CHAPTER 4

# Compensation

The performance of a control system depends on numerous parameters, among which the compensator transfer function $G(s)$ plays a central role. This is the place where you combine poles/zeros to shape the open-loop frequency response you are looking for but also make sure that robustness is ensured over the converter lifetime. This shape is motivated by the transient performance you want once the system operates in closed loop. For instance, in one case, precision and an absolute absence of overshoot is key. In another one, you want the fastest response to a set-point change or an incoming perturbation, sacrificing the precision of the output variable. Having these elements in mind will actually dictate the way you design your compensator. This chapter will explore the possible compensation strategies applied to practical examples such as linear or switching converters.

## 4.1 The PID Compensator

Literature and the Web in particular abound on the subject of PIDs. If you browse the available pages, such as those in [1], you will learn that the first attempts to use this type of compensator date to governor designs, back at the end of the nineteenth century. Industrial applications to navy ships start to take off in the 1920s. Whereas mechanical/pneumatic systems make an extensive use of PID-based systems, power electronics engineers prefer to place poles and zeros.

Rather than again exploring the processing block, we will quickly show what its peculiarities are and how to bridge it with our power conversion world. Already introduced in Chapter 1, a PID is made of three distinct blocks, each processing the error signal with a mathematical treatment: proportional, integral, and derivative. By individually adjusting the coefficient of each block, the designer can tune the behavioral parameters of a control system such as rise time, damping ratio, or response time.

The combination of several mathematical processing blocks can be written in different forms. The control law in its standard form obeys the following equation:

$$v_c(t) = k_p\left(\varepsilon(t) + \frac{1}{\tau_i}\int_0^t \varepsilon(t)dt + \tau_d\frac{d\varepsilon(t)}{dt}\right) \tag{4.1}$$

where $v_c(t)$ is the control signal delivered by the compensator $G$, and $\varepsilon$ is the error level between the output $y$ and the input variable $u$.

- The first term $P = k_p\varepsilon(t)$ is the proportional term. It generates a control signal proportional to the error amplitude. The idea is to generate a corrective signal in proportion to the input/output mismatch amplitude: if the error is large, you expect an energetic correction; if you have a small error, a small corrective action is necessary. $k_p$ is the parameter that lets you tweak the proportional term. If $k_p$ is high, you have a fast response, but risks of overshoot exist. If $k_p$ is small, you have a slow system exhibiting a sluggish response, but overshoots are reduced.

- The second term $I = \frac{1}{\tau_i}\int_0^t \varepsilon(t)dt$ is linked to the error signal integral. It can be seen as an accumulation or an integration over time of the long-term errors or drifts: you generate a corrective signal as long as you sense an error between the input and the output. In theory, the integral term cancels the static error between the setpoint and the output. In transient, the integral term slows down the response and increases the overshoot. You adjust the integral contribution via the coefficient $\tau_i$. It has the dimension of a time constant. Sometimes, as it integrates or accumulates the errors, the integral block can saturate. To prevent this problem, it is possible to add an anti-windup system.
- The third term $D = \tau_d \frac{d\varepsilon(t)}{dt}$ is relative to the differentiation of the error signal—in other words, to the slope of the incoming perturbation or the setpoint change. Should the slope be very steep, this block delivers a large amplitude. For the opposite, a slow-moving perturbation will generate a lower-amplitude corrective signal. You adjust the derivative contribution through the coefficient $\tau_d$ that also has the dimension of a time constant. It can be seen as an anticipation factor by accounting for the variation of the perturbation. In static, as the derivative of the signal is zero, the derivative term has no effect on the output signal. Dynamically, it helps stabilize the output and increases the response speed.

Figure 4.1 shows the general architecture of this PID compensator.

Should you want to develop (4.1), you'll end up with a different expression known as the *parallel PID* formula:

$$v_c(t) = k_p\varepsilon(t) + k_i\int_0^t \varepsilon(t)dt + k_d\frac{d\varepsilon(t)}{dt} \tag{4.2}$$

In this expression, the parameters *ki* and *kd* no longer have the dimension of a time constant:

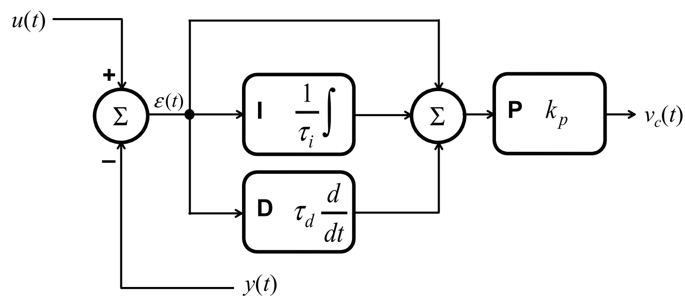



**Figure 4.1** A typical PID implementation showing the three distinct blocks.

$$k_i = \frac{k_p}{\tau_i} \tag{4.3}$$

$$k_d = k_p \tau_d \tag{4.4}$$

The distribution of the $k_p$ term implies a slightly different practical implementation, as shown in Figure 4.2.

In some cases, the whole chain is not necessary and some blocks can disappear. This is the case if you associate a proportional block with an integral block: you create a PI compensator as shown in Figure 4.3.

In this particular case, (4.2) simplifies to

$$v_c(t) = k_p \varepsilon(t) + k_i \int_0^t \varepsilon(t) dt \tag{4.5}$$

Different associations remain possible, like a simple proportional block: you just need a gain $k_p$ for the compensation. Similarly, if you need an integrator, a single integral block is what you will pick.

### 4.1.1 The PID Expressions in the Laplace Domain

The PID control laws given in (4.1) and (4.2) are a continuous-time expression. Small-signal analysis is usually carried in the frequency domain. We will thus apply the Laplace transform to (4.1), remembering that a multiplication by $s$ implies differentiation and division by $s$ implies integration:

$$G(s) = \frac{V_c(s)}{\varepsilon(s)} = k_p \left(1 + \frac{1}{s\tau_i} + s\tau_d\right) \tag{4.6}$$

In this equation, the term $s\tau_d$ is physically improper. If we isolate this derivative term, we have a continuous +1 slope as the frequency increases. The output voltage goes to infinity as $s$ also goes. If $V_D$ is the derivative block output voltage, we have

$$V_D(s) = \tau_d s = \frac{s}{\omega_{z_1}} \tag{4.7}$$

in which $\omega_{z_1} = 1/\tau_d$.

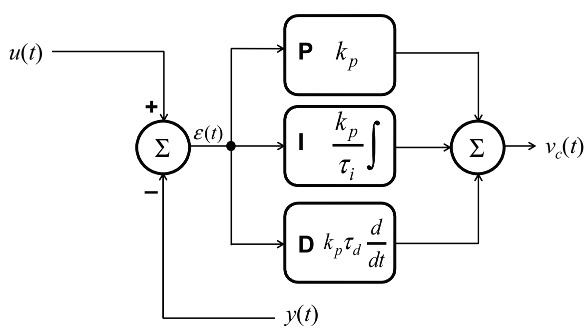



**Figure 4.2** The parallel form of the PID implies the development of (4.1).

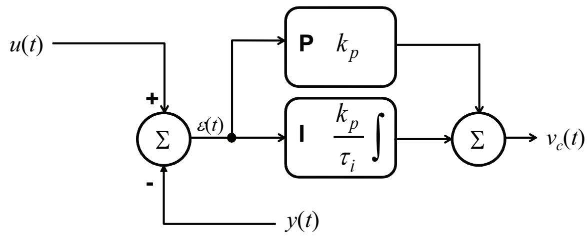



**Figure 4.3** A PI block does not implement the derivative term.

Implementing the derivative term as recommended in (4.6) would bring an extreme sensitivity to incoming noises or perturbations that would corrupt the return chain. To prevent high-frequency noise from coming in, it is common practice to include a pole that safely excludes the upper portion of the frequency spectrum. Figure 4.4 represents both approaches where a pole placed five times above the zero stops the gain excursion ($N = 5$). Including this pole in the PID equation, (4.6) gives birth to the *filtered-PID* equation:

$$G(s) = \frac{V_c(s)}{\varepsilon(s)} = k_p\left(1 + \frac{1}{s\tau_i} + \frac{s\tau_d}{1 + \frac{s\tau_d}{N}}\right) \tag{4.8}$$

If we develop this equation by letting $k_p$ enter the parenthesis, we have again the parallel form, already represented in Figure 4.2 and updated in Figure 4.5.

It is identical in terms of the earlier transfer function of $G$ but introduces individual tuning parameters such as $k_i$ and $k_d$:

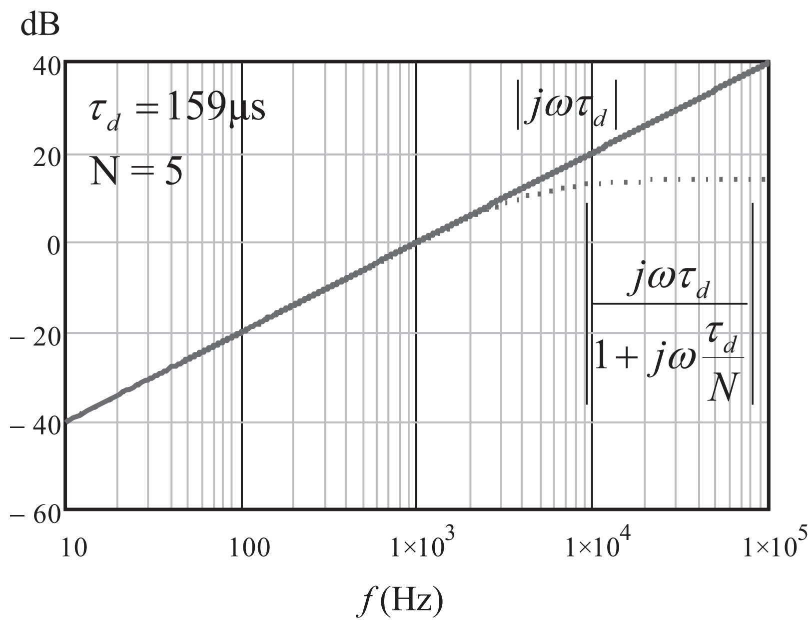



**Figure 4.4** The derivative term builds a +1 slope, potentially bringing the transfer function to infinity as $s$ also goes to infinity. The addition of a high-frequency pole safely clamps the gain excursion.

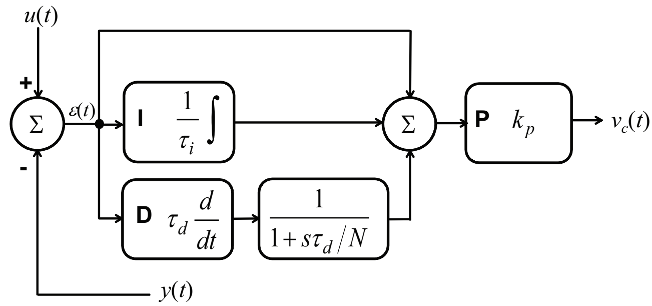



**Figure 4.5** The parallel form filtered PID structure now includes a high-frequency pole.

$$G(s) = \frac{V_c(s)}{\varepsilon(s)} = k_p + \frac{k_i}{s} + \frac{sk_d}{1 + s\dfrac{k_d}{Nk_p}} \tag{4.9}$$

In that case, the links between the coefficients is easy as $k_p$ is unchanged:

$$k_i = \frac{k_p}{\tau_i} \tag{4.10}$$

$$k_d = k_p \tau_d \tag{4.11}$$

Let us now develop (4.8) a little more and factor the terms in a familiar way. We obtain the following equation:

$$\begin{aligned} G(s) = k_p\left(1 + \frac{1}{s\tau_i} + \frac{s\tau_d}{1 + s\dfrac{\tau_d}{N}}\right) &= \frac{1 + s\left(\dfrac{\tau_d}{N} + \tau_i\right) + s^2\left(\dfrac{\tau_d\,\tau_i}{N} + \tau_d\,\tau_i\right)}{s\dfrac{\tau_i}{k_p}\left(1 + \dfrac{\tau_d}{N}s\right)} \\ &= \frac{\left(1 + s/\omega_{z1}\right)\left(1 + s/\omega_{z2}\right)}{\dfrac{s}{\omega_{po}}\left(1 + \dfrac{s}{\omega_{p1}}\right)} \end{aligned} \tag{4.12}$$

This is it—a filtered PID is nothing more than a compensator setting a double zero, an origin pole, and a high-frequency pole! Now it sounds more friendly to us engineers than (4.6), doesn't it? What matters is the relationship to go from the time constant definitions to the poles/zeros placements. Some algebra is necessary to do that.

### 4.1.2 Practical Implementation of a PID Compensator

We, power electronics engineers, place poles and zeros, not individual coefficients as they appear in (4.12). However, for the sake of the study, it is interesting to understand how to bridge a PID compensator to an op amp-based circuitry and learn how to go from one configuration to the other. First, we can redevelop the right side of (4.12) as follows:

$$G(s) = \frac{(1+s/\omega_{z_1})(1+s/\omega_{z_2})}{\dfrac{s}{\omega_{po}}\left(1+\dfrac{s}{\omega_{p_1}}\right)} = \frac{1+s\left(\dfrac{1}{\omega_{z_1}}+\dfrac{1}{\omega_{z_2}}\right)+s^2\left(\dfrac{1}{\omega_{z_1}\omega_{z_2}}\right)}{\dfrac{s}{\omega_{po}}\left(1+\dfrac{s}{\omega_{p_1}}\right)} \tag{4.13}$$

The right side of this equation should match the developed terms of the PID compensator as it appears in (4.12). In other words,

$$\frac{1+s\left(\dfrac{\tau_d}{N}+\tau_i\right)+s^2\left(\dfrac{\tau_d\tau_i}{N}+\tau_d\tau_i\right)}{s\dfrac{\tau_i}{k_p}\left(1+\dfrac{\tau_d}{N}s\right)} = \frac{1+s\left(\dfrac{1}{\omega_{z_1}}+\dfrac{1}{\omega_{z_2}}\right)+s^2\left(\dfrac{1}{\omega_{z_1}\omega_{z_2}}\right)}{\dfrac{s}{\omega_{po}}\left(1+\dfrac{s}{\omega_{p_1}}\right)} \tag{4.14}$$

From this expression, we can identify the individual terms by solving this set of four equations featuring four unknowns:

$$\frac{\tau_d}{N}+\tau_i = \frac{1}{\omega_{z_1}}+\frac{1}{\omega_{z_2}} \tag{4.15}$$

$$\frac{\tau_d\tau_i}{N}+\tau_d\tau_i = \frac{1}{\omega_{z_1}\omega_{z_2}} \tag{4.16}$$

$$\frac{\tau_i}{k_p} = \frac{1}{\omega_{po}} \tag{4.17}$$

$$\frac{\tau_d}{N} = \frac{1}{\omega_{p_1}} \tag{4.18}$$

To go from known poles and zeros to the PID coefficients, we found:

$$\tau_d = \frac{(\omega_{p_1}-\omega_{z_1})(\omega_{p_1}-\omega_{z_2})}{(\omega_{p_1}\omega_{z_1}+\omega_{p_1}\omega_{z_2}-\omega_{z_1}\omega_{z_2})\,\omega_{p_1}} \tag{4.19}$$

$$N = \frac{\omega_{p_1}^2}{(\omega_{p_1}\omega_{z_1}+\omega_{p_1}\omega_{z_2}-\omega_{z_1}\omega_{z_2})\,\omega_{p_1}} - 1 \tag{4.20}$$

$$\tau_i = \frac{\omega_{z_1}+\omega_{z_2}}{\omega_{z_1}\omega_{z_2}} - \frac{1}{\omega_{p_1}} \tag{4.21}$$

$$k_p = \frac{\omega_{po}}{\omega_{z_1}} - \frac{\omega_{po}}{\omega_{p_1}} + \frac{\omega_{po}}{\omega_{z_2}} \tag{4.22}$$

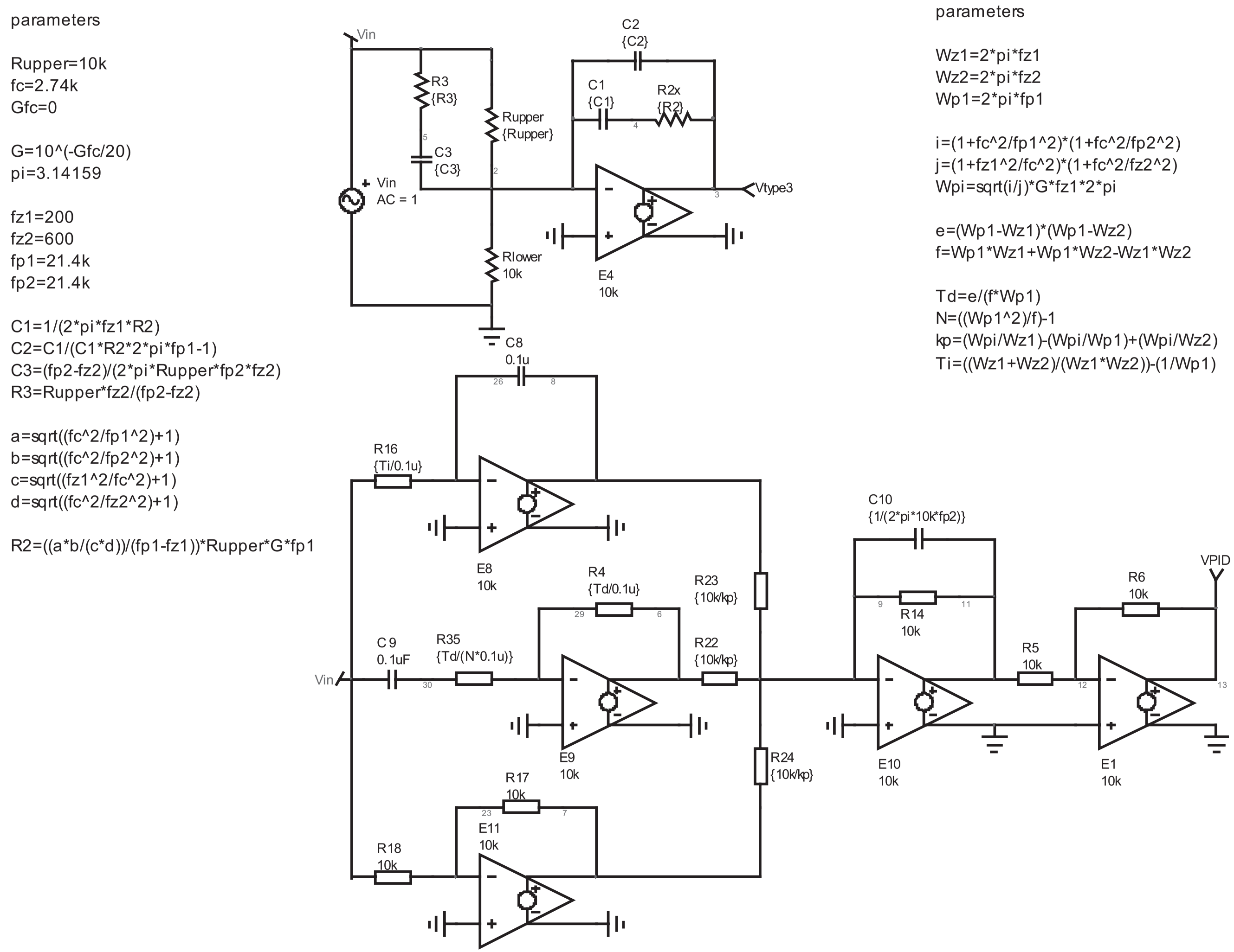



**Figure 4.6** We compare the frequency response of a type 3 compensator classically built with an op amp with that of the equivalent filtered-PID circuit featuring a second high-frequency pole.

To check the validity of these calculations, we have drawn two filters. One is the classical type 3 based on an op amp. A type 3, as we will later see, is made of two zeros, two poles, and an origin pole. To compare the responses, the other filter implements the filtered-PID equation to which a second high-frequency pole $f_{p2}$ has been added. Its location is the same as in the type 3 configuration. Its purpose is to force the gain roll-off at high frequencies, beyond the crossover point. It helps to build a good gain margin but also ensures noise immunity by filtering high-frequency spurious noises.

The comparison SPICE schematic appears in Figure 4.6. Voltage-controlled sources have been used to mimic an op amp behavior, and there is no bias point for this pure ac simulation.

The type 3 compensator organized around $E_4$ places a double zero, an origin pole, and two poles, with one usually at high frequencies. The PID is made of the integral block with source $E_8$, while the filtered derivative term uses source $E_9$. To maintain the inverting sign after the proportional block built with $E_{10}$, a final inverter is added with $E_1$. The polarity of the PID output signal is thus the same as that of the upper circuit using source $E_4$. Please note that all elements are automatically computed in the dedicated parameters windows. An ac signal is injected on the left ($V_{in}$ node), and we can now compare the transfer functions. They appear in Figure 4.7 and confirm the validity of our derivations: the curves perfectly superimpose.

The other way around is also possible, starting from PID coefficients to rebuild a poles-/zeros-based transfer function. The values we have derived are the following ones:

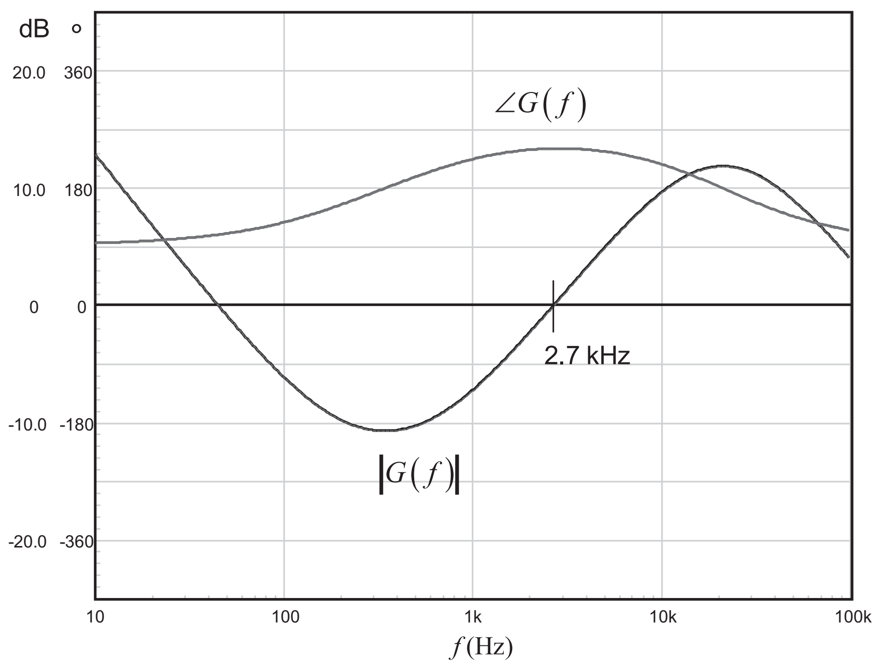



**Figure 4.7** The transfer function from the filtered PID exactly matches that from the op amp-based type 3 compensator. Both curves are perfectly superimposed, confirming the validity of our calculations.

$$f_{z_1} = \frac{\tau_d - \sqrt{-4N^2\tau_d\tau_i + N^2\tau_i^2 - 2N\tau_d\tau_i + \tau_d^2} + N\tau_i}{2\tau_d\tau_i(1+N)2\pi} \tag{4.23}$$

$$f_{z_2} = \frac{\tau_d + \sqrt{-4N^2\tau_d\tau_i + N^2\tau_i^2 - 2N\tau_d\tau_i + \tau_d^2} + N\tau_i}{2\tau_d\tau_i(1+N)2\pi} \tag{4.24}$$

$$f_{p_1} = \frac{N}{2\pi \cdot \tau_d} \tag{4.25}$$

$$f_{po} = \frac{k_p}{2\pi \cdot \tau_i} \tag{4.26}$$

Please note that (4.23) and (4.24) give real zeros if the following conditions are met [2]:

$$N\tau_i - \tau_d \geq 2N\sqrt{\tau_i\tau_d} \tag{4.27}$$

If you find imaginary roots for the zero positions given by (4.23) and (4.24), it simply means that the PID you study implements complex zeros. We will study their effects later in this chapter.

### 4.1.3 Practical Implementation of a PI Compensator

In the case you have built a PI compensator, as presented in Figure 4.3, the compensator formula excludes the derivative term and transforms into a simpler Laplace expression:

$$G(s) = \frac{V_c(s)}{\varepsilon(s)} = k_p + \frac{k_p}{s\tau_i} = k_p\left(\frac{1+s\tau_i}{s\tau_i}\right) = \frac{1+s\tau_i}{s\frac{\tau_i}{k_p}} = \frac{1+s/\omega_z}{s/\omega_{po}} \tag{4.28}$$

If you factor the term $s/\omega_z$ and rearrange the equation, you obtain a familiar format:

$$G(s) = G_0\left(1+\frac{\omega_z}{s}\right) \tag{4.29}$$

Unlike in the complex full PID expression, the gain, the zero, and the pole are immediately identified as:

$$G_0 = \frac{\omega_{po}}{\omega_z} \tag{4.30}$$

$$\omega_z = \frac{1}{\tau_i} \tag{4.31}$$

$$\omega_{po} = \frac{k_p}{\tau_i} \tag{4.32}$$

In the literature, this PI compensator is referred as a type 2a compensator.

The PI expression in (4.28) reveals a pole at the origin associated with a single zero. A possible implementation of such expression appears in Figure 4.8. The PI

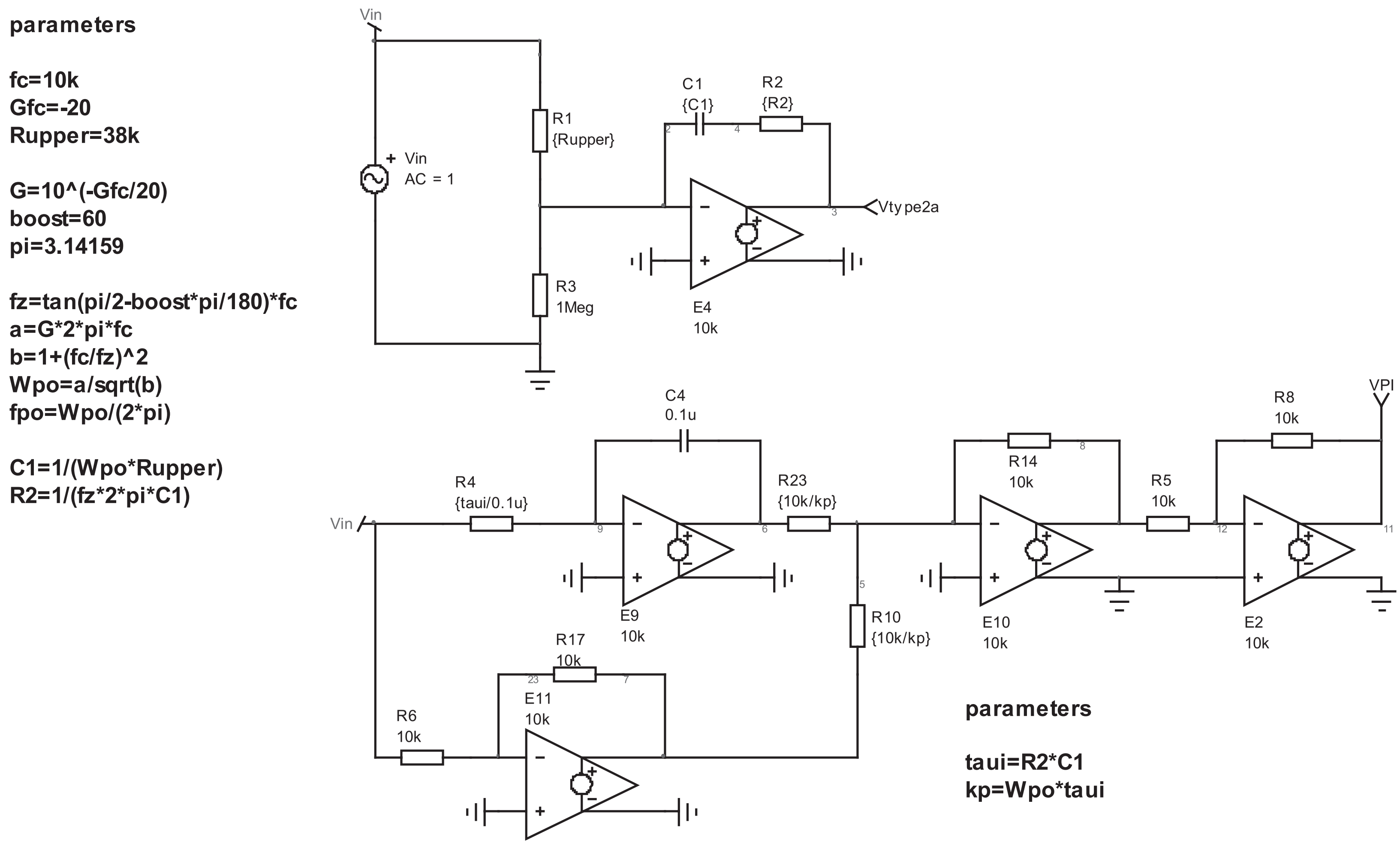



**Figure 4.8** The electrical construction of the PI compensator is simplified due to the absence of the derivative term.

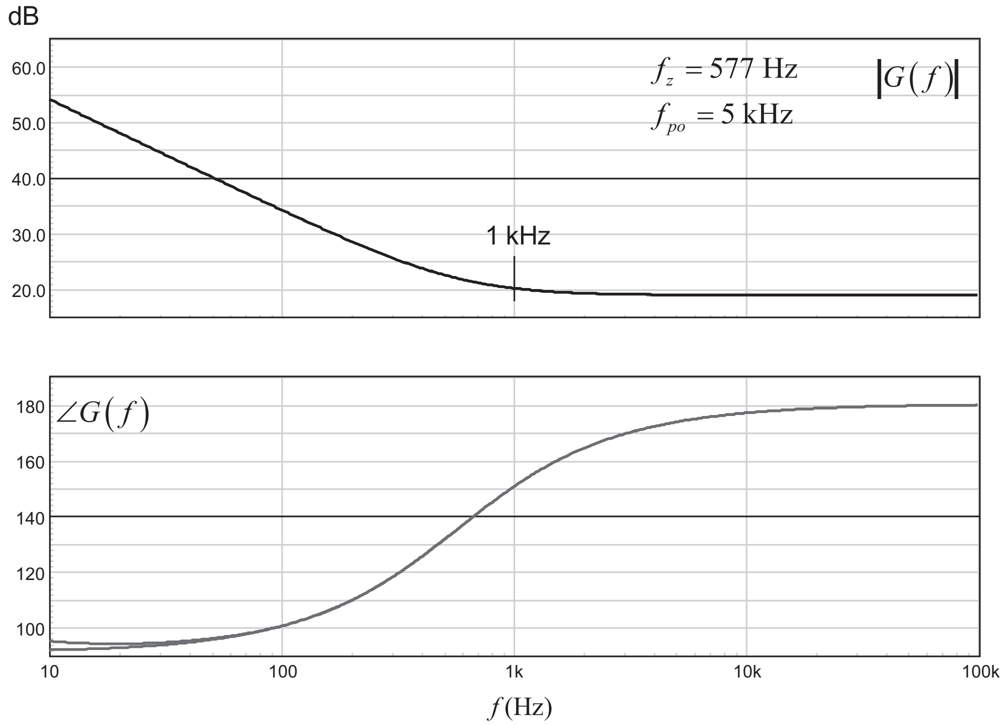



**Figure 4.9** The ac response of the compensator, regardless of whether it is made around a type 2a or a PI compensator, is similar.

compensator coefficients are easily derived and automated in the simulation sheet. The ac response is given in Figure 4.9 and confirms the good agreement between the reference compensator (a type 2a) and the PI circuit.

### 4.1.4 The PID at Work in a Buck Converter

Now that we understand how to build a PID compensator, it is time to put it to work. The idea is to use a buck converter operated in voltage-mode. The schematic of such a switching converter appears in Figure 4.10. The principle of operation is well known. A power switch SW is operated on and off at a pace fixed by an internal clock generator. Typical operating frequencies are between 50 kHz to 1 or 2 MHz for the vast majority of converters. The switch control pattern is produced by the pulse width modulator (PWM) block, just as we studied in Chapter 3. In this technique, the error voltage directly controls the duty ratio and continuously adjusts the switch on-time $t_{on}$ in relation to the operating conditions. Usually, when the error voltage increases, so does the duty ratio, instructing SW to remain closed longer. If SW is permanently closed, you have $V_{in}$ at the diode cathode. Inversely, if SW is permanently open, you have 0. The voltage at the diode cathode is thus a square-wave signal toggling between $V_{in}$ and 0. As we want a dc output voltage, a low-pass filter made of $L$ and $C$ attenuates all unwanted harmonics and the output delivers a dc level equal to the average value of the square-wave signal. Please note that both $L$ and $C$ are affected by parasitic resistive elements, noted $r_L$ and $r_C$,

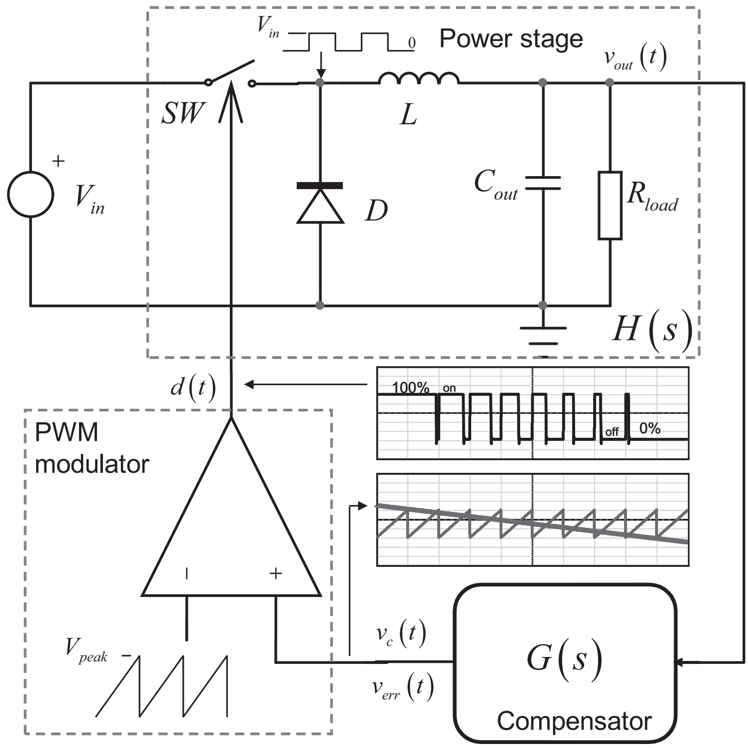



**Figure 4.10** A buck converter meets the output voltage requirement—here 5 V—by adjusting the duty ratio of the power switch.

respectively. They do not appear in this schematic for the sake of clarity. We can show that the output voltage obeys the following relationship:

$$V_{out,avg} = \frac{1}{T_{SW}} \int_{0}^{t_{on}} V_{in} dt = dV_{in} \tag{4.33}$$

If you control the duty ratio $d$ from 0 to 100 percent, you have a means to adjust the output voltage between 0 and $V_{in}$.

The first thing to do when you want to compensate a converter is to get its small-signal transfer function from the control input $V_c$ to the output $V_{out}$. In Figure 4.10, this is $H(s)$, driven by the PWM block. This transfer function can be measured in the lab, derived using small-signal analysis or simply simulated using an average model. We used this last option to plot the ac response of the power stage. The test fixture appears in Figure 4.11, where the capacitor and the inductor are now in series with their respective parasitic elements. We have two distinct storage elements (two state variables), so this is a second-order system. As already demonstrated, the small-signal model of the PWM modulator is simply the inverse of the sawtooth peak amplitude. If you have a 2-V amplitude, the PWM gain is 0.5 or –6 dB. The operating point is selected by biasing the control input to a level that gives you the correct output voltage: 1.05 V applied to $v_c$ gives 5 V in the example. The ac response is that of a second-order system, as shown in Figure 4.12.

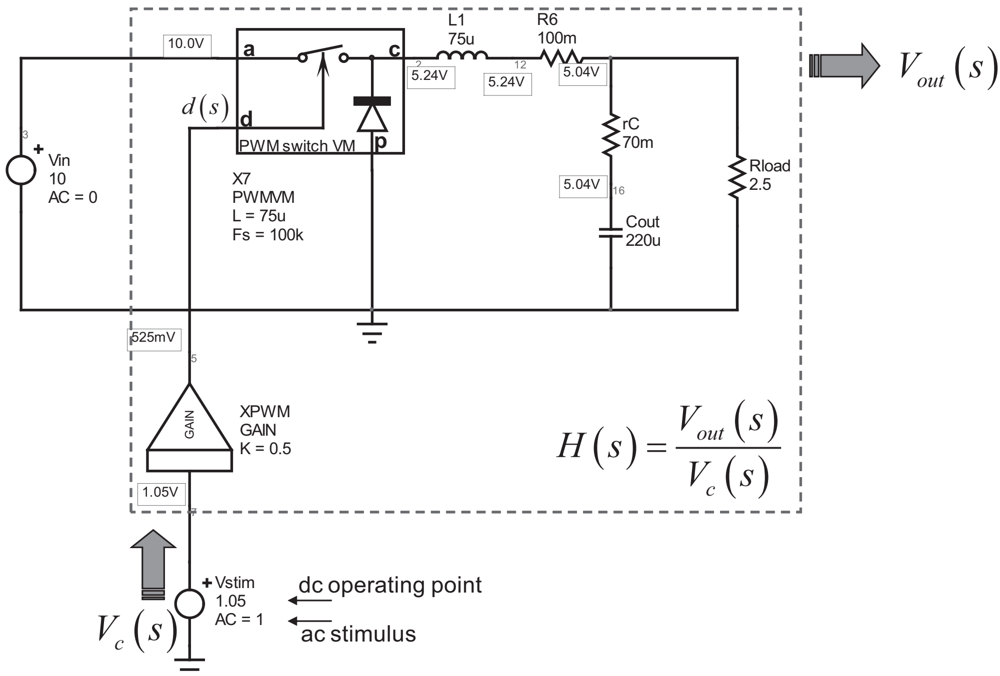



**Figure 4.11** This text fixture uses an average model to extract the small-signal response of the voltage-mode buck converter.

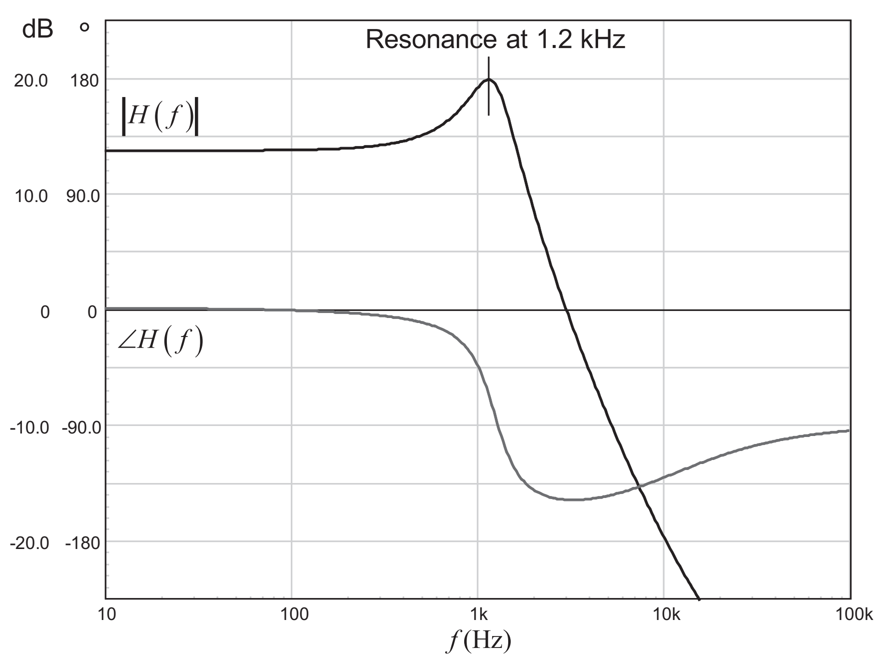



**Figure 4.12** The *LC* filter introduces a resonance in the ac transfer function.

It can be shown that the small-signal analysis carried on the buck converter operated in voltage mode leads to the following equations:

$$H(s) = \frac{V_{out}(s)}{V_c(s)} = H_0 \frac{1 + s/\omega_{z_1}}{\left(\frac{s}{\omega_0}\right)^2 + \frac{s}{Q\omega_0} + 1} \tag{4.34}$$

In which we have

$$\omega_{z_1} = \frac{1}{r_C C} \tag{4.35}$$

$$\omega_0 \approx \frac{1}{\sqrt{LC}} \tag{4.36}$$

$$Q \approx R\sqrt{\frac{C}{L}} \tag{4.37}$$

$$H_0 = \frac{V_{in}}{V_{peak}} \tag{4.38}$$

where $R$, $L$, and $C$ are, respectively, the load resistor ($R_{load}$), the output inductor ($L_1$), and the output capacitor ($C_{out}$) in Figure 4.12. The ESR is designated by $r_C$. $V_{in}$ is the input voltage, and $V_{peak}$ the PWM block sawtooth peak amplitude.

Ideally, we would like a fast transient response, almost no overshoot, and a precise output. In other words, a flat nonpeaking closed-loop gain from input to output is the goal to reach, with a bandwidth of 10 kHz for the sake of the example. To meet this requirement, we must add a PID compensator to the Figure 4.11 converter. The updated schematic using the average model appears in Figure 4.13. In such a representation, the transfer function we study is classically defined as

$$\frac{Y(s)}{U(s)} = \frac{V_{out}(s)}{V_{ref}(s)} \tag{4.39}$$

To obtain a flat ac closed-loop response, let us first study the open-loop expression of our compensated buck. It is the multiplication of the power stage transfer function $H(s)$ derived in (4.34) multiplied by the filtered-PID transfer function $G(s)$ given in (4.12):

$$T_{OL}(s) = H(s)G(s) = H_0 \frac{1 + s/\omega_{z_1}}{\left(\frac{s}{\omega_0}\right)^2 + \frac{s}{Q_0\omega_0} + 1} \frac{1 + s\left(\frac{\tau_d}{N} + \tau_i\right) + s^2\left(\frac{\tau_d \tau_i}{N} + \tau_d \tau_i\right)}{s\frac{\tau_i}{k_p}\left(1 + \frac{\tau_d}{N}s\right)} \tag{4.40}$$

This is a rather complex expression that we can try to simplify. As suggested in [3], if we purposely adjust the double zeros present in the numerator of $G(s)$ to match the position of the double poles that appear in the denominator of $H(s)$, then the expression greatly simplifies. In other words, if we have

$$\left(\frac{s}{\omega_0}\right)^2 + \frac{s}{Q_0\omega_0} + 1 = 1 + s\left(\frac{\tau_d}{N} + \tau_i\right) + s^2\left(\frac{\tau_d \tau_i}{N} + \tau_d \tau_i\right) \tag{4.41}$$

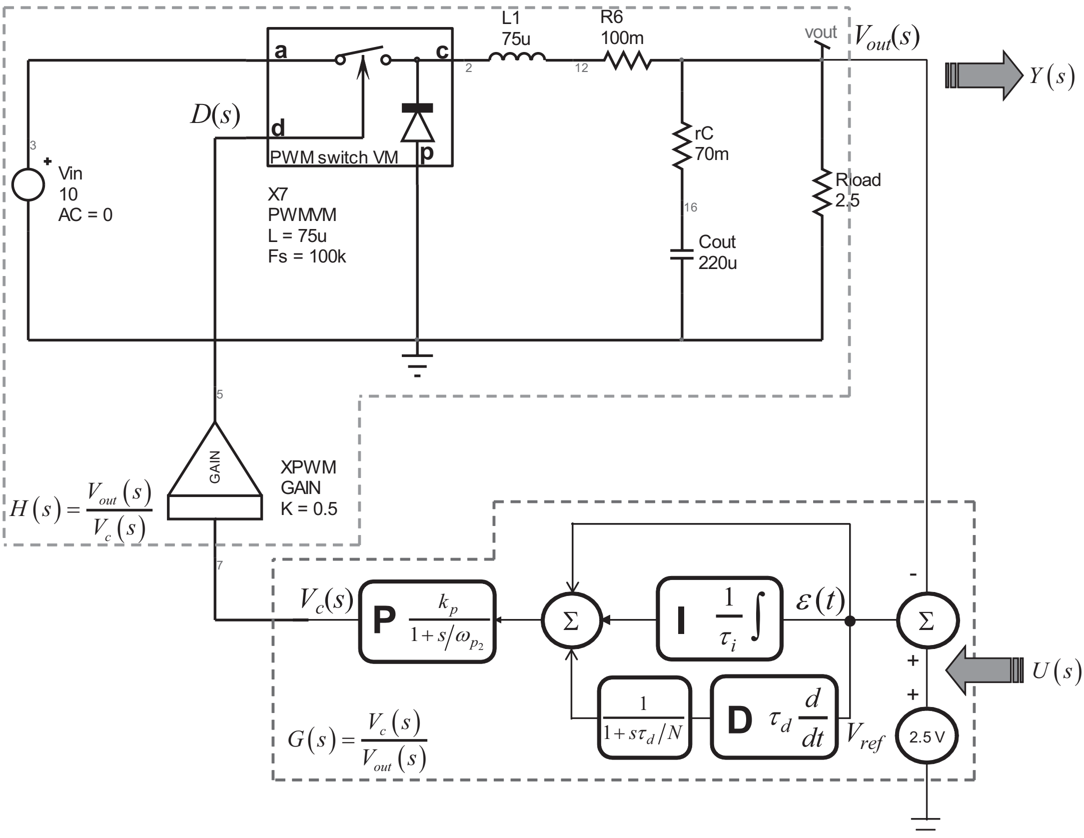



**Figure 4.13** Once the compensator has been added, we obtain a complete buck converter maintaining an output voltage of 5 V. This is obviously a simplified representation.

then the loop gain expression simplifies to

$$T_{OL}(s) = H_0 \frac{1+s/\omega_{z_1}}{s\frac{\tau_i}{k_p}\left(1+\frac{\tau_d}{N}s\right)} \tag{4.42}$$

We know that the closed-loop expression of a unity-return control system is the following:

$$T_{CL}(s) = \frac{T_{OL}(s)}{1+T_{OL}(s)} \tag{4.43}$$

Substituting (4.42) in (4.43) gives

$$T_{CL}(s) = \frac{H_0 \frac{(1+s/\omega_{z_1})}{s\frac{\tau_i}{k_p}\left(1+\frac{\tau_d}{N}s\right)}}{1+H_0 \frac{(1+s/\omega_{z_1})}{s\frac{\tau_i}{k_p}\left(1+\frac{\tau_d}{N}s\right)}} = \frac{1+s/\omega_{z_1}}{1+s\left(\frac{1}{\omega_{z_1}}+\frac{\tau_i}{H_0 k_p}\right)+s^2\left(\frac{\tau_d \tau_i}{N\cdot H_0 k_p}\right)} \tag{4.44}$$

This is our closed expression once the double poles have been neutralized by the double zeros of the PID. If you observe the denominator, you can see that it is following the form of a second-order transfer function:

$$1+s\left(\frac{1}{\omega_{z1}}+\frac{\tau_i}{H_0 k_p}\right)+s^2\left(\frac{\tau_d\,\tau_i}{N\cdot H_0 k_p}\right)=1+\frac{s}{\omega_c Q_c}+\left(\frac{s}{\omega_c}\right)^2 \tag{4.45}$$

Since we want a flat ac response from this second-order transfer function, we can adjust our PID coefficients to match a closed-loop quality factor of 0.5 (coincident poles, no peaking) with a bandwidth of 10 kHz $\omega_c = 62.8$ krad/s. We have four unknown, $\tau_d$, $\tau_i$, $k_p$ and $N$. We thus need four equations. The first two come from the pole/zero neutralization in (4.41). The rest are coming from (4.45) since $Q_c$ and $\omega_c$ are the objectives to reach:

$$\frac{\tau_d}{N}+\tau_i=\frac{1}{\omega_0 Q_0} \tag{4.46}$$

$$\frac{\tau_d\,\tau_i}{N}+\tau_d\,\tau_i=\frac{1}{\omega_0^2} \tag{4.47}$$

$$\frac{1}{\omega_{z1}}+\frac{\tau_i}{H_0 k_p}=\frac{1}{\omega_c Q_c} \tag{4.48}$$

$$\frac{\tau_d\,\tau_i}{N\cdot H_0 k_p}=\frac{1}{\omega_c^2} \tag{4.49}$$

The symbolic extraction gives birth to four quite ugly equations. Sorry, I did not have the courage to give them a little "massage" and make them look more friendly. Mathcad® is to blame!

$$\tau_d=\frac{Q_0 Q_c^2\,\omega_{z1}^2\,\omega_0^2+Q_c^2\,\omega_{z1}\,\omega_0\,\omega_c^2+Q_0 Q_c^2\,\omega_c^4-Q_c\,\omega_{z1}^2\,\omega_0\,\omega_c-2Q_0 Q_c\,\omega_{z1}\,\omega_c^3+Q_0\,\omega_{z1}^2\,\omega_c^2}{\omega_0\,\omega_c(Q_c\,\omega_c-\omega_{z1})(Q_c\,\omega_c^2-\omega_{z1}\,\omega_c+Q_0 Q_c\,\omega_{z1}\,\omega_0)}$$
$$=1.116\text{ ms} \tag{4.50}$$

$$\tau_i=-\frac{Q_c\,\omega_c^2-\omega_{z1}\,\omega_c+Q_0 Q_c\,\omega_{z1}\,\omega_0}{Q\,\omega_{z1}\,\omega_0\,\omega_c-Q_0 Q_c\,\omega_0\,\omega_c^2}=14.6\ \mu\text{s} \tag{4.51}$$

$$N=-\frac{Q_0 Q_c^2\,\omega_{z1}^2\,\omega_0^2+Q_c^2\,\omega_{z1}\,\omega_0\,\omega_c^2+Q_0 Q_c^2\,\omega_c^4-Q_c\,\omega_{z1}^2\,\omega_0\,\omega_c-2Q_0 Q_c\,\omega_{z1}\,\omega_c^3+Q_0\,\omega_{z1}^2\,\omega_c^2}{\omega_0 Q_c\,\omega_{z1}(Q_c\,\omega_c^2-\omega_{z1}\,\omega_c+Q_0 Q_c\,\omega_{z1}\,\omega_0)}$$
$$=72.4 \tag{4.52}$$

$$k_p=\frac{Q_c\,\omega_{z1}(\omega_{z1}\,\omega_c+Q_c Q_0\,\omega_{z1}\,\omega_0-Q_c\,\omega_c^2)}{H_0 Q_0\,\omega_0(\omega_{z1}-Q_c\,\omega_c)^2}=0.178 \tag{4.53}$$

Now that we have our PID coefficients, we can plot the compensator transfer function and the plants for reference. This is what Figure 4.14 shows. The compensator response is rather unusual. We can clearly see an origin pole (infinite gain for $s = 0$) but then a kind of notch appears, exactly at the $LC$ resonant frequency.

With the help of (4.23) through (4.26), we can check the resulting poles and zeros corresponding to the calculated PID. If we look at the zeros placed by the adopted compensation strategy, we have

$$f_{z_1} = \frac{\tau_d - \sqrt{-4N^2\tau_d\tau_i + N^2\tau_i^2 - 2N\tau_d\tau_i + \tau_d^2} + N\tau_i}{2\tau_d\tau_i(1+N)2\pi} = 144.7 - j1.23k \tag{4.54}$$

$$f_{z_2} = \frac{\tau_d + \sqrt{-4N^2\tau_d\tau_i + N^2\tau_i^2 - 2N\tau_d\tau_i + \tau_d^2} + N\tau_i}{2\tau_d\tau_i(1+N)2\pi} = 144.7 + j1.23k \tag{4.55}$$

No wonder a notch appears in the transfer function of $G(s)$: the two zeros are conjugate complex numbers. This actually makes sense since we wanted to neutralize a pair of coincident complex poles located at $\omega_0$. How do you neutralize them? By placing a pair of coincident complex zeros at $\omega_0$. If a double complex pole induces a peak in the transfer function, a double complex zero is seen as a notch in the ac plot. This is exactly what Figure 4.14 shows.

Regarding the additional poles brought by $G$, they are located at

$$f_{p_1} = \frac{N}{2\pi \cdot \tau_d} = 10.3\text{ kHz} \tag{4.56}$$

$$f_{po} = \frac{k_p}{2\pi \cdot \tau_i} = 1.9\text{ kHz} \tag{4.57}$$

Again, it makes perfect sense. The compensator must force the loop gain decrease so that the selected crossover point is reached at the right frequency. What in the plant transfer function opposes a magnitude decrease as the frequency increases? A zero. This is $\omega_{z_1}$ found to be at 10.3 kHz with (4.35). Naturally, the zero (real in this case, a LHP zero) is neutralized by placing a pole $f_{p_1}$ at its frequency. Finally, the 0-dB crossover pole $f_{po}$ is adjusted to make sure the curve is perfectly flat until 10 kHz are reached. The final result appears in Figure 4.15.

The open-loop transfer function is perfect and offers a phase margin of 80°. The closed-loop response is also excellent and shows a flat transmission up to 11.8 kHz, the cutoff frequency. There is absolutely no peaking; we expect an excellent transient response.

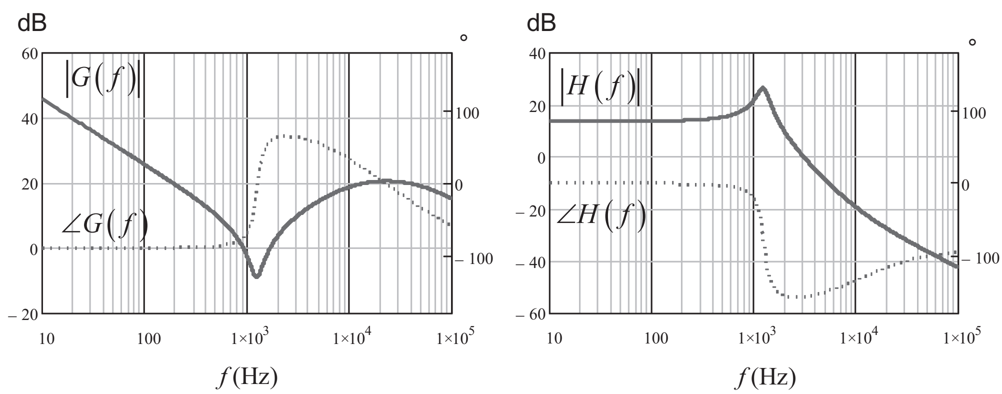



**Figure 4.14** The compensator and the plant transfer functions are plotted using Mathcad.

### 4.1.5 The Buck Converter Transient Response with the PID Compensation

The transient response of our compensated converter can be evaluated in several ways. The simplest one is to use the closed-loop transfer function obtained with (4.43). It describes the relationship between the reference voltage $v_{ref}$ and the output voltage $v_{out}$. To plot the response with Mathcad we apply a step to (4.43) and we take the inverse Laplace transform. However, as this equation is an ac small-signal equation, it does not convey a dc operating point. Suppose we have a 5-V output and we want to check the resulting impact of a 300-mV voltage step on the reference voltage $V_{ref}$. The equation we should plot is thus

$$v_{out}(t) = 5 + \mathcal{L}^{-1}\left(\frac{0.3}{s}\frac{T_{OL}(s)}{1+T_{OL}(s)}\right) \tag{4.58}$$

The resulting waveform appears in Figure 4.16. As expected, it is a perfect nonringing answer, in agreement with Figure 4.15 predictions. The gain is 1 (or 0 dB), justifying the 300-mV output step for a 300-mV input.

This is a purely analytical prediction, and it is interesting to see how it compares to our SPICE model implementing a buck average model. After all, in a regulator application, such as dc-dc/ac-dc converters or linear regulators, the reference voltage is never stepped since we want a constant output voltage. Rather, the output current or the input voltage are perturbed and impose the regulator to reject them in relationship to operating conditions.

The updated simulation schematic appears in Figure 4.17. You can see the average model whose input receives a signal from the B2 source. This source simply clamps the PID error voltage so that the duty ratio voltage does not exceed 100 percent. The PID implementation slightly differs from that in Figure 4.6 but the result is similar. The additional pole $\omega_{p2}$ added through subcircuit X6 does not impact the response because it is placed at high frequencies. This second pole—present in a type 3 compensator—forces frequency roll-off in the upper portion of the spectrum. Indeed, if you look at (4.13), the equation magnitude does not reach 0 as $s$ goes to infinity. With the addition of this extra pole, we make sure the denominator degree is greater than that of the numerator: the function is said to be *proper*.

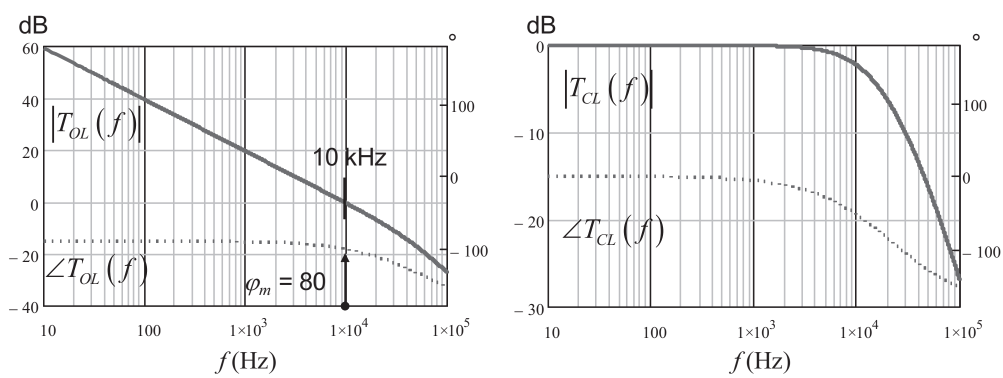



**Figure 4.15** The compensated control system offers a flat response with a 10-kHz crossover frequency.

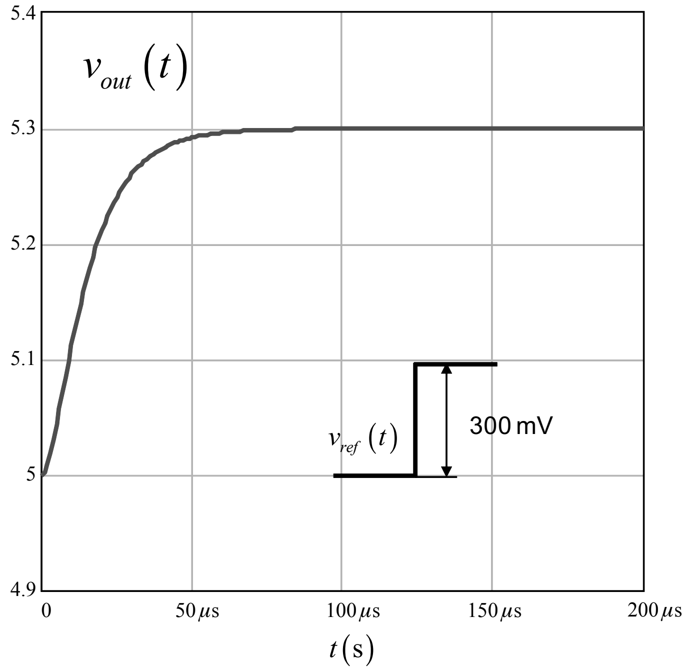



**Figure 4.16** The output transient response further to a 300-mV step on the reference voltage confirms the excellent compensation scheme.

To check the transient response, the load has been replaced by a current source, stepping the current from 1 A to 2 A in 1 µs. The results are displayed in Figure 4.18 and show a stable but oscillatory waveform. It really contradicts the Figure 4.15 plot showing a low-$Q$ flat closed-loop transfer function. From the transient result, we can derive the quality factor value using the formula we have seen in Chapter 3:

$$Q = \sqrt{\left(\frac{\pi}{\ln k}\right)^2 + \frac{1}{4}} = \sqrt{\left(\frac{\pi}{\ln\frac{21.5}{48.4}}\right)^2 + \frac{1}{4}} = 3.9 \tag{4.59}$$

It is far from our 0.5 design target! Also, if you look carefully, the oscillations do not correspond to a 10-kHz signal (our crossover point) but to the 1.2-kHz $LC$ network resonant frequency. So where is the problem?

### 4.1.6 The Setpoint Is Fixed: We Have a Regulator!

In Chapter 1, we detailed the definition of a control system where the output must faithfully follow a setpoint change. In a switching or linear converter case, the setpoint is the reference voltage $V_{ref}$ that the output must match, scaled up or down by a divider ratio. When operating, $V_{ref}$ never changes (unless it is an adjustable output, of course) and the system must deliver a stable and precise output despite incoming perturbations. In that case, we talk about a regulator as introduced in Chapter 1. Such a system can be modeled in a way where perturbations appear. This model has already been presented before in its simplified form. The updated version that is shown in Figure 4.19 now includes the input voltage as another perturbation.

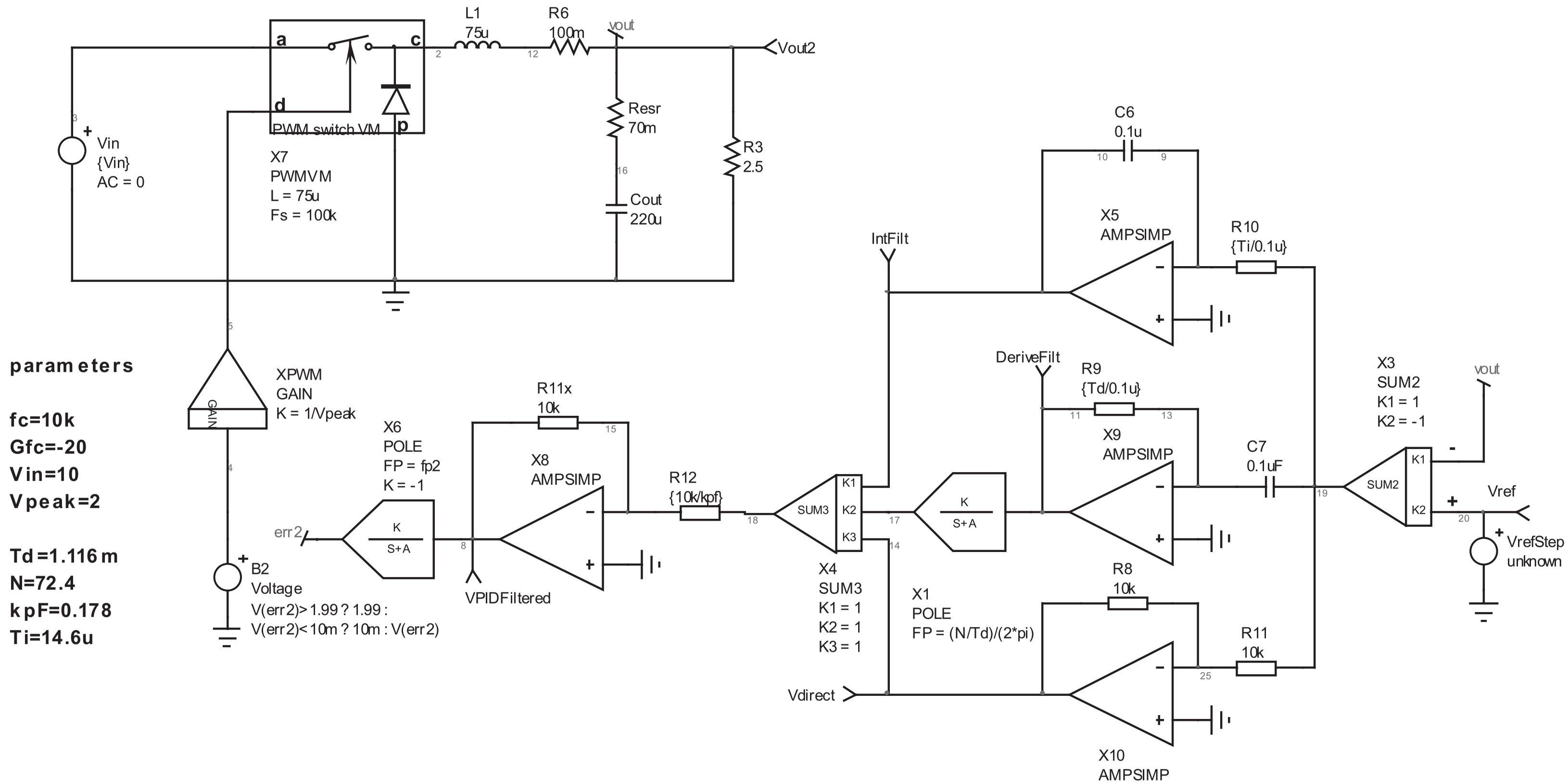



**Figure 4.17** The complete simulation schematic implements the buck model with the full PID chain. The coefficients are those we already computed.

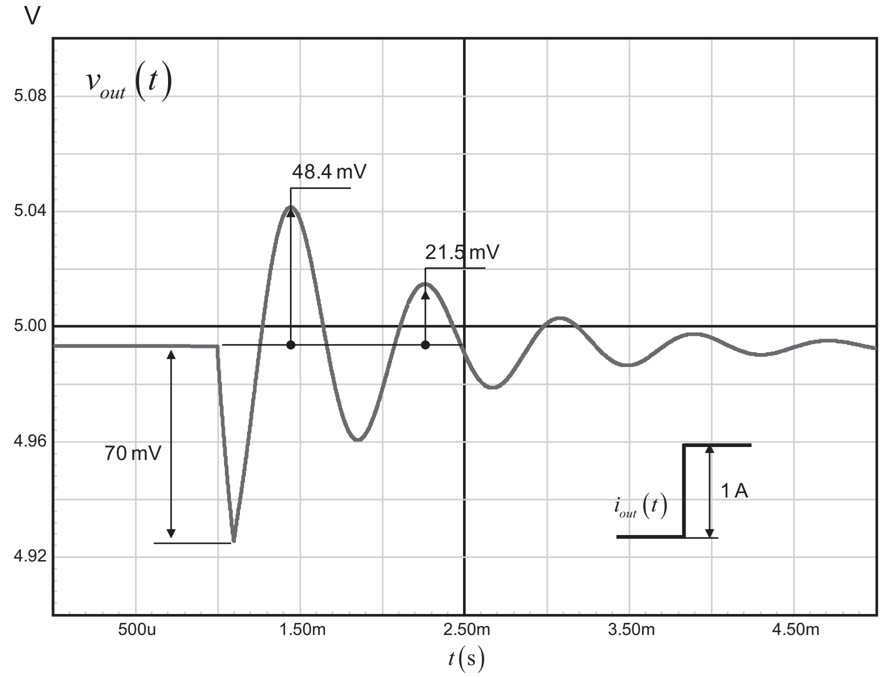



**Figure 4.18** The transient response to an output current step is stable but oscillatory.

In a previous chapter, we have seen that in a system like that of Figure 4.19 all perturbations were multiplied by the sensitivity function $S$. Therefore, without applying the superposition theorem, we can directly write the output voltage equation of Figure 4.19 system:

$$V_{out}(s) = V_{ref}(s)\frac{T(s)}{1+T(s)} - Z_{out}(s)I_{out}(s)\frac{1}{1+T(s)} + V_{in}(s)G_{V_{in}}\frac{1}{1+T(s)} \tag{4.60}$$

In reality, as the reference voltage $V_{ref}$ is fixed, its ac value $\hat{v}_{ref}$ is 0. The same comment applies to the input voltage $V_{in}$: when we step the output, we assume that the input voltage does not change. The previous equation defining the output voltage can thus be rewritten as

$$V_{out}(s)\Big|_{\hat{v}_{ref}=0,\hat{v}_{in}=0} = -Z_{out}(s)I_{out}(s)\frac{1}{1+T(s)} \tag{4.61}$$

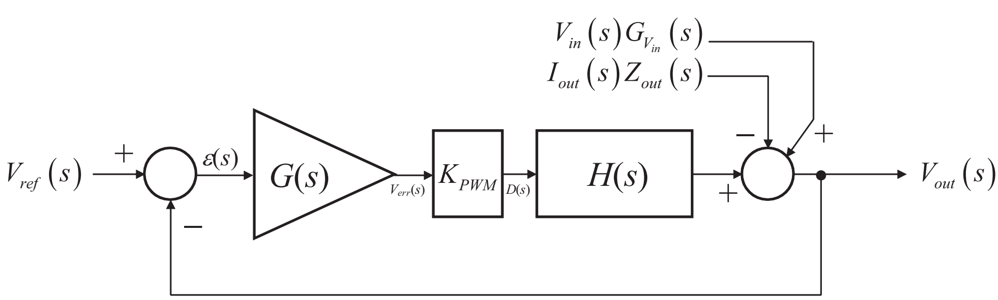



**Figure 4.19** A regulator operates with a fixed setpoint constantly fights the incoming perturbations.

This is the small-signal deviation obtained when a step load is applied to the output. As you can read, the output impedance plays the main role, together with the sensitivity function. The transfer function from $V_{ref}$ to $V_{out}$ is not pertinent in this case. Despite Bode predicting a good answer, we just missed the main point in our regulator example: the output impedance $Z_{out}$ fixes the transient response to a load step. Studying the transfer function $V_{out}(s)/V_{ref}(s)$ is simply not enough; we must always check the magnitude of $Z_{out}(s)$.

### 4.1.7 A Peaky Output Impedance Plot

With the help of the SPICE circuit from Figure 4.17, we can easily plot the output impedances in an open- or closed-loop condition. We just add a 1-A ac current source in parallel with the load, and we directly obtain the output impedance by plotting the output voltage vector. This is what Figure 4.20 shows you.

As you can see, despite the PID action, the output impedance natural peaking ($Z_{out,OL}$) is not tamed at all. Can we check the resulting quality factor from this plot? It is certainly difficult to measure it directly from the graph. However, by looking at Appendix 4B, we can see that a relationship links the quality factor $Q$ to the group delay $\tau_g$. To use it, we must first display the closed-loop output impedance argument and calculate the corresponding group delay. The results appear in Figure 4.21 and show a group delay of 1.015 ms. Applying (4.206) derived in Appendix 4B, we estimate the quality factor to a value of:

$$Q = \tau_g \pi f_0 = 1200 \times 1.015m \times 3.14159 = 3.82 \tag{4.62}$$

This is in good agreement with the measured value from the transient step in Figure 4.18. The difference is that (4.62) is obtained from a small-signal graph where the system is linear. Equation (4.59) comes from the transient response of a system perhaps operated in a nonlinear zone, thus potentially leading to corrupted results.

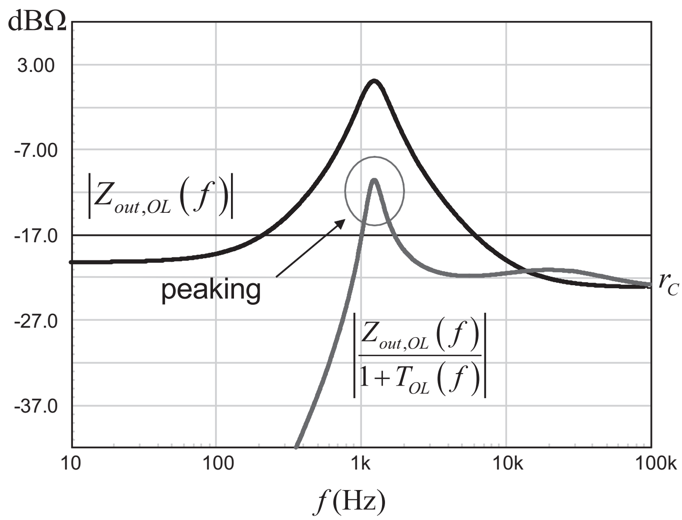



**Figure 4.20** The closed-loop output impedance is still peaky despite the PID presence.

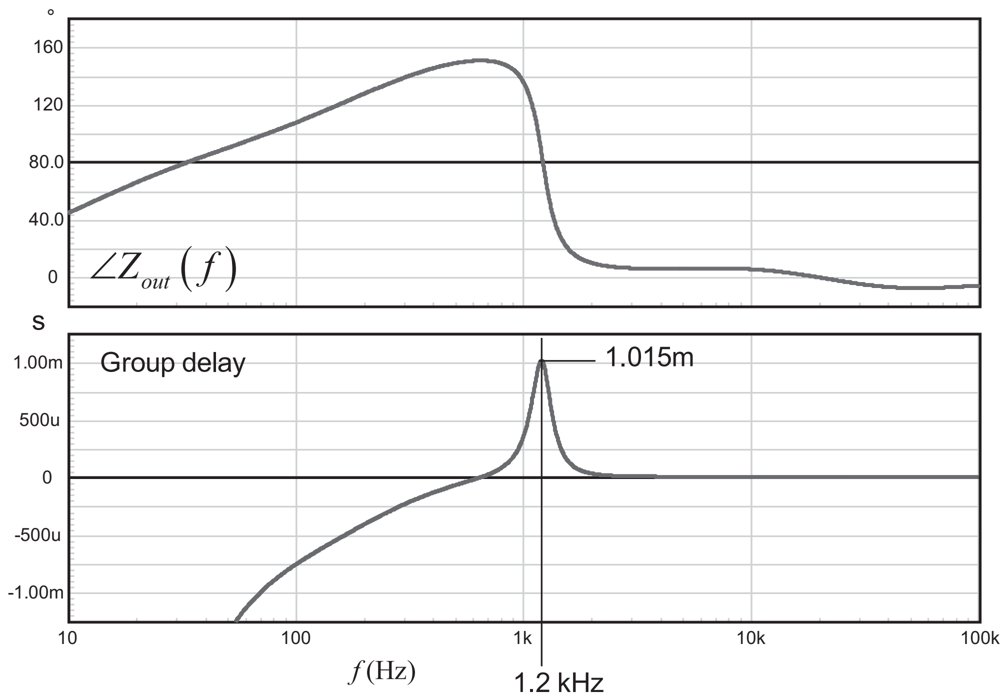



**Figure 4.21** By ac sweeping the output impedance and determining its group delay, we have a means to determine the quality factor.

These measurements confirm that the output impedance of our closed-loop converter is that of a second-order system featuring a $Q$ of 3.9. No wonder it gives an oscillatory response! We have compared the response of the PID-stabilized buck of Figure 4.17 with that of a *RLC* circuit also featuring a $Q$ of 3.9. Figure 4.22 shows that both resulting waveforms perfectly agree. The PID compensator we

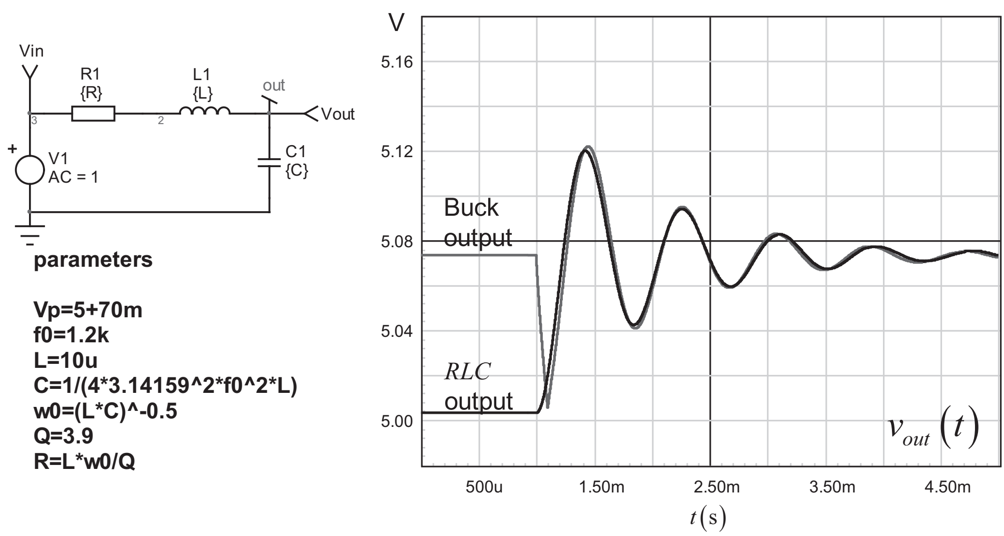



**Figure 4.22** The compensated buck behavior is similar to that of an undamped *RLC* network affected by a quality factor of 3.9.

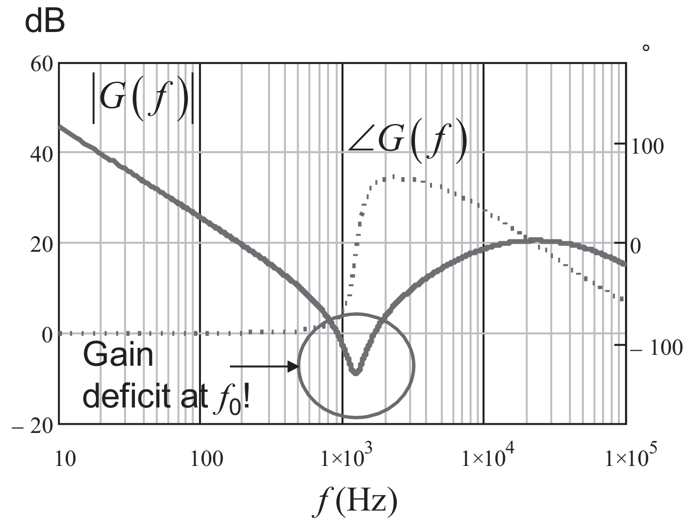



**Figure 4.23** In the compensator block, the presence of a conjugate pair of zero reduces the gain at resonance—the opposite of what should be done!

selected did not tame the naturally peaking open-loop impedance of the buck converter. Let's see why and how to remedy that.

From (4.61), we learned that the transient response of our switching converter directly depends on its closed-loop output impedance characteristics. To make them as low as possible, it is in our interest to grow the loop gain $T(s)$ so that the perturbations—the output current step, in our case—are efficiently rejected. To be more precise, we must ensure that sufficient gain exists at the resonant frequency so that efficient damping occurs. Unfortunately, in our PID compensator, to perfectly compensate the double pole at $f_0$, the equations system leads us to also place a double zero at this frequency via the compensator $G$. However, because of the quality factor greater than 0.5, the complex poles conjugate pair asked for a complex conjugate pair of zeros for a complete cancelation. If a complex conjugate pair of poles peaks, a complex conjugate pair of zeros works as a rejector: the gain dips at the resonance, whereas it would need to increase! This is the region we highlighted in Figure 4.23.

It is clear from this graph that a classical PID approach applied *stricto sensu* to the case of a buck converter does not give the right results (i.e., a fast, nonringing response). Let's see a more general method that will give better results.

## 4.2 Stabilizing the Converter with Poles-Zeros Placement

PID control with individual coefficients calculation is often used for the control of complex plants (e.g., by using Ziegler and Nichols experimental processes, as described in [4]). In switching or linear regulators designs, engineers rarely use the PID method we described. If computing the PID compensator individual coefficients leads to positioning poles and zeros as shown in (4.54) to (4.57), designers prefer to directly place these elements in order to tailor the system response to their needs. For instance, without going through a complex polynomial analysis as we did in the

PID example, you can simply select the crossover frequency of your choice and the exact phase margin you expect at this point by properly placing the poles and zeros. You can then shift one zero or one pole and see the effect on the response speed or the recovery time, for instance. At the end, you will end up with a configuration that a PID algorithm could have recommended, but the steps to get there, in the author's opinion, are faster and simpler.

### 4.2.1 A Simple Step-by-Step Technique

As a starting point, you need the plant transfer function $H(s)$ to learn about the system you want to stabilize. We have seen how to get it from the buck converter in a previous paragraph (e.g., by applying the method described in Figure 4.11). Other methods exist, such as analytical analysis (equation-based small-signal model), laboratory experiments with a network analyzer, or even a Ziegler-Nichols approximation (good results for well-damped systems). From this graph, you need two pieces of data: the plant magnitude and the argument at the selected crossover frequency, $|H(f_c)|$ and $\angle H(f_c)$. These elements will tell you how to shape the compensator frequency response so that you obtain the desired crossover frequency while phase and gain margins values are within the limits you have fixed. However, this is not enough. Observe the magnitude and phase plots to detect the presence of peaks or notches and make sure the selected crossover point is far away from a resonance or in a zone where the phase lag does not excessively degrade. Failure to damp resonating peaks leads to an oscillatory response as we experienced with the PID-compensated buck.

The compensator is the place where you will arrange poles, zeros, and gains/attenuations to shape the loop gain $T$. This compensator uses an active element, usually an operational amplifier—an op amp—but a lot of industrial projects can implement different types of amplifiers, such as the TL431 (open-collector self-contained op amp with a reference voltage), a shunt regulator (a kind of active Zener circuit), or even an operational transconductance amplifier (OTA). Very often, in industrial applications, these processing blocks transmit the error signal via an isolating device, an opto-coupler, adding complexity to the final transfer equation. We will cover all these structures in detail in dedicated chapters.

Despite the usage of different types of amplifiers, the compensator ac shaping process follows a common agenda:

1. Identify on the plant transfer function $H(s)$ the magnitude and argument at the crossover frequency $f_c$ you have selected.
2. In the compensator, place a pole at the origin to offer a high gain at dc and efficiently reject the perturbations. Input voltage and output current are perturbations in a switching or linear regulator. A high dc gain ensures a good audio or input rejection, while it also guarantees the lowest (theoretically null) dc static error on the output. This is the integrating block in our PID expression. This origin pole is found in most of compensators. However, we will later see a compensation scheme in which there is no origin pole. For instance, output impedance shaping for high-speed dc-dc regulators implements a pure proportional architecture without an origin pole.

3. As a complement to the origin pole, identify the position of poles, zeros, and gain/attenuation points to (a) force crossover at the selected frequency $f_c$, (b) correct the plant phase response by locally building phase boost, and (c) ensure the targeted open-loop phase margin. Depending on the needed correction, you can deal with three types of compensators, bringing a phase boost from 0° to 180°. They are designated as types 1, 2, and 3 and can be declined with op amps, TL431, and so on.
4. Once the compensator has been designed, plot the loop gain response $T(s)$ and verify that the crossover frequency associated with margins (phase and gain) are within acceptable limits. Check that sweeping the input voltage and the load does not degrade the margins. Do the same check with parasitic element changes, such as capacitor ESRs. Finally, a load step will tell you if the response looks like what you were expecting.

### 4.2.2 The Plant Transfer Function

Obtained by a SPICE simulation, two examples of plant transfer functions appear in Figure 4.24 and Figure 4.25. The first one is a damped current-mode flyback converter operated in continuous conduction mode (CCM). The first thing is to identify a crossover frequency $f_c$. We have seen some guidelines in a previous chapter where $f_c$ was chosen based on the transient response to a load step. In this example, we have selected 5 kHz. Looking at the graph, we do not see resonating peaks on the magnitude curve, and the phase gently stays above –90° up to a fairly high frequency. No hidden traps then and the 5 kHz number does not seem to be

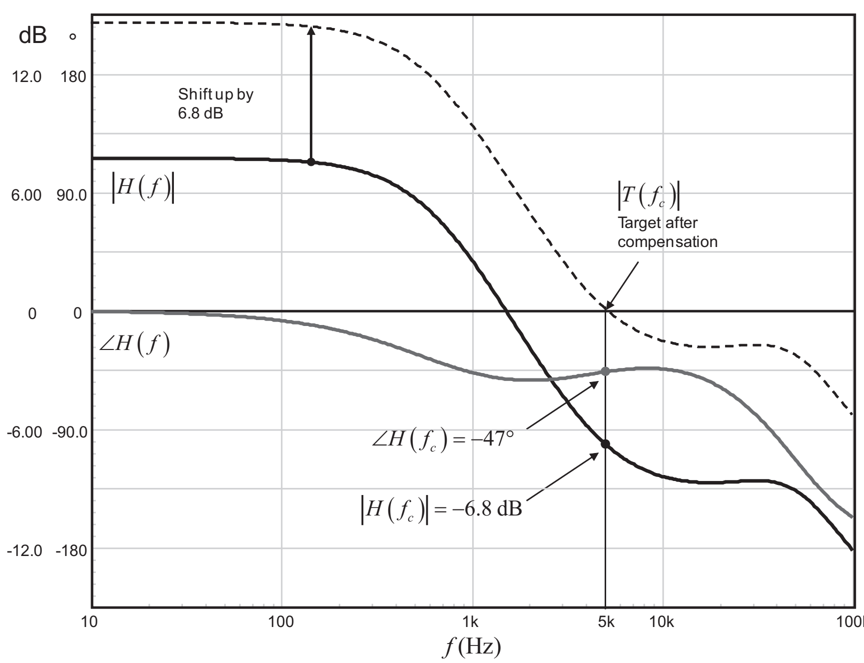



**Figure 4.24** This is a current-mode flyback CCM converter transfer function. To obtain a 5-kHz crossover frequency, $G$ must compensate a gain deficit at this frequency.

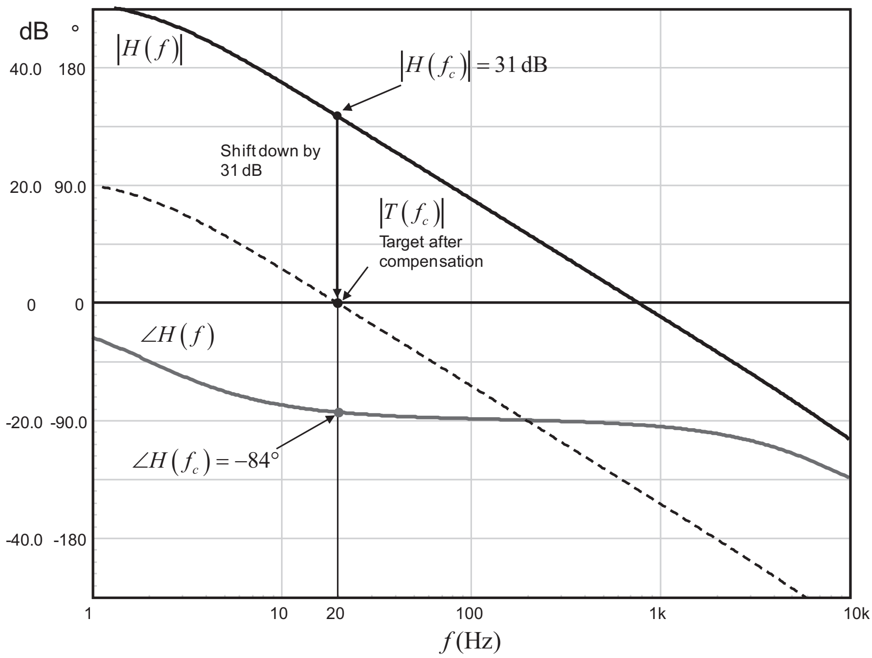



**Figure 4.25** With this transfer function of a BCM power factor correction boost converter, the designer must shape the compensator G to compensate an excess of gain at the selected 20-Hz crossover frequency.

a difficult goal to accomplish. What is the plant magnitude value at 5 kHz? From the graph, we read a gain of –6.8 dB. Since we want a 5-kHz crossover frequency, it means that the compensator magnitude $|G|$ at 5 kHz must exactly be 6.8 dB so that $|H(5\text{ kHz}) \cdot G(5\text{ kHz})|$, the open-loop gain $T$, is exactly 1 or 0 dB at this point. In this particular example, we will shift the curve up to compensate the gain deficit at the selected crossover point.

However, there are situations where you observe an excess of gain at crossover rather than a deficit. This is what happens in Figure 4.25 for a power factor corrector (PFC) operated in borderline conduction mode (BCM). The crossover frequency is usually selected at a low value to reject the 100/120-Hz output ripple that would pollute the error signal otherwise. A 20-Hz crossover frequency can be a number PFC designers deal with. From the graph, we observe a 31-dB gain excess at 20 Hz. Therefore, we must shift the curve down by 31 dB: the compensator magnitude $|G|$ at 20 Hz must be –31 dB so that $|H(20\text{ Hz}) \cdot G(20\text{ Hz})|$ is 1 or 0 dB.

### 4.2.3 Canceling the Static Error with an Integrator

We have learned that the loop gain phase lag at the crossover frequency $\angle T(f_c)$ must keep away from the 360° limit; otherwise, an oscillatory response with a pronounced overshoot is obtained. The distance to this limit is the phase margin noted $\varphi_m$. Very often, the phase margin represents a design target (e.g., 70° could be asked by your customer or your project manager). In fact, 90° is not an unusual target in space/military designs. Sometimes, conditional stability can be banned, and your customer will ask for bench measurements to show compliance with his demand. It is thus your duty as a design engineer to shape the compensator response and reach the crossover frequency goal together with other requirements such as phase margin. To obtain these numbers, we must first understand what a classical compensator is made of. As the lowest static error is usually wanted, an integrating block is necessary. Please note it is not always the case as sometimes a static error is accepted to get rid of the integrator block. As seen in the PID section, this block is identified in the compensator transfer function as the origin pole. An origin pole is a simple division by $s$ in the compensator transfer function:

$$\frac{V_{out}(s)}{V_{in}(s)} = G(s) = \frac{1}{s} \tag{4.63}$$

At dc (or at the origin of the $x$-axis if you prefer), when $s$ equals 0, the gain is simply infinite. With an infinite gain at dc, theory tells us that the static error (the dc or steady-state difference between the output and the setpoint) is canceled. In reality, there is a limit brought by the op amp open-loop gain $A_{OL}$ (80–90 dB, for instance). This limit will surely induce a static error but with high open-loop gain values, this error is usually negligible. As we said, some designers actually want a static error and do not place an origin pole. This is the case for high-speed dc-dc converters for motherboards, as we will later see in an example. In the case of our origin pole, the phase lag brought by (4.63) is computed as follows:

$$\lim_{\omega \to 0} \arg \frac{V_{out}(j\omega)}{V_{in}(j\omega)} = \lim_{\omega \to 0} \arg\left(-j\frac{1}{\omega}\right) = \arctan(-\infty) = -\frac{\pi}{2} \tag{4.64}$$

In this equation, we can see that an origin pole brings a permanent phase lag of 90°, independent from frequency. This is an important result: every time you identify a pole at the origin in a transfer function, you have a phase lag of 90° that will cumulate with that of other poles and zeros when present. Should you have two origin poles $\left(G(s) = \frac{1}{s^2}\right)$, the phase lag will be 180°.

You can see an integrator in Figure 4.26 made around a simple voltage-controlled voltage source. If we consider an infinite gain, the transfer function of such a configuration is simply

$$G(s) = -\frac{Z_f}{Z_i} = -\frac{\frac{1}{sC_1}}{R_1} = -\frac{1}{sR_1C_1} \tag{4.65}$$

If we consider the cutoff angular frequency $\omega_{po}$ to be $1/R_1C_1$, then (4.65) can be rewritten as

$$G(s) = -\frac{\omega_{po}}{s} \tag{4.66}$$

Replacing $s$ with $j\omega$, we can extract the magnitude of this expression:

$$|G(j\omega)| = \left|0 + j\frac{\omega_{po}}{\omega}\right| = \frac{\omega_{po}}{\omega} = \frac{f_{po}}{f} \tag{4.67}$$

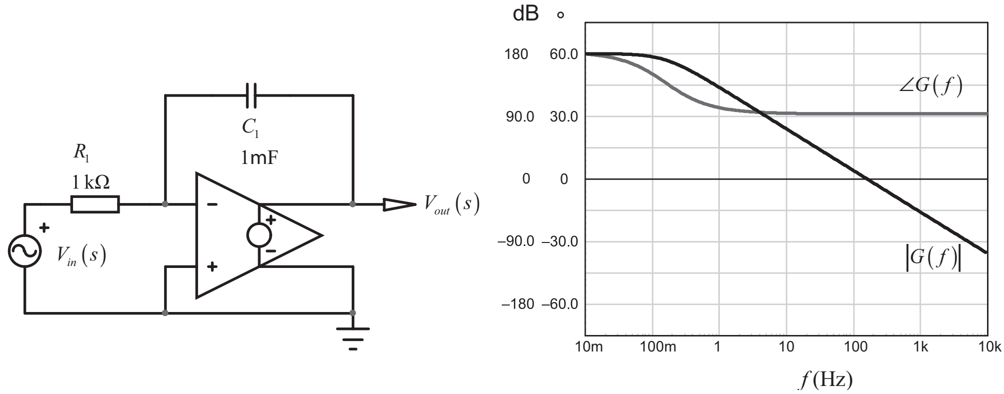



**Figure 4.26** A simple integrator built with an op amp in an inverting configuration brings a total phase lag of 270° or a phase lead of 90°.

The argument is obtained from (4.66); now considering the inverting sign brought by the op amp:

$$\arg G(j\omega) = \arg\left(0 + j\frac{\omega_{po}}{\omega}\right) = \tan^{-1}(\infty) = \frac{\pi}{2} \tag{4.68}$$

When $\omega$ approaches zero, the gain goes infinite. When $\omega$ reaches $\omega_{po}$, the so-called 0-dB crossover pole, the gain is 1 or 0 dB. However, in reality, the op amp exhibits a finite open-loop gain usually noted $A_{OL}$. If we now consider it, we show in Appendix 4D that the transfer function becomes

$$G(s) \approx \frac{A_{OL}}{1 + s/\omega_{p_1}} \tag{4.69}$$

where $A_{OL}$ is the open-loop gain of the op amp (1,000 or 60 dB in the magnitude graph) and $\omega_{p_1}$ defined by

$$\omega_{p_1} = \frac{\omega_{po}}{A_{OL}} \tag{4.70}$$

The magnitude of the integrator is computed by replacing $s$ with $j\omega$:

$$|G(j\omega)| = \frac{A_{OL}}{\sqrt{1 + \left(\dfrac{\omega}{\omega_{p_1}}\right)^2}} \tag{4.71}$$

In dc, when $s$ approaches 0, the integrator gain does not reach infinity but is clamped to $A_{OL}$. This is what is displayed in the right side of Figure 4.26. In the same figure, you can observe a phase lead of 90° or a lag of 270°. Both angles are equal, as we can add or subtract $\pm 2k\pi$ to any angle without affecting its value. If we subtract 360° from 90°, we obtain –270°, which physically makes better sense. In effect, a negative phase, by convention, illustrates a delay: in the time domain, the output of an op amp can appear only later than its input stimulus. We will therefore use this latter approach to manipulate the various stability limits: an inverting integrator will delay the phase by 270°. The simulator, however, applies a mathematical treatment to the numbers it manipulates and ignores whether 360° must be added or subtracted. You just need to keep in mind that both –270° and 90° refer to the same angle when analyzing phase charts.

These tangent calculations can sometimes be tricky; this is why I wrote a dedicated appendix. Tangent lovers, please proceed to Appendix 4C.

### 4.2.4 Adjusting the Gain with the Integrator: The Type 1

This is the notion of 0-dB crossover pole already tackled in a previous chapter. In (4.67), the term $f_{po}$ relates to a frequency at which the gain magnitude is 1 or 0 dB. By changing its value, you have a means to modify the gain you need at another frequency point (e.g., the crossover frequency). Suppose you will use a simple integrator to close your converter loop. Assume the crossover frequency is 20 Hz and the gain excess at this point is 23 dB. You must adjust $f_{po}$ so that a 20-Hz modulation brings an attenuation of –23 dB or $10^{-\frac{23}{20}} = 70.8\ m$. In other words, you have to solve

$$\frac{f_{po}}{20} = 0.0708 \tag{4.72}$$

It implies placing the 0-dB crossover pole at

$$f_{po} = 20 \times 0.0708 \approx 14 \text{ Hz} \tag{4.73}$$

This is typically a compensation scheme that could fit power factor correction circuits where the crossover frequency is purposely put low to avoid reacting on the output ripple. Its schematic appears in Figure 4.27. In the technical literature, it is referred as a type 1 compensator: no phase boost, an origin pole for a static large gain, and a certain amount of gain or attenuation at a selected frequency. As already indicated, this circuit introduces a permanent phase lag of 270°.

The test fixture in Figure 4.27 implements an auto-bias circuit made of the voltage-controlled voltage source $E_1$. Its purpose is to bias the error amplifier at a level where it is operated in a linear way, far away from its lower or upper stops. You adjust this level via the source $V_2$. It is put to around 2.5 V in this example. $E_1$ thus biases the divider network at the calculated level. Since we automated our calculations for a 400 V output, the bias point on $R_1$ right terminal confirms it. The capacitor value is obtained from (4.65) and is simply

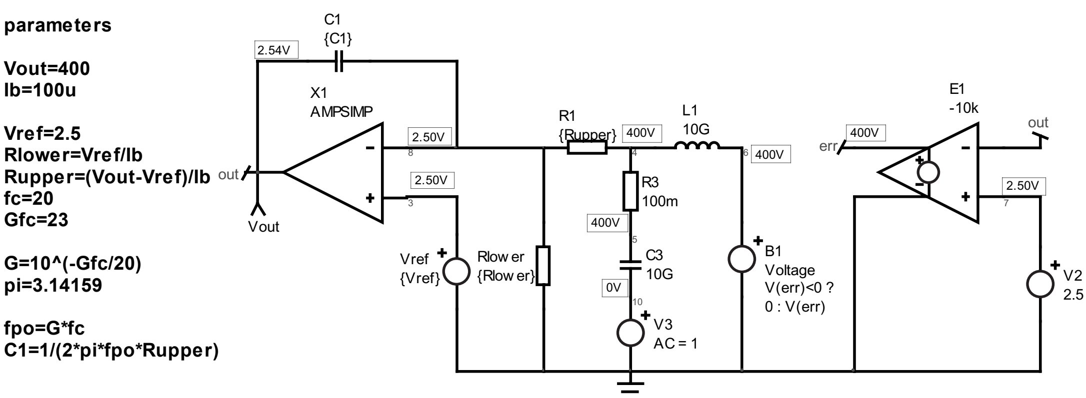



Figure 4.27 This circuit automates a type 1 compensator where the divider network is calculated based on bias current requirements. The cutoff frequency is adjusted through capacitor $C_1$.

$$C_1 = \frac{1}{2\pi R_{upper} f_{po}} \tag{4.74}$$

With an upper resistor calculated at 4 MΩ, the capacitor value is 28 nF or 33 nF for the next normalized value. The ac plot from Figure 4.28 confirms the validity of our calculations. The phase is fixed to 90° or –270°, no phase boost at all.

In this example, the 0-dB crossover pole position is a means to adjust the gain to any value at the selected frequency. We will see in the following examples how this can be coupled to other expressions.

### 4.2.5 Locally Boosting the Phase at Crossover

We know that an origin pole is needed if we want the lowest static error on the output. However, as observed in Figure 4.26, the phase brought by this arrangement is absolutely flat. It means that if you add an integrator to an existing design, you simply bring an additional phase lag of 270° and an infinite gain at dc—that is all. Should you need to improve the phase at some particular point (crossover, for instance), the integrator alone is of no help at all. A poor phase margin is due to an excessive phase lag of the transmission chain around the crossover frequency. To improve the situation, we must counteract the phase lag by adding some phase lead somewhere in the chain: the compensator $G$ is the place to do it.

The upper side of Figure 4.29 shows an argument example of the plant we want to stabilize. At the selected 4-kHz crossover frequency, the plant lags by 71°.

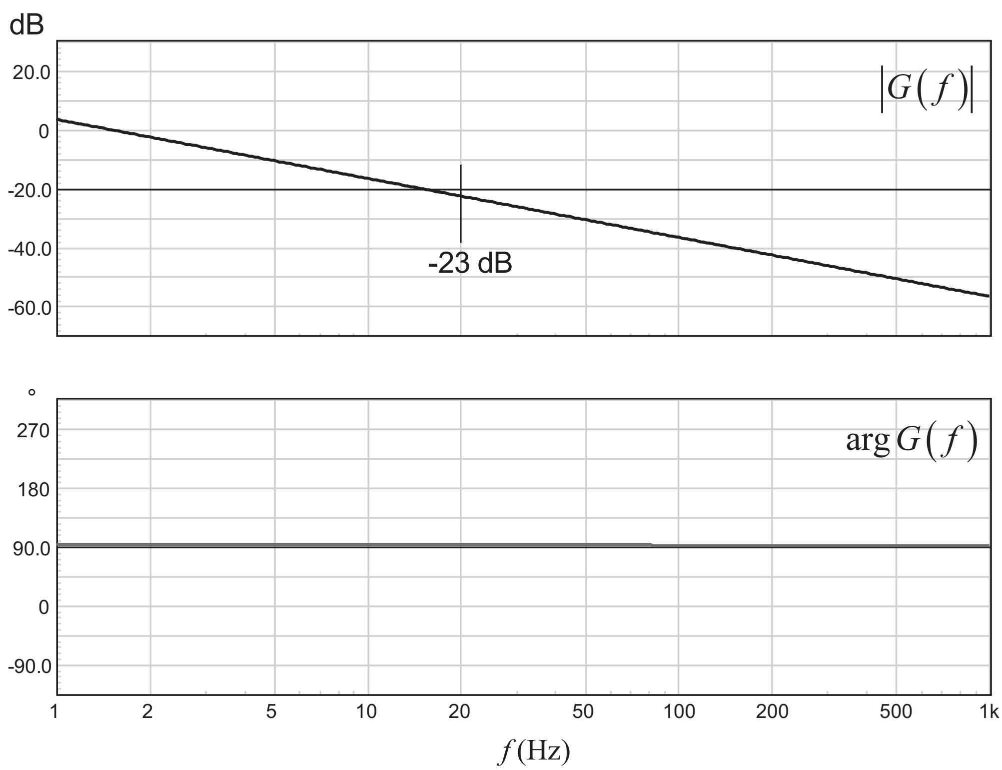



**Figure 4.28** The ac plot confirms an attenuation of 23 dB at the selected 20-Hz frequency.

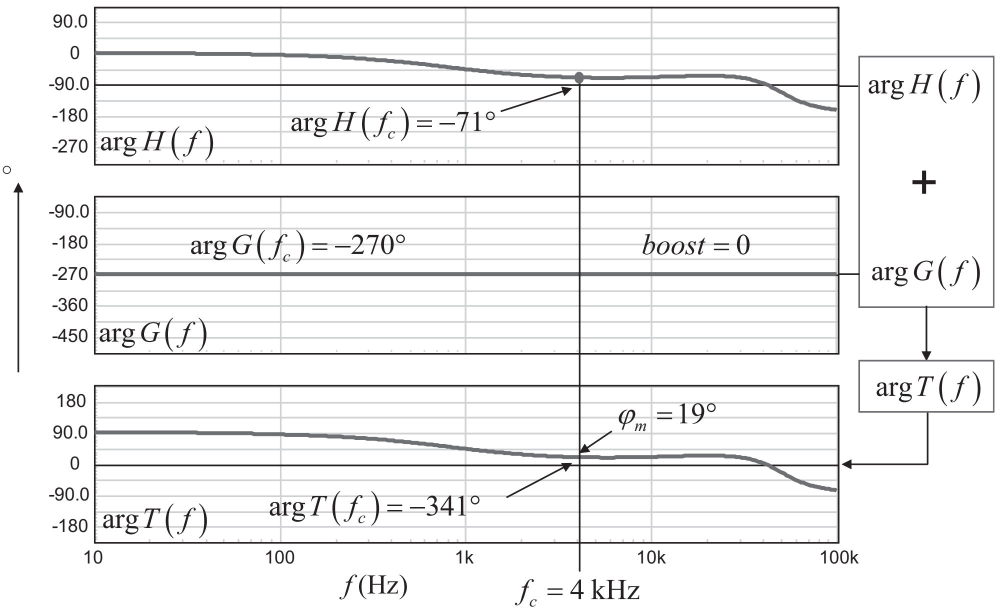



**Figure 4.29** Boosting the phase means locally decreasing the plant phase lag by providing phase lead at the crossover point: this is the so-called phase boost.

To obtain the smallest static error, we insert a pole at the origin. We know from the previous paragraph that the addition of an origin pole via an inverting integrator lags the phase by –270°. Therefore, once this integrator is put in series with the plant, we have a total phase lag of

$$\arg T(4\text{ kHz}) = -71° - 270° = -341° \tag{4.75}$$

If we would keep the integrator and close the loop, we would obtain a phase margin equal to

$$\varphi_m = 360° - 341° = 19° \tag{4.76}$$

This is what Figure 4.29 shows you in its lowest part as you simply sum the plant argument to that of the compensator. Please note that the phase margin is measured as the distance between the loop gain argument and the –360° limit or 0° as –360° is a complete turn. We measure 19°.

If our design target is 70°, we are far from this number. To correct this figure, we need to reduce the loop gain phase lag at crossover so that its distance to the –360° line (or 0°) is equal to our phase margin. The necessary increase, or *phase boost*, is part of the following equation:

$$\arg T(f_c) - 270° + boost = -360° + \varphi_m \tag{4.77}$$

Solving for the *boost* value, we have

$$boost = -360° + \varphi_m - \arg T(f_c) + 270° = \varphi_m - \arg T(f_c) - 90° \tag{4.78}$$

In our example, we would need a phase boost of

$$boost = \varphi_m - \arg T(f_c) - 90° = 70 + 71 - 90 = 51° \tag{4.79}$$

It means that the compensator $G$ must be arranged so that its argument at crossover (4 kHz) is no longer –270° (as the sole integrator would provide) but should be

$$\arg G(4\text{ kHz}) = -270° + 51° = -219° \tag{4.80}$$

Once the compensator is tailored to exhibit this phase, new simulation results showed in Figure 4.30 confirm the calculations and the proper phase margin: we have a robust design.

### 4.2.6 Placing Poles and Zeros to Create Phase Boost

In the integrator transfer function, the argument is flat to –270° along the frequency axis. To change this argument and actually reduce the phase lag at some point, what can we do? We can place a zero in the compensator. We know that the +1-slope of a zero is accompanied by a phase starting from 0 at dc and linearly increasing to 90°. Indeed, this phase lead, well placed on the frequency axis, will bend the integrator phase lag and make it decrease as wished. A zero is defined by the following expression:

$$G(s) = 1 + \frac{s}{\omega_z} \tag{4.81}$$

The magnitude of such transfer function is found by replacing $s$ by $j\omega$:

$$|G(j\omega)| = \left|1 + s\frac{\omega}{\omega_z}\right| = \sqrt{1 + \left(\frac{\omega}{\omega_z}\right)^2} \tag{4.82}$$

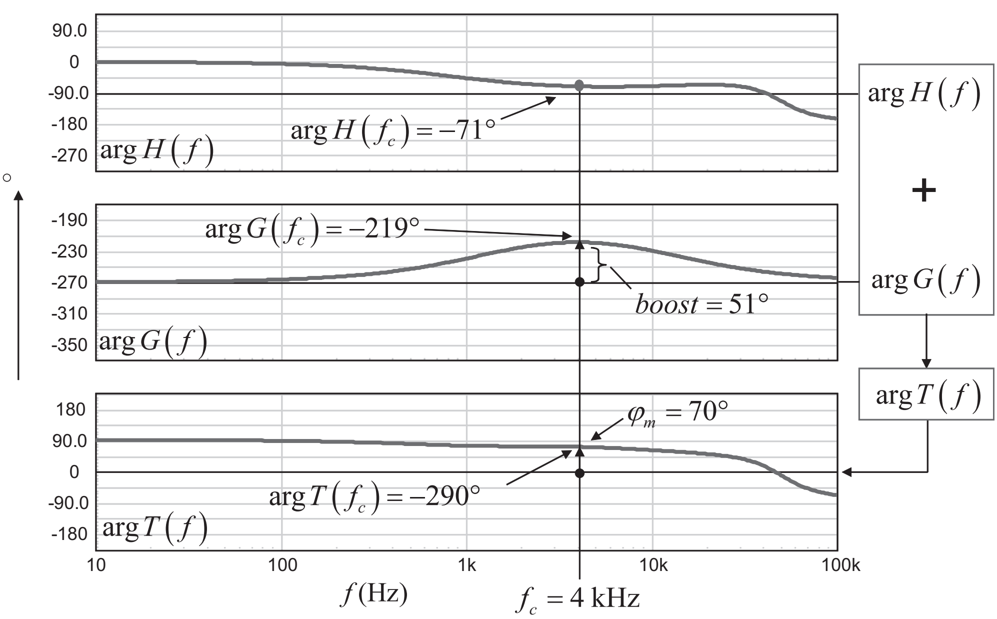



**Figure 4.30** By locally decreasing the phase lag or boosting the phase at the crossover, we obtain the required phase margin.

The zero argument is derived as

$$\arg G(j\omega) = \tan^{-1}\left(\frac{\omega}{\omega_z}\right) \tag{4.83}$$

If we plot (4.82) and (4.83), Figure 4.31 displays the ac response of a zero, placed at 1.4 kHz in this example.

As observed on the graph, the zero is able to bring a phase lead that will oppose the phase lag of the plant, to the benefit of stability. However, we want the phase boost to be calibrated to a certain amount: 51° in the design example of Figure 4.30. Therefore, at some point on the frequency axis, when the wanted boost is obtained, the phase lead must be brought to zero. Also, the zero alone would bring the compensator gain to infinity as frequency increases. We limit its action by adding a pole.

The expression of a pole is the following one:

$$G(s) = \frac{1}{1 + s/\omega_p} \tag{4.84}$$

To compute its magnitude, we replace $s$ with $j\omega$:

$$|G(j\omega)| = \frac{1}{\sqrt{1 + \left(\frac{\omega}{\omega_p}\right)^2}} \tag{4.85}$$

The pole argument is derived as

$$\arg G(j\omega) = \arg(1) - \arg\left(1 + j\frac{\omega}{\omega_p}\right) = -\tan^{-1}\left(\frac{\omega}{\omega_p}\right) \tag{4.86}$$

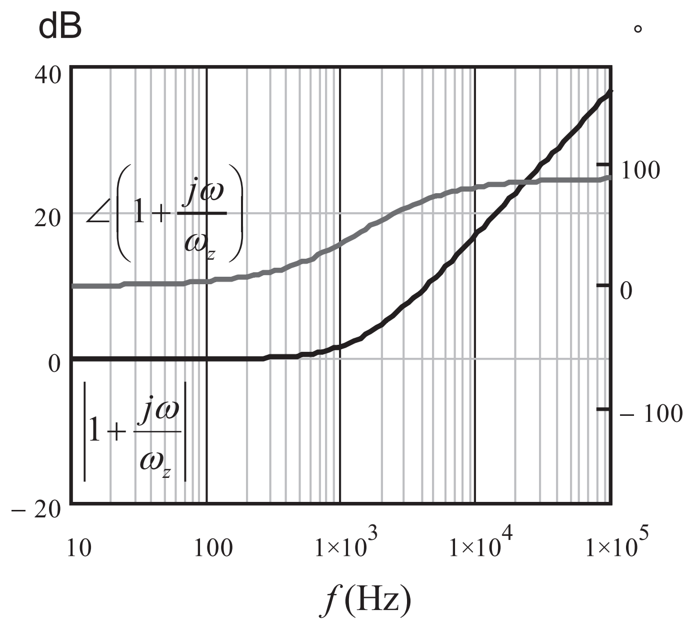



**Figure 4.31** The ac response of a zero placed at 1.4 kHz: the phase starts to increase before the zero position and continues toward 90°.

If we now plot (4.85) and (4.86), we obtain the drawing of Figure 4.32. We can see a phase starting from 0 and linearly decreasing toward the –90° asymptote.

In the first case, with the zero, the phase is only growing positively (phase lead), while the pole only brings negative phase (phase lag). As we want a certain amount of phase lead—actually, our phase boost—we can combine a zero and a pole together to exactly calibrate the amount of phase boost we want.

When we associate a pole and a zero, also called a single pole/zero pair, the total argument before either one starts to kick in is 0°. If the zero is placed before the pole, the phase starts to lead as we progress along the frequency axis. This what we showed in Figure 4.31. If the pole now enters in action, its phase lag counteracts that of the zero and the phase starts to drop (Figure 4.32). Ultimately, as the zero argument asymptote is 90° and that of the pole –90°, the phase contribution of the combined pole-zero is 0°. The combined transfer function of the pole-zero combination is as follows:

$$G(s) = \frac{1 + s/\omega_z}{1 + s/\omega_p} \tag{4.87}$$

The magnitude is extracted by replacing $s$ with $j\omega$ and is the quotient of the numerator magnitude by that of the denominator:

$$\left|G(j\omega)\right| = \frac{\sqrt{1 + \left(\dfrac{\omega}{\omega_z}\right)^2}}{\sqrt{1 + \left(\dfrac{\omega}{\omega_p}\right)^2}} \tag{4.88}$$

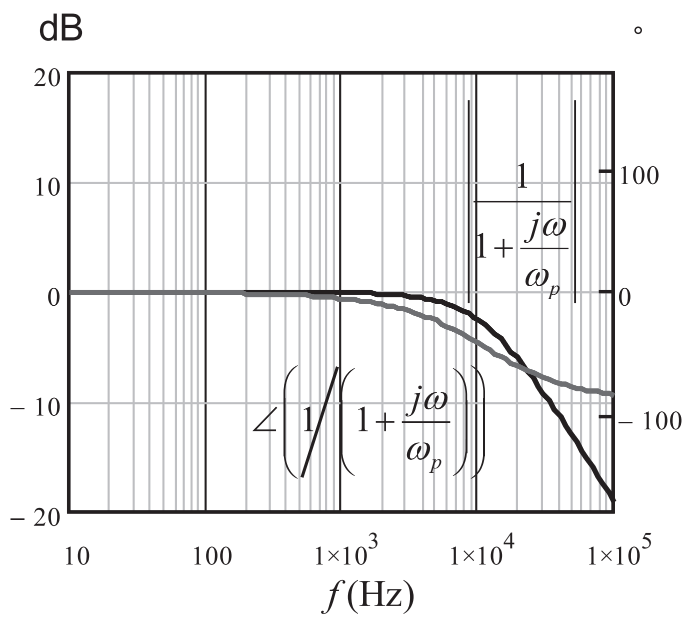



**Figure 4.32** The ac response of a pole placed at 11.4 kHz: the phase starts to lag before the pole position and decreases toward –90°.

