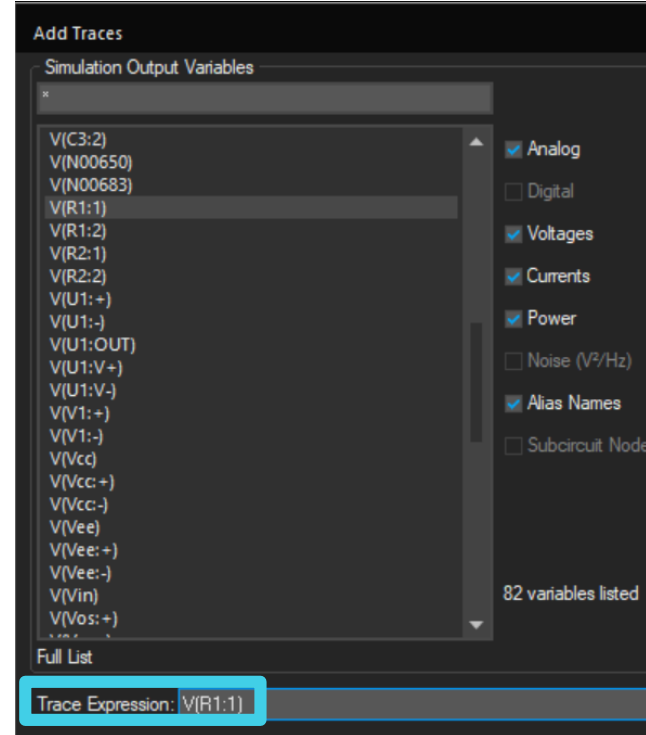
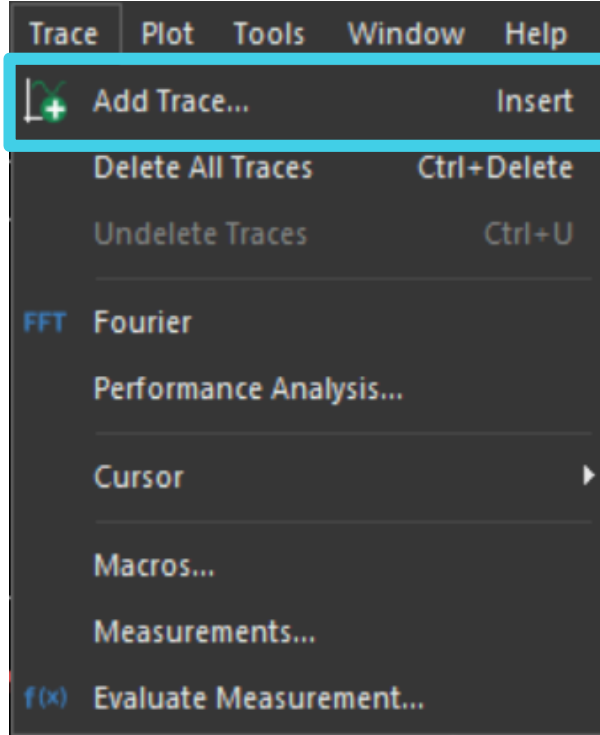
Delete traces

- Delete a trace by left clicking trace name in legend and pressing “Delete” key
 - To select multiple traces, hold “Ctrl”



Add multiple traces to plots

- Navigate to Trace > Add Trace
- Select desired trace then OK



Thanks for your time!

To find more **PSpice® for TI**
technical resources and search products,
visit **[ti.com/tool/PSPICE-FOR-TI](https://www.ti.com/tool/PSPICE-FOR-TI)**.