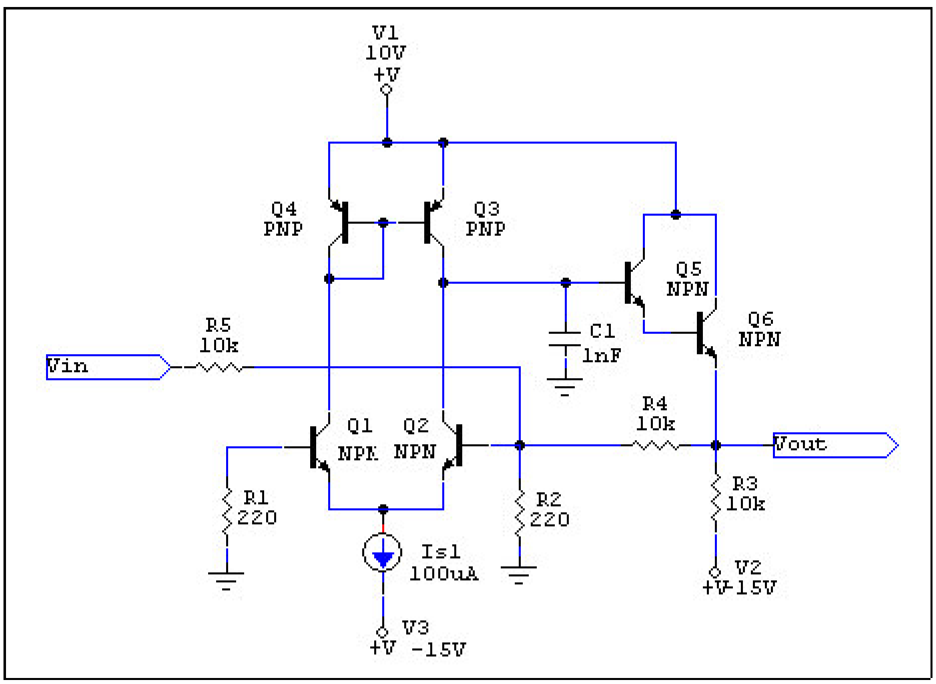
SSM2040 Filter Analysis Part 1 - Ryan Williams

http://www.sdiy.org/destrukto

The following analysis is used to determine the maximum bias current needed for the discrete OTA cells in the ssm2040 filter (René Schmitz version) to achieve a 20KHz cutoff frequency. This information will then be used to set a current limiting resistor in the voltage to current converter. Then an analysis of the voltage to current converter is used to verify that the filter has a 1V/octave response. Some steps are skipped, especially in the OTA cell. The derivation of the equations used to describe these circuit blocks is described elsewhere (check my web site). A few other assumptions are made about the reader's knowlege, but I'll point these out later.

$Kohm := 1000ohm$ $\quad$ $Mohm := 1000Kohm$ $\quad$ $mA := .001A$ $\quad$ $uA := .001mA$

$nF := 10^{-9}F$ $\quad$ $V_t := .026V$ $\quad$ $nA := .001uA$



The above circuit shows a simplified version of the OTA block used in the SSM-2040. The +5V core is replaced by ground to simplify the analysis slightly. The current source (Is1) represents the bias current for the amplifier. The goal here is to determine the range of current that will be useful for musical applications. The OTA core itself (Q1,Q2,Q3,Q4) converts the differential voltage at the bases of Q1 and Q2 to an output current at Q3 and Q2's collectors. The output current is scaled by the bias current (Is1). With the addition of the capacitor (C1) and the darlington follower (Q5,Q6) the circuit forms a current controlled integrator with negative feedback taken from Vout. The circuit for the OTA is very similar to that of the CA3080 and LM13700. There are several derivations of these equations on the net so I won't deal with that now but you can check my website for links to some papers that will help. The equation for the output current into C1 is as follows:

$$I_o = I_{abc} \cdot \tanh\left[\frac{-(V_{in} + V_{out}) \cdot \frac{220ohm}{10Kohm + 220ohm}}{2 \cdot V_t}\right]$$

The Vt variable equals 0.026V at room temperature (again see my site for links talking about this). Since our signal input is fairly small (due to the attenuating resistors 220ohm and 10K) we can approximate the tanh function. This will simplify the analysis significantly. For small values tanh(x)=x so our new equation for Io looks like this:

$$I_o = 19.21 \cdot I_{abc}\left[(-V_{in} - V_{out}) \cdot \frac{220}{10Kohm + 220ohm}\right]$$

The 19.21 constant comes from the 1/(2*Vt). Next the basic equation for a capacitor current/voltage relationship gives this:

$$I = C \cdot \frac{dV}{dt} \qquad \Rightarrow \qquad I_o = 1nF \cdot \frac{d}{dt} V_c$$

I have labeled Vc as the voltage across the capacitor. I also assume that the initial voltage across the capacitor at time t=0sec is 0V. The output voltage is approximated as two diode drops below the capacitor voltage but we will see shortly that this makes no difference in the output voltage.

$$V_{out} = V_c - 1.2V$$

Plugging the equations for Vout and Io into the OTA equation gives this:

$$1 \cdot nF \cdot \frac{d}{dt}(V_{out} + 1.2V) = 19.21 \cdot I_{abc}\left[(-V_{in} - V_{out}) \cdot \frac{220ohm}{10 \cdot Kohm + 220ohm}\right]$$

The next step assumes you have some knowlege of the laplace transform, and frequency response in general. I have used the laplace transform to solve the differential equation above for Vout as a function of Vin and Iabc. This is not the only way to solve such and equation but since we need our result in the frequency domain (as opposed to the time domain) this is my preffered method. I'll spell it out in a few steps incase you are not familiar with the laplace transform. If this is completely new to you I suggest reading a little bit online. try google: "solving differential equations with the laplace transform" and probably some hits on "frequency response"

$$\text{laplace}\left[1 \cdot nF \cdot \frac{d}{dt}(V_{out} + 1.2V)\right] = s \cdot 1nF \cdot V_{out}$$

$$\text{laplace}\left[19.21 \cdot I_{abc}\left[(-V_{in} - V_{out}) \cdot \frac{220 \cdot ohm}{10 \cdot Kohm + 220 \cdot ohm}\right]\right] = 19.21 \cdot I_{abc}\left[(-V_{in} - V_{out}) \cdot \frac{220 \cdot ohm}{10 \cdot Kohm + 220 \cdot oh}\right.$$

The first laplace transform is of the left hand side of the equation. The second shows the right hand side. This is one of the most simple laplace transforms you would ever do. Generally they are much more complicated. The 1.2V on the left hand side is gone because the laplace transform of a constant is 0 (for time t>0sec). The result on the right hand side is exactly what we started with. The transfer function of the circuit in the frequency domain can now be found:

$$19.21\cdot\frac{220\cdot ohm}{10\cdot Kohm + 220\cdot ohm} = 0.414 \qquad s\cdot 1nF\cdot V_{out} = 0.414\cdot I_{abc}\cdot\left(-V_{in} - V_{out}\right)$$

$$\frac{V_o}{V_{in}} = \frac{-0.414\cdot I_{abc}}{1nF\cdot s + 0.414\cdot I_{abc}}$$

should be in this form =>

$$\frac{V_o}{V_{in}} = \frac{-1}{\frac{s}{\omega_c} + 1}$$

$$\frac{V_o}{V_{in}} = \frac{-1}{\left(\frac{1nF}{0.414\cdot I_{abc}}\right)\cdot s + 1}$$

