

README file for LTSpice_opt program

LTSpice_opt is a Matlab program that uses an iterative optimization approach to design analog filters. It is designed to be used in conjunction with the popular circuit simulator LTSpice. It embeds an LTSpice simulation inside the powerful Matlab nonlinear least-squares optimizer (lsqnonlin). It works as follows;

- The user provides a target frequency and/or phase response in Matlab, and a circuit topology in LTSpice with some initial component values.
- The user provides a list of which circuit instances the optimizer is allowed to vary.
- The optimizer then iteratively adjusts those component values, running a simulation for every pass through the lsqnonlin algorithm, in an attempt to reduce the error between the target frequency response and the simulated response.
- Once the optimizer has finished, a new schematic is generated with the optimized component values. During the schematic generation process, each component value is quantized to a user-defined tolerance.

Why is this capability useful? Don't we already know how to design filters?

Traditional filter design uses standard circuit topologies such as Sallen-and-Key or multiple-feedback op-amp based active filters. In these cases, given a "standard" filter shape such as a Butterworth or Chebychev, a high-order filter may be factored into 2nd-order sections, and an op-amp circuit can be used for each of those sections. This design procedure is quite straightforward and has not changed for many years. However, there are many cases where this approach fails;

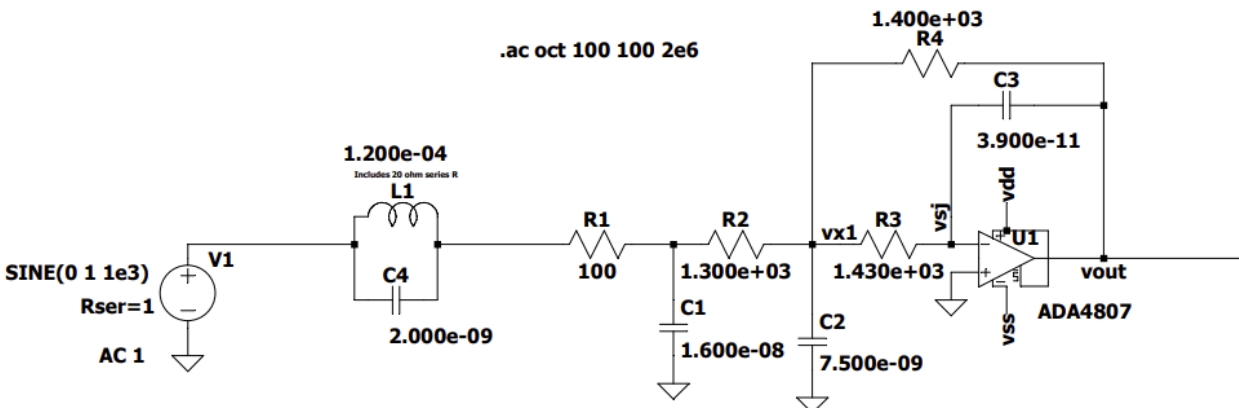
- **The desired filter shape is not a traditional shape** such as Butterworth or Chebychev.
 - For example, the filter may need to compensate for some other part of the system that has a non-flat frequency response, while simultaneously attenuating other frequency regions.
- **The need to reduce power/area by combining multiple filter sections into a single op-amp circuit.**
 - This leads to non-conventional circuit topologies that have very messy closed-loop formulas, and it becomes very difficult to solve for the component values.

- The application operates at frequencies where finite op-amp gain-bandwidth degrades the frequency response.
 - Calculating the effects of finite gain-bandwidth on the frequency response is quite complicated, especially in cases where the gain/phase response deviates from the traditional single-pole model. By running optimizer simulations in LTSpice, the actual target op-amp may be included in the simulation. This yields a solution that inherently attempts to compensate for finite gain-bandwidth effects.

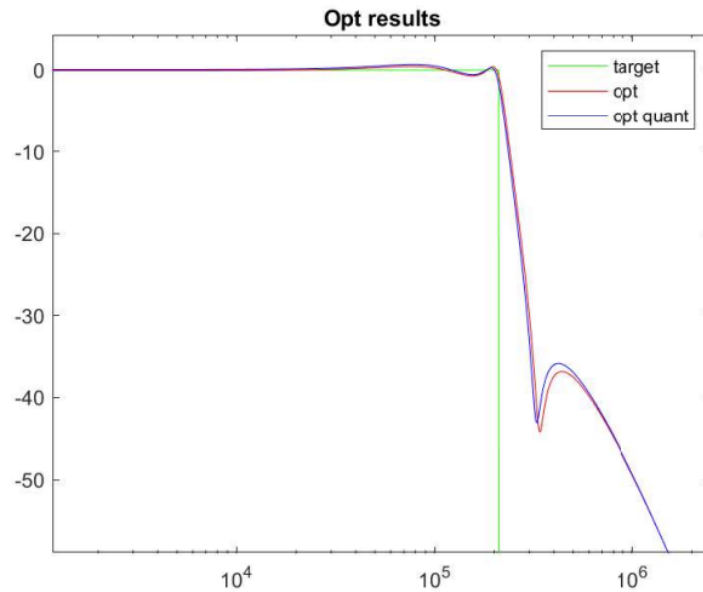
Example circuit and results

The optimized circuit of “example2.asc” (included in this distribution) is shown below. Note that while one could obtain the transfer function of this circuit by manual or automatic means, it is not in a form that would match any conventional lowpass filter shape. By using the passband/stopband/don’t-care band approach, we can still obtain a lowpass shape that is quite sharp for a single op-amp filter.

Schematic (post-optimization)



Simulation results



Matlab files included in the gitHub repo;

LTSpice_opt.m

- the main application

simControl.m

- edited by the user, includes;

path to the LTSpice executable

path to the LTSpice circuit simulation directory

file name of the LTSpice circuit that will be optimized

LTSpice variable that produces the output that will be matched to the target

(example, 'V(vout)')

list of instance names in the schematic that the optimizer will adjust

min values of the instances above

max values of the instances above

tolerance of the instances above

Match Mode; match amplitude only, phase only, or both
LTSpice simulation time; how long should Matlab wait for the LTSpice simulation to finish?

setTarget.m

- user-defined target frequency response and error weighting entry

update_schematic.m

- writes new schematic with optimized component values

optLTspice.m

- optimizer error-evaluation function

LTSpice2Matlab

- reads LTSpice output file (.raw format). From Matlab File Exchange site

round63.m

- quantizes component values to selected tolerance level. From Matlab File Exchange site

LTspice Files files included in gitHub repo

Example1.asc - a 3rd-order single-op-amp inverting lowpass filter.

Example2.asc - a complicated single-op-amp filter including an LC stopband notch

Installation of the files

Place all the “.m” files in a directory where you will run Matlab.

Place all the .asc files in a directory where you will run LTSpice.

The full-path location of these directories needs to be entered in “simControl.m” (see step “3” below).

Running the examples

The main application is started by running "LTspice_opt.m". At the top of the file you will see a variable called "example", which can be set to either 1 or 2 to run one the examples (corresponding to LTspice schematics :example1.asc" or "example2.asc"). This variable is passed to simControl.m and setTarget.m to automatically set all the required parameters for the example circuit. When running your own circuit, the example variable may be set to 0, or you can comment out the corresponding code in simControl.m and setTarget.m (and add your own parameters).

How to run your circuit;

1) Enter a schematic in LTSpice. Set the simulation command to do a frequency response sweep

example, '.ac oct 100 100 2e6'.

Run the simulation from within LTSpice to make sure there are no errors, and plot the frequency response of the output node. Note that the frequencies used in the LTSpice simulation are passed into the "setTarget.m" module as the variable "freqx". The target response must be computed at these same frequencies.

2) Identify the component instance names that you want the optimizer to adjust. Note that making ALL the components adjustable is usually a bad strategy. For example, for most filters it is possible to scale the R's by some factor k , and also scale the C's by $1/k$, without changing the frequency response. If all the R's and C's are adjustable, then the optimizer may drift off into an undesirable range of values, so it makes sense to pick one component that is not in the list of adjustable values, or alternatively restrict the range of one component to values that are acceptable.

3) Fill in the simControl.m file. This file is heavily commented and it should be fairly clear how to enter your design.

The first step is to fill in the paths to your Matlab working directory (where you placed the .m files) as well as the LTspice working directory (where you placed the .asc files).

Next, you need to fill in the information about which schematic instances will be adjusted by the optimizer. For each component listed that will be

optimized, you must enter a min and max value. In most cases, the spread of min and max values should be fairly wide, to avoid limiting the optimizer; however, in some cases it makes sense for a particular component to have a more narrow allowable spread. For example, the min value of an input resistor may need to be large enough so it can be easily driven, but the maximum value may be limited by noise or bias current considerations.

In addition to the instance names, min values, and max values, the user should fill in the tolerance of the newly-adjusted components. This is specified in standard "E" format (number of values per decade), as defined in the comment section of the file. Note that the tolerance quantization is done "outside" the least-squares loop, as part of the schematic-generation process.

Note that the above information is entered into Matlab "Cell arrays". The examples in the file should make it clear how to enter this information.

The "LTSpice_simtime" parameter is the amount of time that Matlab will wait for LTSpice to finish a sim. Typical values are 0.5 to 1.5 seconds, but this depends on the speed of your computer and the complexity of the circuit. If the wait time is too short, the program may crash. However, if it's too long, the optimizer will operate slowly. The Matlab code will retry the simulation if it fails, and the number of restarts is reported in the command window during the optimization phase.. If there are more than a few restarts, it's wise to increase the LTSpice_simtime parameter.

The "matchMode" parameter may be set to amplitude-only (1), phase-only (2), or both amplitude and phase (3).

4) Edit the setTarget.m file. This file contains a frequency vector called "freqx" that will be imported from the main program, and will contain all the frequencies that are used in the LTSpice sim. You must generate a vector called "target" with the dimensions as freqx. "Target" should contain the desired magnitude response (note, NOT in dB) of your filter at the corresponding frequencies in freqx.

Also in setTarget.m is a vector called "errWeights". This vector controls how much to weight the error at the corresponding frequencies in the freqx vector. It is initialized to all 1's, which may be adequate for many designs. However, when the filter design contains some combination of passbands, stopbands, and transition bands, it may be useful to set the weighting to 0 in the transition bands, and to some larger number in the stopbands. A rule of thumb is that the ratio of linear passband ripple to linear stopband ripple should equal the ratio of the stopband weights to the passband weights.

Note that lsqnonlin is a least-squares optimizer, and therefore cannot be expected to return an equiripple design. However, the user can experiment with running the algorithm multiple times, and for each new run, adjusting weighting factors in regions where the ripple is too high. This can eventually result in a nearly equiripple design.

5) Run the main program LTSpice_opt. Before running the main program it is advisable to shut down any open instances of LTSpice, as well as any other applications that may load the CPU.

If everything is set correctly, you will soon see a series of updates in the Matlab command window, which tracks the algorithm progress. For each iteration of lsqnonlin, the component values are displayed, along with the rms error in the frequency response versus the target. Depending on many factors, the optimizer may run for only a few minutes, or for much longer if the design is complicated.. Go have a cup of coffee!

In addition to the updated component values, the window displays the cumulative number of simulations as well as the number of simulation restarts. A restart can occur when something causes the simulation to take longer than normal. This could be due to a number of factors involving the operating system as well as any cloud-based automatic backup systems, which may temporarily lock up a file. If the ratio of the number of restarts to the total number of cumulative simulations exceeds more than a few percent, it is recommended to increase the LTSpice_simtime variable in the file simControl.m.

When the algorithm has finished, a plot is displayed of the target response, the optimized response with arbitrary-precision components, and the optimized response with components that were quantized according to user input as described earlier. A new schematic is also generated with the quantized optimized component values, with a schematic name the same as the original but with "opt" appended to the name. It is advisable to open this schematic and run the simulation from LTSpice to make sure that everything has worked as expected.

Acknowledgements

This development was inspired by a program written more than 40 years ago by **Mark Davis**, who was my co-worker at the time at dbx Inc. Mark is an MIT PHd graduate, and has made extensive contributions to the field of audio during the

course of his career. His original program ran in a DOS command window and was 100% text-based. Mark used it to design crossover filters for loudspeakers (where the target response came from a speaker measurement program), while others used it to design non-standard op-amp circuits. I hope this program proves to be as useful as the original!