Testing op amp tools for their active filter design accuracy and dynamic range

Michael Steffes - December 08, 2017

The major op amp suppliers continue to improve and update their online design tools. Here, three of the major tools will be applied to the same relatively simple 2nd order multiple feedback (MFB) example design. The tools will be navigated as best they can to the same response targets and the RC values scaled to the same range. While a lot of focus goes into sensitivities and response spreads due to component tolerances, every design first moves off the nominal target at the standard value selection step. Those tools that support actual op amp implementations, will also depart from nominal due to the typical gain bandwidth product (GBP or GBW) for the op amp selected in implementation. This review will assess three final design accuracy metrics in two parts. This first part will set up the response targets and navigate to the solutions, while the second part will compare these metrics:

1. With standard RC values forced onto the design and a minimum GBP op amp selected, how nearly does the typical value response match the desired target response shape?
2. The MFB design flows can reduce the noise gain peaking. Some tools do this better than others and an SNR analysis for each output will illustrate those differences.
3. Since the tools deliver slightly different noise gain profiles over frequency, the minimum loop gain in the passband will vary. All other things being equal, higher loop gain will give lower distortion. A comparison of noise gains over frequency subtracted from the op amp Aol will give the different loop gains for the designs where the minimum values will be compared.

Author background and disclaimers

The author has participated in the development and introduction of over 80 high speed amplifier products spanning five suppliers and 32 years. Having delivered literally hundreds of active filter designs to designers worldwide, and leading the online op amp tools development for Intersil, the internal nuances of common op amp based designs has been a perennial area of interest and incremental improvement.

The design tools considered here are regularly changed and improved. This two-part investigation will be making a best good faith effort to exercise the public tools available in Nov. 2017. Not only is the intent to show some fine scale differences in the results, but to show methodologies to assess the merits of a particular design. Enroute, some tool navigation notes will be necessary. The author has made every effort to understand and exercise the tools in the best way possible. The aim is to arrive at a set of RC values from each of the tools to get very similar final results. Then to apply those RC values to a set of simulations using the same op amp model where the only differences in performance can be attributed to the RC values delivered.
Target active filter response

A single 2\textsuperscript{nd} order low pass filter stage will be selected for design. The MFB (or Rauch) topology will be selected as representative of many current active filter designs using fully differential amplifiers (FDAs). The MFB requires a voltage feedback amplifier (VFA) where precision VFA-based FDAs have been one of the most active new product areas in recent years. FDAs are not well supported by the current tools, but the design results illustrated here using VFA op amps can be applied to FDA designs as well (1). System designs applying the most recent FDAs often include an MFB filter implementation. Those design requirements span a wide range of gains (0.1V/V to 200V/V) where a few examples can be found in (2) sections 9.1.4 and 9.2.1. While attenuating MFB designs using FDAs are common in practice, it does not appear that the current tools support gain magnitudes <1 for the MFB.

The aim here is to target a moderately difficult active filter stage with slight peaking and assess fine scale differences in the results delivered by the different tools. As will be shown in Part 2, the MFB has a noise gain shape that includes peaking due to both the desired filter peaking and noise gain zeroes that vary in the results delivered by the different tools. Here, a modest 1dB peaking target in the response shape will allow the separate noise gain zero peaking to be easily seen. Most op amp-based filter literature seems to assume a gain of 1V/V design for simplicity. Here a more challenging gain of 10V/V (20dB) design will be targeted with an F\textsubscript{−3dB} at 100kHz. The targeted small signal filter frequency response will be:

1. DC gain of –10V/V (20dB)
2. Small signal response peaking of 1dB
3. Small signal response –3dB frequency at 100kHz.

This discussion will focus on the small signal gain response and how well the tools hit that target vs. an ideal response. Nominal deviations in the f\textsubscript{0} and Q < 0.5% should be expected due to standard value RC selections. The requisite equations to describe all this follow.

Ideal 2\textsuperscript{nd} order low pass equation:

\[
\frac{V_o}{V_i} = \frac{A\omega_0^2}{s^2 + s\frac{2\omega_0}{Q} + \omega_0^2}
\]  

Equation 1

Where

1. A is the DC gain
2. \(\omega_0\) is the characteristic or natural frequency (\(\omega_n\)). Will mainly use frequency (f\textsubscript{0} = \(\omega_0/2\pi\)) in Hz.
3. Q is the degree of complexity for the target poles, Q>0.707 will start to show peaking.

From this, all the important relationships can be delivered.

Peaking in dB (this is the expected maximum peaking above the DC gain)

\[
Peaking_{dB} = 20\log\left[\sqrt{\frac{Q}{1 - \frac{1}{4Q^2}}}\right]
\]  

Equation 2
This can be solved for Q given a target or simulated maximum peaking. Converting the dB peaking to a linear peaking over the DC gain (called α here), and solving for Q gives Equation 3:

\[
Q = \frac{\alpha}{\sqrt{2} \left(\alpha + \sqrt{\alpha^2 - 1}\right)}
\]

Equation 3

A 1dB peaking target is a linear gain increase of 1.122. Placing that into Equation 3 gives a target Q of 0.957. From the simulated response shapes delivered by the different tools, Equation 3 will be used to extract the actual Q in the nominal (standard value but exact RC values) designs.

The frequency at which the peak gain should occur is given by Equation 4:

\[
 f_{\text{peak}} = f_0 \left(1 - \frac{1}{2Q^2}\right)
\]

Equation 4

The frequency at which the small signal response will be down −3dB is given by Equation 5. Using the required Q to hit a 1dB peak, Equation 5 can be used to target a characteristic frequency, \(f_o\). For the 100kHz \(f_{-3\text{dB}}\) target with a Q=0.957, the \(f_o\) solves to 80.26kHz. Going back to Equation 4 with this gives an expected \(f_{\text{peak}} = 54.08kHz\).

\[
 f_{-3\text{dB}} = f_0 \left(1 - \frac{1}{2Q^2}\right) + \sqrt{\left(1 - \frac{1}{2Q^2}\right)^2 + 1}
\]

Equation 5

Using the simulated peaking to extract the Q by Equation 3, Equations 4 and 5 can then be used with the actual Q to get two estimates of the simulated \(f_o\) where those will be averaged to assess closeness of fit in the results delivered by the different tools.

A final key equation to use with one of the tools is the frequency at which the peaked response passes back through the DC gain, called the passband or cutoff frequency, \(f_{\text{cutoff}}\) in a Chebyshev design flow. Solving Equation 6 for the required \(f_{\text{cutoff}}\) to get a 100kHz \(f_{-3\text{dB}}\) using a Q=0.957 gives \(f_{\text{cutoff}} = 76.47kHz\).

\[
 \frac{f_{-3\text{dB}}}{f_{\text{cutoff}}} = \frac{1}{\sqrt{2}} \left[1 + \sqrt{1 + \frac{(2Q^2)^2}{(2Q^2 - 1)^2}}\right]
\]

Equation 6

Summarizing these nominal targets:

1. DC gain = -10V/V (20dB) (the MFB is inverting gain, but only magnitudes used in design)
2. Peaking = 1dB (linear peaking of 1.122) implies a Q = 0.957.
3. \(f_{-3\text{dB}}\) = 100kHz
4. \(f_o\) = 80.26kHz
5. \(f_{\text{peak}} = 54.08kHz\)
Implementing the 2nd order MFB design

Implementing the 2nd order MFB design

Once we have the desired response shape described above, there are several common implementation approaches. 3rd party tools like Schematica (3) offer a wide range of implementation options where here only the MFB topology will be investigated. Most op amp supplier tools have reduced their offerings to the MFB or Sallen-Key filter (SKF) topologies as being most suitable to board level designs using their op amps or FDAs.