The transfer function is a simple first order lowpass filter. The SSM2040 filter uses 4 stages of this block to create a 4th order filter, but the cutoff frequency of all 4 of them is the same. To find the cutoff frequency I have put the transfer function so that the denominator is in the form (omega*s+1) where the cutoff frequency is found as omega/(2*pi):

$$f_c\left(I_{abc}\right) := \frac{\dfrac{0.414\cdot\frac{I_{abc}}{A}}{10^{-9}}}{2\pi}\cdot Hz$$

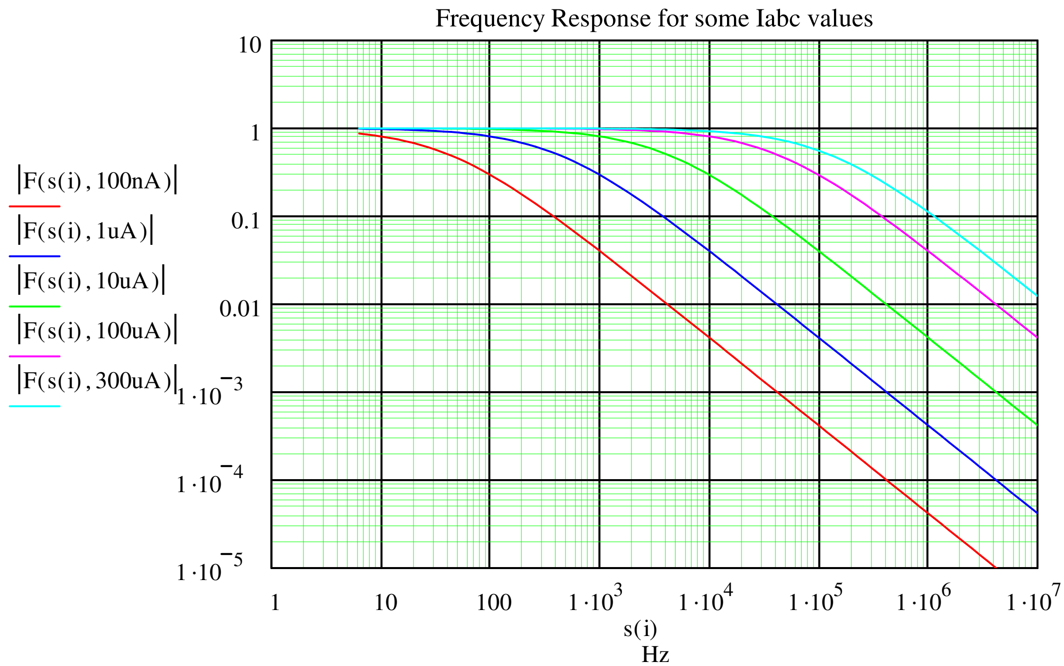
$$f_c(1nA) = 0.066\,Hz$$

$$f_c(300uA) = 19.767\,KHz$$

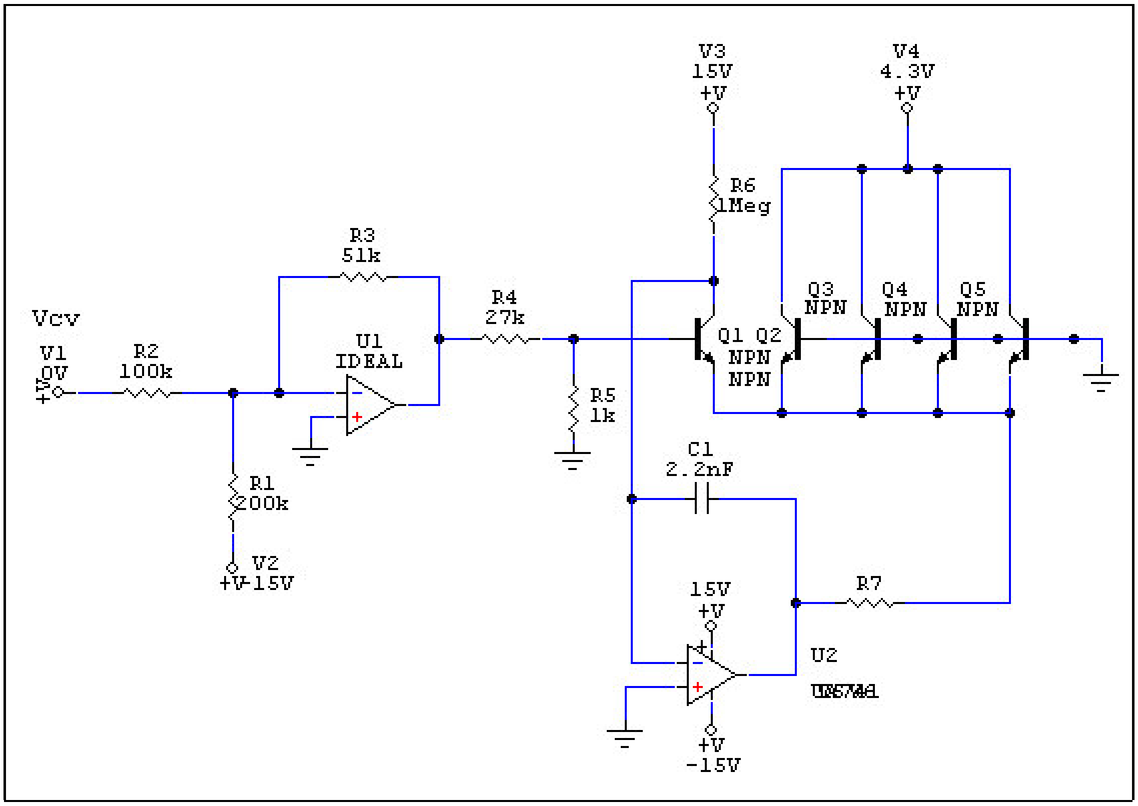
I have solved fc for some Iabc values to find the minimum and maximum usable bias currents. The minimum should be somewhere around 1nA and the maximum should be around 300uA. In the original circuit the current was limited to a few mA which is far too large for this circuit. Now, just for fun, I have plotted the frequency response of the filter for several different values of Iabc. The following steps are only used to scale the plot appropriately.

$$min := 2\pi \qquad max := 2\pi\cdot 10^{7} \qquad n := 100 \qquad i := 0..n \qquad r := \ln\left(\frac{max}{min}\right)$$

$$F\left(s, I_{abc}\right) := \frac{-1}{\left(\dfrac{10^{-9}}{0.414\cdot\frac{I_{abc}}{A}}\right)\cdot s + 1} \qquad s(i) := min\cdot e^{i\cdot\frac{r}{n}}\cdot Hz$$



We have now seen that the bias current should be limited to 300uA. If the current is not limited, then high CV values can cause pops in the audio output. I have seen this on my filter and this kind of behavior (or destruction) might be expected from an IC OTA as well. The currents I am talking about are about 2.5mA which is way to high. The filter has some CV feedthrough at higher bias currents. This is seen as a DC offset at the filter's output. It is likely caused by the low precision OTA circuit used. The pops I heard on my filter may have been partially due to the fed through CV rapidly modulating the output when the bias current reached some large value. At the high end of the CV potentiometer range, the current changes very quickly with even the smallest adjustment.



The above circuit is the current sink that generates Iabc for each of the OTAs. The 4 NPN collectors (Q2,3,4,5) are Iabc1, 2, 3, and 4. The resistor R7 (not given a value yet) is used to limit the maximum current of the current sinks (more on this later). First we find the output voltage of the opamp (U1) as a function of the Vcv control voltage:

$$V_{opamp}(V_{cv}) := 15 \cdot V \cdot \frac{51 \cdot Kohm}{200 \cdot Kohm} - V_{cv} \cdot \frac{51 \cdot Kohm}{100 \cdot Kohm}$$