The argument is simply the difference between (4.83) and (4.86):

$$\arg G(j\omega) = \tan^{-1}\left(\frac{\omega}{\omega_z}\right) - \tan^{-1}\left(\frac{\omega}{\omega_p}\right) \tag{4.89}$$

We have plotted (4.87) in Figure 4.33 where the zero was placed at 1.4 kHz and the pole at 11.4 kHz. As expected, the combined action of the pole and the zero creates a localized phase lead at a certain frequency. If the pole and the zero are coincident, they perfectly neutralize each other: flat 0-dB magnitude over frequency and 0° contribution. As you split both pole and zero, you start to create phase lead in between. When they are split apart so that the pole kicks in after the zero argument has reached its asymptotic value, the boost is maximum to 90°. In Figure 4.34, we have placed a pole at $x$ times the zero position and looked at the total argument for various values of $x$:

$$f_p = f_z \cdot x \tag{4.90}$$

You can see the boost building up as the pole moves away from the zero.

Now, where to position the pole and the zero so that the phase boost occurs exactly at the crossover frequency? In other words, when we place a pole and a zero, at what frequency does the phase boost peak, before it returns to zero? We can easily obtain that answer by deriving (4.89) and looking at the frequency that cancels the result:

$$\frac{d}{df}\left(\tan^{-1}\left(\frac{f}{f_z}\right) - \tan^{-1}\left(\frac{f}{f_p}\right)\right) = \frac{1}{f_z\left(\frac{f^2}{f_z^2}+1\right)} - \frac{1}{f_p\left(\frac{f^2}{f_p^2}+1\right)} = 0 \tag{4.91}$$

Solving this equation shows that the phase boost peaks at the geometric means of the pole/zero frequencies:

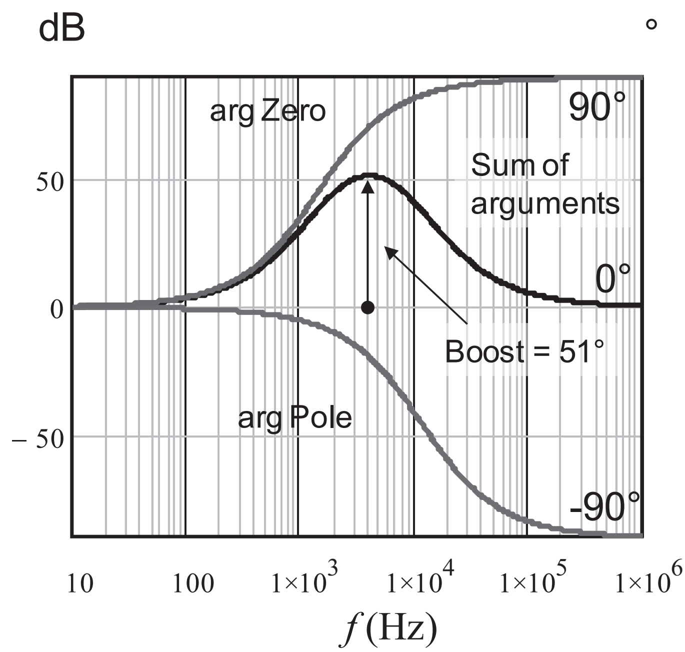



**Figure 4.33** When a pole and a zero are combined, we create a calibrated amount of phase lead.

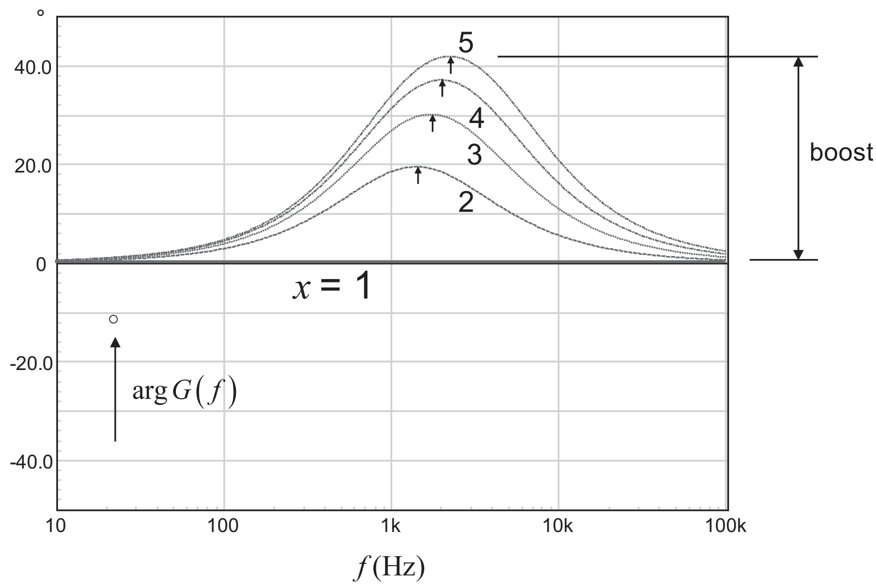



**Figure 4.34** By splitting the pole and the zero apart, you can adjust the phase lead to your needs.

$$f_{max} = \sqrt{f_p f_z} \tag{4.92}$$

With our zero at 1.4 kHz and the pole at 11.4 kHz, the peak occurs at:

$$f_{max} = \sqrt{1.4k \times 11.4k} = 4 \text{ kHz} \tag{4.93}$$

as confirmed by Figure 4.33.

### 4.2.7 Create Phase Boost up to 90° with a Single Pole/Zero Pair

We have learned that combining a pole and a zero provides a phase boost adjustable from 0° to 90°. As we want this boost to occur at the crossover frequency to locally compensate a phase margin deficit, we need equations to properly place the pole and the zero. As both are currently unknown, we actually need two equations:

$$boost = \tan^{-1}\left(\frac{f_c}{f_z}\right) - \tan^{-1}\left(\frac{f_c}{f_p}\right) \tag{4.94}$$

and

$$f_c = \sqrt{f_p f_z} \tag{4.95}$$

From (4.95), we can extract the zero definition:

$$f_z = \frac{f_c^2}{f_p} \tag{4.96}$$

We can substitute this definition in (4.94):

$$boost = \tan^{-1}\left(\frac{f_p}{f_c}\right) - \tan^{-1}\left(\frac{f_c}{f_p}\right) \tag{4.97}$$

To help solve this equation, we can introduce a coefficient $k$ and write $k = f_p/f_c$ and rewrite (4.97):

$$boost = \tan^{-1}(k) - \tan^{-1}\left(\frac{1}{k}\right) \tag{4.98}$$

We must now remember a trigonometric formula involving the arctangent:

$$\tan^{-1}(k) + \tan^{-1}\left(\frac{1}{k}\right) = \frac{\pi}{2} \tag{4.99}$$

We can extract $\tan^{-1}(1/k)$ from this equation and reinject the result into (4.98). Once it is done, we obtain

$$2\tan^{-1}(k) = boost + \frac{\pi}{2} \tag{4.100}$$

Solving for $k$, we simply have

$$k = \tan\left(\frac{boost}{2} + \frac{\pi}{4}\right) \tag{4.101}$$

This expression is nothing else than the "$k$ factor," introduced by Dean Venable in the 1990s [5].

Knowing that $k = f_p/f_c$, we obtain our pole definition:

$$f_p = k \cdot f_c = \tan\left(\frac{boost}{2} + \frac{\pi}{4}\right) f_c \tag{4.102}$$

And the zero is derived from (4.96):

$$f_z = \frac{f_c}{k} = \frac{f_c}{\tan\left(\frac{boost}{2} + \frac{\pi}{4}\right)} \tag{4.103}$$

When the crossover frequency is identified together with the necessary phase boost, you can select the pole and the zero position using these derivations. The limit given by the method lies in the placement of the crossover frequency at the exact geometric means between the pole and the zero. What if you need to place the pole to counteract an identified zero in the transfer function? In that case, you can no longer apply the previous synopsis. No need to panic; (4.94) is still valid to place the remaining pole or zero at the right position. Let's assume you have to place a zero $f_z$ at 800 Hz and the crossover frequency $f_c$ is 8 kHz. As the needed phase boost is 55°, where do you place the pole? You simply extract $f_p$ from the rearranged equation (4.94), and you are done:

$$\tan^{-1}\left(\frac{f_c}{f_p}\right) = \tan^{-1}\left(\frac{f_c}{f_z}\right) - boost \tag{4.104}$$

To solve this equation, we can use the following trigonometric relationship:

$$\tan(A - B) = \frac{\tan A - \tan B}{1 + \tan A \tan B} \tag{4.105}$$

If we apply the formula to (4.104), we have an equation in the form of

$$\frac{f_c}{f_p} = \frac{\dfrac{f_c}{f_z} - \tan(boost)}{1 + \dfrac{f_c}{f_z}\tan(boost)} \tag{4.106}$$

Solving for $f_p$ gives us

$$f_p = \frac{f_z f_c + \tan(boost) f_c^2}{f_c - f_z \tan(boost)} \tag{4.107}$$

If we place the zero at 800 Hz with a crossover frequency at 8 kHz, and we need a 55° phase boost, then the pole should be placed at

$$f_p = \frac{800 \times 8k + \tan(55°) \times 8k^2}{8k - 800 \cdot \tan(55°)} = 14.2 \text{ kHz} \tag{4.108}$$

The result is confirmed by the plot that appears in Figure 4.35. Should you fix the pole instead, the zero would be located at the following value:

$$f_z = \frac{f_c f_p - \tan(boost) f_c^2}{f_c + f_p \tan(boost)} \tag{4.109}$$

### 4.2.8 Mid-Band Gain Adjustment with the Single Pole/Zero Pair: The Type 2

We know how to place the single pole/zero pair to meet the phase boost requirement at crossover. How do we now adjust the compensator to provide the needed gain or attenuation at crossover? We simply marry the single pole/zero pair with an origin pole such as the one described by (4.66). In that case, (4.87) becomes:

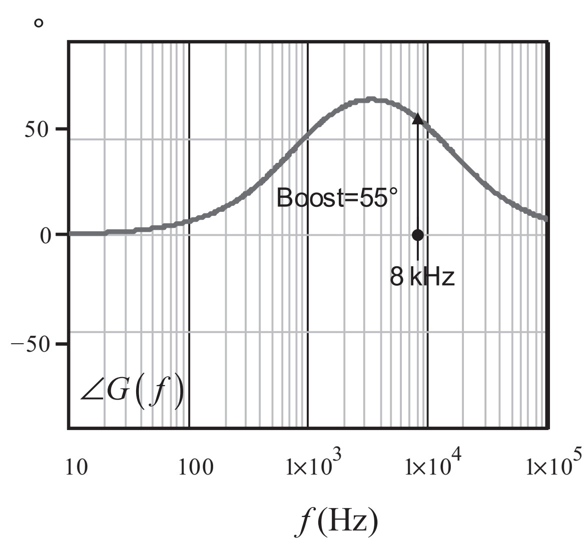



**Figure 4.35** The crossover frequency is no longer in the geometric means of the pole/zero position, but the phase boost is what we wanted.

$$G(s) = -\frac{1+s/\omega_z}{\frac{s}{\omega_{po}}(1+s/\omega_p)} \tag{4.110}$$

Please note the "–" sign presence as we now deal with an inverting op amp-based compensator.

This equation does not fit the transfer function format described in Chapter 2. Let's rework it by factoring $s/\omega_z$ in the numerator:

$$G(s) = -\frac{s}{\omega_z}\frac{\left(\frac{\omega_z}{s}+1\right)}{\frac{s}{\omega_{po}}(1+s/\omega_p)} \tag{4.111}$$

The 0-dB crossover pole $\omega_{po}$ can go to the numerator, and a simplification by $s$ is possible:

$$G(s) = -\frac{s}{\omega_z}\frac{\omega_{po}}{s}\frac{\left(\frac{\omega_z}{s}+1\right)}{(1+s/\omega_p)} = -\frac{\omega_{po}}{\omega_z}\frac{1+\omega_z/s}{1+s/\omega_p} = -G_0\frac{1+\omega_z/s}{1+s/\omega_p} \tag{4.112}$$

In this expression, the term $G_0$ is called the mid-band gain and is equal to

$$G_0 = \frac{\omega_{po}}{\omega_z} \tag{4.113}$$

As $\omega_z$ is fixed by the amount of needed phase boost, you will place $\omega_{po}$ depending on the desired gain/attenuation you want at crossover. We have built a type 2 compensator. The design flow is rather simple: you select the pole and zero based on the needed phase boost at the crossover frequency, and you adjust the gain or attenuation at $f_c$ using (4.113). Figure 4.36 shows how it works on an asymptotic construction.

A type 2 can be assembled in a lot of different ways, whether you use op amps, TL431, shunt regulators, or so on. The classical type studied in textbooks is made around an op amp and appears in Figure 4.37.

We won't spend more time on this structure, as Chapter 5 is dedicated to it and goes into the details of numerous configurations. Let's familiarize ourselves with the type 2 in a quick design example.

### 4.2.9 Design Example with a Type 2

Let's assume we want to stabilize a power supply that has a gain deficit of 18 dB at a 5-kHz selected crossover frequency. The necessary phase boost is 68°. From (4.101) and (4.102), we place a pole at

$$f_p = \tan\left(\frac{boost}{2}+\frac{\pi}{4}\right)f_c = \tan\left(\frac{68°}{2}+45°\right)\times 5k = 25.7\text{ kHz} \tag{4.114}$$

The zero is placed at

$$f_z = \frac{f_c}{\tan\left(\frac{boost}{2}+\frac{\pi}{4}\right)} = \frac{5k}{\tan\left(\frac{68°}{2}+45°\right)} = 972\text{ Hz} \tag{4.115}$$

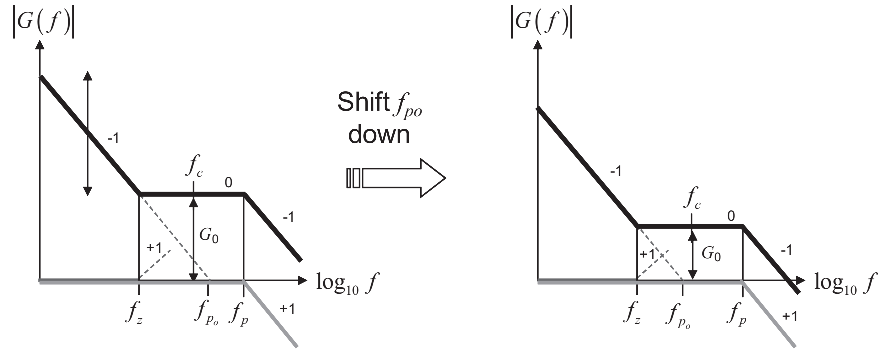



**Figure 4.36** Adjusting the 0-dB crossover pole position (while the pole/zero pair is untouched) gives a way to tweak the mid-band gain $G_0$. Here the mid-band gain reduction is achieved by shifting down the 0-dB crossover pole.

Now the 0-dB crossover pole: we need an 18-dB gain at 5 kHz. From (4.113) and (4.115), it must be placed at

$$f_{po} = f_z \cdot 10^{G_0/20} = 972 \times 8 \approx 7.8\ \text{kHz} \tag{4.116}$$

If we now asymptotically construct the resulting response, we have the diagram of Figure 4.38.

We have simulated this type 2 configuration, and Figure 4.39 plots the ac response typical of that type of architecture.

The calculation methodology for Figure 4.37 components is treated in detail in Chapter 5 and will not be detailed here.

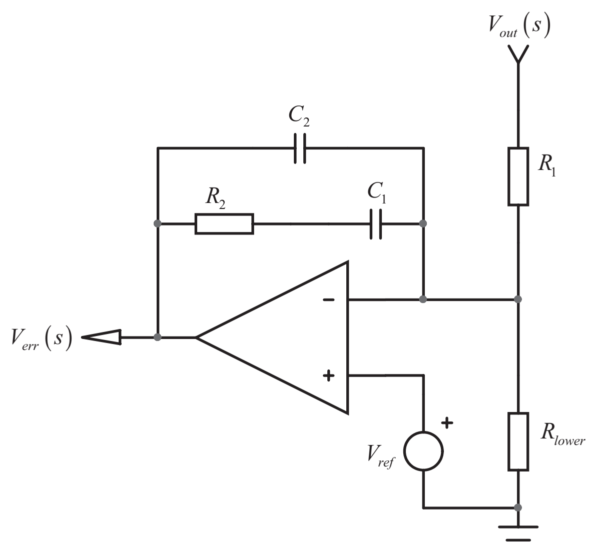



**Figure 4.37** A type 2 compensator made around an op amp.

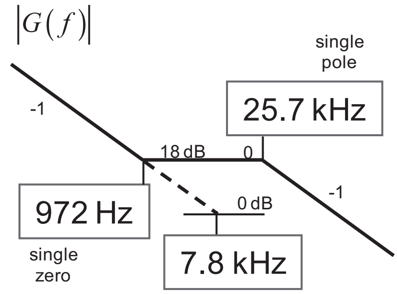



**Figure 4.38** The asymptotical response of the designed type 2 compensator.

### 4.2.10 Create Phase Boost up to 180° with a Double Pole/Zero Pair

The maximum boost brought by a pole/zero combination is 90°. This is easily seen in Figure 4.33, and it occurs when the pole and zero are split far away from each other. There are designs, however, that require more than a 90° phase boost. In that case, you will have to combine two zeros and two poles, also called a double pole/zero pair. The principle is the same as with the single pole/zero pair except that the double zero phase asymptote is now 180° and that of the double pole is –180°. When they are far away from each other, the maximum phase boost becomes 180°.

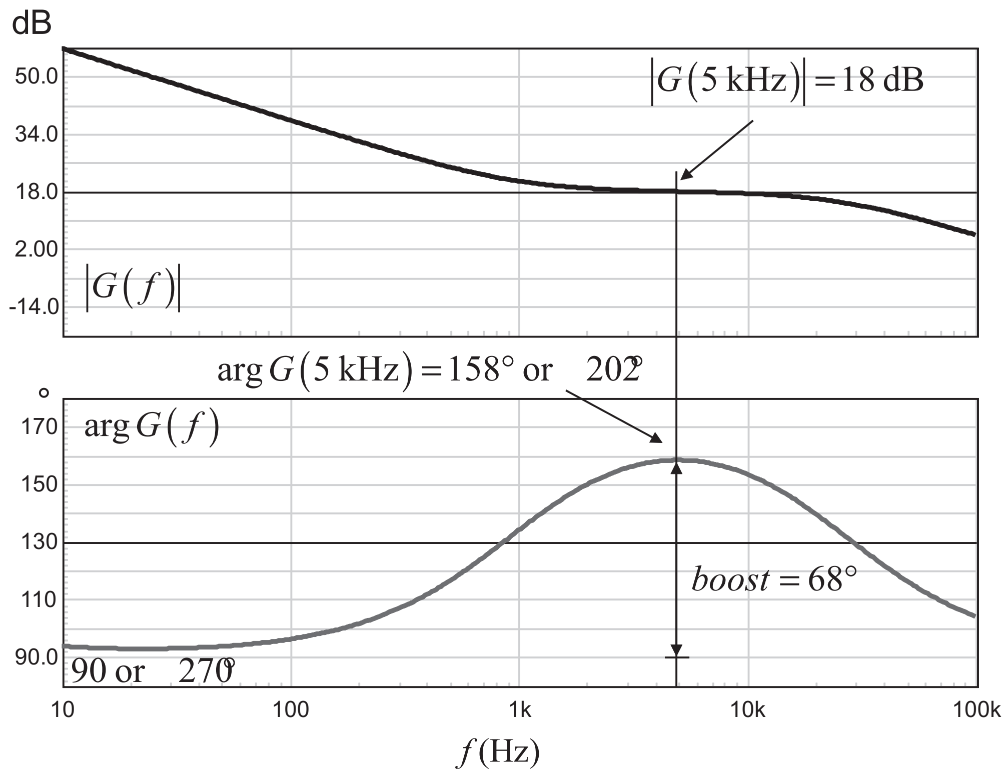



**Figure 4.39** A SPICE simulation of the designed type 2 confirms the targets are reached.

The combined transfer function of the double pole-zero combination is as follows:

$$G(s) = \frac{(1+s/\omega_{z_1})(1+s/\omega_{z_2})}{(1+s/\omega_{p_1})(1+s/\omega_{p_2})} \tag{4.117}$$

The magnitude is extracted by replacing $s$ with $j\omega$ and is the quotient of the numerator magnitude by that of the denominator:

$$|G(j\omega)| = \frac{\sqrt{1+\left(\frac{\omega}{\omega_{z_1}}\right)^2}\sqrt{1+\left(\frac{\omega}{\omega_{z_2}}\right)^2}}{\sqrt{1+\left(\frac{\omega}{\omega_{p_1}}\right)^2}\sqrt{1+\left(\frac{\omega}{\omega_{p_2}}\right)^2}} \tag{4.118}$$

The argument is simply the difference between numerator argument and that of the denominator:

$$\arg G(j\omega) = \tan^{-1}\left(\frac{\omega}{\omega_{z_1}}\right) + \tan^{-1}\left(\frac{\omega}{\omega_{z_2}}\right) - \tan^{-1}\left(\frac{\omega}{\omega_{p_1}}\right) - \tan^{-1}\left(\frac{\omega}{\omega_{p_2}}\right) \tag{4.119}$$

If we assume that the two zeros and the two poles are coincident (e.g., a double pole and a double zero), then (4.91) can be reformulated with $f$ rather than $\omega$:

$$\frac{d}{df}\left(2\tan^{-1}\left(\frac{f}{f_{z1,2}}\right) - 2\tan^{-1}\left(\frac{f}{f_{p1,2}}\right)\right) = \frac{2}{f_z\left(\frac{f^2}{f_{z1,2}^2}+1\right)} - \frac{2}{f_p\left(\frac{f^2}{f_{p1,2}^2}+1\right)} = 0 \tag{4.120}$$

Solving for $f$ shows that the phase boost also peaks at the geometric means of the double pole-double zero location:

$$f_{\max\,boost} = \sqrt{f_{z1,2}\,f_{p1,2}} \tag{4.121}$$

As we did for the pole/zero pair, we can calculate the phase boost brought by the combination. The phase boost is obtained from (4.119), which is rearranged considering the double pair:

$$boost = 2\tan^{-1}\left(\frac{f_c}{f_{z1,2}}\right) - 2\tan^{-1}\left(\frac{f_c}{f_{p1,2}}\right) = 2\left(\tan^{-1}\left(\frac{f_c}{f_{z1,2}}\right) - \tan^{-1}\left(\frac{f_c}{f_{p1,2}}\right)\right) \tag{4.122}$$

This expression has already been solved in (4.99) and (4.100). The difference is simply the factor 2 in the right term, leading to the final expression of $k$ for the double pole/zero pair:

$$k = \tan\left(\frac{boost}{4} + \frac{\pi}{4}\right) \tag{4.123}$$

The definition derived in [5] features a squared right term in (4.123), probably chosen for convenient expressions using $k$ in the compensator components calculations. The poles and zeros positions with respect to $k$ are, however, similar:

$$f_{p_{1,2}} = k \cdot f_c \tag{4.124}$$

$$f_{z_{1,2}} = \frac{f_c}{k} \tag{4.125}$$

In the previous example, a single pole/zero pair brought a phase boost of 51° at 8 kHz. We have now placed a double zero at 1.4 kHz and a double pole at 11.4 kHz. As confirmed by Figure 4.40, the phase boost is doubled to 102°.

There are some cases where the double zero is fixed, and you must adjust one pole to match the needed phase boost. It can, for instance, be the case for a voltage-mode buck converter where the designer usually places the double zero $f_{z_{1,2}}$ at the $LC$ filter resonant frequency and a pole $f_{p_2}$ at half the switching frequency for noise immunity. The remaining pole is adjusted to meet the phase boost. Extracted from (4.119), its position is simply

$$f_{p_1} = \frac{f_c}{\tan\left(2\tan^{-1}\left(\frac{f_c}{f_{z_{1,2}}}\right) - boost - \tan^{-1}\left(\frac{f_c}{f_{p_2}}\right)\right)} \tag{4.126}$$

Assume a crossover frequency of 10 kHz, a double zero located at 1.2 kHz, and a high-frequency pole placed at 50 kHz. Suppose we need a phase boost of 130°; then, if we follow (4.126), this second pole must be placed at

$$f_{p_1} = \frac{10k}{\tan\left(2\tan^{-1}\left(\frac{10k}{1.2k}\right) - 120° - \tan^{-1}\left(\frac{10k}{50k}\right)\right)} = 14.3\text{ kHz} \tag{4.127}$$

We have entered (4.119) in Mathcad and plotted the argument response based on the previous poles/zeros locations. The result appears in Figure 4.41.

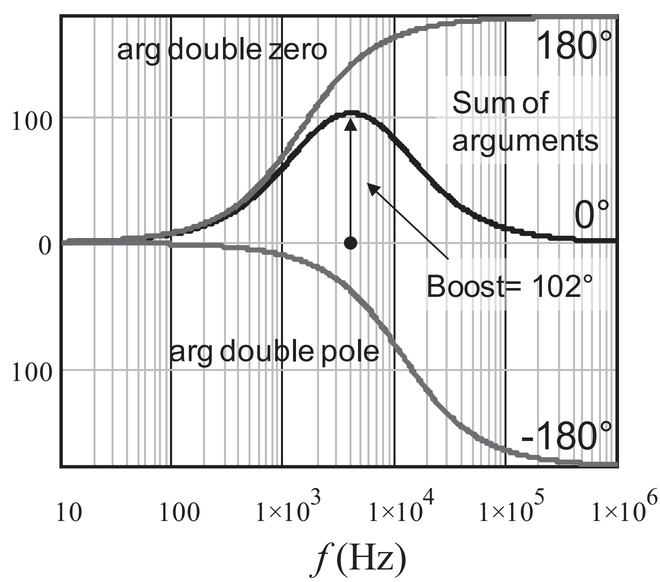



**Figure 4.40** The double pole/zero pair doubles the phase boost of that given by a single pole/zero pair.

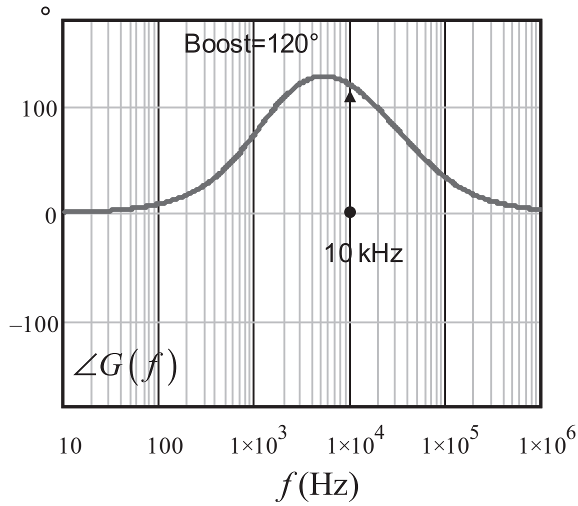



**Figure 4.41** The phase boost at 10 kHz confirms the target at 120°.

### 4.2.11 Mid-Band Gain Adjustment with the Double Pole/Zero Pair: The Type 3

The mid-band gain adjustment in the type 2 compensator was made using the 0-dB crossover pole position. In our double pole/zero pair, the same method is used: an origin pole is inserted in (4.117) to create the type 3 compensator:

$$G(s) = -\frac{(1+s/\omega_{z1})(1+s/\omega_{z2})}{\frac{s}{\omega_{po}}(1+s/\omega_{p1})(1+s/\omega_{p2})} \tag{4.128}$$

Please note the "–" sign presence as we now deal with an inverting op amp-based compensator.

This equation does not fit the transfer function format described in Chapter 2. Let's rework it by factoring $s/\omega_{z1}$ in the numerator:

$$G(s) = -\frac{s}{\omega_{z1}}\frac{\left(\frac{\omega_{z1}}{s}+1\right)(1+s/\omega_{z2})}{\frac{s}{\omega_{po}}(1+s/\omega_{p1})(1+s/\omega_{p2})} \tag{4.129}$$

The 0-dB crossover pole $\omega_{po}$ can go to the numerator and a simplification by $s$ is possible:

$$G(s) = -\frac{\omega_{po}}{s}\frac{s}{\omega_{z1}}\frac{\left(\frac{\omega_{z1}}{s}+1\right)(1+s/\omega_{z2})}{(1+s/\omega_{p1})(1+s/\omega_{p2})} = -G_0\frac{\left(\frac{\omega_{z1}}{s}+1\right)(1+s/\omega_{z2})}{(1+s/\omega_{p1})(1+s/\omega_{p2})} \tag{4.130}$$

in which

$$G_0 = \frac{\omega_{po}}{\omega_{z1}} \tag{4.131}$$

Figure 4.42 shows an asymptotic construction of a type 3 magnitude.

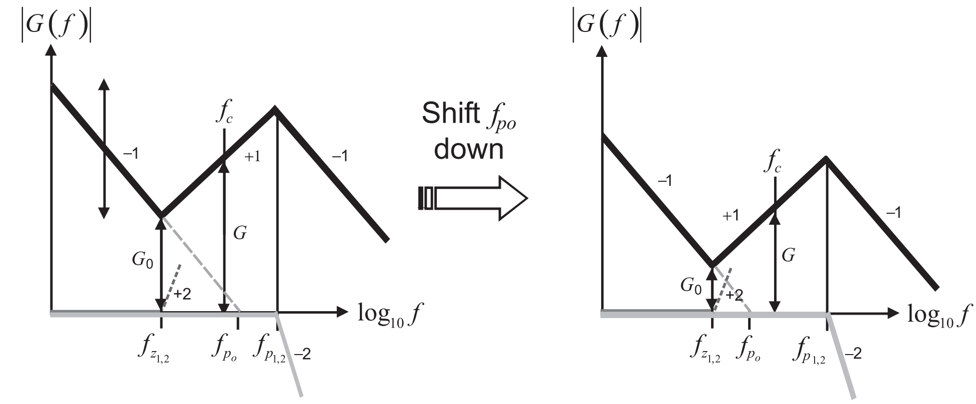



**Figure 4.42** By changing the position of the 0-dB crossover pole, you have a means to adjust the mid-band gain value to crossover at the right frequency.

As $\omega_{z1}$ is fixed by the amount of needed phase boost, you will place $\omega_{po}$ depending on the required gain/attenuation you want at crossover. We have built a type 3 compensator. The design flow is rather simple: you select the poles and zeros based on the needed phase boost at the crossover frequency, and you adjust the gain or attenuation at $f_c$. Unfortunately, (4.131) cannot be used alone to calculate the 0-dB crossover pole position. Why? Because, as you can see in Figure 4.42, the crossover occurs after the two zeros have kicked in, in the middle of a +1 slope. The case differs from that of the type 2 where the slope was unchanged (0 slope) at the point the zero was positioned. The gain $G$ at this position shown in Figure 4.42 is not $G_0$ and must account for the position of the double pole/zero pair. The formula derives from (4.130):

$$\omega_{po} = G \cdot \omega_{z1} \frac{\sqrt{1+\left(\frac{\omega_c}{\omega_{p1}}\right)^2}\sqrt{1+\left(\frac{\omega_c}{\omega_{p2}}\right)^2}}{\sqrt{1+\left(\frac{\omega_{z1}}{\omega_c}\right)^2}\sqrt{1+\left(\frac{\omega_c}{\omega_{z2}}\right)^2}} \tag{4.132}$$

If we consider a double coincident poles/zeros pair, the formula becomes

$$\omega_{po} = G \cdot \omega_{z1,2} \frac{\left(\omega_{p_{1,2}}^2 + \omega_c^2\right)}{\omega_{p_{1,2}}^2 \sqrt{\left(\frac{\omega_{z1,2}}{\omega_c}\right)^2 + 1}\sqrt{\left(\frac{\omega_c}{\omega_{z1,2}}\right)^2 + 1}} \tag{4.133}$$

In this expression, $G$ is the wanted gain or attenuation at crossover.

A type 3 can be assembled in a lot of different ways, regardless of whether you use op amps, TL431, shunt regulators, or so on. The classical type studied in textbooks is made around an op amp and appears in Figure 4.43.

The calculation of the components values appears in Chapter 5, dedicated entirely to the op amp architectures, and will not be repeated here.

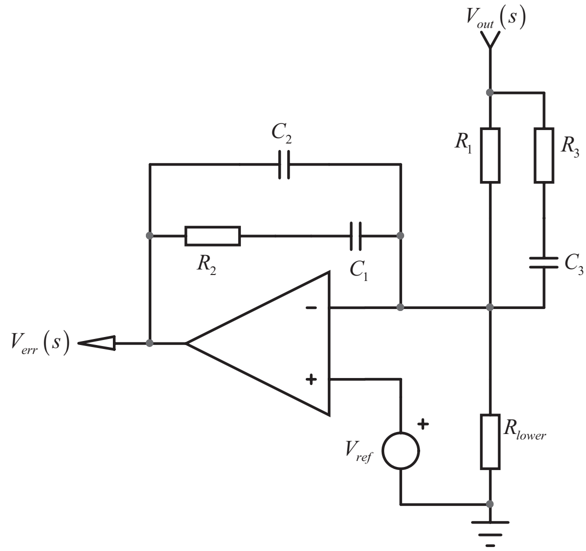



**Figure 4.43** A type 3 compensator uses the same basis as a type 2 to which an *RC* network is added in parallel with the upper resistor $R_1$.

### 4.2.12 Design Example with a Type 3

In this example, we assume a converter whose crossover frequency must be set to 5 kHz. At this frequency, the plant shows a gain deficit of 10 dB. The required phase boost is 158°. From the formula given in (4.124), we can calculate the position of the double pole:

$$f_{p_{1,2}} = \tan\left(\frac{boost}{4} + \frac{\pi}{4}\right) f_c = \tan\left(\frac{158°}{4} + 45°\right) \times 5k \approx 52 \text{ kHz} \tag{4.134}$$

The double zero position comes easily with (4.125):

$$f_{z_{1,2}} = \frac{f_c}{\tan\left(\frac{boost}{4} + \frac{\pi}{4}\right)} = \frac{5k}{\tan\left(\frac{158°}{4} + 45°\right)} \approx 480 \text{ Hz} \tag{4.135}$$

The gain $G$ at 5 kHz must be 10 dB. Applying (4.132) or (4.133) will tell us to position the 0-dB crossover pole at the following value:

$$\begin{aligned} f_{po} &= G \cdot f_{z_{1,2}} \frac{\left(f_{p_{1,2}}^2 + f_c^2\right)}{f_{p_{1,2}}^2 \sqrt{\left(\frac{f_{z_{1,2}}}{f_c}\right)^2 + 1} \sqrt{\left(\frac{f_c}{f_{z_{1,2}}}\right)^2 + 1}} \\ &= 10^{\frac{10}{20}} \times 480 \times \frac{52k^2 + 5k^2}{52k^2 \times \sqrt{1 + \left(\frac{480}{5k}\right)^2} \times \sqrt{1 + \left(\frac{5k}{480}\right)^2}} \approx 146 \text{ Hz} \end{aligned} \tag{4.136}$$

The asymptotic construction of the compensator is given in Figure 4.44.

We simulated this compensator and the ac response is shown in Figure 4.45. It confirms the calculations we made.

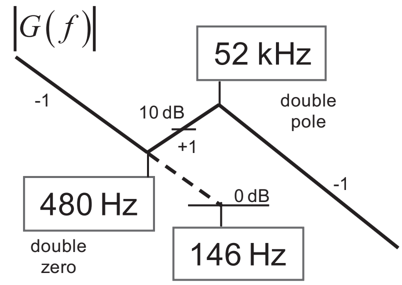



**Figure 4.44** The asymptotic construction of the type 3 shows the positions of the poles and zeros pairs.

### 4.2.13 Selecting the Right Compensator Type

Figure 4.46 depicts the possible architectures with their respective ac responses. You select the compensator type based on the converter you have to stabilize.

- Type 1: as its phase boost is zero, it can be used only in converters where the plant phase lag is minimal at the selected crossover frequency. Assume your plant exhibits a phase lag of 40° at the crossover frequency. Then, if you add another –270° from the type 1, you end with a total phase lag of 310°, giving a phase margin of 50°. However, as with any integral term, it will bring a response with overshoot that can sometimes be too severe. This type is widely used in PFC applications.
- Type 2: this is the most popular structure in current-mode control ac-dc or dc-dc converters. With a phase boost up to 90°, it perfectly suits the compensation needs of current-mode converters such as flyback, forward, boost, and buck-boost. It will also work for voltage-mode control types of converters,

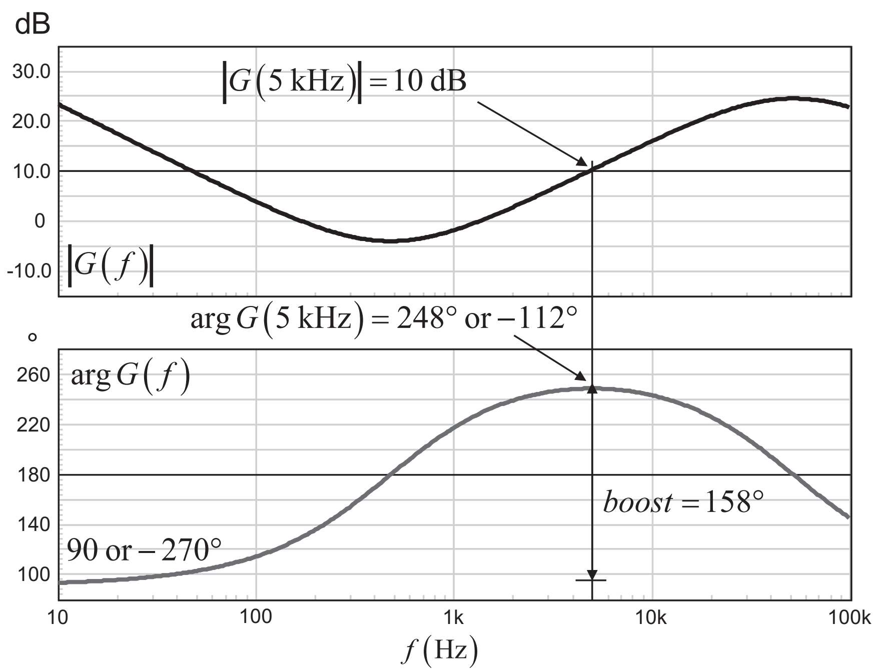



**Figure 4.45** The SPICE simulation of the compensator shows the right gain at 5 kHz and the correct phase boost.

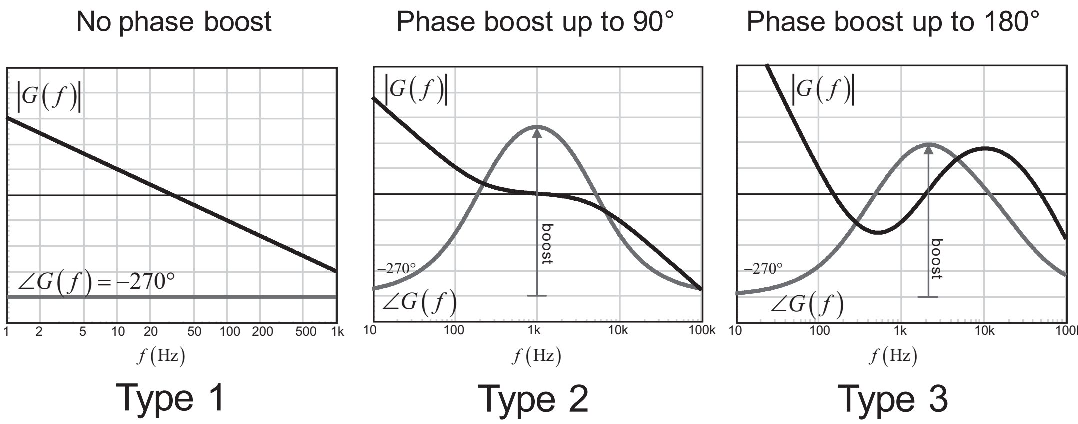



**Figure 4.46** This is a brief summary of the possible architectures with their respective ac responses.

but the limited boost will restrict its usage to discontinuous mode only. The type 2 is sometimes declined in 2a or 2b, building PI or filtered-proportional compensators, respectively. They are covered in Chapter 5.

