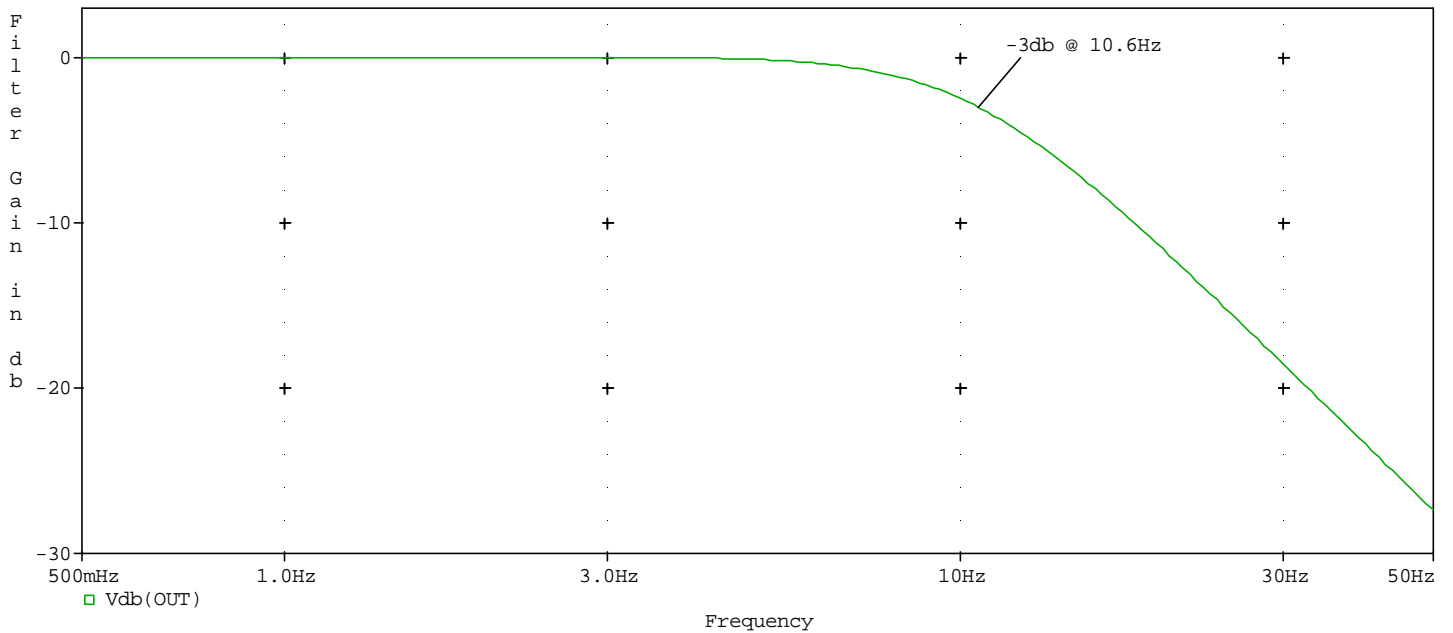
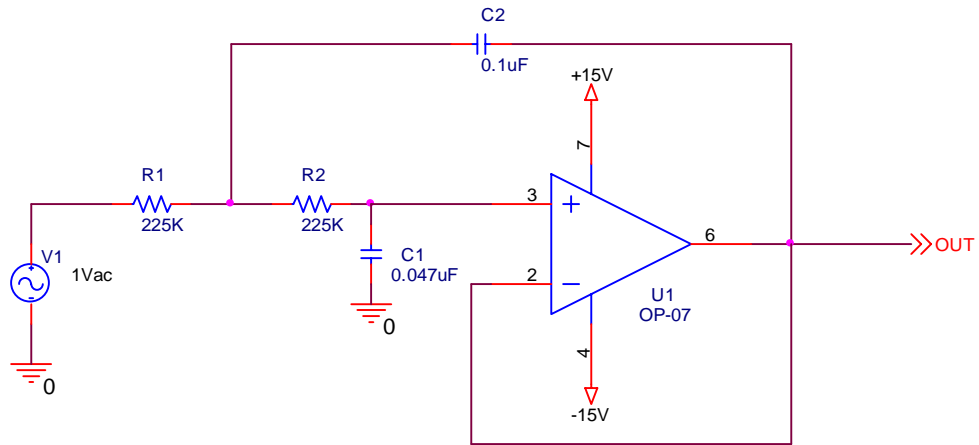


UNITY GAIN 10Hz LOW-PASS FILTER



Using SPICE as a Design Aid

Simulation is very effective in the initial design phase to try out different ideas and circuit configurations. If the simulation is done carefully, the breadboard has a good likelihood of working correctly without significant modifications.

When a circuit has been designed and tested in SPICE, then a breadboard can be built. When the simulation and the actual results correlate, then the circuit can be easily altered in SPICE to perform many different types of analysis. Testing your ideas using simulation can save many hours at the test bench.

The above example illustrates the design and simulation of a unity gain low pass filter. This circuit topology may easily be used for a wide range of low pass filter applications. In this circuit, we found the test results correlated closely with the spice analysis.

A simple inexpensive low pass filter added to an analog front end can increase the performance of the overall circuit. It is highly desirable to select a low as possible cutoff frequency for transducer interface filtering.

Low pass filtering should be used as close to the front end as possible in a system. A filter of this type is generally used to significantly reduce both noise and high frequency errors.

By tailoring the values of capacitance and resistance, a variety of response characteristics can be achieved. In order to achieve a maximally flat (Butterworth response) filter choose $R1 = R2$ and $C2 = 2C1$.

It is much easier to try and optimize the circuit in SPICE than it is to repeatedly modify the breadboard. Quick substitutions of the op amp and components can be made in SPICE and the results immediately viewed.

SPICE Simulation file:

```
UNITY GAIN 10Hz LOW-PASS FILTER
.AC DEC 100 .5Hz 50Hz
.TF V(4) V1
V+ 5 0 DC 15V
V- 6 0 DC -15V
C1 3 0 0.047UF
C2 2 4 0.1UF
R1 1 2 225K
R2 2 3 225K
V1 1 0 AC 1
U1 3 4 5 6 4 OP-07
.OP
.LIB
.PROBE
.END
```