The voltage at the base of the Q1 is:

$$V_{base}(V_{cv}) := V_{opamp}(V_{cv}) \cdot \frac{1Kohm}{27Kohm + 1Kohm}$$

To find the Iabc current I'll look at Q1 with the opamp circuit formed around it (U2) plus one of the Iabc NPN transistors (Q2 for example). The other transistors have the same base-emitter voltage and collector current. The trick to this circuit, is that the collector current through Q1 is held constant by the opamp (U2). Q1 is in the negative feedback path of U2, Because the opamp will keep the voltage at it's inputs equal, the noninverting input will always be held at 0V (unless the opamp output exceeds it's range). The current trough Q1 can then be found as:

$$I_{c1} := \frac{15V}{1Mohm}$$

The equation for an NPN transistor's collector current is:

$$I_c = I_s \cdot e^{\frac{V_{be}}{V_t}}$$

where Is is the saturation current, Vbe is the base emitter voltage and Vt is the voltage at room temperature.

This equation generates an exponential current as a function of it's base emitter voltage. This is not satisfactory for our purposes because both Is and Vt change with temperature. Is is is probably not precisely given in a data sheet but might be somewhere around 10^-15. This value changes quite a bit with temperature. The changes in Vt are fairly small and will be ignored for now. They can be corrected with a PTC ("tempco") resistor or electronically with a more complicated circuit. To solve the Iabc current we need to do a little trick which cancels out Is from the equation. Dividing I2 by I1 will achieve this. note: this step could be a subtraction but due to log rules, division is equivalent:

$$\frac{I_{c2}}{I_{c1}} = \frac{I_s \cdot e^{\frac{V_{b1}-V_{e1}}{V_t}}}{I_s \cdot e^{\frac{V_{b2}-V_{e2}}{V_t}}}$$

The Is variables in each transistor should be the same if we use two matched transistors on the same chip (the CA3046 or HFA3046 in our case). If the transistors are hand matched and thermally connected (glued together) then we can usually make the same assumption. Because the emitter voltages of both transistors are equal, and the base of Q2 is grounded, we can simplifiy that equation to:

$$I_{c2} = I_{c1} \cdot e^{\frac{-V_{b1}}{V_t}}$$

now set Iabc=Ic2. Ic1, and Vt are plugged in as constants and Iabc is found to be:

$$I_{abc}(V_{cv}) := I_{c1} \cdot e^{\frac{-V_{base}(V_{cv})}{V_t}}$$

$I_{abc}(2V) = 318.233\,nA$ $\qquad$ $I_{abc}(1V) \cdot 2 = 315.886\,nA$

The two currents shown above Iabc at 2V CV and 2*(Iabc at 1V) are used to check whether the bias current will allow 1V/octave. We can see that the current at 2V CV is almost right at twice the current at 1V and that is very close to 1V/octave. If resistor tolerances are off then the value could be slightly different. I would reccomend using 1% resistors and possibly decreasing R4 (27K) just a small amount to allow greater than 1V/octave. Then the CV attenuator pots could set 1V/octave. A trimmer could be used as well if very good scaling is important. This may be the case if you are planning to tune the oscillating frequency of the filter to a VCO.

$I_{abc}(1.793V) = 275.275\,nA$ $\qquad$ $f_c(I_{abc}(1.793V)) = 18.138\,Hz$

$I_{abc}(11.793V) = 303.539\,uA$ $\qquad$ $f_c(I_{abc}(11.793V)) = 20\,KHz$

The above values show that 1.793V to 11.793V CVs cover just about the whole audio range. The panel control of the current source (not shown) can be adjusted to output between -15V and +15V. The extremes of this range are not usable. fc for these two values are shown below:

$f_c(I_{abc}(-15V)) = 1.411 \times 10^{-4}\,Hz$ well below the audio range.

$f_c(I_{abc}(15V)) = 189.124\,KHz$ well above the audio range.