The MFB implementation circuit is shown in Figure 1. Here, the RC numbering is following (4) where the tools all vary on their RC numbering convention. This one follows Dr. Budak’s “Passive and Active Network Analysis and Synthesis” Figure 10-10, page 350.

![Figure 1: Basic MFB op amp implementation circuit.](image)

The performance equations for this circuit using the numbering shown above will be given by these equations - pulled from (4).

Laplace transfer function,

\[
\frac{V_o}{V_i} = \frac{-1}{C_1 C_2 R_2 R_3 s^2 + \frac{1}{C_1 R_2 R_3} \left( R_3 + R_2 \left( 1 + \frac{R_2}{R_1} \right) \right) + \frac{1}{R_1 R_2 C_1 C_2}} \quad \text{Equation 7}
\]

Setting \( s = 0 \) (DC) will give the DC gain as

\[
A_v(\text{DC}) = -\frac{R_1}{R_3} \quad [\text{in V/V}] \quad \text{Equation 8}
\]

Matching up the terms in Equation 7 to the general form of Equation 1 will give

\[
\omega_0 = \sqrt{\frac{1}{R_1 R_2 C_1 C_2}} \quad [\text{in radians}] \quad \text{Equation 9}
\]
Once a design tool has delivered its RC values, if the tool is not adjusting for the op amp gain bandwidth product (GBP or GBW), putting those values into these three equations will show what the nominal design should be delivering assuming an ideal op amp. Again, exact RC solutions not adjusting for GBW will hit the targets exactly whereas standard values will move off slightly. If the tool is adjusting for GBW, the values will appear to be missing the targets if placed into Equations 8 to 10 and only by simulation with the op amp model will the actual response shape be extracted.

Reference 4 goes on to propose an implementation flow that attempts to improve the dynamic range delivered in a 2nd order MFB low pass filter. It does this by scaling the resistors up as far as allowed before they start to make a meaningful impact in the total output noise power over what the op amp noise terms will be generating. It then goes on to set some key element ratios to reduce the noise gain peaking. That normally shows up as designs with R2>R3. Solutions that deliver results where R2<R3 (in Figure 1) will be showing a higher noise gain peaking than necessary. An improved version of the implementation strategy described in (4) deliver the MFB solutions from the Intersil online design tool (5). The details of those implementation flows are beyond the scope of this investigation. Those solutions are unique to each tool and rarely published. Comparing their results, not necessarily knowing the details of each, is the intent here.

**Navigating the different tools to deliver the desired design.**

There are numerous active filter design tools available. Here, only three (FilterPro and the new TI Webench Filter Designer are combined as one tool) of the most recently updated op amp supplier tools will be exercised. Some of the 3rd party tools (3) offer many nice features but are not well linked to the implementation op amp selection. Most of the tools start out with exact RC solutions - often with standard value C’s chosen first in an E24 series, then exact values for the three R’s. If the tool is also using an ideal op amp in the design (infinite bandwidth), those initial results will hit the target shape exactly. Going to actual standard value R’s and finite bandwidth amplifiers will pull the nominal response shape away from the target by some degree. The designs delivered will be simulated using both the vendor’s op amp (where possible) and then the same op amp to compare how well the nominal design is hitting the target – the nominal mismatch to target is before any component tolerancing is even applied.

Since the eventual aim is to run simulations with an actual op amp model, an important 1st step is to pick those devices. Starting from the required GBW (or GBP), the most recently updated tool from ADI (6) shows both a minimum GBW without an adjustment for the implementation device GBW and a much lower GBW if the designer selects to adjust the RC’s for the typical device GBW. Implementing the target design in the ADI tool and navigating to where it allows you to pick devices will show these numbers at the bottom:

**Recommended GBW = 15.6MHz**

**Required GBW = 3.89MHz**

The Intersil Active Filter Designer (5) at one time adjusted the RC’s for the op amp GBW where this same design implemented there reports a minimum required GBW = 4.56MHz. That RC adjustment flow for GBW appears to be disabled and the recommended GBW out of the ADI tool will be used as

\[
Q = \frac{\sqrt{\frac{C_2}{C_1}}}{\sqrt{\frac{R_1}{R_2}} + \frac{1}{\sqrt{\frac{R_1}{R_3}}} + \frac{1}{\sqrt{\frac{R_2}{R_3}}}} \quad \text{[unitless]}
\]

Equation 10
a minimum. The TI tools (7) show a very conservative minimum GBW = 76.8MHz and do not adjust for op amp GBW. Using a minimum GBW device should be saving power. That will be done here using approximately the recommended value of 16MHz from the ADI tool for all 3 tools.

To get RC values delivered by each of the tools, we would ideally select an op amp with the same GBW and noise in each of the 3 products lines. The ADI and Intersil tools scale the R’s by the selected op amps input noise terms. While it would certainly be possible to deliver a very low spot noise design using a device like the 1nV/√Hz ISL28190 (8), that would create a few problems.

1. Scaling the R’s down so low to be comparable to a 1nV/√Hz input voltage noise will cause the input resistor (R3 in Figure 1) to become very low. That resistor is the load seen by the prior stage and making this very low will create problems for that prior stage. Here, an input R > 250Ω is a reasonable target to balance loading and noise considerations.

2. To do comparison designs, a similar input voltage noise that is available in a low power device from all 3 suppliers would be preferable (6 to 7nV/√Hz will be the target).

Table 1 shows the 3 devices (9,10,11) selected for similar noise and a GBW in the 12 to 18MHz region plus an additional 150MHz device from TI, OPA300 (13), to test the simulations with a device exceeding the > 77.9MHz recommended in the TI tools. The Intersil tool does not show a device very close to those selected from ADI and TI, but the ISL28110 has similar noise, just a bit low on the GBW and not 5V capable – the Intersil tool is apparently delivering ideal op amp RC values, so the low bandwidth does not matter. But the Intersil solutions do scale the resistors for the selected devices’ input noise terms.

Table 1 Selected devices and parameters

<table>
<thead>
<tr>
<th>Supplier</th>
<th>Device</th>
<th>Part Number</th>
<th>Gain</th>
<th>Pole</th>
<th>3dB Bandwidth</th>
<th>GBW</th>
<th>Noise</th>
<th>Total Q</th>
<th>Stage Noise</th>
<th>Stage Noise</th>
<th>GBW Adjusted</th>
<th>Frequency</th>
<th>Noise Adj.</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>LTC</td>
<td>11080002</td>
<td>A6240</td>
<td>2.6</td>
<td>10</td>
<td>16.6</td>
<td>6.0</td>
<td>1.0</td>
<td>1.0</td>
<td>3</td>
<td>2</td>
<td>2</td>
<td>200kHz</td>
<td>2</td>
<td>LTC4250</td>
</tr>
<tr>
<td>ADI</td>
<td>ADI3000</td>
<td>A3000</td>
<td>0.6</td>
<td>4.0</td>
<td>11.0</td>
<td>1.3</td>
<td>2.5</td>
<td>2.6</td>
<td>1</td>
<td>0</td>
<td>0</td>
<td>100kHz</td>
<td>0</td>
<td>ADI2459</td>
</tr>
<tr>
<td>Intersil</td>
<td>ISL28100</td>
<td>ISL28190</td>
<td>2.7</td>
<td>0.5</td>
<td>10.3</td>
<td>0.5</td>
<td>3.5</td>
<td>3.5</td>
<td>3</td>
<td>3</td>
<td>3</td>
<td>100kHz</td>
<td>3</td>
<td>Intersil</td>
</tr>
<tr>
<td>Intersil</td>
<td>ISL28100</td>
<td>ISL28110</td>
<td>2.7</td>
<td>0.5</td>
<td>10.3</td>
<td>0.5</td>
<td>3.5</td>
<td>3.5</td>
<td>3</td>
<td>3</td>
<td>3</td>
<td>100kHz</td>
<td>3</td>
<td>Intersil</td>
</tr>
</tbody>
</table>

Note that that reported SSBW at Gmin = 1 in this table is not necessarily the actual GBW that is more useful for MFB design. Now step through each of the tools to deliver the RC values.