- Type 3: with its phase boost capability up to 180°, the type 3 is often used in voltage-mode or direct duty ratio converters. Easily declined in op amp-based architectures, it becomes more complicated to build with OTA- or TL431-based compensators.

### 4.2.14 The Type 3 at Work with a Buck Converter

Now that we have seen how to place poles and zeros independently from tweaking individual PID parameters, it is time to apply the technique to the buck converter already presented in Figure 4.11. The main problem with the PID compensator example was that the cancellation of the resonating double poles led to the placement of a conjugate zeros pair, giving rise to a gain decrease in the compensator at exactly the resonant frequency (the notch in Figure 4.23). If the control-to-output response was good (see Figure 4.16), the perturbation rejection showed an oscillatory response because of an undamped output impedance. To ensure proper damping of the output impedance, it is important to always verify that sufficient gain exists at the resonant frequency (if there is one in the plant function, of course) so that the resulting closed-loop output impedance quality factor is less than one. With a voltage-mode converter, the classical compensation method works as follows:

1. Select a crossover frequency $f_c$ that is at least three to five times away from the resonating peak $f_0$ (never before the peak or a similar problem would occur by gain deficiency at resonance).
2. Place a real zero pair at the resonance frequency. This time, the zeros are real and not conjugate. We will see that these zeros can be split to increase the stability in light load conditions.
3. If the ESR-linked zero appears before crossover, neutralize it as we did with the PID by placing a pole right at its location. If the zero appears far away from the bandwidth, simply place the pole at half of the switching frequency. This first pole can then be moved to adjust the phase margin at the wanted value if necessary.

4. To force gain decrease at high frequency and ensure gain margin exists, place a second pole sufficiently high so that its presence does not hamper phase margin. It is usually placed at half of the switching frequency.

The converter transfer function in Figure 4.12 shows a resonant frequency at 1.2 kHz. To respect what we said from the previous point 1, we have to select a crossover frequency beyond 5 kHz. Let us adopt 10 kHz in this example, without considering undershoot specifications for the sake of simplicity. As the phase lag from the plant is not far from 180°, we are going to need an amount of phase boost greater than 90° if we want a decent phase margin. This is the role of the type 3 compensator we have already described.

The transfer function of such a compensator is as follows:

$$G(s) = -G_0 \frac{\left(1+\frac{\omega_{z_1}}{s}\right)\left(1+\frac{s}{\omega_{z_2}}\right)}{\left(1+\frac{s}{\omega_{p_1}}\right)\left(1+\frac{s}{\omega_{p_2}}\right)} \tag{4.137}$$

in which $G_0 = \frac{\omega_{po}}{\omega_{z_1}}$.

To apply the compensation strategy to our voltage-mode buck converter, we will place the double zeros at 1.2 kHz and will adjust the first pole around the plant zero position—as already recommended by (4.56) as a matter of fact—to tweak the phase margin to our target. With a 100-kHz operating frequency, the second pole will be arbitrarily placed at 50 kHz. This pole ensures further gain decrease at high frequencies and improves noise immunity. Now, where do we place the 0-dB crossover pole? Its position depends on the selected crossover frequency and the plant gain (or deficiency) at this frequency: $\omega_{po}$ will be placed to exactly force crossover at $f_c$. The principle is to shift up (or down) the plant magnitude $H$ at $f_c$ plot by the compensator magnitude $G$ at $f_c$ so that $|G(f_c)H(f_c)| = 1$. We have seen that in Figure 4.24 and Figure 4.25.

To force crossover at 10 kHz, we look at the plant transfer function $H(s)$, unveiled in Figure 4.14. From this curve, we can extract the gain deficiency at 10 kHz: around –20 dB. The phase lag is also extracted to be in the vicinity of 130°. We can also precisely derive these values from (4.34), for example, in an automated sheet:

$$|H(10k)| = H_0 \frac{\sqrt{1+\left(\frac{f_c}{f_{z_1}}\right)}}{\sqrt{\left(1-\frac{f_c^2}{f_0^2}\right)^2+\left(\frac{f_c}{f_0 Q}\right)^2}}$$

$$= \frac{10}{2.5}\frac{\sqrt{1+\left(\frac{10k}{10.3k}\right)}}{\sqrt{\left(1-\frac{10k^2}{1.24k^2}\right)^2+\left(\frac{10k}{1.24k\times 1.45}\right)^2}} = 0.108 \text{ or } -19.3\text{ dB} \tag{4.138}$$

$$\angle H(10k) = \tan^{-1}\left(\frac{f}{f_{z_1}}\right) - \tan^{-1}\left[\frac{f_c}{f_0 Q}\frac{1}{\left(1-\frac{f_c^2}{f_0^2}\right)}\right]$$

$$= \tan^{-1}\left(\frac{10k}{10.3k}\right) - \tan^{-1}\left[\frac{10k}{1.24k \times 1.45}\frac{1}{\left(1-\frac{10k^2}{1.24k^2}\right)}\right] = -134° \quad (4.139)$$

Since the plant attenuation at 10 kHz is –19.3 dB, the compensator $G$ must provide an amplification of exactly 19.3 dB at this frequency. As we want a phase margin of 70° at 10 kHz, what phase boost must we design the compensator for? We will use (4.78) for that purpose:

$$boost = \varphi_m - \arg T(f_c) - 90° = 70 + 134 - 90 = 114° \quad (4.140)$$

Following the compensation strategy unveiled a few lines earlier, we are going to place a double zero at the $LC$ network resonant frequency (1.2 kHz):

$$f_{z_1} = f_{z_2} = 1.2 \text{ kHz} \quad (4.141)$$

One pole will be placed at 50 kHz for noise immunity purposes:

$$f_{p_2} = 50 \text{ kHz} \quad (4.142)$$

The remaining pole can be placed straight at the ESR zero (10.3 kHz), and you will check if the resulting phase margin suits your needs. You can also compute the position of this pole to exactly meet the 114° phase boost suggested by (4.140). If we stick to the second option, we use the boost formula given by (4.119):

$$\arg G(f_c) = boost = \tan^{-1}\left(\frac{f_c}{f_{z_1}}\right) + \tan^{-1}\left(\frac{f_c}{f_{z_2}}\right) - \tan^{-1}\left(\frac{f_c}{f_{p_1}}\right) - \tan^{-1}\left(\frac{f_c}{f_{p_2}}\right) \quad (4.143)$$

$$\tan^{-1}\left(\frac{10k}{f_{p_1}}\right) = 2\tan^{-1}\left(\frac{10k}{1.2k}\right) - \tan^{-1}\left(\frac{f_c}{f_{p_2}}\right) - 114° = 166.3° - 11.3° - 114° = 41° \quad (4.144)$$

From which we can obtain the pole position:

$$f_{p_1} = \frac{f_c}{\tan(41°)} = \frac{10k}{0.87} = 11.5 \text{ kHz} \quad (4.145)$$

We are going to use the template we already tested in Figure 4.17. This time, the PID parameters are tweaked to place the previous real poles and zeros we have derived. The 0-dB crossover pole is computed in the integral term $k_i$, which naturally depends on the wanted gain at crossover, 19.3 dB in our case. In this particular case, as we have split the poles, (4.136) needs to be updated to account for this fact:

$$f_{po} = G \cdot f_{z_1} \frac{\sqrt{\left(\frac{f_c}{f_{p_1}}\right)^2 + 1}\sqrt{\left(\frac{f_c}{f_{p_2}}\right)^2 + 1}}{\sqrt{\left(\frac{f_{z_1}}{f_c}\right)^2 + 1}\sqrt{\left(\frac{f_c}{f_{z_2}}\right)^2 + 1}}$$

$$= 10^{\frac{20}{20}} \times 1.2k \times \frac{\sqrt{1+\left(\frac{10k}{11.3k}\right)^2} \times \sqrt{1+\left(\frac{10k}{50k}\right)^2}}{\sqrt{1+\left(\frac{1.2k}{10k}\right)^2} \times \sqrt{1+\left(\frac{10k}{1.2k}\right)^2}} \approx 1.9 \text{ kHz} \qquad (4.146)$$

The complete template appears in Figure 4.47. This time, the PID coefficients are derived from the poles/zeros positions we fix, and we use definitions given in (4.19) through (4.22) to parameterize the PID blocks.

The ac responses brought by the compensator and the compensated loop gain are, respectively, given in Figure 4.48 and Figure 4.49.

It is now interesting to look at the closed-loop output impedance since we know that the transient response to a load step will depend upon this figure. It is given in Figure 4.50. As you can see, the peaking found in the classical PID compensation has gone. The curve is very smooth, and we expect a good transient response without significant overshoot.

We have stepped the buck converter output from 1 A to 2 A in a 100-$\mu$s slew-rate. The response is given in Figure 4.51. The undershoot is small (0.74 percent), and the spurious oscillations are gone. This is a well-compensated design.

Until now, the transient response and ac analysis have been carried on a converter operated in the continuous conduction mode (CCM). This mode simply states that the inductor current does not return to zero within one switching cycle. When the load is getting lighter, the inductor fully demagnetizes within a switching cycle and the converter is told to operate in the discontinuous conduction mode (DCM). This new operating mode, for a voltage-mode converter, drastically changes its ac response. This is what is shown in Figure 4.52. The gain is reduced and the transition frequency drops from 3 kHz in CCM to around 150 Hz in DCM. Regarding the asymptotic phase lag, it approaches 180° for a CCM converter but remains below 90° for the DCM operation. What is the problem then? Well, we have compensated our buck converter in the worst load condition case (at minimum input voltage) so that the phase margin meets the design target. What happens if the converter now transitions into DCM? SPICE is extremely helpful as the models derived in [6] are auto-toggling: they automatically transition between CCM and DCM and deliver the correct ac response. The loop gain ac response of the CCM-compensated buck converter now operated in DCM is given in Figure 4.53.

From this picture, we can see that the phase margin degrades significantly when the buck converter enters DCM. For a 500-Ω load ($I_{out}$ = 10 mA), the phase margin drops to 40°. How can we improve the situation? By splitting the zero pair located at the resonant frequency. We are going to push one zero above the resonant frequency—at 3 kHz—while the other one will shift down to 300 Hz. These values can be analytically derived

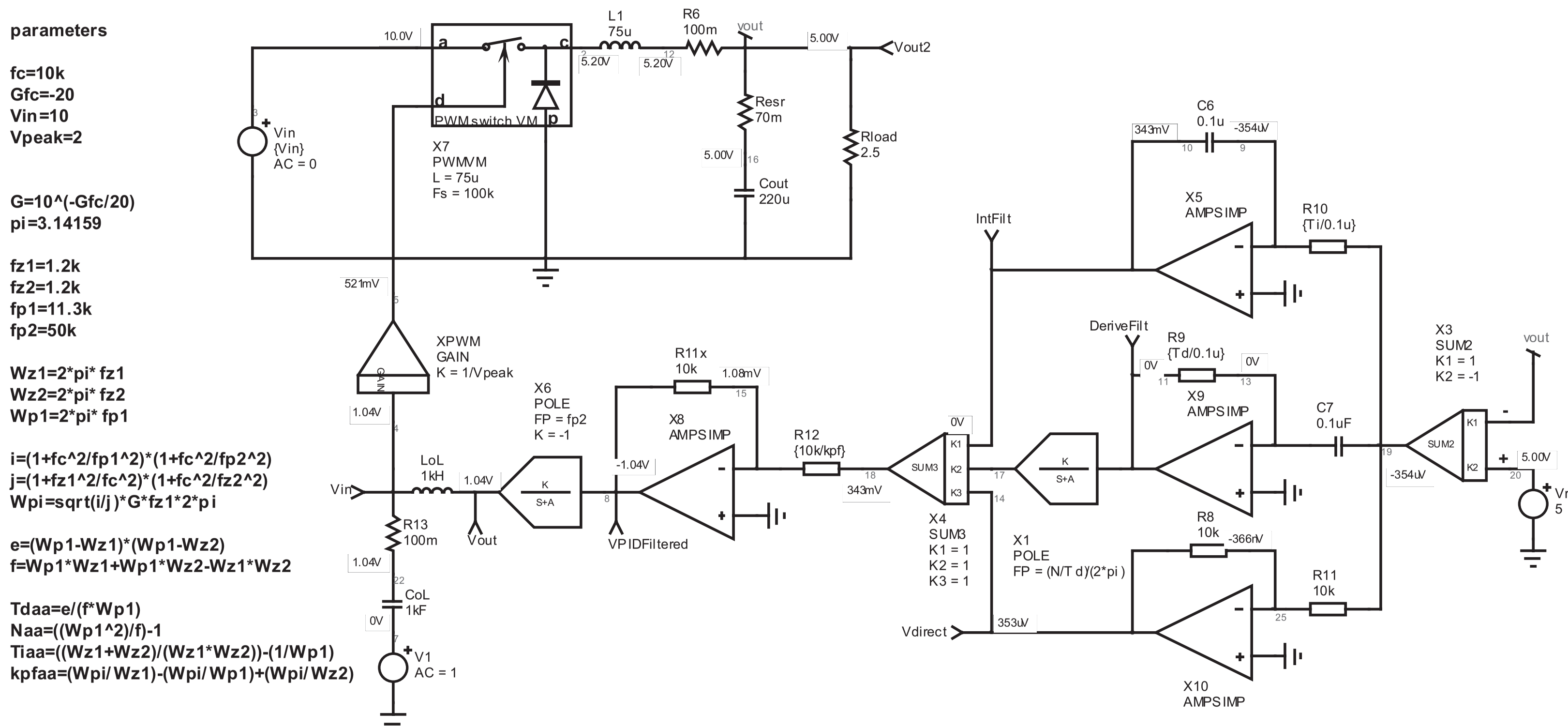



**Figure 4.47** The simulation template shows the filtered-PID compensator whose coefficients are now computed based on the wanted real poles and zeros.

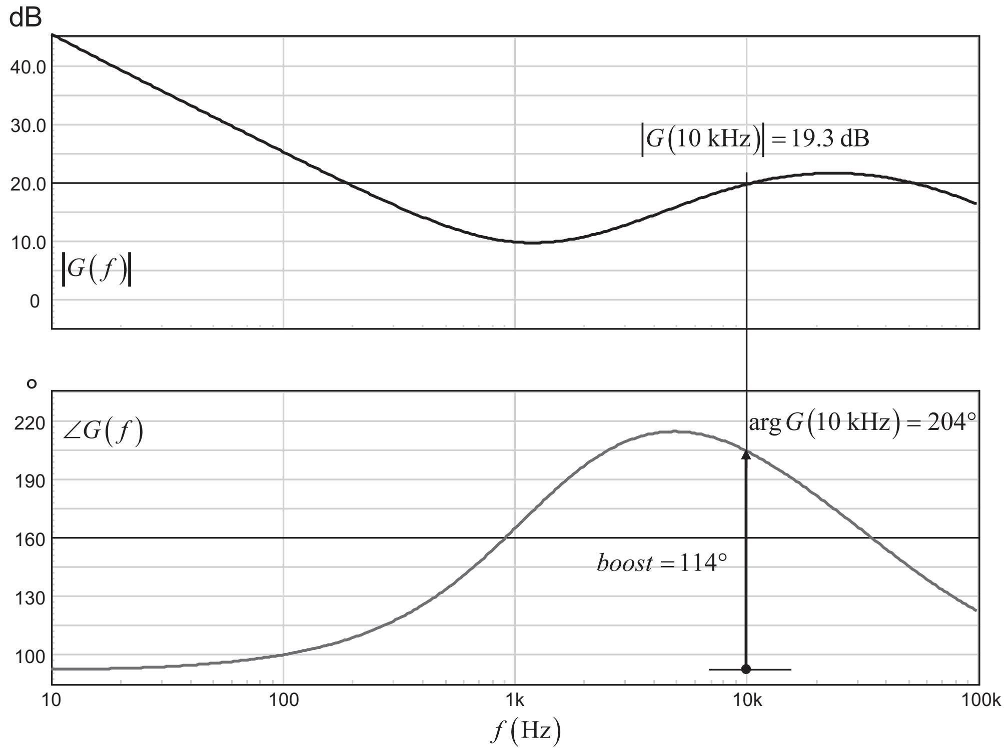



**Figure 4.48** The compensator boosts the phase by 114°, as expected.

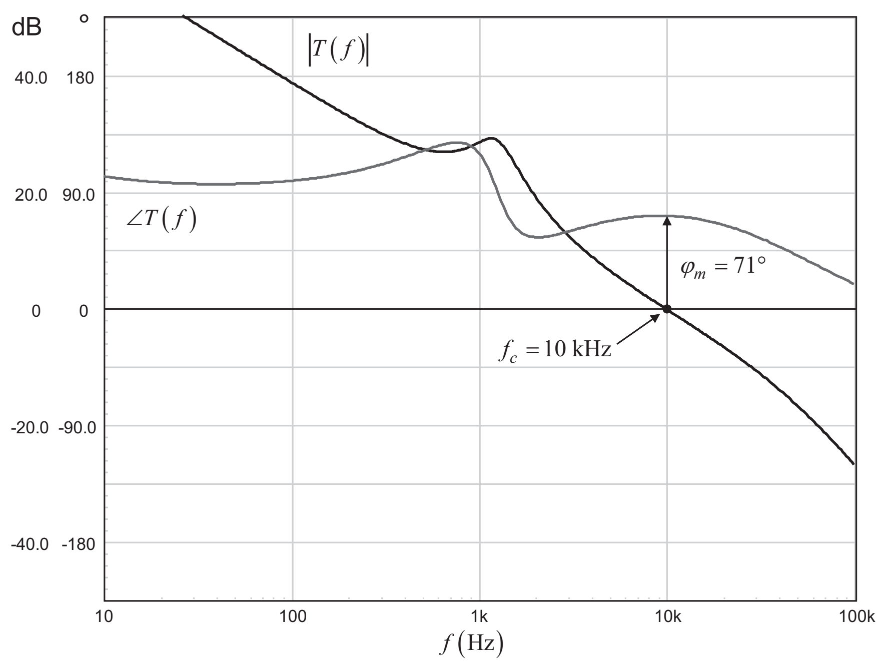



**Figure 4.49** The phase margin is 70° as expected at a 10-kHz crossover frequency.

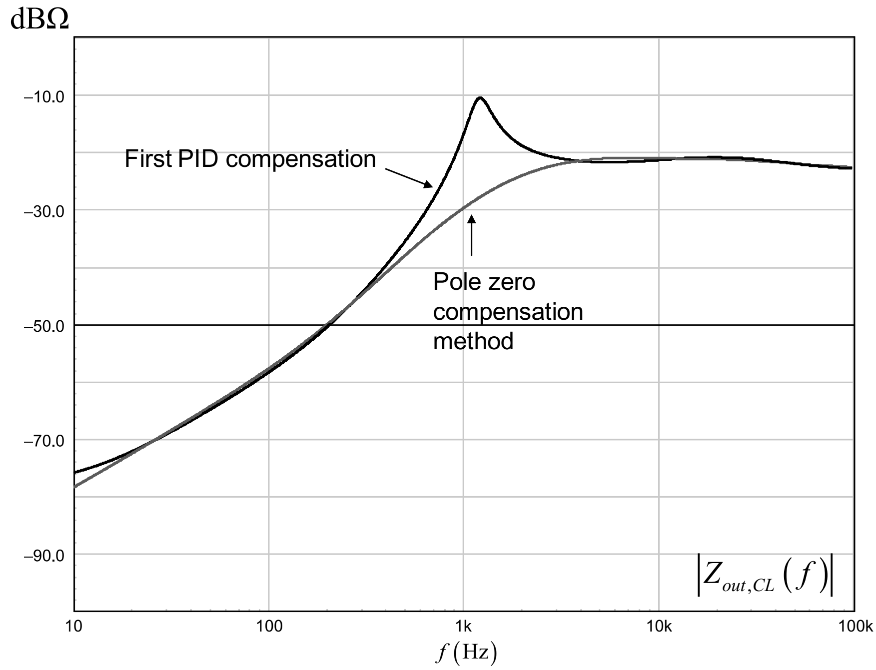



**Figure 4.50** The closed-loop output impedance of the compensated buck converter does not reveal any peak.

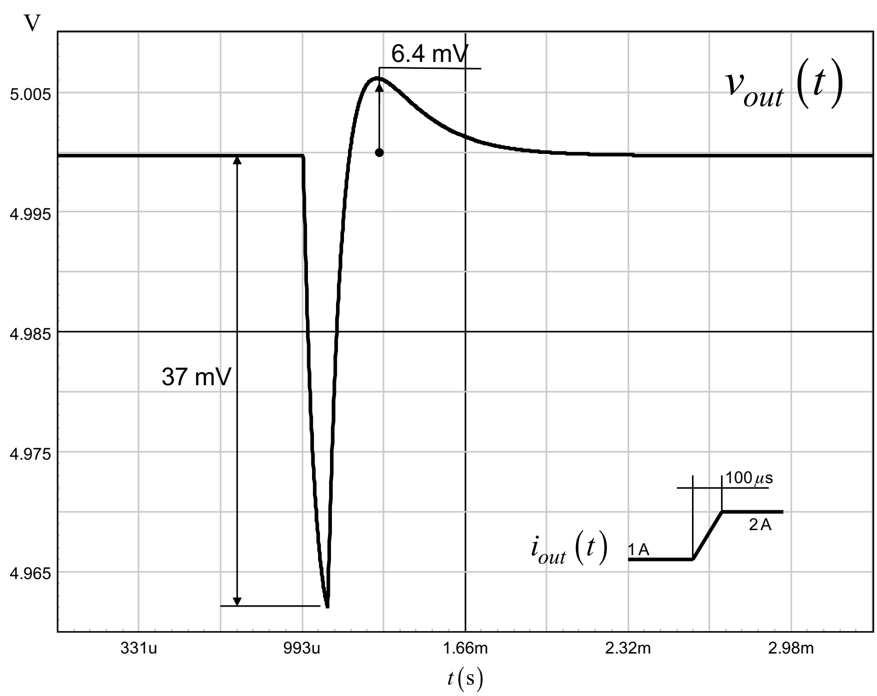



**Figure 4.51** The overshoot keeps extremely small when the compensated buck converter is subjected to a load step.

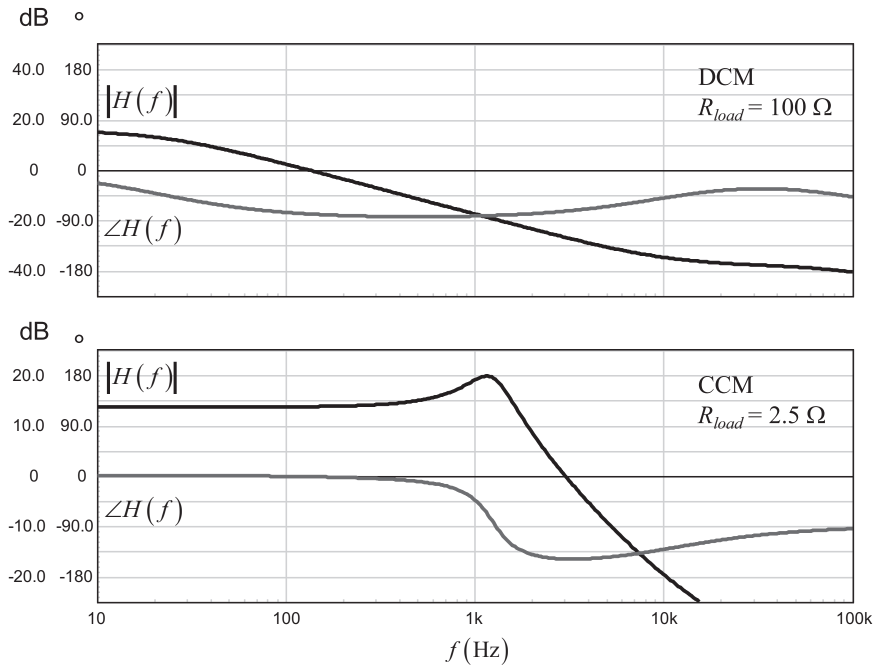



**Figure 4.52** When operated in DCM, the second-order CCM operated buck converter turns into a first-order converter.

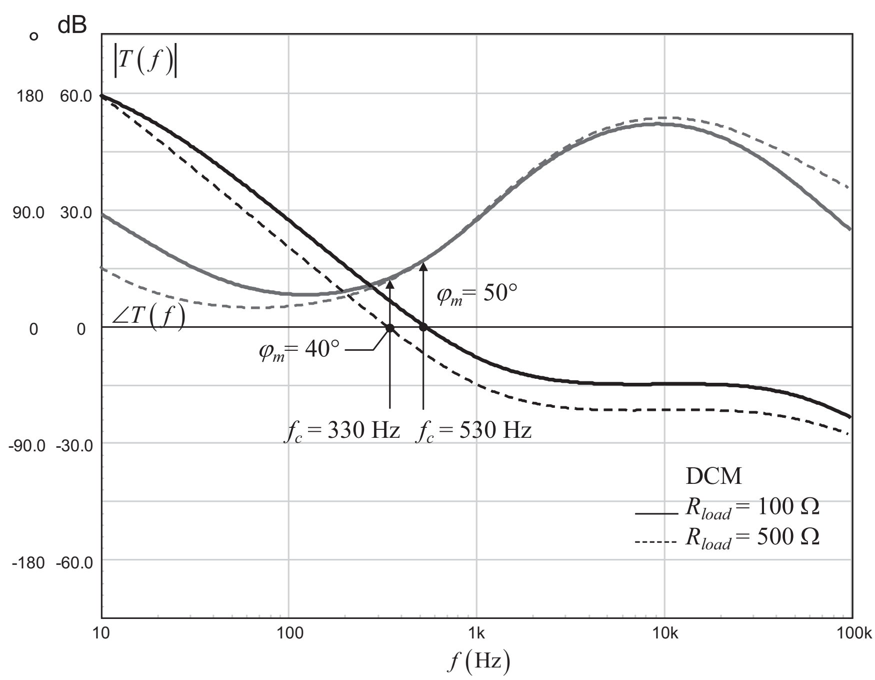



**Figure 4.53** When transitioning in DCM, because of the drastic plant gain change, the crossover frequency is reduced and the phase margin at this new point becomes questionable.

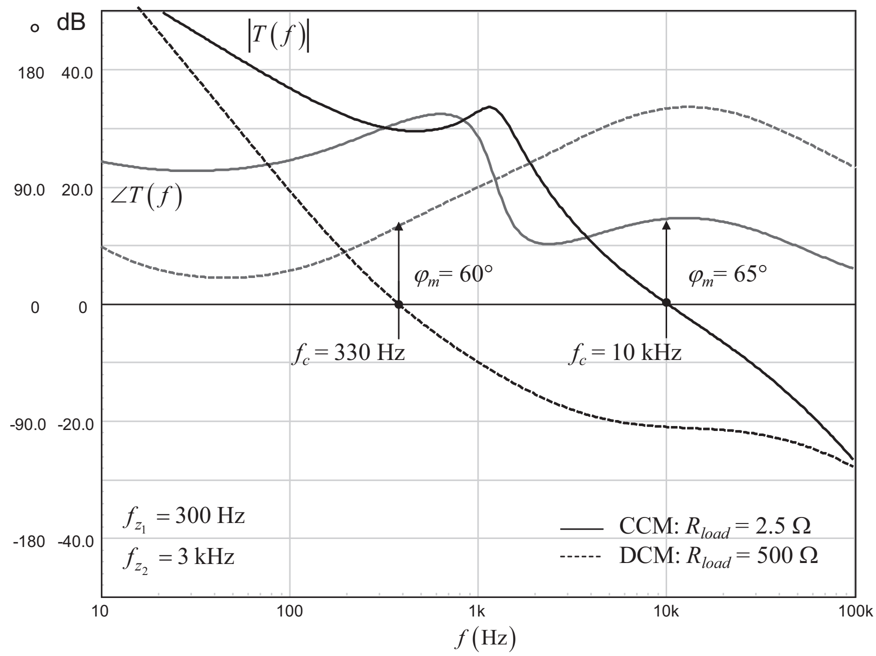



**Figure 4.54** By splitting the zeros, you improve the phase margin in DCM.

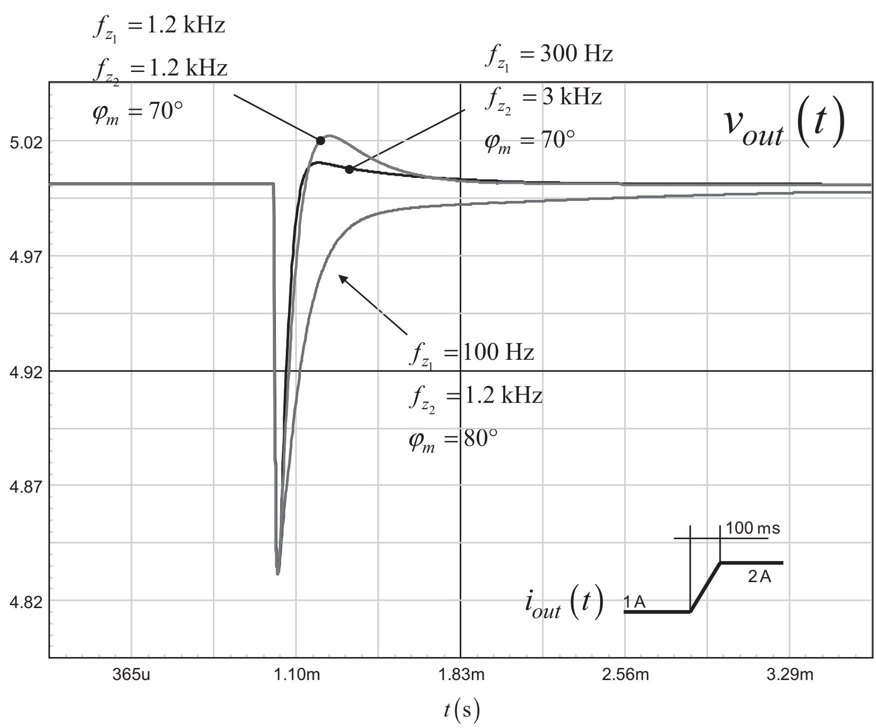



**Figure 4.55** Splitting the zeros affects the transient response in CCM, mainly recovery time and the overshoot.

| | Frequency | Overshoot | Settling time | Phase margin |
|---|---|---|---|---|
| $f_{z_1}$ | ↗ | ↗ | faster | ↘ |
| | ↘ | ↘ | slower | ↗ |
| $f_{z_2}$ | ↗ | ↗ | faster | ↘ |

**Figure 4.56** This array shows how splitting the zeros can change the transient response of the affected converter.

but the exercise is iterative by nature since splitting these zeros affects both the CCM and DCM ac responses: should you tweak the phase margin in DCM, you must verify what it becomes in CCM. A SPICE simulator is actually the best to quickly run the exercise and check both responses in a few milliseconds.

We have simulated the converter with the new zeros position (300 Hz and 3 kHz), and Figure 4.54 shows you the results: the phase margin in DCM is much better than before, unfortunately to the detriment of ac response in CCM: the phase margin drops by 6° (see Figure 4.49) but remains at a comfortable value of 65° though.

In Figure 4.55, we have tested three different transient responses to the 1A load step and compared them to the reference one, the 1.2-kHz zero pair. When a zero is going down the frequency axis, it improves the phase margin and reduces the overshoot. However, it increases the recovery time. On the contrary, if this zero is shifted up, the converter recovers faster but overshoots. This is normal, as we learned in Chapter 2 that the open-loop zeros (those you put in the compensator, $f_{z1}$, $f_{z2}$ in this example) turn into closed-loop poles. If you put a zero at a low frequency to boost the phase, you will naturally slow down the transient response. The array in Figure 4.56 tells you how moving the zeros affects the closed-loop performance.

The transient response resulting from this compensation scheme differs from that obtained with the PID calculation methodology: the response is nonoscillatory but there is a rather deep overshoot, typical of systems including an integral term. Certain applications do not tolerate these undershoots and ask for stringent regulation limits. This is the case for high-speed dc-dc converters for motherboards. For this type of application, the two methods we described cannot be implemented.

## 4.3 Output Impedance Shaping

In the previous methods, we have seen how the converter fights the sudden current increase or release: an undershoot and an overshoot are typical of an inductive-shaped output impedance. If you look back to Figure 4.50, you can indeed see an impedance growing as the frequency also increases, confirming an inductive-like output impedance. Capitalizing on this shape, [7] offers a simplified representation of a closed-loop buck converter impedance, associating an equivalent inductance and the output capacitor ESR. The schematic appears in Figure 4.57.

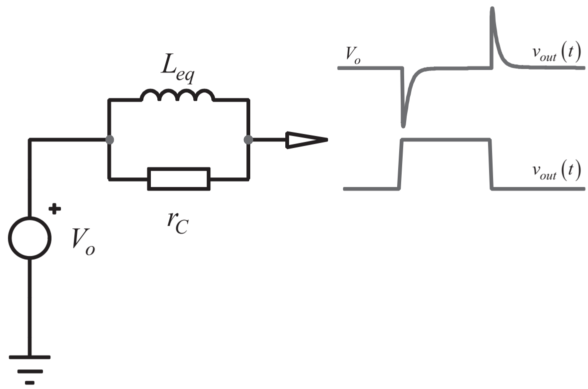



**Figure 4.57** The simplified equivalent model of the buck converter associates an inductor and a resistor.

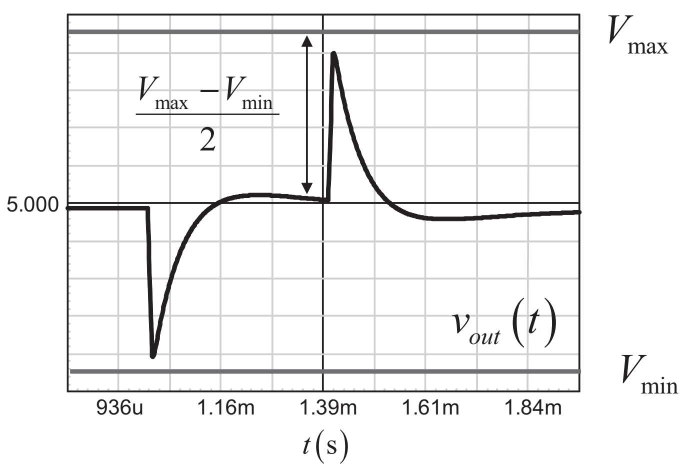



**Figure 4.58** The designer must limit both the undershoot and the overshoot to stay within the specifications.

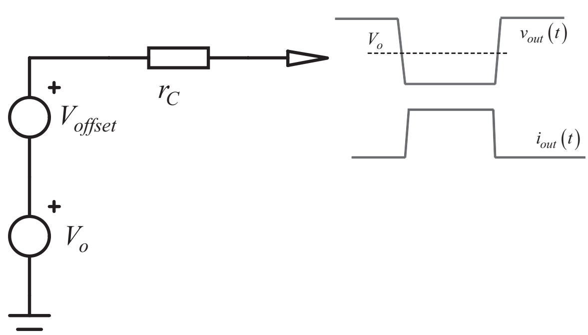



**Figure 4.59** If the output impedance becomes purely resistive, the output signal is no longer affected by large under-/overshoots.

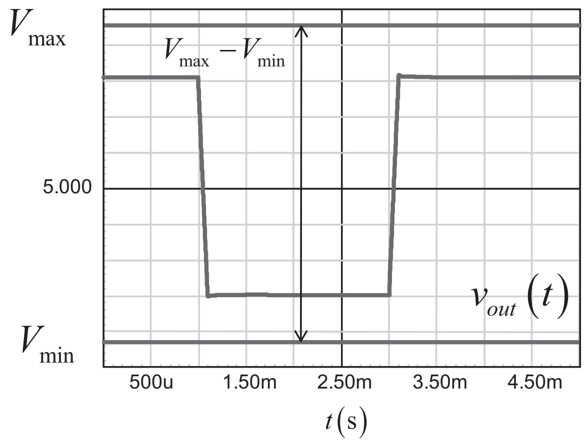



**Figure 4.60** A way to limit the voltage excursions is to shape the output impedance and make the inductive term disappear.

