# NCV8901xx generic SPICE ac simulation model

This model is intended to be used to run ac simulations (only), to help design loop compensation.
It is valid for the following parts: NCV890100, NCV890130, NCV890101 and NCV89131.
It has been tested on various spice simulators (SystemVision, Simetrix, LTspice to name a few).
It already includes the ac stimulus source, as well as the inductor.
The inductor value and series resistance are parameters for the model: they must either be added as user parameters when creating the symbol, or directly fed to the simulator with a .PARAM command.

External components that must be added for a complete, closed-loop simulation are:
- Input voltage source
- Output capacitor with series resistance
- Output load
- Feedback divider
- Compensation components (series resistance-capacitance to ground)

Examples of test benches for the model:
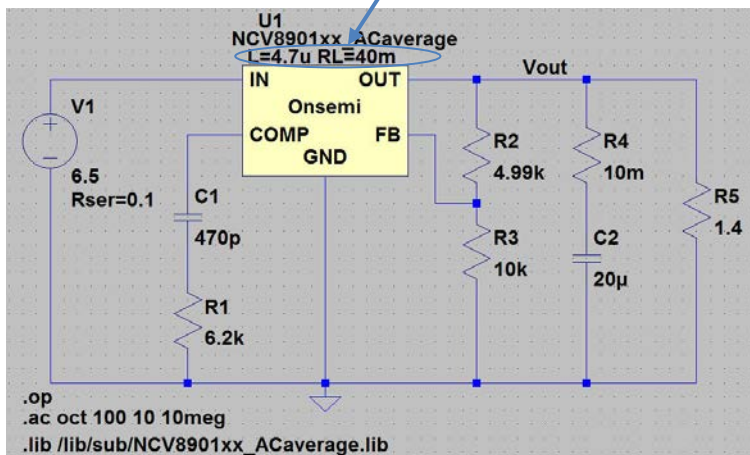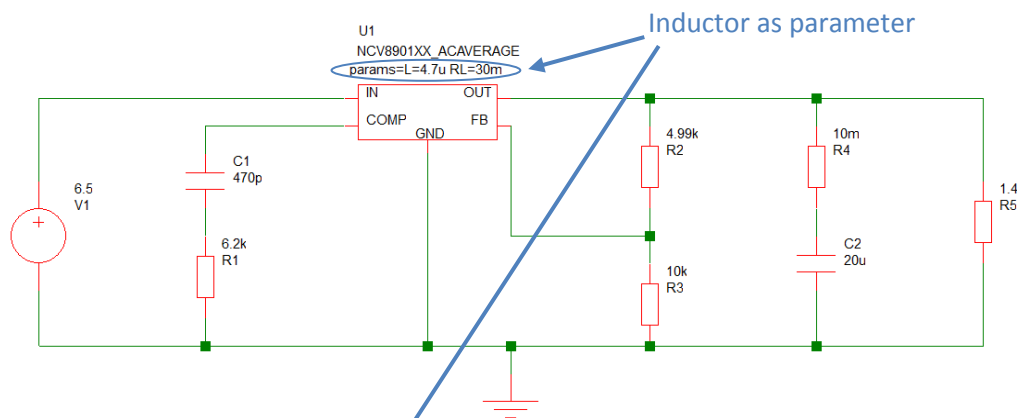


Always check that the bias operating point is correct (expected output voltage). If the output voltage is not correct, the Bode plot will be wrong!

To obtain a Bode plot of the closed-loop system, plot the COMP node: