There are some discrepancies between the outputs of my Simulink and Pspice models. The main reason for this difference is the capacitors. Each capacitor has a time domain to charge and discharge. The calculation says that the product of R and C should be 1 in order to achieve a perfect integrator integrating from x' to x. But when you change the C values everything changes. Simulink model is mathematically true. But pspice model is experimental. For example just in the beginning of the simulink model the y-axis value increase. In contrast in pspice model it first just a little bit decreases and then increases. The perfect fit for R and C values is 1k ohm resistor and 1m F capacitor. I obtained this values by trial and error. By the way 75 nodes limit was problem at first because I used 4 ideal opamps with VCVS's. But then I used a non-inverting integrator which returns a positive value not a negative value. This solved the problem.

Aydın Uzun   2015401210