The combined action of the inductor and the resistor generates the typical signal represented on the output. The shape is typical of a converter involving an integral term in its compensation chain. As observed, the undershoot and the overshoot can be quite large and exceed the allowable band. This what we shown in Figure 4.58, where margin barely exists.

To get rid of these large excursions around the target output voltage, the best would be suppress the inductive term from Figure 4.57. This is the approach proposed in Figure 4.59.

In this configuration, the targeted level is purposely shifted by an offset to position the output voltage exactly in the middle of the peak-to-peak excursion. This way, the allowable excursion doubles compared to the previous approach. Figure 4.60 portrays the resulting waveform. The method is called *adaptative voltage positioning*.

### 4.3.1 Making the Output Impedance Resistive

The closed-loop output impedance $Z_{out,CL}$ of the voltage-mode converter is defined by its open-loop output impedance $Z_{out,OL}$ divided by the open-loop gain $T_{OL}$:

$$Z_{out,CL}(s) = \frac{Z_{out,OL}(s)}{1 + T_{OL}(s)} \tag{4.147}$$

The open-loop output impedance has already been derived in Appendix 4A and obeys the following expression:

$$Z_{out,OL}(s) = R_0 \frac{\left(1 + \frac{s}{\omega_{z1}}\right)\left(1 + \frac{s}{\omega_{z2}}\right)}{\left(\frac{s}{\omega_0}\right)^2 + \frac{s}{\omega_0 Q} + 1} \tag{4.148}$$

The loop gain $T_{OL}$ is made of the plant transfer function $H(s)$ and the compensator transfer function $G(s)$:

$$T_{OL}(s) = H(s)G(s) \tag{4.149}$$

The buck transfer function is that of a second-order system with a zero linked to the output capacitor ESR:

$$H(s) = H_0 \frac{1+\dfrac{s}{\omega_{z_1}}}{\left(\dfrac{s}{\omega_0}\right)^2 + \dfrac{s}{\omega_0 Q} + 1} \tag{4.150}$$

The idea is to shape the closed-loop output impedance and make it look like a resistance over the frequency spectrum. To fulfill this goal, there is one circuit upon which we can act: the compensator $G$. This is the place where we will arrange poles or zeros to make the closed-loop output impedance look resistive. What resistance, by the way? The output capacitor equivalent series resistance, $r_C$, is a good choice, as it is the value toward which the impedance heads at high frequency (Figure 4.61). Equation (4.147) can now be updated to reflect this choice:

$$r_C = \frac{Z_{out,OL}(s)}{1+T_{OL}(s)} = \frac{Z_{out,OL}(s)}{1+H(s)G(s)} \tag{4.151}$$

Substituting (4.148) and (4.150) into (4.151), we have

$$r_C = R_0 \frac{\left(1+\dfrac{s}{\omega_{z_1}}\right)\left(1+\dfrac{s}{\omega_{z_2}}\right)}{\left(\dfrac{s}{\omega_0}\right)^2 + \dfrac{s}{\omega_0 Q} + 1} \frac{1}{1+H_0 \dfrac{1+\dfrac{s}{\omega_{z_1}}}{\left(\dfrac{s}{\omega_0}\right)^2 + \dfrac{s}{\omega_0 Q} + 1} G(s)} \tag{4.152}$$

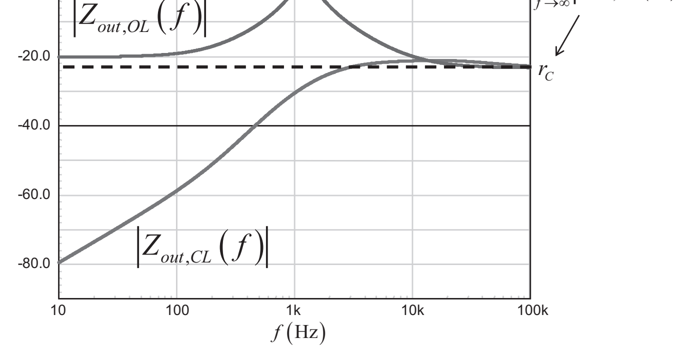



**Figure 4.61** When you compare the buck converter open and closed-loop output impedances, they both converge to a fixed value at high frequency, the output capacitor ESR, $r_C$.

In which $G(s)$ must be shaped to meet the resistive output impedance goal. If we extract the compensator transfer function from (4.152), we obtain

$$G(s) = \frac{R_0\left(1+\frac{s}{\omega_{z_2}}\right)}{r_C H_0} - \frac{\left(\frac{s}{\omega_0}\right)^2 + \frac{s}{Q\omega_0} + 1}{H_0\left(1+\frac{s}{\omega_{z_1}}\right)} \tag{4.153}$$

At first glance, I am not able to guess the frequency response of such a transfer function. To gain insight on this expression, we can plot it with Mathcad®. To run this exercise, let us consider the following component values:

$L = 75\ \mu\text{H}$

$r_L = 300\ \text{m}\Omega$

$r_C = 30\ \text{m}\Omega$

$C_{out} = 220\ \mu\text{F}$

$R_{load} = 2.5\Omega$

$\omega_{z_1} = r_L/L = 4$ krad/s so $f_{z_1} = 636$ Hz

$\omega_{z_2} = 1/(r_C C_{out}) = 151$ krad/s so $f_{z_2} = 24.1$ kHz

$R_0 = r_L \,||\, R_{load} = 268\ \text{m}\Omega$

$\omega_0 \approx 1/\sqrt{LC_{out}} = 7.78$ krad/s so $f_0 \approx 1.2$ kHz

$Q \approx \sqrt{L/C_{out}}/(r_L + r_C) = 1.77$

$H_0 = V_{in}/V_{peak} = 10/2 = 5$

The plot appears in Figure 4.62.

I confess that I did not expect this magnitude shape when looking at (4.153), but this is it. You can see a dc gain less than 5 dB and a zero kicking around 500 Hz. Then the 1-slope breaks into a 0-slope as the pole appears around 20 kHz. The shape fits the following equation:

$$G(s) = K_0 \frac{1+\frac{s}{\omega_{zG}}}{1+\frac{s}{\omega_{pG}}} \tag{4.154}$$

The fun part now is to identify the pole and the zero definition, as well as the gain $K_0$ with the elements part of (4.153). From these definitions, the authors of [8] did a great job and came up with the following relationships:

$$K_0 = \frac{r_L - r_C}{H_0 r_C} = 1.8 \tag{4.155}$$

$$a = \frac{r_L}{\omega_{z_1}\omega_{z_2}} - \frac{r_C}{\omega_0^2} = 47.7p \tag{4.156}$$

$$b = r_L\left(\frac{1}{\omega_{z_1}} + \frac{1}{\omega_{z_2}}\right) - \frac{r_C}{Q\omega_0} = 74.2u \tag{4.157}$$

$$c = r_L - r_C = 0.27 \tag{4.158}$$

$$\omega_{p_G} = \omega_{z_1} = 4\,\text{krad/s},\, f_{p_G} = f_{z_1} = 24.1\,\text{kHz} \tag{4.159}$$

$$\omega_{z_G} = \frac{b - \sqrt{b^2 - 4ac}}{2a} = 3.64\,\text{krad/s},\, f_{z_G} = 580\,\text{Hz} \tag{4.160}$$

If we plot (4.154) fed with the previous gain, pole, and zero positions and compare it to the original plot from Figure 4.62, the agreement is good, as seen in Figure 4.63.

Now that the transfer function for the compensator is known, we need to find an implementation. The op amp architecture offered in Figure 4.64 seems like a good fit.

To calculate the component values, we need to derive the transfer function of this compensator. Given the inverting configuration, the first expression is

$$G(s) = \frac{V_c(s)}{V_{out}(s)} = -\frac{R_1}{Z_1(s)} \tag{4.161}$$

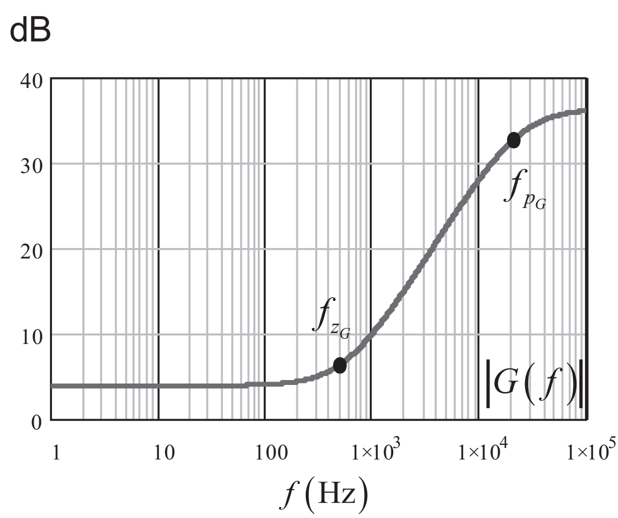



**Figure 4.62** The required transfer function combines a pole and a zero but no origin pole: no integral term.

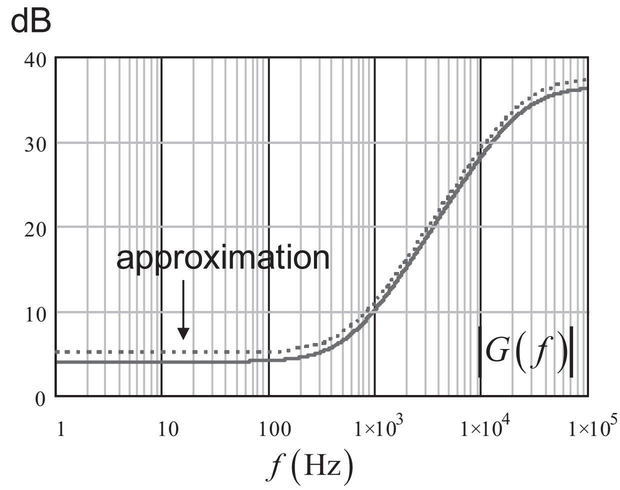



**Figure 4.63** The agreement between the original curve and the approximation is good.

The impedance $Z_1$ is made of a series-parallel combination:

$$Z_1 = \frac{R_2\left(R_3 + \dfrac{1}{sC_1}\right)}{R_2 + \left(R_3 + \dfrac{1}{sC_1}\right)} \tag{4.162}$$

If we substitute this definition in (4.161), then develop and rearrange the final expression, we obtain

$$G(s) = -\frac{R_1}{R_2}\frac{[1 + sC_1(R_2 + R_3)]}{1 + sR_3C_1} = -K_0\frac{1 + \dfrac{s}{\omega_{zG}}}{1 + \dfrac{s}{\omega_{pG}}} \tag{4.163}$$

in which we have

$$K_0 = \frac{R_1}{R_2} \tag{4.164}$$

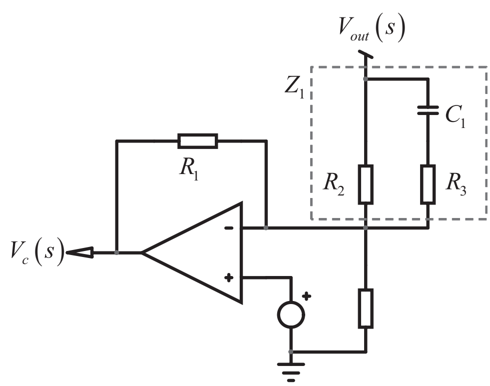



**Figure 4.64** The simple pole/zero combination can be easily implemented with an op amp.

$$\omega_{z_G} = \frac{1}{C_1(R_2 + R_3)} \tag{4.165}$$

$$\omega_{p_G} = \frac{1}{R_3 C_1} \tag{4.166}$$

If you fix $R_1$ to 10 kΩ (for instance) and solve for the three remaining unknowns $C_1$, $R_2$, and $R_3$, you should obtain the following results:

$$R_3 = \frac{R_1 \omega_{z_G}}{K_0(\omega_{p_G} - \omega_{z_G})} = 137\ \Omega \tag{4.167}$$

$$R_2 = \frac{R_1}{K_0} = 5.5\ \text{k}\Omega \tag{4.168}$$

$$C_1 = \frac{K_0(\omega_{p_G} - \omega_{z_G})}{R_1 \omega_{p_G} \omega_{z_G}} = 48\ \text{nF} \tag{4.169}$$

If we assign these values to (4.163), we obtain the Bode plot presented in Figure 4.65, a copy of that displayed in Figure 4.63.

Having the compensator transfer function on hand, we can now plot the final output impedance as defined by (4.147). This graph is shown in Figure 4.66. As observed, the closed-loop output impedance is perfectly resistive along the frequency axis and shows a value of –30.4 dBΩ, which is around 30 mΩ, our output capacitor ESR. The proposed scheme works well on paper and has transformed our inductive open-loop output impedance into a resistive closed-loop output impedance.

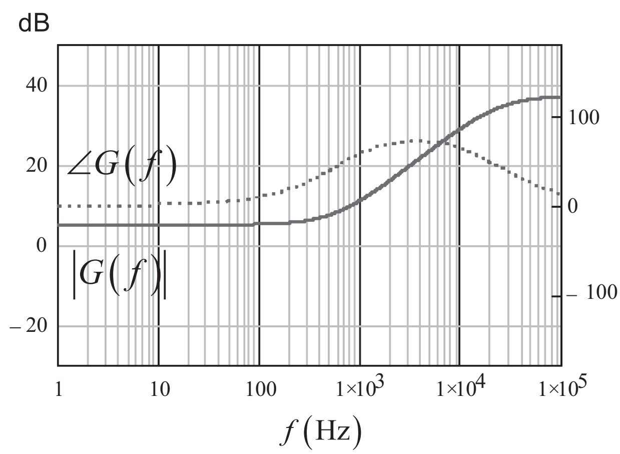



**Figure 4.65** The compensator Bode plot does not include an integral term (no origin pole).

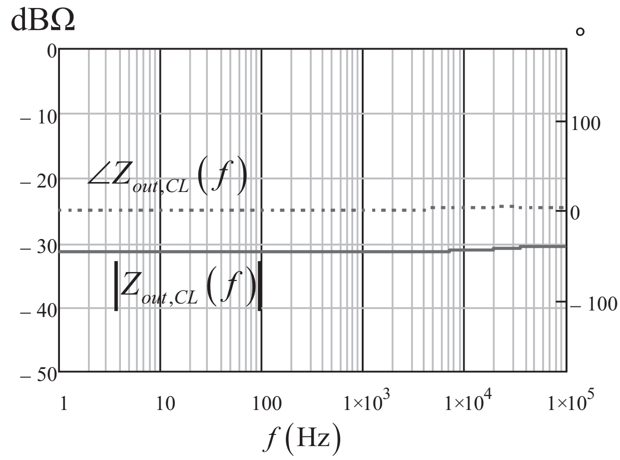



**Figure 4.66** The output impedance is perfectly resistive as expected!

We can run a simulation to check the validity of our calculations. The simulated schematic appears in Figure 4.67. On the left, the compensator values are automatically calculated to place the pole and zero recommended by (4.159) and (4.160). Let us look at the ac transfer function to check the crossover frequency and the phase margin at this point. The ac response is given in Figure 4.68. We can see a crossover frequency slightly above 20 kHz and a phase margin of 90°: this is a rock-solid design!

To check the effects of this compensation on the closed-loop impedance, we are going to add a 1-A ac source in parallel with the load, while the loop opening network made of *LoL* and *CoL* is removed. The plot proposed in Figure 4.69 confirms the resistive nature of the output impedance. It is time to test the transient response of the whole converter. The response to a 1-A step is given in Figure 4.70. It is a perfect square wave whose amplitude is exactly the 30-mΩ ESR multiplied by the 1 A current step.

Is the solution a panacea? Well, if we take a closer look at the open-loop ac response from Figure 4.68, we can see a rather low dc gain, below 20 dB. We learned that the dc gain directly impacts the output static error. If we program the op amp divider network to get 5V, despite the precision of the reference voltage, an open-loop dc gain of 10 will not guarantee a 5-V output. To compensate for this problem, we have purposely shifted the target by altering resistor $R_7$ on the simulation schematic. It purposely shifts up the output target to reach 5 V, and the dc bias shows this to work. If you change the input conditions, this low gain will not shield the output from moving. This voltage-mode implementation of a resistive output impedance can work only if you have a stable input voltage. This is the case for high-speed dc-dc converters for motherboards where the designers generate a 3.3 V or a 1.5 V output directly from the 12V rail. In applications where the input voltage widely varies, this voltage mode solution does not work, and you will need to consider the current-mode version. This is the subject of [8, 9]. The founding equations differ from that of the voltage-mode structure, but the principle remains the same.

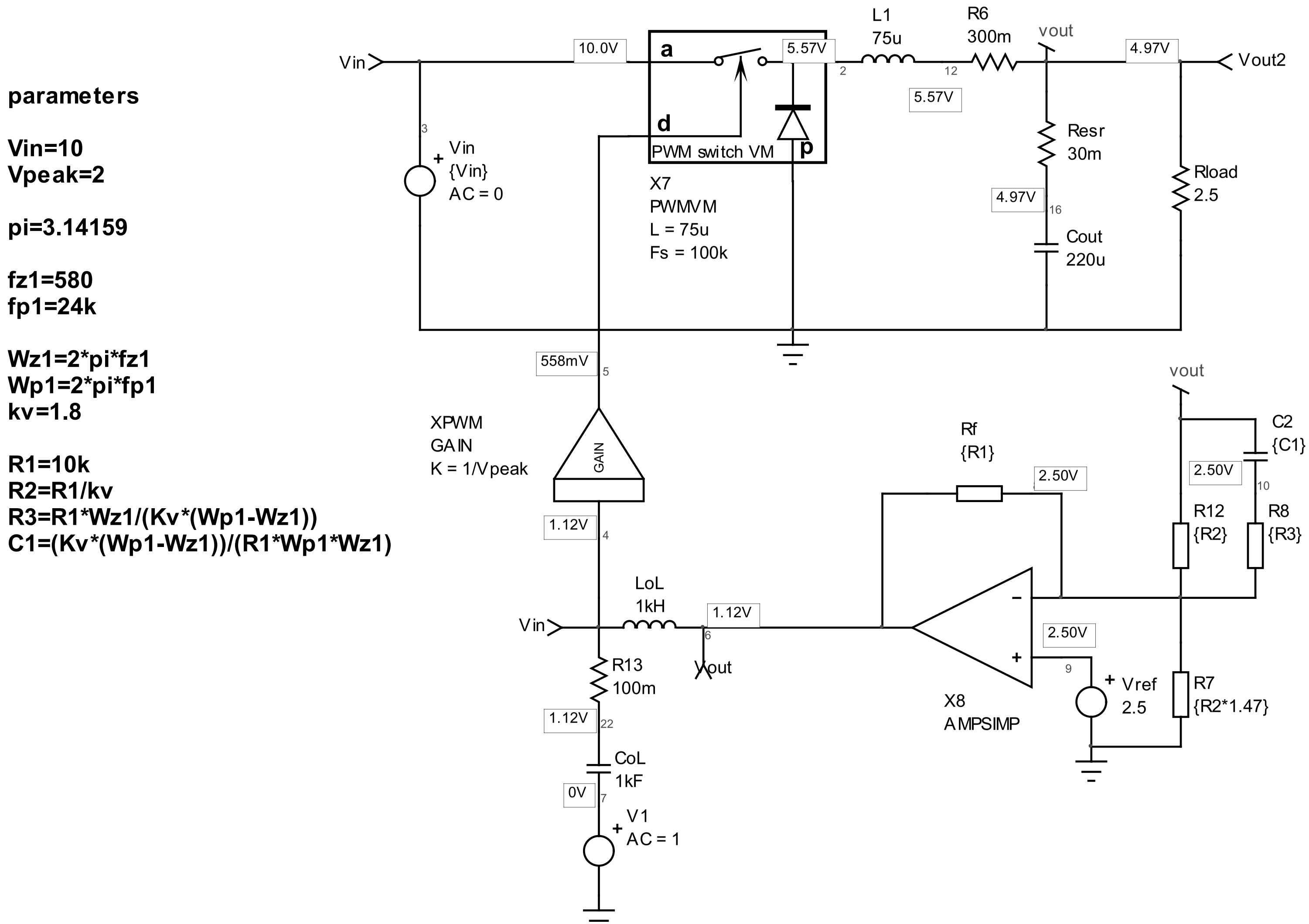



**Figure 4.67** The simulation circuit shows the buck converter whose loop is now closed with the Figure 4.64 circuit.

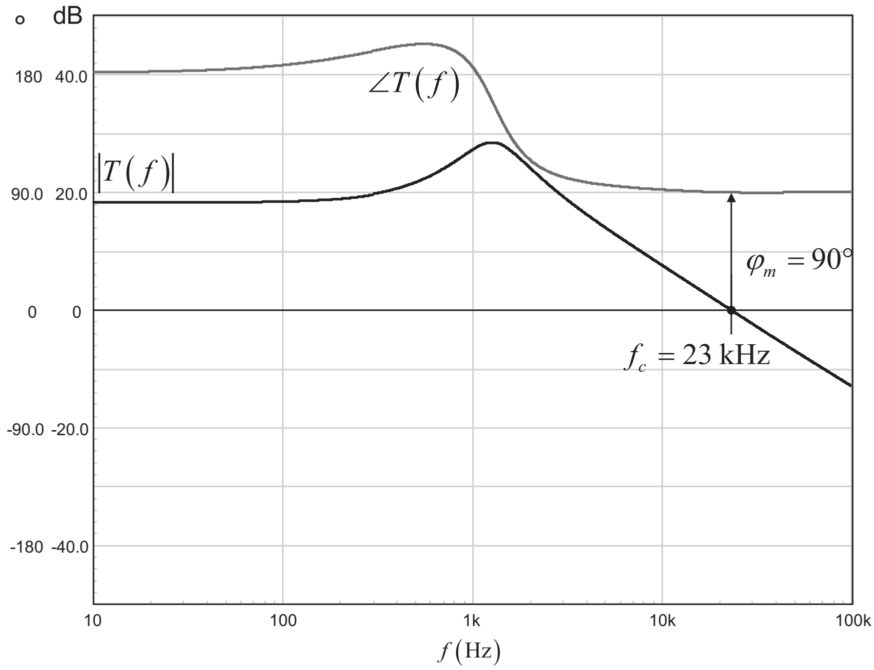



**Figure 4.68** The ac response shows a good phase margin despite a high crossover frequency.

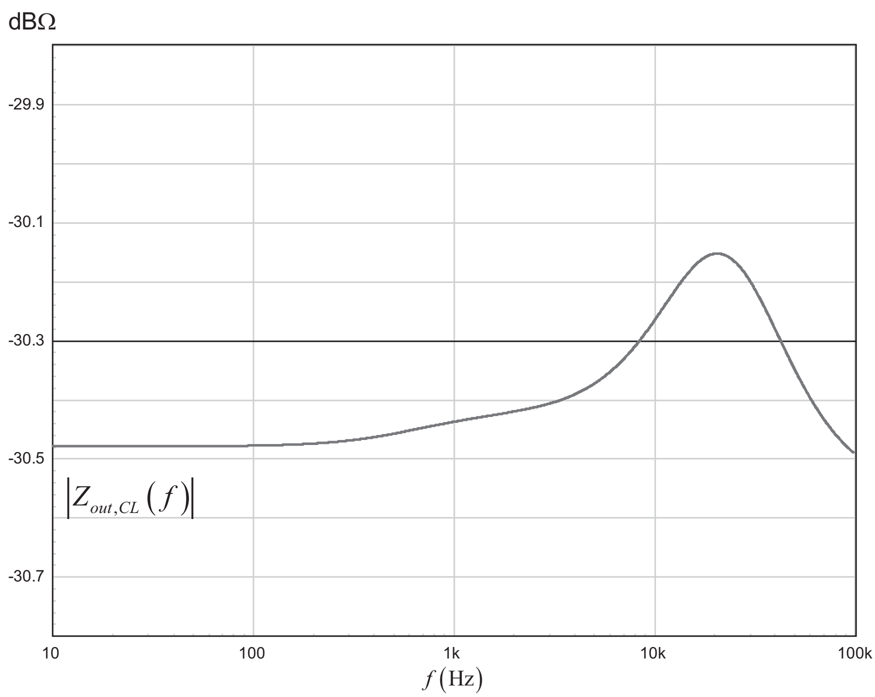



**Figure 4.69** The ac simulation of the output impedance confirms our analytical calculations: it is almost constant at –30 dBΩ.

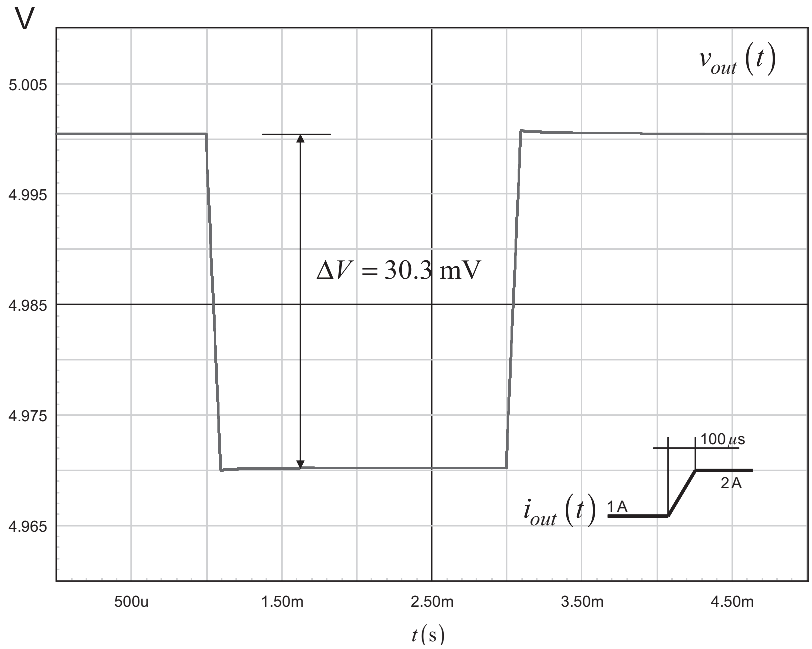



**Figure 4.70** As expected, there is no over- or undershoot on this waveform. The total deviation is exactly the current multiplied by the ESR value.

## 4.4 Conclusion

I was taught PID at university in a rather complex way. There were no connections to pole-zero placement that I was more familiar with. In this chapter, after several algebraic manipulations, we found that a filtered-PID was actually a type 3 compensator, a well-known circuitry for those who compensate converters. Compensating a switching converter by focusing only on the setpoint-to-output transfer function is offering an incomplete view of the transmission chain. Most of our converters are actually regulators, and the setpoint (the reference voltage) is fixed when the converter undergoes a load change. What matters to the transient response is thus the output impedance expression. This is the element that dictates the response and will tell you if your design will deliver a nonringing response. We discovered this fact by blindly applying a PID-based compensation method to the voltage-mode buck converter under study. The result was not convincing, leading to an oscillatory response. We obtained a nonpeaking output impedance with the classical pole/zero placement technique, and the step response was perfectly damped. In both methods, we concentrated our efforts on stabilizing the setpoint-to-output transfer function, without specifically looking at the output impedance. In the third example, we explicitly looked at its expression and worked the compensator to make it fully resistive. The result is a nice square-wave signal, without over-/undershoots. This technique is widely used in high-speed dc-dc converters for personal computers and can be applied in situations where a stringent specification concerns the load-step response.

Now that we reviewed a few of the available compensation techniques, we are going to discover how to build compensators with operational amplifiers.

## References


[1] Wikipedia contributors, "PID Controller," http://en.wikipedia.org/wiki/PID_controller, last accessed June 3, 2012.

[2] Besançon-Voda, A., and S. Gentil. "Régulateurs PID Analogiques et Numériques," *Techniques de l'ingénieur*, R7416, 1999.

[3] Retif, J.-M. "Automatique Regulation," 4ième année GE, Institut National des Sciences Appliquées, 2008.

[4] Ziegler, J. G., and N. B. Nichols. "Optimum Settings for Automatic Controllers," *Trans. ASME*, Vol. 64, 1942, pp. 759–768.

[5] Venable, D. "The k-Factor: A New Mathematical Tool for Stability Analysis and Synthesis," *Proceedings of Powercon 10*, 1983, pp. 1–12.

[6] Basso, C. *Switch-Mode Power Supplies: SPICE Simulations and Practical Designs*, New York: McGraw-Hill, 2008.

[7] Erisman, B., and R. Redl. "Modify Your Switching Supply Architecture for Improved Transient Response," *EDN Magazine*, November 11, 1999.

[8] Yao, K., M. Xu, Y. Meng, and F. Lee. "Design Considerations for VRM Transient Response Based on the Output Impedance," *IEEE Proceedings*, Vol. 18, No. 6, November 2003.

[9] Erisman, B., and R. Redl. "Optimizing the Load Transient Response of the Buck Converter," *APEC'1998 Proceedings*, Vol. 1, pp. 170–176.


## Appendix 4A: The Buck Output Impedance with Fast Analytical Techniques

An output impedance is a transfer function. It is made of an excitation signal—a current source—that produces a voltage across the impedance it sweeps, the response signal. This known principle is found in Figure 4.71.

As already indicated, the output impedance expression must follow the form indicated in the picture: a static ohmic expression followed by an *s*-quotient without dimension. First, we start with the dc term, $R_0$. It is found by putting all storage elements in their dc states: the capacitors are open and the inductors short circuited. You obtain the simple circuit of Figure 4.72.

From that drawing, we see immediately that the dc term is simply:

$$R_0 = r_L \parallel R_{load} \tag{4.170}$$

Now, observing Figure 4.71, do we see branches that would prevent the excitation from reaching the output? In other terms, what could prevent the excitation current from generating a voltage across $R_{load}$? Well, if $R_{load}$ was simply short

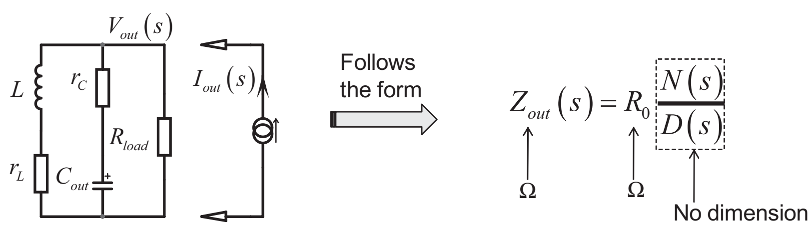



**Figure 4.71** An output impedance is one of the six possible transfer functions.

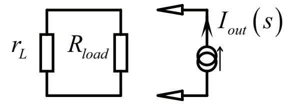



**Figure 4.72** When all capacitors are open and inductors short circuited, the circuit greatly simplifies.

circuited by the parallel networks, the voltage would be zero. What, in this parallel network, could be a short circuit? Potentially the $RL$ and $RC$ branches. Mathematically, you can then write two equations as follows:

$$sL + r_L = r_L\left(1 + s\frac{L}{r_L}\right) = 0 \tag{4.171}$$

and

$$r_C + \frac{1}{sC} = \frac{sr_cC + 1}{sC} = 0 \tag{4.172}$$

Solving these simple equations gives us two zeros expressions:

$$\omega_{z_1} = \frac{r_L}{L} \tag{4.173}$$

$$\omega_{z_2} = \frac{1}{r_C C} \tag{4.174}$$

We are now halfway to our impedance definition:

$$Z_{out}(s) = R_0 \frac{(1 + s/\omega_{z_1})(1 + s/\omega_{z_2})}{D(s)} \tag{4.175}$$

The most complex part of the process is the derivation of denominator $D(s)$. Theory tells us that this denominator solely depends on the network structure, not on the excitation. If the excitation signal does not play a role when you derive the denominator, put it to zero in the network you study: short the excitation voltage or open the excitation current. For our impedance case, the current source just leaves the picture. Once $D(s)$ is derived, it appears as such in the remaining transfer functions like input impedance, input to output gain, and so on. Hey, we have it already from (4.34) then! That's right, but for the sake of the exercise, let us try to find it using the fast analytical techniques described in [1]. What does $D(s)$ look like? This is a two-storage element circuit—thus, a second-order network. It must obey the following expression:

$$D(s) = 1 + a_1 s + a_2 s^2 \tag{4.176}$$

As explained in the referenced work, the general form of the coefficient of $s^k$ in the previous equation ($a_1$, $a_2$) must be of dimension $(Hz)^{-k}$ so that the denominator remains dimensionless. These coefficients sum all the combined network time constants equal to $R_xC$ or $L/R_y$, where resistors $R_x$ or $R_y$ are the respective resistances seen at the capacitive or the inductive ports under some particular configurations we will see. These configurations involve so-called dc and high-frequency states defined as follows:

- Dc state: the capacitor is simply open (its impedance is infinite at $s = 0$), while the inductor is a short circuit.
- High-frequency state: in this case, the capacitor is now a short-circuit, and the inductor becomes open.

According to [1], the possible terms for a second-order network denominator are

- $\tau_1 + \tau_2$ for $a_1$. Dimension is time (s) or $Hz^{-1}$.
- $\tau_1 \tau_2'$ or $\tau_1' \tau_2$ are the two possible choices for $a_2$, where $\tau_1$ or $\tau_2$ are the time constants from earlier. Dimension is squared time ($s^2$) or $Hz^{-2}$. $\tau_1'$ $\tau_2'$ will be defined in a few lines.
- In all cases, the time constants $\tau$ are of the form $RC$ or $L/R$.

Applying definitions from [1], the denominator formula for a second-order system is given by

$$D(s) = 1 + s(\tau_1 + \tau_2) + s^2(\tau_1 \tau_2') \tag{4.177}$$

equivalent to

$$D(s) = 1 + s(\tau_1 + \tau_2) + s^2(\tau_1' \tau_2) \tag{4.178}$$

For $a_1$, we are first looking at the resistance $R$ seen from the inductor port when the capacitor is put in its dc state (open circuited). The sketch appears in Figure 4.73. The resistance seen at the inductor port is straightforward and equal to

$$R = r_L + R_{load} \tag{4.179}$$

Leading to

$$\tau_1 = \frac{L}{r_L + R_{load}} \tag{4.180}$$

Now, we look at the capacitor port resistance when the inductor is in its dc state. This is what Figure 4.74 represents. In this case, the equivalent resistance seen at the capacitor port is simply

$$R = r_C + r_L \parallel R_{load} \tag{4.181}$$

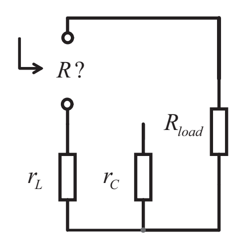



**Figure 4.73** The capacitor is open to find the inductor driving resistance.

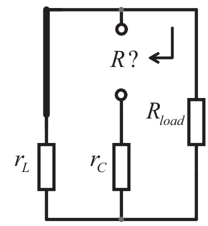



**Figure 4.74** An inductor in its dc state is a short circuit.

thus

$$\tau_2 = C(r_C + r_L \,||\, R_{load}) \tag{4.182}$$

Following the definition of (4.177), the first coefficient $a_1$ is defined by

$$a_1 = \tau_1 + \tau_2 = \left( \frac{L}{r_L + R_{load}} + C[(r_L \,||\, R_{load}) + r_C] \right) \tag{4.183}$$

The coefficients for $a_2$ require a little more attention. We have seen that we have to evaluate $\tau_1\tau_2'$ or $\tau_1'\tau_2$, where $\tau_1$ and $\tau_2$ are the same time constants evaluated for $a_1$. Therefore, we can either decide to find how $\tau_1$ (involving $L$) combines with $\tau_2'$ (involving $C$) or see how $\tau_2$ (involving $C$) combines with $\tau_1'$ (involving $L$). Either combination must lead to a similar result for $a_2$. Let's see the first option where we select $\tau_1$ and must find the resistance seen at port $C$ when $L$ is in its high-frequency state (open circuited). The equivalent schematic appears in Figure 4.75.

The resistance seen at the capacitor port in this case is simply

$$R = r_C + R_{load} \tag{4.184}$$

therefore,

$$\tau_2' = (r_C + R_{load})C \tag{4.185}$$

If we choose the second option, where we select $\tau_2$, we must find the resistance seen at port $L$ while $C$ is in its high-frequency state (short circuit). The equivalent schematic appears in Figure 4.76.

The resistance is defined by a simple series-parallel association:

$$R = r_L + r_C \,||\, R_{load} \tag{4.186}$$

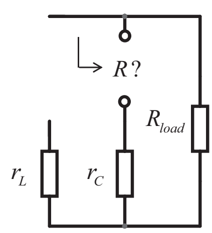



**Figure 4.75** The inductor in its high-frequency state is an open circuit and leaves the picture.

which gives us

$$\tau_1' = \frac{L}{r_L + r_C \parallel R_{load}} \tag{4.187}$$

Following (4.177), we have our expression for coefficient $a_2$:

$$a_2 = \tau_1\tau_2' = \frac{L}{r_L + R_{load}} C(r_c + R_{load}) = LC\frac{r_c + R_{load}}{r_L + R_{load}} \tag{4.188}$$

which is equivalent to

$$a_2 = \tau_1'\tau_2 = \frac{L}{r_L + R_{load} \parallel r_C} C[(r_L \parallel R_{load}) + r_C] = LC\frac{r_L \parallel R_{load} + r_C}{r_L + R_{load} \parallel r_C} = LC\frac{r_c + R_{load}}{r_L + R_{load}} \tag{4.189}$$

If you develop this expression, it is similar to that in (4.188). We are all set; the complete denominator function is thus:

$$D(S) = 1 + s\left(\frac{L}{r_L + R_{load}} + C[(r_L \parallel R_{load}) + r_C]\right) + s^2 LC\frac{r_c + R_{load}}{r_L + R_{load}} \tag{4.190}$$

We now can update (4.175). We obtain our final output impedance definition:

$$Z_{out}(S) = (r_L \parallel R_{load})\frac{\left(1 + s\frac{r_L}{L}\right)(1 + sr_C C)}{1 + s\left(\frac{L}{r_L + R_{load}} + C[(r_L \parallel R_{load}) + r_C]\right) + s^2\left(LC\frac{r_C + R_{load}}{r_L + R_{load}}\right)} \tag{4.191}$$

It can be put under the familiar form

$$Z_{out}(S) = R_0\frac{(1 + s/\omega_{z1})(1 + s/\omega_{z2})}{1 + \frac{s}{\omega_0 Q} + \left(\frac{s}{\omega_0}\right)^2} \tag{4.192}$$

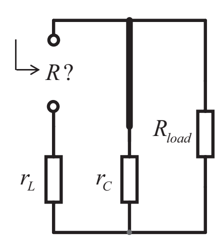



**Figure 4.76** In this choice, we keep $\tau_2$ and look at the resistance seen from the inductive port while $C$ is a short circuit.

in which we have

$$R_0 = r_L \,||\, R_{load} \tag{4.193}$$

$$\omega_{z_1} = \frac{r_L}{L} \tag{4.194}$$

$$\omega_{z_2} = \frac{1}{r_C C} \tag{4.195}$$

$$\omega_0 = \frac{1}{\sqrt{LC}}\sqrt{\frac{r_L + R_{load}}{r_C + R_{load}}} \tag{4.196}$$

$$Q = \frac{LC\omega_0(r_C + R_{load})}{L + C[r_L r_C + R_{load}(r_L + r_C)]} \tag{4.197}$$

If we now consider $r_C$ and $r_L$ equal to zero, (4.196) and (4.197) simplify to (4.36) and (4.37).