The panel control could be modified to limit the minimum and maximum Vcv values but I'll leave this alone because it will be nice to add or subtract and offset from the panel if needed. the pot is usuable over most of it's range (-7 or -8V to +12V). Instead the current limiting resistor R7 should be adjusted. To find a reasonable value we take the sum of all the Iabc values and Ic1, then divide the biggest possible voltage drop across R7 by the sum of these currents. I have added 200uA to each value of Iabc (an arbitrary number I chose to account for any error). We can work down from there to find a resistor value that exists. As long as the bias currents stay within about 600uA then I don't think there will be many problems.

$$I_{c1} + 4\cdot 300uA = \frac{0.7V - (-13.5V)}{R} \qquad R := \frac{-0.7V + 13.5V}{I_{c1} + 4\cdot 500uA}$$

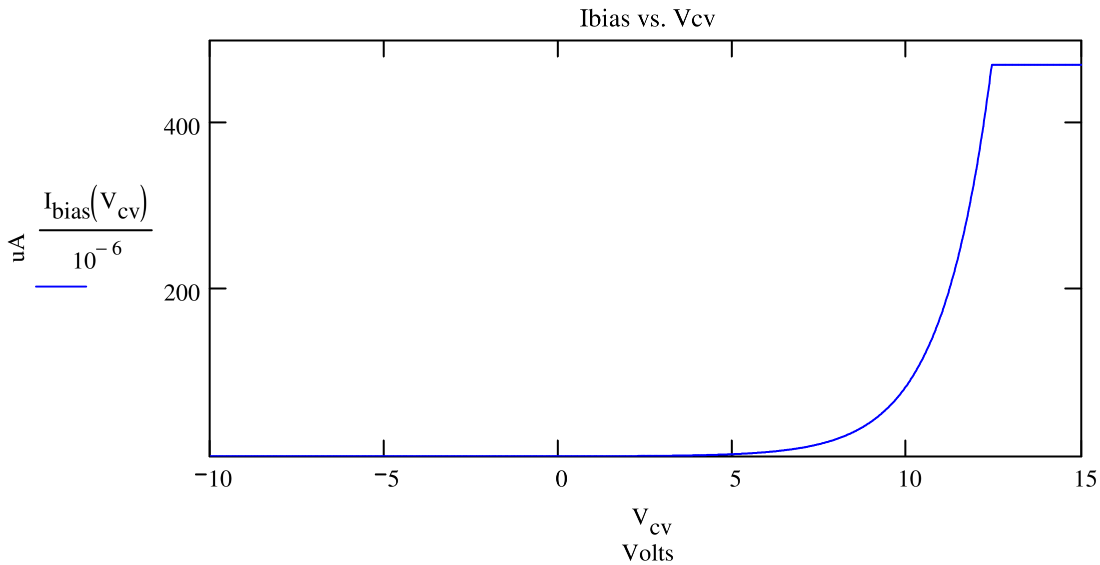
$R = 6.352\,Kohm$

The 13.5V shown is for a TL072. That is about the lowest voltage that a TL072 can output before clipping (assuming a +-15V supply). -0.7V is one diode drop below ground at the emitter of each NPN transistor. I have chosen a resistor of 6.8Kohm and the current through each transistor will be limited to:

$R := 6.8Kohm$ $\qquad$ $\frac{(13.5 - .7)\cdot V}{4\cdot R} = 470.588\,uA$ max bias current

a plot of the bias current vs Vcv is shown below.

$$I_{bias}(V_{cv}) := \text{if}\left(I_{abc}(V_{cv}) < 470.588uA, I_{abc}(V_{cv}), 470.588uA\right)$$



Yes, I did cheat and manually clip the plot at 470uA. One other point to note is that a normal 0..+5V control voltage cannot sweep the filter across it's entire audio range. If this is wanted, then a smaller input resistor on the U1 opamp would achieve this. Perhaps for the second CV input. The first would be labeled 1V/oct and the second could be labeled CV1.