Analog Devices’ Filter Wizard

Analog Devices’ Filter Wizard (6)

This tool has been significantly changed and upgraded over the last few years. It is much simpler on the front end with a combined LTC and ADI op amp table (315 devices) showing detailed parameters. It does not, however, allow direct target pole entry and the “fewer stages” slider steps across the desired 1.00dB peaking. We will use the 1.04dB peaking solution with F_{3dB} = 100kHz and compare to that ideal shape. Two solutions will come out of this tool – one without GBW adjustment (where the fit errors will be both standard values snaps and op amp GBW) and one with the GBW adjustment selected where a closer fit should result even using the higher recommended GBW vs the minimum reported. Setting up the targets and selecting the LTC6240 for implementation gives the following two solutions. This tool also provides a simulated results download to excel which will be used for fit comparisons using the LTC6240 macromodel. The designs are being developed using +/-2.5V supplies to simplify later simulations.
Going back and selecting the GBW adjustment features makes these slight adjustments to the RC values. Note the C’s are in an E24 series while the R’s are E96. The tolerance on those C’s can be selected separately from the series where 2% NPO (or C0G) capacitors in E24 steps are readily available.

Both solutions show an input resistor >250Ω where that is < the resistor inside the loop – this will produce lower noise gain peaking inside the circuit vs. solutions where the input resistor is > the resistor inside the loop.

**Texas Instruments' Webench Filter Designer** (7)

There are actually two tools that have been available from TI. The original FilterPro tool first appeared from Burr Brown in 1991 and has been updated regularly to version 3.1. This tool downloads to your computer and offers many very nice filter selection tradeoff features. It does not help you select an op amp and executes designs only with an ideal op amp. Filterpro (12) as of August 2017 was removed from the TI website in favor of the more recent Webench Filter Designer.
Since FilterPro resides locally on a vast number of designer desktops, it can still be used for illustration purposes. It seems the newer Webench Filter Designer uses the core design code from FilterPro but adds considerable front end solution optimization features and does try to provide a device for the design. To get the desired 2nd order MFB design, these tools both provide a 1dB peaked Chebyshev option where the Passband or Cutoff Frequency \( f_c \) must be entered as 76.49kHz to get a 100kHz \( F_{-3dB} \) design in FilterPro. That frequency becomes the passband, \( f_p \), frequency in the Webench version.

Working first with Filterpro, since it is not linking the RC solutions to a particular device, it initially delivers a very low resistor value solution driven by a 1nF feedback capacitor selection. That is easily changed by changing the feedback capacitor value down to be more in the range shown in the ADI tool. Going with a 300pF feedback capacitor value delivers this design.

![Figure 4](image)

**Figure 4** Filterpro with its C1 reduced from 1nF to 300pF

Note in this case the resistor inside the loop is < than the input resistor. This solution will yield higher noise gain peaking than the ADI solution. Filterpro offers a simulated data download feature where its simulated response with an ideal op amp can be compared to the target response shape.

Simply stepping that C1 in **Figure 4** around, different solutions with the inside the loop resistor > than the input resistor can be generated. Use the redesign (changing the CapSeedValue) features in the Webench filter designer (7) to generate that 2nd TI design in **Figure 5**. Adjusting the “Optimizer” knob to the right would have a similar effect.
A number of interesting issues are highlighted on this page:

1. The cutoff frequency is now not the one entered into the design targets page (which is called \( f_p \), passband here), but instead is now the natural or characteristic frequency (the same that will be used in the Intersil tool).
2. The 77MHz min GBW is very high vs alternate tools and it says no amplifier found. It seems this tool still needs to be updated with the higher speed devices where the 150MHz OPA300 (13) would have been a valid solution to that minimum 77MHz target.
3. The resistor inside the loop is now > input resistor – this will give a lower noise gain peaking than the earlier solution using the 300pF feedback C in the FilterPro result.