## Reference

[1] Erickson, R. W. "The n Extra Element Theorem," CoPEC, http://ecee.colorado.edu/~ecen5807/course_material/EET, last accessed June 3, 2012.

## Appendix 4B: The Quality Factor from a Bode Plot with Group Delay

It can be sometimes difficult to extract the quality factor from a Bode plot whose magnitude weakly peaks. This is the case of the graph in Figure 4.77, where the transient response overshoot indicates a quality factor beyond 0.5. However, the flat magnitude plot makes it difficult to measure.

Let us now have a look at the Bode plot generated from a second-order network where $Q$ has been purposely swept from a low value of 0.6 up to 10. This is shown in Figure 4.78 in which the magnitude starts to peak when $Q$ exceeds 1. In all cases, the phase starts from 0° in dc down to –180° reached in the upper portion of the frequency spectrum. However, you can notice that the phase rate of change depends on the quality factor value: at low $Q$s, the phase smoothly drifts to –180°, whereas it moves in a much sharper way as $Q$ increases.

In other terms, we can clearly see that the phase curve slope increases with $Q$. Therefore, there must be a relationship between this slope and the quality factor. This relationship is the *group delay* noted $\tau_g$ and defined as

$$\tau_g = -\frac{d\varphi(\omega)}{d\omega}[S] \tag{4.198}$$

The group delay has a dimension of time and is a well-known parameter in optics or in audio. In audio, the group delay evaluates the phase nonlinearity as a signal propagates through a filter, a pair of wires, a loudspeaker, and so on. If

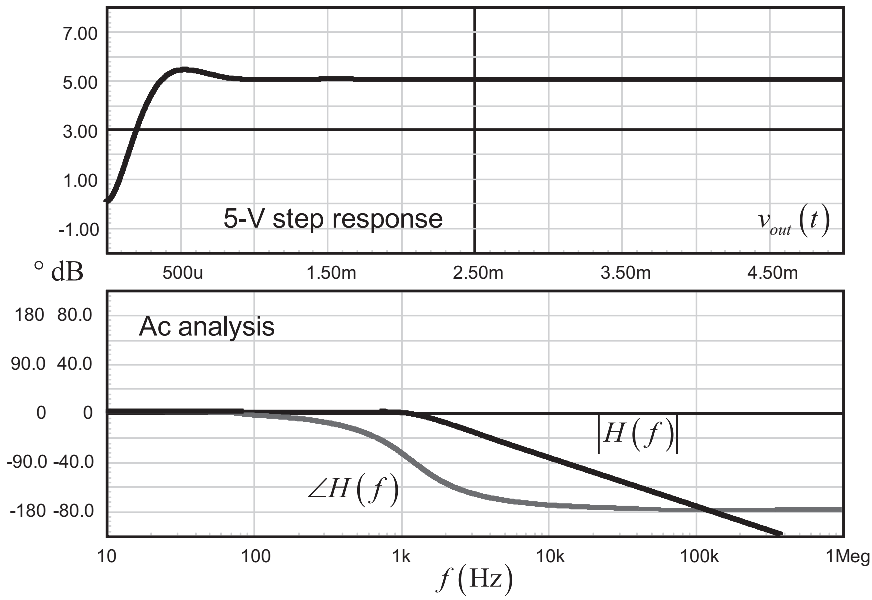



**Figure 4.77** In this graph, it is difficult to extract the quality factor value, despite a value above 0.5 given the overshoot in the time-domain response.

the phase variation is linear over a certain band of frequencies, the group delay is constant or uniform and the frequency components of the signal within the considered band are equally treated: the signal is simply shifted in time and its integrity preserved. For the opposite, if the phase suddenly changes at certain frequencies, the group delay peaks, indicating phase distortion at these points. The ears or a

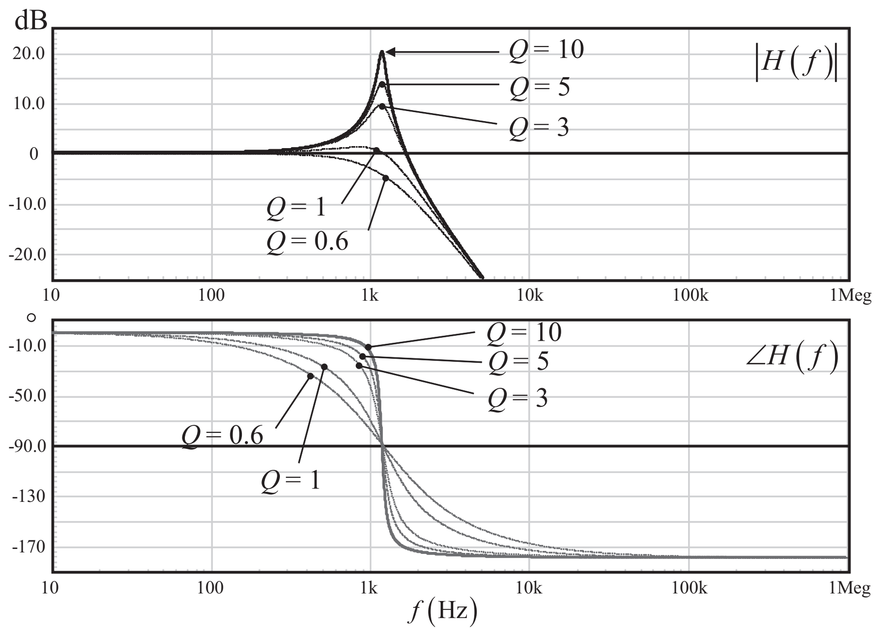



**Figure 4.78** As you can see in this plot, the phase rate of change increases as *Q* does.

measurement instrument will then reconstruct the signal by combining frequency components affected by different delays. This is obviously a situation detrimental to the signal integrity, resulting in audio distortion. Of course, if you play (loud) Motorhead's "The Ace of Spades," none will notice it and more distortion will actually be enjoyable!

To see if a relationship links the quality factor to group delay, let us study the transfer function of a second-order network:

$$H(s) = \frac{1}{1 + \frac{s}{\omega_0 Q} + \left(\frac{s}{\omega_0}\right)^2} \tag{4.199}$$

If we replace $s$ by $j\omega$, we can rewrite this function, separating the real and imaginary parts:

$$H(j\omega) = \frac{1}{\left(1 - \frac{\omega^2}{\omega_0^2}\right) + j\frac{\omega}{\omega_0 Q}} = \frac{1}{x + jy} \tag{4.200}$$

From this definition, the magnitude and the argument come easily:

$$|H(\omega)| = \frac{1}{\sqrt{x^2 + y^2}} = \frac{1}{\sqrt{\left(1 - \frac{\omega^2}{\omega_0^2}\right)^2 + \left(\frac{\omega}{\omega_0 Q}\right)^2}} \tag{4.201}$$

$$\angle H(\omega) = \tan^{-1}\left(\frac{y}{x}\right) = \tan^{-1}\left[\frac{\omega}{\omega_0 Q}\frac{1}{\left(1 - \frac{\omega^2}{\omega_0^2}\right)}\right] \tag{4.202}$$

We can now apply the group definition stated in (4.198) through (4.202):

$$\tau_g(\omega) = -\frac{d}{d\omega}\tan^{-1}\left[\frac{\omega}{\omega_0 Q}\frac{1}{\left(1 - \frac{\omega^2}{\omega_0^2}\right)}\right] = \frac{Q\omega_0(\omega^2 + \omega_0^2)}{Q^2\omega^4 - 2Q^2\omega^2\omega_0^2 + Q^2\omega_0^4 + \omega^2\omega_0^2} \tag{4.203}$$

At the resonance, where $\omega = \omega_0$, the phase change is the sharpest. This equation simplifies to

$$\tau_g = \frac{2Q}{\omega_0} \tag{4.204}$$

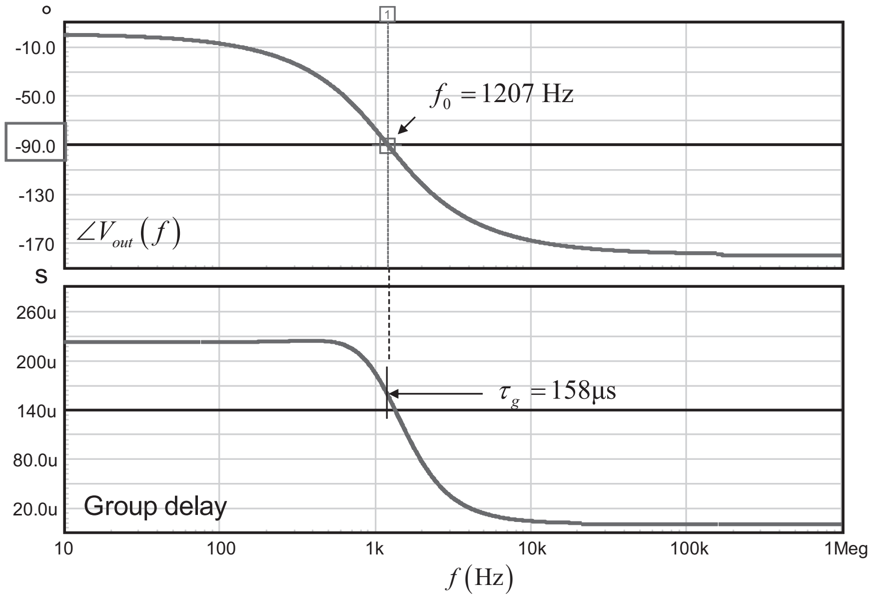



**Figure 4.79** A 0.6-*Q* brings a rather flat magnitude response without peaking. Group delay measurement helps you to compute the *Q* quite easily.

From this expression, we have our relationship linking $Q$ to the group delay at the resonance:

$$Q = \frac{\tau_g \omega_0}{2} = \tau_g \pi f_0 \tag{4.205}$$

Let us see if we can easily apply the concept to a Bode plot. We have simulated the transfer function of an $RLC$ filter tuned to 1.2 kHz, affected by a quality factor of 0.6. As shown in Figure 4.79, the transfer function argument crosses the –90° line and indicates the resonant frequency value. We have plotted the group delay by the execution of a simple script in Intusoft's Intuscope graphical tool. Under Cadence's Probe, you will use the mathematic function $G$. Type $VG(3)$ and Probe will display the group delay of node 3, for instance. From the graph, we measure a group delay of 158 $\mu$s at 1.2 kHz. Applying the formula derived in (4.205), we have

$$Q = \tau_g \pi f_0 = 1207 \times 158u \times 3.14159 = 0.599 \tag{4.206}$$

This is exactly the value we have chosen for the $RLC$ filter $Q$.

## Appendix 4C: The Phase Display in Simulators or Mathematical Solvers

The way a simulator or a mathematical solver displays the phase can sometimes puzzle people who are not used to manipulating complex numbers. As a reminder, a complex number is defined as

$$z = x + jy \tag{4.207}$$

where $x$ and $y$ are, respectively, the real and imaginary parts of the complex number $z$. Such a number and its conjugate $\overline{z}$ are represented as vectors in Figure 4.80's so-called Argand plane. The $x$-axis represents the real part, while the $y$-axis is the imaginary part. Please note that the angle $\alpha$ is positive in quadrant I (we turn counterclockwise, or positively) whereas in quadrant IV, this angle is considered negative as we turn clockwise, or negatively.

From this drawing, we can obtain the magnitude and the argument of the considered number. The *magnitude* of $z$, also found in textbooks as *modulus*—"module" in French—is obtained by applying Pythagorean geometry. We say magnitude and not *amplitude* because the length of a vector can only be positive:

$$|z| = \sqrt{x^2 + y^2} \tag{4.208}$$

In loop control theory, when we cascade transfer function blocks expressed in Laplace notation, we also manipulate complex numbers since we replace $s$ with $s = j\omega$ in harmonic analysis. If we have cascaded blocks $G$ and $H$, then the loop gain magnitude $|T|$ is the product of the individual magnitudes:

$$|T(s)| = |G(s)H(s)| = |G(s)| \cdot |H(s)| \tag{4.209}$$

Of course, if the individual amplitudes $G$ and $H$ are converted in decibels, like in a Bode plot, you would simply sum them to obtain that of $T$ since $\log(ab) = \log(a) + \log(b)$.

For a quotient in a transfer function, the magnitude is the quotient of the individual magnitudes. For instance, if we have

$$G(s) = \frac{N(s)}{D(s)} \tag{4.210}$$

then

$$|G(s)| = \frac{|N(s)|}{|D(s)|} \tag{4.211}$$

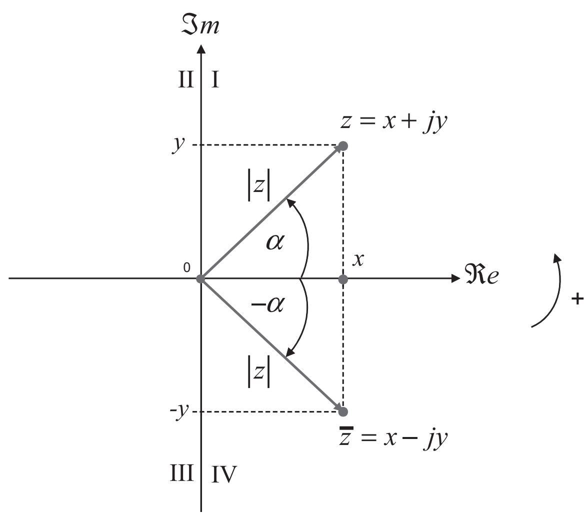



**Figure 4.80** The complex number $z$ and its conjugate expression.

The argument is obtained in a similar way if we look at the angles shown in the graph. The tangent of $\alpha$ is classically obtained by dividing the angle opposite side (the sinus amplitude) by the angle cosinus:

$$\tan(\alpha) = \frac{\sin\alpha}{\cos\alpha} = \frac{y}{x} \tag{4.212}$$

As $\alpha$ is the argument of $z$ that we want, we extract it by using the arctangent function:

$$\arg(z) = \operatorname{atan}\left(\frac{y}{x}\right) \text{ or } \tan^{-1}\left(\frac{y}{x}\right) \tag{4.213}$$

If we apply these definitions to our cascaded gain blocks $H$ and $G$, the argument of $T$ is simply the sum of these arguments:

$$\arg T(s) = \arg\left[G(s)H(s)\right] = \arg G(s) + \arg H(s) \tag{4.214}$$

For the quotient in (4.210), the argument is the difference in arguments between that of the numerator and that of the denominator:

$$\arg G(s) = \arg N(s) - \arg D(s) \tag{4.215}$$

These properties are used extensively throughout the chapters to compute and evaluate various transfer functions magnitude and arguments.

### Calculating the Tangent

The tangent function works on an angle and returns a height. This is the height of a straight line starting from the $x$-axis, tangent to the unit circle, and intersecting with a line starting from the plan origin while forming the angle of interest. Such a simple representation appears in Figure 4.81.

In this picture, you see the plane divided in four subdivisions called quadrants. Each corresponds to a certain combination of $x$ and $y$ signs in the Argand plane. For instance, in quadrant I, $x$ and $y$ are both positive, while in quadrant II $y$ is still

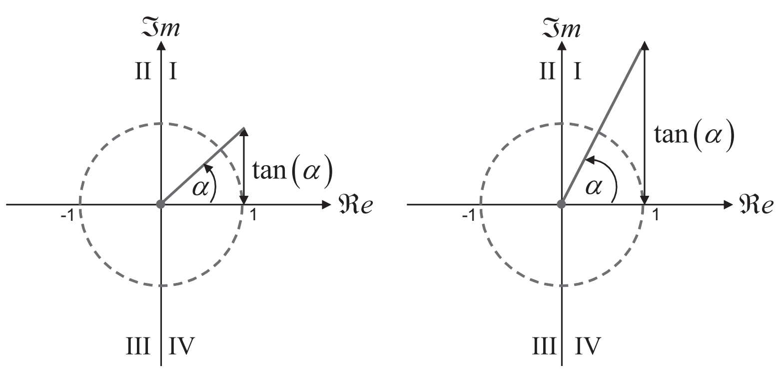



**Figure 4.81** The tangent function is defined for an angle $\alpha$ such that $-90° < \alpha < 90°$ or $-\frac{\pi}{2} < \alpha < \frac{\pi}{2}$.

positive but $x$ is negative. In quadrant I, as the angle α widens and approaches 90° or $\pi/2$, the tangent function goes to infinity as $\cos(\pi/2)$ in (4.212) equals 0. The same occurs when the angle approaches –90° or $-\pi/2$ in quadrant IV. Now what value does the tangent function return for angles appearing in quadrants II or III, for instance, 120° or 225°? To understand the answer, it is interesting to plot the sine/cosine functions and compute the tangent according to (4.212). This is what Figure 4.82 shows. The left side of the picture describes the sine and cosine functions, while the angle changes from 0° to 360° at which point, a complete revolution has been made since 0° and 360° refer to the same location on the circle. The right side of the picture displays the trigonometric circle on which some typical angles are reported. $\alpha_1$ to $\alpha_5$ are positive angles, since they start from 0 and increase while turning counter-clockwise.

In this picture, we can see that the tangent exists for the angle of $\alpha_1$ and returns a height $h_1$ located in quadrant I. When $\alpha_2$ is reached (90° or $\pi/2$), we are at the sine peak while the cosine is 0. Obviously, at this point, we have a division by zero and the tangent is infinite or undefined: this is a discontinuity in the function. Now, when we progress further on the trigonometric circle and pass the 90° angle, we reach the angle $\alpha_3$. At this point, the cosine function has become negative, making the tangent jump to a negative value calculated as $h_2$ and now located in quadrant IV. If we further increase the angle and eventually reach $\pi$, the tangent returns 0,

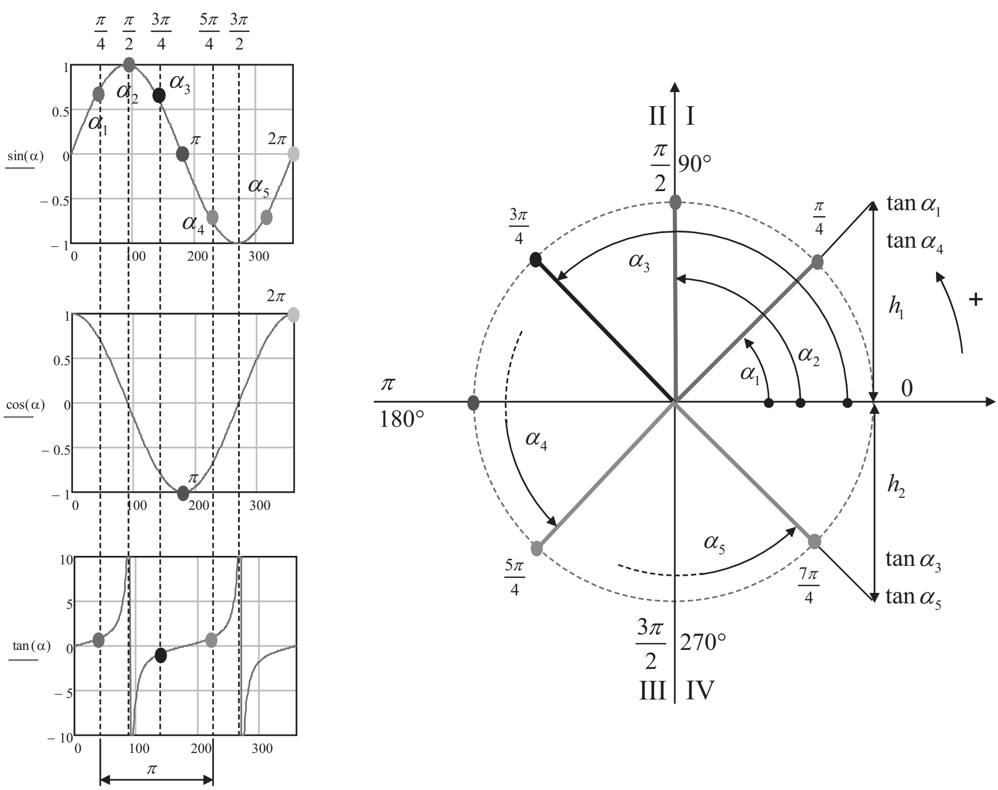



**Figure 4.82** When the angle approaches 90°, the tangent goes to infinity, and as the angle continues to widen, the tangent becomes negative: we have jumped to the other side of the circle.

$\sin(\pi) = 0$. As the angle positively progresses in quadrant III, the tangent builds up positively again in quadrant I and gives $h_1$ at the angle of $\frac{5\pi}{4}$ or 225°. The tangent increases until an angle of $\frac{3\pi}{2}$ or 270° is reached, leading to an infinite value. Beyond this point, the tangent is negative again and as the angle opens to $\frac{7\pi}{4}$ or 315°, the tangent returns a height $h_2$ located in quadrant IV.

### Accounting for the Quadrant

To illustrate the ambiguity with a more practical case, we can feed our calculator with the angles values we used in degrees and compute the tangent height $h$. This is what we will get:

$\tan(45°) = 1$

$\tan(135°) = -1$

$\tan(225°) = 1$

$\tan(315°) = -1$

For two different angles, 45°/225° or 135°/315°, distant from 180°, the results returned by the tangent function are similar. Since the arctangent is the reciprocal of the tangent and returns an angle, we have an ambiguity: is $\tan(\alpha) = 1$ coming from an angle of 45° or 225°? Arctangent will always return an angle comprised between –90° and 90°. If your complex number or vector lies in quadrants I or IV, the arctangent will return the correct value. If the vector lies in quadrant II or III, there is a problem and arctangent will return the wrong angle. Mathematically, we say that the tangent function is not bijective: for a given tangent value, two

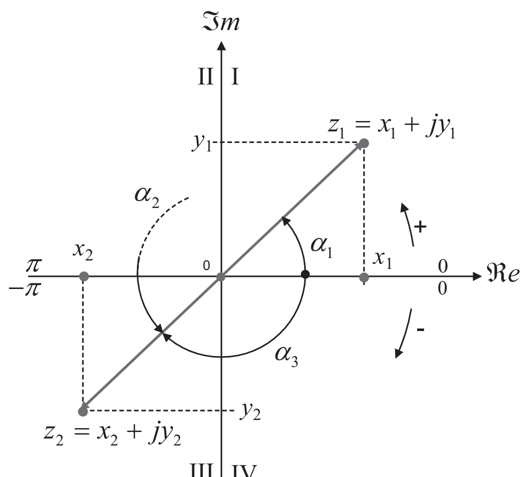



**Figure 4.83** Computing the argument of these two complex numbers will return a similar angle if no precautions are taken.

possible angles exist. A simple illustration appears in Figure 4.83. If you compute the argument of these complex numbers where $x_1 = -x_2 = 1$ and $y_1 = -y_2 = 1$, (4.213) will incorrectly return the same angle for both vectors while they are obviously different:

$$\arg(z_1) = \tan^{-1}\left(\frac{1}{1}\right) = 45° \tag{4.216}$$

$$\arg(z_2) = \tan^{-1}\left(\frac{-1}{-1}\right) = 45° \tag{4.217}$$

This can also be easily verified as the periodicity (or the modulo) of the tangent is $\pi$. We can add $\pi$ to any angle, as it won't affect its tangent calculation:

$$\tan(\alpha + \pi) = \frac{\sin(\alpha + \pi)}{\cos(\alpha + \pi)} = \frac{-\sin\alpha}{-\cos\alpha} = \tan\alpha \tag{4.218}$$

In Figure 4.83, we have considered angles belonging to the closed-open interval $[0°, 360°)$ or $[0, 2\pi)$. These are the angles $\alpha_1$ and $\alpha_2$ in Figure 4.83. In the field of complex numbers, however, the so-called *principal argument* of a complex number is defined in the open-closed interval $(-\pi, \pi]$ or $(-180°, 180°]$. Therefore, we will consider positive arguments from 0 to π in quadrants I and II, $y > 0$, while turning counter-clockwise. For the opposite, we will consider negative arguments from 0 to less than $-\pi$ in quadrants III and IV, $y < 0$, while turning clockwise. Thus, in Figure 4.83, assuming $\alpha_1 = 45°$, the correct answer for the argument of $z_2$ is found by subtracting $\pi$ (we turn clockwise) to $\alpha_1$: $\alpha_3 = \alpha_1 - 180° = -135°$.

The reason why 360° or $2\pi$ is excluded from the angle definition range comes from the fact that 0° and 360° refer to identical positions on the circle. So when an angle opens beyond 359.99°, the next position is ... 0°. The same applies to the range $(-\pi, \pi]$ or $(-180°, 180°]$ as $-\pi$ and $\pi$ (or −180° and 180°) refer to a similar position on the circle. Applying this convention, if $y_2$ is 0 in Figure 4.83, the magnitude of $z_2$ becomes $x_2$ (−1, a negative real number) and its argument is $\pi$ or 180°.

This technique will be widely used when calculating the argument of inverting compensators. It is important that you feel comfortable with it. Assume you have a zero and an origin pole wired in an inverting configuration. The transfer function is (please note the negative sign):

$$G(s) = -\frac{\omega_{po}}{\omega_z}\left(1 + \frac{\omega_z}{s}\right) \tag{4.219}$$

The argument of the compensator is

$$\arg G(j\omega) = \arg\left(-1 - \frac{\omega_z}{j\omega}\right) = \arg\left(-1 + j\frac{\omega_z}{\omega}\right) = \tan^{-1}\left(-\frac{\omega_z}{\omega}\right) = -\tan^{-1}\left(\frac{\omega_z}{\omega}\right) \tag{4.220}$$

As you can see, the complex number lies in quadrant II ($x = -1$); therefore, the correct answer is obtained by adding $\pi$ to (4.220):

$$\arg G(\omega) = \pi - \tan^{-1}\left(\frac{\omega_z}{\omega}\right) \tag{4.221}$$

This formula can be further tweaked. We need to use a formula also popular in this book:

$$\tan^{-1}(x) + \tan^{-1}\left(\frac{1}{x}\right) = \frac{\pi}{2} \tag{4.222}$$

From which we can state

$$\tan^{-1}\left(\frac{1}{x}\right) = \frac{\pi}{2} - \tan^{-1}(x) \tag{4.223}$$

Substituting (4.223) in (4.221), we have

$$\arg G(\omega) = \pi - \tan^{-1}\left(\frac{\omega_z}{\omega}\right) = \pi - \left[\frac{\pi}{2} - \tan^{-1}\left(\frac{\omega}{\omega_z}\right)\right] = \frac{\pi}{2} + \tan^{-1}\left(\frac{\omega}{\omega_z}\right) \tag{4.224}$$

At dc, the argument is $\frac{\pi}{2}$ or –270°, and, for high frequency, the argument is π or –180°. A similar result is obtained from (4.221).

### Improving the Arctangent Function

To return the right angle, solvers and simulators must know the quadrant in which the evaluated complex number resides. One known function is *atan*2 and is implemented in Mathcad®. The function observes the signs of $y$ and $x$ and, depending on the quadrant, applies the following scaling factors as detailed in [1]:

$$\operatorname{atan2}(x,y) = \begin{cases} \tan^{-1}\left(\frac{x}{y}\right) & x > 0 \\ \pi + \tan^{-1}\left(\frac{x}{y}\right) & y \geq 0, x < 0 \\ -\pi + \tan^{-1}\left(\frac{x}{y}\right) & y < 0, x < 0 \\ \frac{\pi}{2} & y > 0, x = 0 \\ -\frac{\pi}{2} & y < 0, x = 0 \\ \text{undefined} & y = 0, x = 0 \end{cases} \tag{4.225}$$

The *atan*2 function will return angles moving from $\pi$ to $-\pi$ with a discontinuity as the angles approach $-180°$ or $-\pi$. When $x = 0$, where the tangent has an infinite positive or negative height, the function properly returns the angle of $\pm\frac{\pi}{2}$. A possible implementation of *atan*2 is proposed by [1] and gives adequate results:

$$\operatorname{atan}2(x,y) = 2\tan^{-1}\left(\frac{y}{\sqrt{x^2+y^2}+x}\right) \tag{4.226}$$

To compare these functions, we have plotted *atan* and *atan*2 from the sine and cosine waveforms already used in Figure 4.82. The plots appear in Figure 4.84.

The upper right corner shows the result delivered by *atan* and gives an angle bounded between $-\frac{\pi}{2}$ and $\frac{\pi}{2}$: the result is displayed *modulo* $\pi$ or with a periodicity of $\pi$ as also shown in Figure 4.82.

The lower-right corner shows the results computed by the *atan*2 function. As expected, the phase nicely increases to 180° and suddenly jumps to $-179°$ ($-180$ or $-\pi$ is excluded). This time, the result is displayed *modulo* 360 or $2\pi$. Some simulators display the phase in this mode. However, in some cases, you would prefer to map these results in the segment $[0,360)$ or $[0,2\pi)$ for an easier reading. A possible way to program this in Mathcad® is as follows:

$$360_{\text{map}}(x,y) := \left|\begin{array}{l}\operatorname{atan2}(x,y)+2\pi \text{ if } \operatorname{atan2}(x,y)<0\\ \operatorname{atan2}(x,y) \text{ otherwise}\end{array}\right.$$

The result must then be multiplied by $360/2\pi$ for a proper display in degrees.

If you do not want to type these lines, the built-in function *angle* gives similar results as shown in Figure 4.85 in which we used Figure 4.84's sine and cosine functions. The discontinuity has gone, and the graph shows a linearly increasing angle.

When you approach 360°, the function jumps back to zero, indicating a complete circle revolution.

### Phase Display in a SPICE Simulator

In some simulation packages, the graphical analysis tool does the 0°–360° mapping automatically for you. This is the case for Intusoft's graphical analysis tool Intuscope, where the function *phaseextend* is implemented. This software routine observes the phase discontinuity in the *atan*2 function and applies the strategy of the $360_{\text{map}}$ function. There is a slight difference, though, as it authorizes the phase to extend beyond $-360°$ if necessary. Let us see how this technique applies to different circuits.

As a first example, we have selected a simple op amp-based inverter drawn in Figure 4.86.

The gain of such a simple cell is easily derived, and we find

$$|G(s)| = \left|\frac{V_{out}(s)}{V_{in}(s)}\right| = -\frac{R_f}{R_i} \tag{4.227}$$

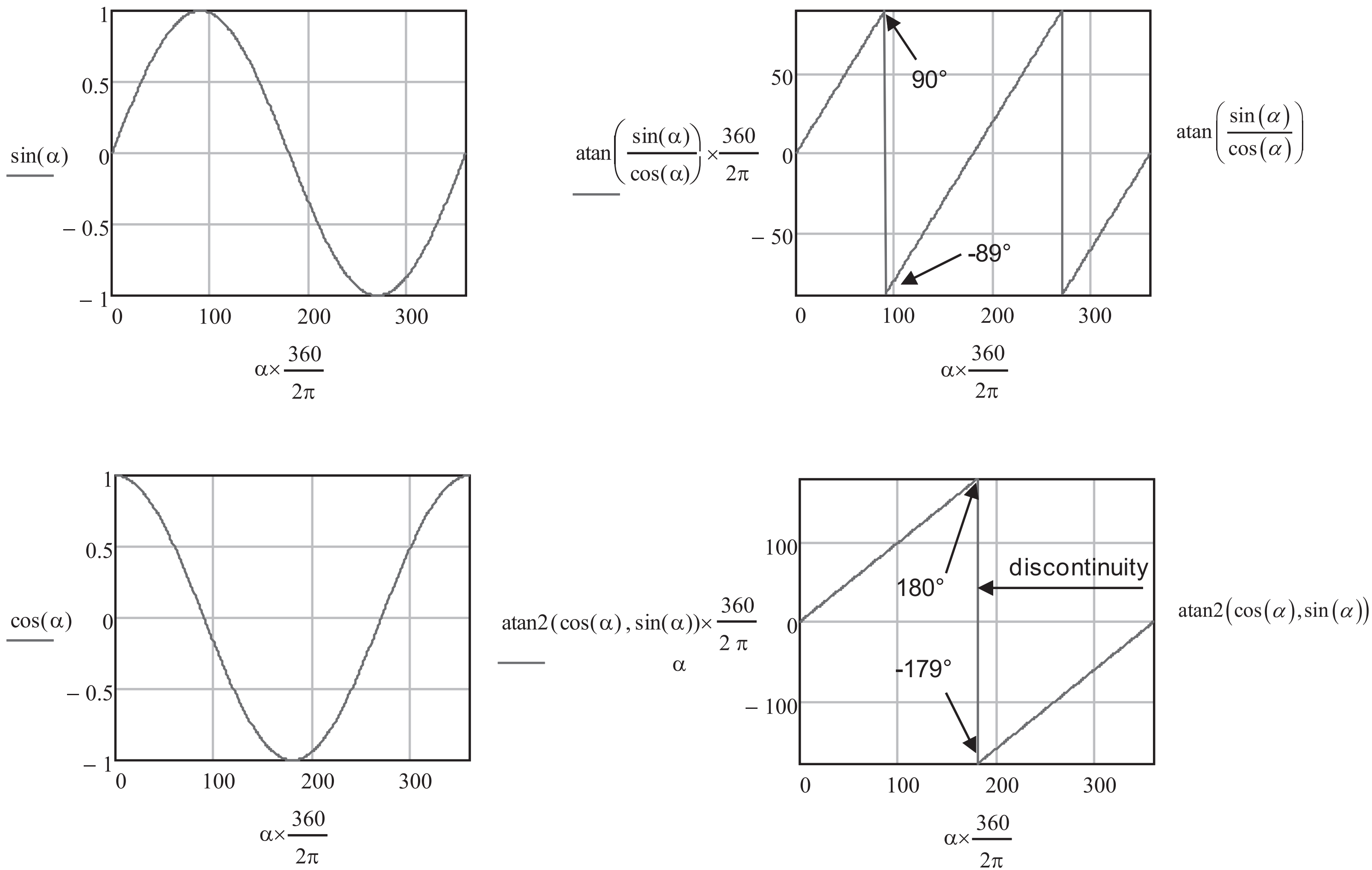



**Figure 4.84** If the *atan* expression returns an angle between –90° and 90°, *atan2* accounts for the quadrant and returns an answer corrected as expressed in (4.225).

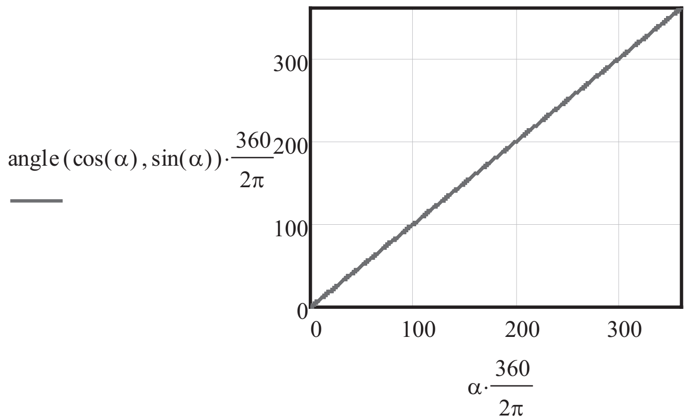



**Figure 4.85** The function *angle* automatically maps angles into the [0,2$\pi$) segment.

The mathematical argument of this transfer function is the argument of a negative real (no imaginary part as when $y_2 = 0$ in Figure 4.83) and equals

$$\arg\left(-\frac{R_f}{R_i}\right) = \pi \tag{4.228}$$

The simulation results for equal resistors values (–1 gain) are given in Figure 4.87. The upper section of the figure confirms the calculation with a displayed 180° phase.

However, at school, we learned that the phase brought by an inverting circuit was –180° and not 180° as expressed by (4.228). Could the simulator be wrong?

Mathematically, talking about an angle of 180° or –180° is the same: it refers to a similar angle in the polar plane, as an angle is always defined $\pm k2\pi$ or $\pm k \times 360°$: should you add or subtract a complete circle revolution to an angle, you return to the same starting point on the circle. If you add –360° to 180°, you obtain –180° as shown in the lower section of Figure 4.87. If this option gives more physical sense to the representation of $G(s)$ in (4.227), both diagrams in Figure 4.87 are, however, mathematically equal.

