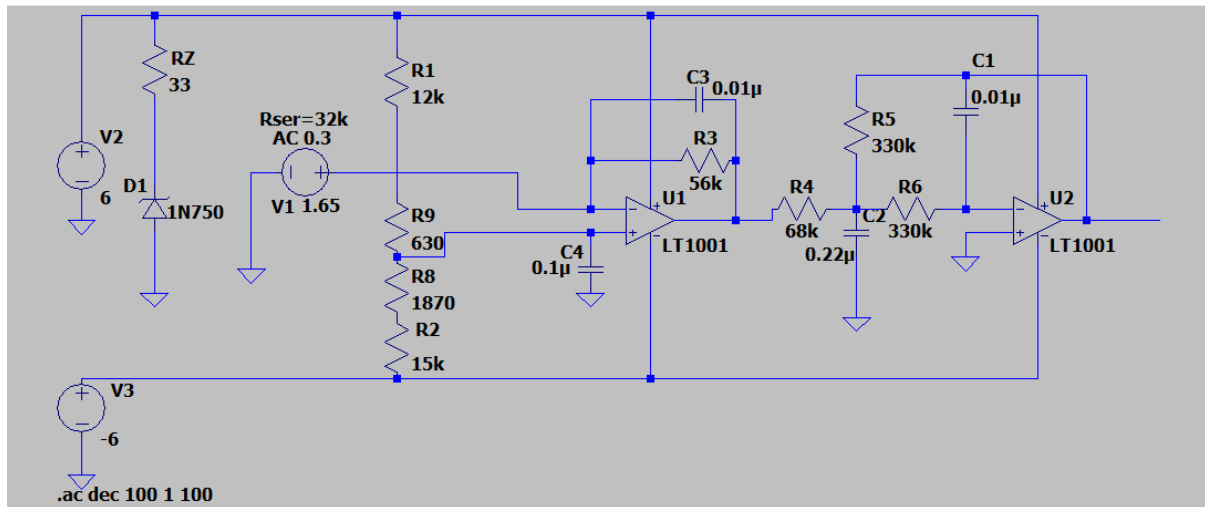


Simulation test circuit in LT-SPICE



The simulation of circuit was done in Ltspice (from Linear Technology) software package from the SPICE family with a graphical input of circuit diagrams and a graphical output presenting the simulation results before building the circuit hardware. In place of sensor a voltage source with a series resistance $R_s=32\text{ k}\Omega$ has been used which replicates the sensor.