Since the design is using an Ideal amplifier, the RC solutions should hit the design targets with an error only attributable to standard value snaps. This new tool does not appear to allow a simulated data download, so the only path to response shape data is to simulate it in FilterPro (ideal op amp) or TINA (14) using the LMP7711 model. This device is quite a bit slower than recommended in the TI tools, so some deviation from target should be expected.

**Intersil’s Filter Designer (5)**

The Intersil tool offers a manual entry option where the desired DC gain, \( Q \), and \( f_c \) can be directly entered and a solution generated. The 5V supply requirement eliminates the ISL28110 as a selection, but, unlike the ADI tool, you are allowed to force the design to an unsuitable device and continue to the next step of generating RC values. Doing that with the ISL28110 gives this schematic on the design summary page.
The Intersil tool does not appear to offer direct simulation data download, so the only path to fit assessment will be to use these values in TINA with the LMP7711. The RC’s here appear to be assuming an ideal op amp. Using these values with the much higher speed OPA300 should improve the fit. The Intersil solution shows the R2 element much larger than the input resistor, R3. This should give the lowest noise gain peaking of the solutions shown so far. The Intersil tool does provide a redesign feature where C2 and C1 can be reset with new R solutions generated. We will continue, however, with the first cut solution shown above using the LMP7711 and OPA300 TINA models.

Conclusions and what’s next?

It is apparent that considerable effort from the op amp vendors has been applied to improving their online active filter design tools. The three newest shown here have improved considerably over the last 5 years where a much more concerted effort to tie requirements to implementation op amps has been made. Table 2 summarizes some of the features across each of the four tools used to generate the designs here.

Table 2 Some of the features for each of the tools
To actually build the designs, each of the tools are required to adjust to standard RC values. It is likely some tools do that better than others. Also, for the ADI feature to adjust for GBW, a better nominal fit to target should be expected once the detailed response shapes are generated with the op amp models and compared to the ideal target. Part 2 will show those comparisons and then go on to generate the spot and integrated noise comparisons. It will conclude comparing noise gain shapes over frequency with the minimum loop gains delivered by each of the RC solutions generated here.

Editor's note: See the second part of this article [here](#).

References

1. “Using the infinite-gain, MFB filter topology in fully differential active filters”, Thomas Kuehl, 3Q2009, TI Analog Applications Journal
2. TI, THS4551 “Low Noise, Precision, 150-MHz, Fully Differential Amplifier”
3. Filter Wiz Lite from Schematica
5. Entry page for the Intersil online op amp design tools. Login required
6. Entry page for the Analog Devices online Active Filter Wizard. Login required
7. Entry page for the Texas Instruments Webench Filter Designer. Login required
9. TI/NSM LMP7711 “Single and Dual Precision, 17MHz, Low Noise, CMOS Input Amplifiers”
10. ADI/LTC LTC6240 “Single/Dual/Quad 18MHz, Low Noise, Rail-to-Rail Output, CMOS Op Amps”
11. Intersil ISL28110 “Precision Low Noise JFET Operational Amplifiers”
12. FilterPro users guide updated 2011
13. TI/BurrBrown OPA300 “Low-Noise, High-Speed, 16-Bit Accurate, CMOS OPERATIONAL AMPLIFIER”
14. TINA simulator available from DesignSoft for <$350 for the Basic Plus edition. Includes a wide range of vendor op amps and is the standard platform for TI op amp models

Related articles:

- [Design second- and third-order Sallen-Key filters with one op amp](#)
- [Advanced considerations for gain and Q sequencing in multistage lowpass active filters](#)
- [Designing RC active filters with standard-component values](#)
- [TI improves its active filter design tool](#)
- [Active-filter circuit and oscilloscope inspect a Class D amplifier's output](#)