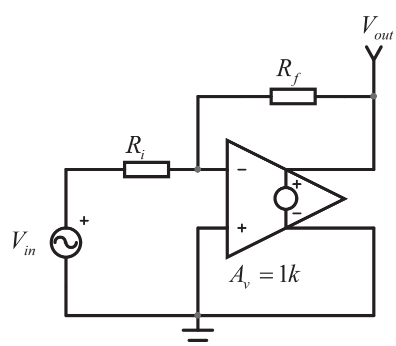



**Figure 4.86** A simple inverter made around a voltage-controlled control source.

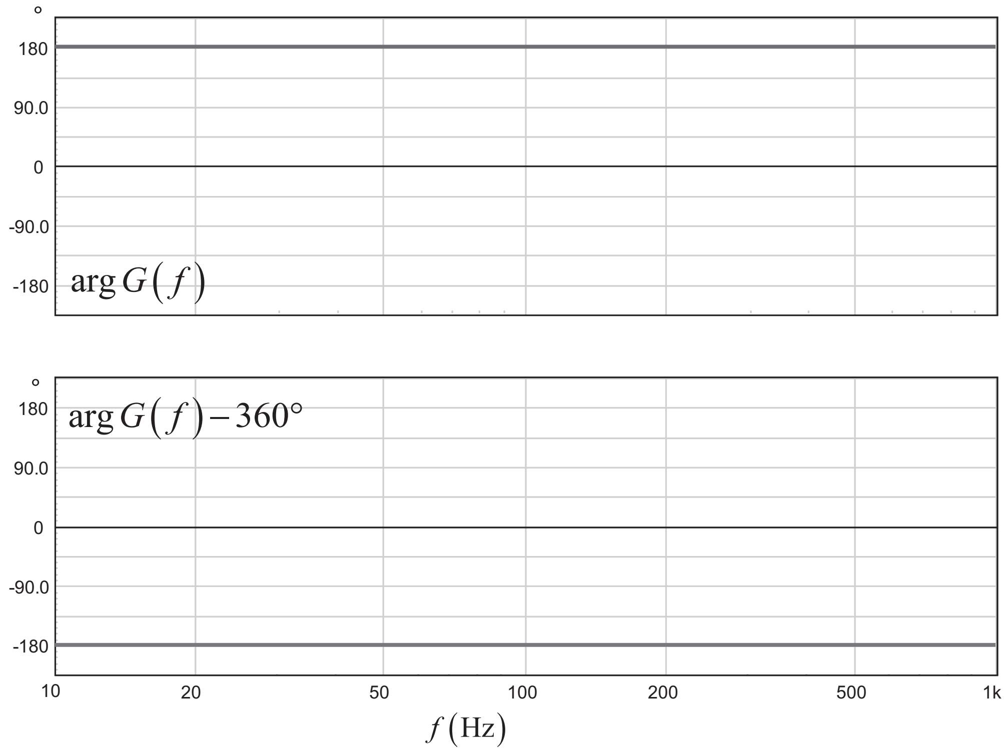



**Figure 4.87** The simulator returns the mathematical value of the argument: 180° or $\pi$.

In Figure 4.89, we plotted the signals coming out from an inverting integrator. We can say that $w_2$ lags $w_1$ by 270°, but if we select a period on the right side of the picture, we can also say that $w_2$ leads $w_1$ by 90°! Again, both statements are mathematically equal: $90° = -270° + 360°$. If you plot the phase response of this inverting integrator, what will SPICE display? Well, the inverter argument is 180° according to (4.228) and the argument of the origin pole is –90°. Summing these two values will give a displayed phase of 90°, equivalent to an argument of –270°, as confirmed by Figure 4.89. We will use the second notation in our phase calculations for the sake of compliance with most of the theory books. The negative phase notation emphasizes, after all, the fact that the time-domain response of the considered network is always delayed compared to the excitation signal.

Most of simulation packages use the function *atan*2 to display an argument signed between $-\pi$ (excluded) and $\pi$. Some graphical investigation tools, such as Intuscope, enhance the *atan*2 function with a software routine known as *phaseextend* that computes the phase down to –360° and beyond. It works more or less like the $360_{map}$ routine we detailed. When you start an ac analysis, the simulation engine calculates the argument of the transfer function from the very first frequency points. The returned argument is that of the function *atan*2, signed between $-\pi$ (excluded) and $\pi$. As the ac sweep proceeds, if the function senses an argument discontinuity between adjacent simulation points—for instance, when a jump occurs from –179° to 180° (or vice versa)—*phaseextend* does its job and smoothes the discontinuity,

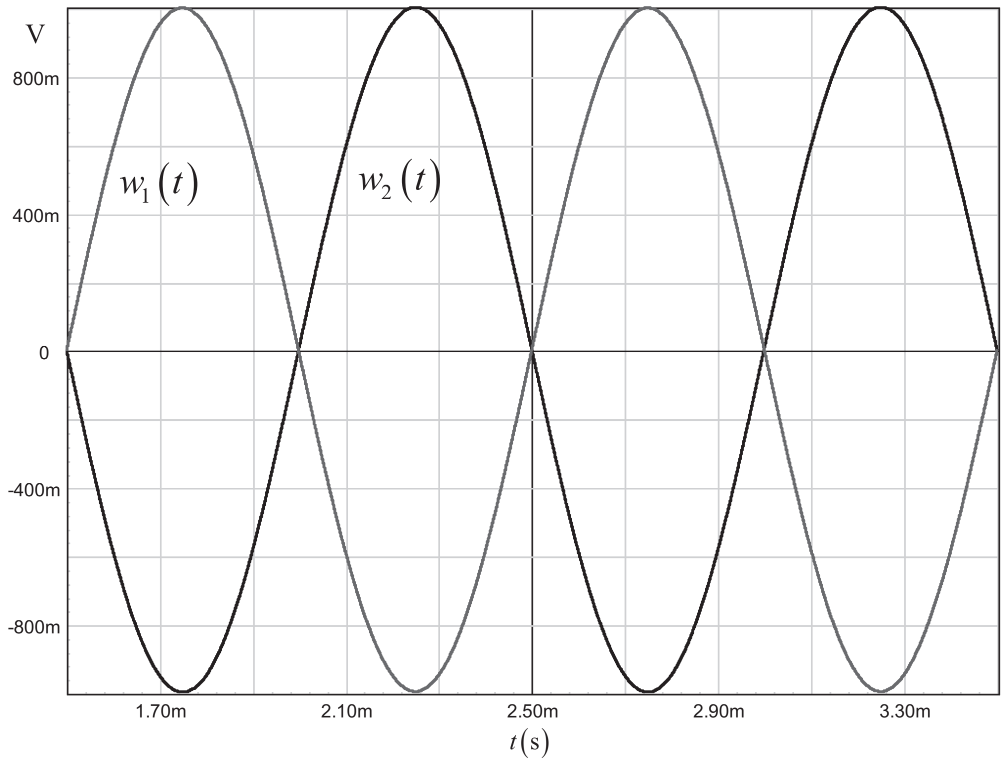



**Figure 4.88** These two signals are out of phase. However, can you say that waveform 1 precedes waveform 2 by 180° or waveform 2 lags waveform 1 by 180°?

returning an argument down to –360°. Let us check this theory on the Figure 4.90 third-order *RC* network on which we launched an ac sweep.

From the simulation data, we can use the Intuscope built-in calculation engine and apply macros to the real and imaginary portions of the $V_{out}$ vector. This is what we did to compare the results returned by the classical arctangent function and the function described by (4.226). The results are plotted in Figure 4.91. In the lower section, the *atan* function returned results signed between $-\frac{\pi}{2}$ and $\frac{\pi}{2}$, as expected. In the middle section, we implemented (4.226), and it nicely returned an argument signed between $-\pi$ (excluded) and $\pi$. You can observe that the phase toggling point in the returned value occurred a little before 400 Hz. Finally, in the upper section of the figure, the phase lags from 0° down to 270° (we have three poles) using the *phaseextend* function that efficiently smoothes the reported discontinuity around 400 Hz. Please note that the phase starts from 0 since at 1 Hz, where the ac sweep started, the returned argument was 0.

In a new simulation, we did not start the ac sweep from 1 Hz, but from 400 Hz. At this frequency, the simulation engine evaluated the argument returned by *atan*2 to be already beyond 180°. This explains why Intuscope now displays the phase from 180° rather than 0° as in the previous case. This is what you can see in Figure 4.92, and it is correct.

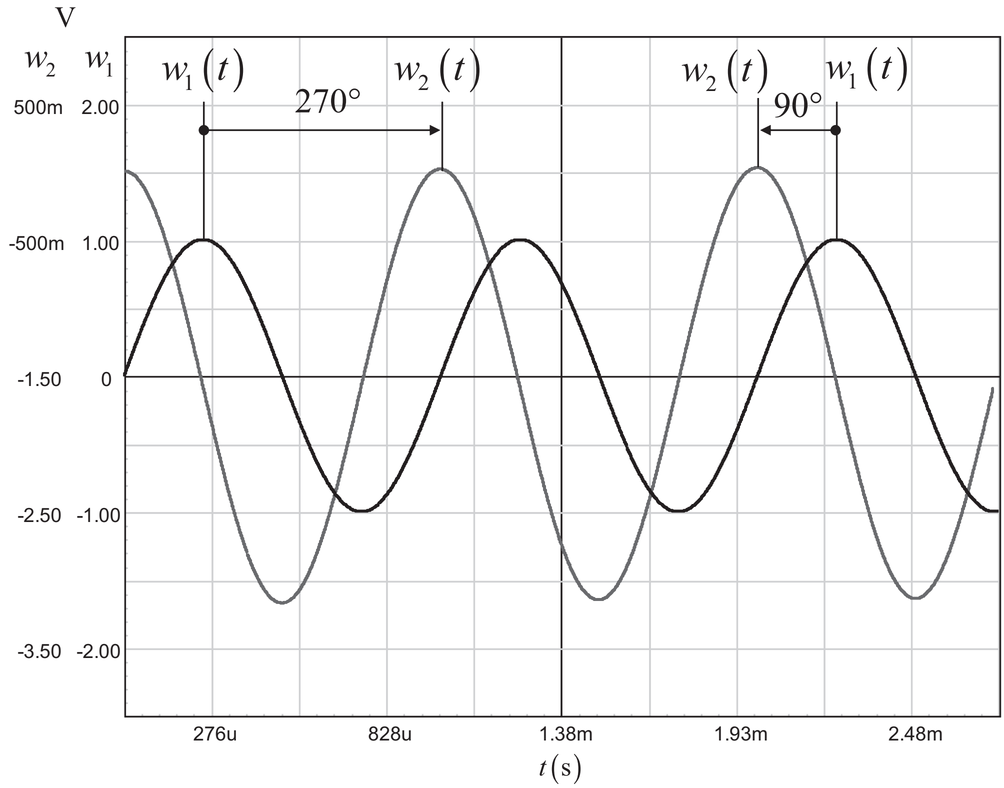



**Figure 4.89** In this picture, you can say $w_2$ leads $w_1$ by 90° or $w_2$ lags $w_1$ by 270°; it is a similar statement!

## Conclusion

An engineer who is not familiar with the intricacies of circuit simulation can often be puzzled by phase or argument evaluations. We have seen that engineering judgment is necessary to interpret the results displayed by the graphical viewer. The change in the start frequency is a typical example where attention is required to understand the answer.

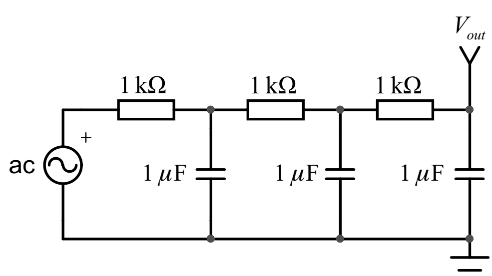



**Figure 4.90** This third-order *RC* network has a 0 argument at dc and its phase smoothly shifts to –270°.

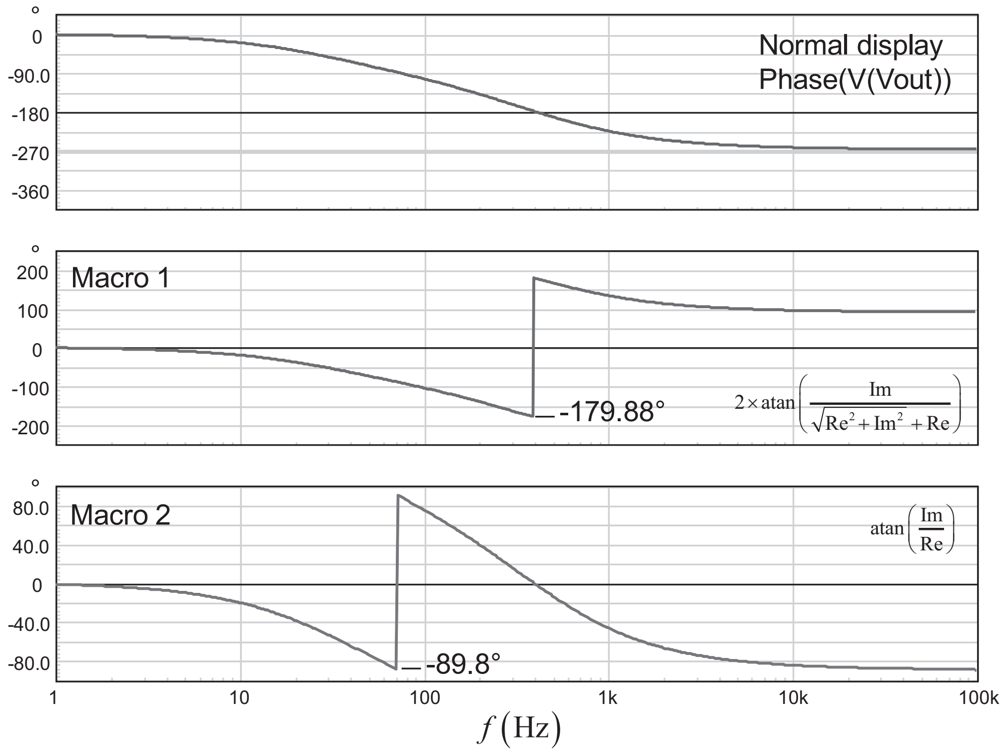



**Figure 4.91** The *phaseextend* function built in Intuscope displays the phase down to –270°. Using macros, we can nicely reproduce the results predicted in Figure 4.84.

## Reference

[1] Wikipedia contributors, "atan2," http://en.wikipedia.org/wiki/Atan2, last accessed June 3, 2012.

## Appendix 4D: Impact of Open-Loop Gain and Origin Pole on Op Amp-Based Transfer Functions

An operational amplifier is often considered a perfect device, featuring an infinite open-loop gain and infinite bandwidth. In reality, the open-loop gain is finite, and designers place a pole at low frequency for stability purposes. The typical ac response of such op amp, a $\mu$A741, for instance, appears in Figure 4.93 and shows a typical open-loop gain $A_{OL}$ of 106 dB together with a low-frequency pole $f_p$ placed below 10 Hz.

When we consider an op amp to build a compensator or a simple amplifier, we usually do not consider the origin pole and the open-loop gain, assuming they can be neglected. However, it is always interesting to understand how these parameters can affect the final performance, especially if they are moving from lot to lot in production or if they change with temperature. Data-sheet indications showing the wide spread of a parameter should always trigger your interest in investigating

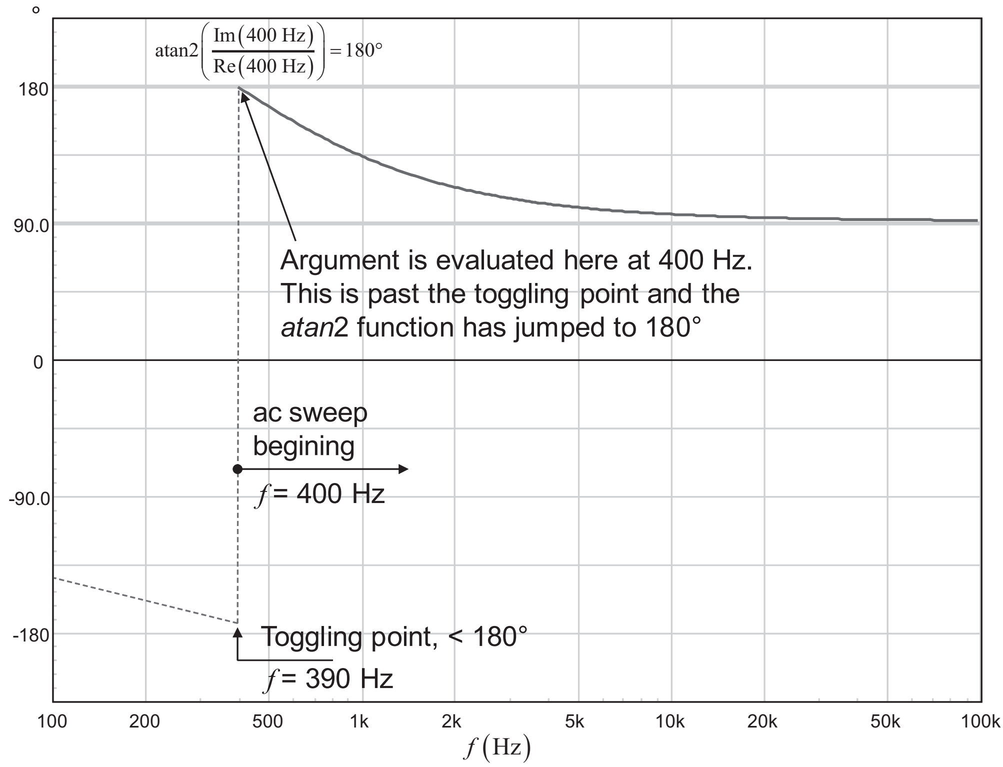



**Figure 4.92** The displayed phase starts from 180° because the simulator evaluates the argument at a frequency point (400 Hz), where the *atan*2 function has already jumped to 180°.

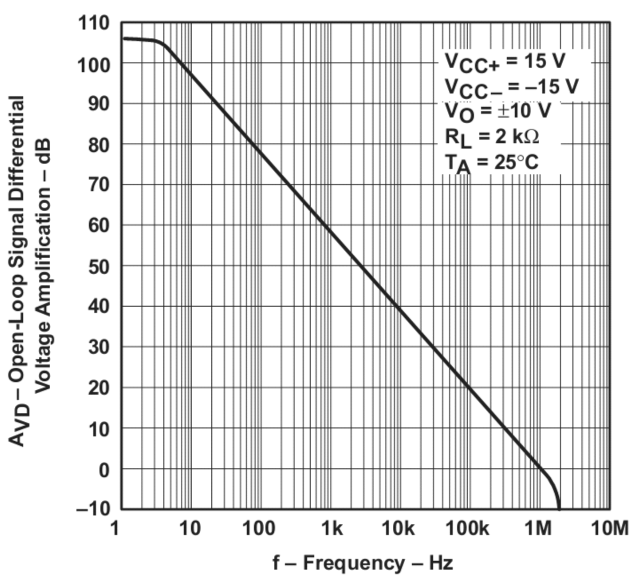



**Figure 4.93** The open-loop gain is around 106 dB typically for this $\mu$A741 from TI, and the low frequency pole appears below 10 Hz.

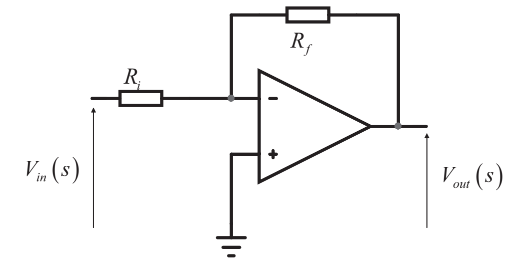



**Figure 4.94** A simple inverter built around an op amp.

the consequences of its variations upon the final design performance. For instance, the $\mu$A741 gain is typically indicated to be 200k (106 dB) but can potentially go down to 20k (86 dB) in some cases. To understand how this variation will affect the design, you must run the analytical analysis and assess the final impact. From the derived equations, you will, as design engineer, decide to either include and compensate for the effects or simply ignore them.

Let us consider the circuit proposed in Figure 4.94. You recognize a simple inverter whose gain depends on $R_f$ and $R_i$. We know that the gain of such a circuit is $-R_f/R_i$, considering an infinite open-loop gain and no low-frequency pole. To see the effects of these elements, we need to make them appear in the op amp equivalent circuit. A simplified representation including these new comers appears in Figure 4.95.

To calculate the transfer function of such an arrangement, we need a few equations. Let's start with the output voltage, $V_{out}$:

$$V_{out}(s) = A_{OL} \cdot \varepsilon(s) \frac{1}{1+\frac{s}{\omega_p}} \tag{4.229}$$

The error voltage, $\varepsilon$, is equal to the difference between the negative and positive inputs:

$$\varepsilon = V_{(+)} - V_{(-)} \tag{4.230}$$

As the positive input is grounded, let's apply the superposition theorem to this linear network and derive the negative input node voltage. The negative input voltage

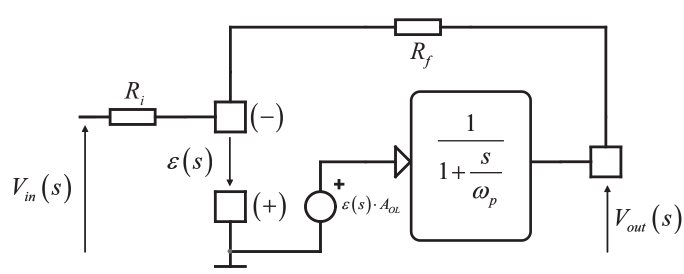



**Figure 4.95** When including the open-loop gain and the origin pole, the circuit slightly complicates.

is the sum of the voltage seen at this pin when $V_{out}$ is grounded plus the voltage seen at the same pin when $V_{in}$ is grounded. Therefore, we have

$$\varepsilon(s) = -\left[V_{out}(s)\frac{R_i}{R_i + R_f} + V_{in}(s)\frac{R_f}{R_i + R_f}\right] \tag{4.231}$$

If we substitute this value into (4.229), re-arrange the expression, and extract the ratio $V_{out}/V_{in}$, we obtain the following equation:

$$G(s) = \frac{V_{out}(s)}{V_{in}(s)} = -\frac{R_f}{\frac{R_f + R_i}{A_{OL}}\left(1+\frac{s}{\omega_p}\right) + R_i} \tag{4.232}$$

This transfer expression does not really conform to the format introduced in Chapter 2. To gain insight into the function, it must be rearranged to fit the following equation:

$$G(s) = G_0 \frac{1}{1+\frac{s}{\omega_{peq}}} \tag{4.233}$$

where $G_0$ is the low-frequency or dc gain and $\omega_{peq}$ the new pole brought by the structure.

Let us develop and rearrange the (4.232) denominator:

$$D(s) = R_i + \frac{R_f + R_i}{A_{OL}} + s\left(\frac{R_i + R_f}{A_{OL}\omega_p}\right) \tag{4.234}$$

From this expression, we can factor the left term $R_i + \frac{R_f + R_i}{A_{OL}}$:

$$D(s) = \left(R_i + \frac{R_f + R_i}{A_{OL}}\right)\left[1 + s\left(\frac{R_i + R_f}{A_{OL}\omega_p}\right)\frac{1}{\left(R_i + \frac{R_f + R_i}{A_{OL}}\right)}\right] \tag{4.235}$$

If we develop and factor the right term, we have

$$D(s) = \left(R_i + \frac{R_f + R_i}{A_{OL}}\right)\left[1 + s\frac{R_i + R_f}{\omega_p(R_f + R_i + A_{OL}R_i)}\right] \tag{4.236}$$

The complete expression can be rewritten as follows:

$$G(s) = -\frac{R_f}{R_i + \frac{R_f + R_i}{A_{OL}}}\frac{1}{1 + s\frac{R_i + R_f}{\omega_p(R_f + R_i + A_{OL}R_i)}} \tag{4.237}$$

in which

$$G_0 = -\frac{R_f}{R_i + \frac{R_f + R_i}{A_{OL}}} \tag{4.238}$$

and the equivalent pole is

$$\omega_{peq} = \omega_p \frac{R_f + R_i + A_{OL} R_i}{R_i + R_f} \tag{4.239}$$

In these expressions, when the open-loop gain is really high, the inverter gain $G_0$ is solely set by $R_f$ and $R_i$ and equals $-R_f/R_i$. This is, by the way, the benefit of feedback where, despite variations of the open-loop gain $A_{OL}$, the transmission gain depends solely on the external elements $R_f$ and $R_i$. With high open-loop gains, the equivalent pole is relegated to infinity where it can be neglected.

Let's go through a quick example to see if these statements are always valid. Suppose we want to build a 30-kHz bandwidth inverter featuring a gain of 10. For this purpose, we will choose resistors such as $R_f$ = 10 kΩ and $R_i$ = 1 kΩ. With a $\mu$A741 dc-gain of 200k and a low-frequency pole located at 10 Hz, we obtain the following values from (4.238) and (4.239):

$$G_0\Big|_{A_{OL}=200k} = -9.999 \tag{4.240}$$

$$f_{peq}\Big|_{A_{OL}=200k} \approx 182 \text{ kHz} \tag{4.241}$$

From these values, we can see that we will have a flat response up to 182 kHz with a gain of 10, as expected. The ac response appears in Figure 4.96(a), and it is flat at 30 kHz: the buffer works as expected.

If we now use the lower op amp open-loop gain $A_{OL}$ of 20k, the new values for the transmission gain and the pole are the following ones:

$$G_0\Big|_{A_{OL}=20k} = -9.995 \tag{4.242}$$

$$f_{peq}\Big|_{A_{OL}=20k} \approx 18.2 \text{ kHz} \tag{4.243}$$

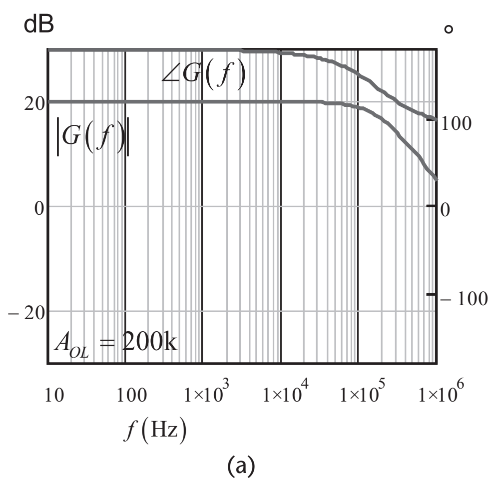



**Figure 4.96** When the op amp open-loop gain falls to 20k, the inverter gain is almost unchanged, but the ac response is seriously affected. The pole has shifted below 20 kHz, and the expected flat ac response up to 30 kHz is lost.

The ac response is given in the Figure 4.96(b) where you can see that the buffer ac response now suffers beyond 10 kHz. Your design is no longer able to pass the specification. The phase lag can bring instability if this buffer is included in a compensator chain, for instance.

What can you do then? You can select an op amp with lower open-loop gain variations, but it is not unusual to find ratios of 1 to 3 or even more, as we have seen. You can also choose a high-speed type of op amp. This device features a moderate open-loop gain but the designers pushed the low-frequency pole way up the frequency axis. For instance, National-Semiconductor LM6132BI data sheet says the typical open-loop gain is 15k, which is much lower than the $\mu$A741 minimum value. To calculate the position of the low-frequency pole, we can use another parameter given by op amp manufacturers, the gain bandwidth product GBW. It is defined as

$$\mathrm{GBW} = \mathrm{A}_{OL} \times f_p \tag{4.244}$$

where $A_{OL}$ is the op amp open-loop gain and $f_p$ is the point where the ac magnitude drops by 3 dB. This number is constant, given the single-pole response of the compensated op amp: when the amplitude drops by 10 (–20 dB), the frequency has increased also by 10 (a decade). For the LM6132BI, the typical GBW product is 10 MHz. From this value, we can extract the position of the low-frequency pole:

$$f_p = \frac{\mathrm{GBW}}{A_{OL}} = \frac{10Meg}{15k} = 667 \text{ Hz} \tag{4.245}$$

If we plug the new open-loop gain of 15k and the 667-Hz pole position in (4.239), our inverter circuit with a gain of 10 will have a new pole situated at 910 kHz. When the open-loop gain drops to 6k, as indicated in the data sheet, together with a minimum GBW of 7 MHz ($f_p$ = 1.1 kHz), the new pole will be located at 600 kHz. We are safe with our 30-kHz bandwidth requirements.

### The Integrator Case

If you now consider an integrator as drawn in Figure 4.26, the resistance $R_f$ in (4.232) is replaced by the capacitor impedance:

$$Z_f = \frac{1}{sC_1} \tag{4.246}$$

while $R_i$ remains in place. Developing (4.232) accounting for these changes gives

$$G(s) = \frac{\dfrac{1}{sC_1}}{\dfrac{\dfrac{1}{sC_1} + R_1}{A_{OL}}\left(1 + \dfrac{s}{\omega_p}\right) + R_1} = \frac{A_{OL}\omega_p}{s + \omega_p + s^2 R_1 C_1 + sR_1C_1\omega_p + sA_{OL}R_1C_1\omega_p} \tag{4.247}$$

Rearranging this equation by factoring $\omega_p$ and considering $\omega_{po} = 1/R_1C_1$ leads to

$$G(s) = \frac{A_{OL}}{1 + s\left(\dfrac{1}{\omega_p} + \dfrac{1}{\omega_{po}} + \dfrac{A_{OL}}{\omega_{po}}\right) + s^2\left(\dfrac{1}{\omega_p \omega_{po}}\right)} \tag{4.248}$$

This is a second-order equation. In the denominator second term, if we consider a large open-loop gain $A_{OL}$, the expression simplifies to

$$G(s) \approx \frac{A_{OL}}{1+s\left(\frac{A_{OL}}{\omega_{po}}\right)+s^2\left(\frac{1}{\omega_p \omega_{po}}\right)} \tag{4.249}$$

We can rewrite this expression and make it fit the classical second-order polynomial form:

$$G(s) = \frac{1}{1+\frac{s}{\omega_0 Q}+\left(\frac{s}{\omega_0}\right)^2} \tag{4.250}$$

We need to identify the terms $\omega_0$ and $Q$ with (4.249) and (4.250):

$$\omega_0 = \sqrt{\omega_{po}\omega_p} \tag{4.251}$$

$Q$ is obtained by solving the simple equation

$$\frac{A_{OL}}{\omega_{po}} = \frac{1}{Q\omega_0} \tag{4.252}$$

which gives

$$Q = \frac{\omega_{po}}{A_{OL}\omega_0} \tag{4.253}$$

Knowing that the open-loop gain $A_{OL}$ is large, $Q$ is naturally a very small value.

The roots or the poles of the denominator are obtained by using a formula introduced in Chapter 3, (3.35):

$$s_1, s_2 = \frac{\omega_0}{2Q}\left(\pm\sqrt{1-4Q^2}-1\right) \tag{4.254}$$

From (4.253) we extract $\omega_0$:

$$\omega_0 = \frac{\omega_{po}}{A_{OL}Q} \tag{4.255}$$

If we substitute this definition in (4.254), we obtain the first root:

$$s_1 = \frac{\omega_{po}}{A_{OL}} \frac{\left(\sqrt{1-4Q^2}-1\right)}{2Q^2} \tag{4.256}$$

Considering first order terms only, we know that

$$(1+x)^a \approx 1+ax \tag{4.257}$$

(4.256) can be rewritten as

$$s_1 \approx \frac{\omega_{po}}{A_{OL}} \frac{1-\frac{4Q^2}{2}-1}{2Q^2} = -\frac{\omega_{po}}{A_{OL}} \tag{4.258}$$

The pole $\omega_1$ is located at

$$\omega_1 \approx \frac{\omega_{po}}{A_{OL}} \tag{4.259}$$

The second pole is calculated with $s_2$:

$$s_2 = \omega_0 \frac{\left(-\sqrt{1-4Q^2}-1\right)}{2Q} \tag{4.260}$$

Applying (4.257) and considering $Q$ as a small quantity, we have

$$s_2 \approx -\omega_0 \frac{\left(1-\frac{4Q^2}{2}\right)-1}{2Q} = \omega_0\left(\frac{2Q^2-2}{2Q}\right) \approx -\frac{\omega_0}{Q} \tag{4.261}$$

If we substitute in this equation the definitions from (4.251) and (4.253), we have

$$s_2 = -\frac{\sqrt{\omega_{po}\omega_p}}{\frac{\omega_{po}}{A_{OL}\sqrt{\omega_{po}\omega_p}}} \tag{4.262}$$

Extracting the magnitude of this root leads to the second pole definition:

$$\omega_2 \approx \omega_p A_{OL} \tag{4.263}$$

$\omega_1$ is nothing but the 0-dB crossover pole divided by the op amp open-loop gain $A_{OL}$, whereas the second pole is the gain bandwidth product GBW defined in (4.244). If we neglect the high-frequency pole, the transfer function simplifies to

$$G(s) \approx \frac{A_{OL}}{1+\frac{s}{\omega_1}} \tag{4.264}$$

Let's assume the following component values were used to build Figure 4.26, which now includes the op amp open-loop gain and a low-frequency pole located at 30 Hz:

$R_1 = 10\ \text{k}\Omega$

$C_1 = 0.1\ \mu\text{F}$

$A_{OL} = 10000$

$f_p = 30\ \text{Hz}$

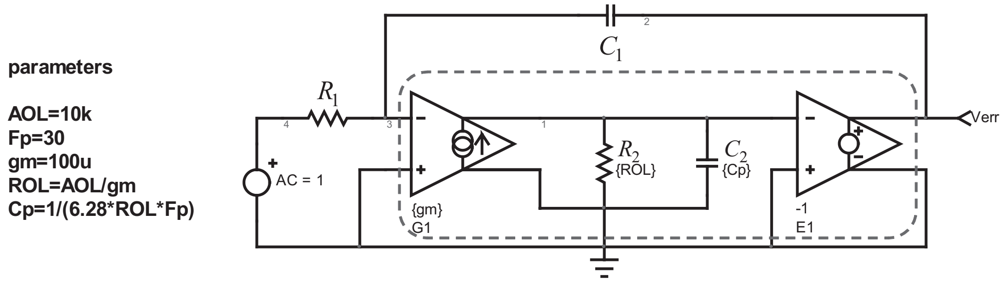



**Figure 4.97** A simple setup will tell us if our equations are correct.

If we plot the ac response from such an integrator, then from (4.259) and (4.263) we should see inflexion points at the following frequencies:

$$f_{po} = \frac{1}{2\pi R_1 C_1} = \frac{1}{6.28 \times 10k \times 0.1u} = 159\text{ Hz} \tag{4.265}$$

$$f_1 = \frac{\omega_{po}}{2\pi A_{OL}} = \frac{1}{2\pi R_1 C_1 A_{OL}} \approx 16\text{ mHz} \tag{4.266}$$

$$f_2 = f_p A_{OL} = 30 \times 10000 = 300\text{ kHz} \tag{4.267}$$

To verify these numbers, we have built a simple ac setup as shown in Figure 4.97. The first stage is a voltage-controlled current source affected by a transconductance $g_m$ of 100 $\mu$S. Multiplied by the resistor $R_2$, it gives the open-loop gain of 10,000. The low-frequency pole is contributed by $C_2$ and set to 30 Hz.

The input source performs an ac sweep, and the results appear in Figure 4.98. The poles are exactly at the predicted positions.

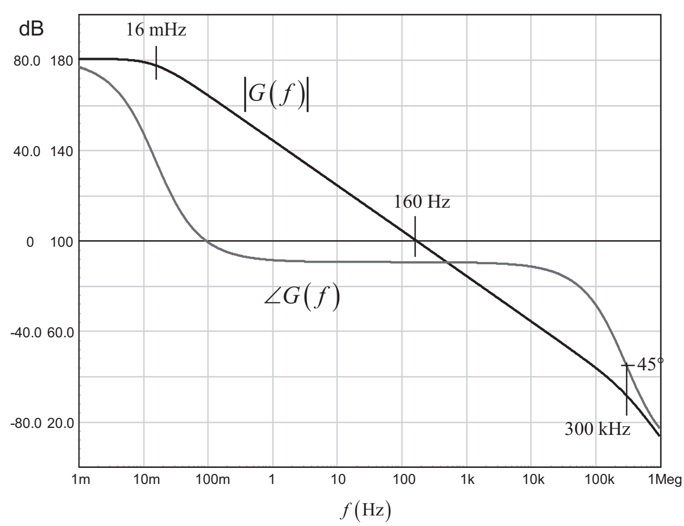


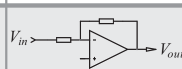
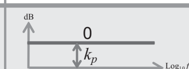
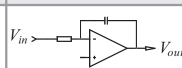
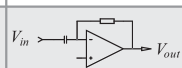
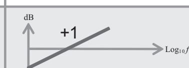
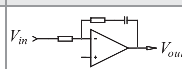
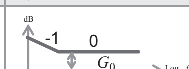
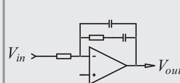
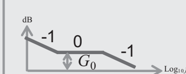
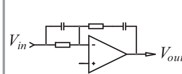
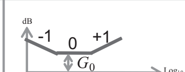
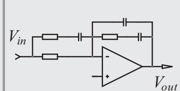
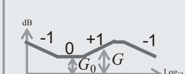

**Figure 4.98** This plots confirms that the op amp-based integrator featuring a low-frequency pole and a finite open-loop gain is a second-order system.

This appendix shows that you must look at the ac op amp characteristics if you plan to design high-speed power converters where a high crossover frequency is needed. In this case, you will need to consider the combination open-loop gain/low-frequency pole and check whether or not they impact your design. For that purpose, a dispersion parameter can be affected to the op amp open-loop gain, especially if you run a Monte Carlo analysis in SPICE.

## Appendix 4E: Summary of Compensator Configurations

Table 4E.1 gives the correspondence between P, I, and D individual blocks with op amp-based compensators.

**Table 4E.1** Summary of Compensator Configurations and Their Transfer Functions

| Action Mode | Basic Element | Transfer Function | Implementation | Bode Plot $\lvert G(s)\rvert$ | Type |
|---|---|---|---|---|---|
| Proportional | P | $G(s) = -k_p$ |  |  | |
| Integral | I | $G(s) = -k_i \dfrac{1}{s}$ |  |  | 1 |
| Derivative | D | $G(s) = -sk_d$ |  |  | |
| Proportional Integral | PI | $G(s) = -G_0 \dfrac{1+s/\omega_{z_1}}{s}$ |  |  | 2a |
| Proportional Integral + 1st-order lag | $PI_1$ | $G(s) = -G_0 \dfrac{1+s/\omega_{z_1}}{s}\dfrac{1}{1+s/\omega_{p_1}}$ |  |  | 2 |
| Proportional Integral Derivative | PID | $G(s) = -G_0 \dfrac{(1+s/\omega_{z_1})(1+s/\omega_{z_2})}{s}$ |  |  | 3a |
| Proportional Integral Derivative + 2nd order lag | $PID_2$ | $G(s) = -G_0 \dfrac{(1+s/\omega_{z_1})(1+s/\omega_{z_2})}{s(1+s/\omega_{p_1})(1+s/\omega_{p_2})}$ |  |  | 3 |