TND6238/D
Rev. 2, OCTOBER – 2021

Control Loop Design and Easy Verification Method

© Semiconductor Components Industries, LLC, 2017

October, 2021 – Rev. 2
Publication Order Number:
TND6238/D

Control Loop Design and Easy Verification Method

Prepared by: **Didier Balocco**
Technical Marketing Engineer

ABSTRACT

This paper presents a simple methodology applied to measure and optimize the control loop of a switching system. After a brief introduction to control loop theory and stability criteria, such system evaluation is explained. The use of a PWM simulation model is shown in practice to predict loop stability, together with a quick overview of obtainable results. A straightforward method to implement control loop measurement on a real environment is presented, followed by an optimization method using standard calculation−tools.

INTRODUCTION

The method presented here is based on work done in my previous company. When I joined that company, stability issues were happening time to time. When the converters' manufacturing started after the design phase, that instability signs were sometime observed due to some lot−to−lot differences. The rework and all the subsequent requalification time of faulty units were high and management was complaining. To get rid of these stability issues, I developed a simple measurement setup with an optimization method that I will present here.

With that method, we have measured many converters during the design phase. We could set enough gain and phase margins to solve stability issues before going in full production.

The idea was to be proactive instead of being reactive.

The method uses a very basic tool set that is generally available in all laboratories. The setup is simple, does not require a lot of expertise and can be applied everywhere. Using this method, analyzing the loop response in the customer system or in the field is also made possible.

LOOP THEORY AND STABILITY CRITERIA

System Block Diagram

A feedback loop in a system can be model with the following well−known block diagram.

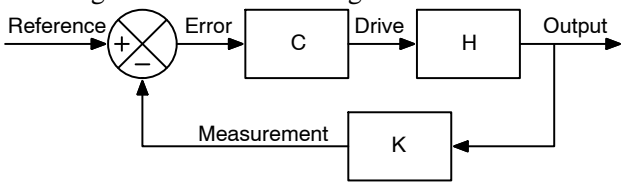


Figure 1. Feedback Loop Block Diagram

H block represents the *plant* or the system to control. It delivers the *Output* to be controlled by the feedback loop. The *K* block measures and scales the system output before comparing it with the *Reference*. The result of the scaling is called *Measurement* and the comparison result is called the *Error*. C (for *Compensator*) amplifies the comparison error in order to get the expected value in the output. It generates the *Drive* signal that controls the plant *H*.

System Equations

If we write some equations describing the system relationships, we obtain:

$$\begin{cases} \text{Error} = \text{Reference} - \text{K} \times \text{Output} \\ \text{Output} = \text{C} \times \text{H} \times \text{Error} \end{cases} \quad \text{(eq. 1)}$$

If we solve it, we can write the output equation:

$$\text{Output} = \frac{\text{H} \times \text{C}}{1 + \text{H} \times \text{K} \times \text{C}} \text{ Reference} \quad \text{(eq. 2)}$$

Normally, in the *Compensator C*, we build an integrator (or we place a pole at the origin) to reduce the static output error. In this case, the quasi−static gain (or dc gain) is infinite, so we get:

$$\lim_{\omega \to 0} |\text{C}(\omega)| = \infty \quad \text{(eq. 3)}$$

Substituting (3) in (2), we can obtain the output value in static or DC:

$$\text{Output} = \frac{1}{\text{K}} \times \text{Reference} \quad \text{(eq. 4)}$$

This equation is well known and used to scale the output value with a resistive divider like in a TL431 application, for example. Using the set of equations in (1), we can derive two more formulas. First, we can extract the *Measurement* as a function of the Error. It is like opening the system by excluding the comparator.

$$\text{Measurement} = \text{H} \times \text{K} \times \text{C} \times \text{Error} \quad \text{(eq. 5)}$$

We can recognize the system open−loop equation as defined in the literature:

$$\text{T} = \text{H} \times \text{K} \times \text{C} \quad \text{(eq. 6)}$$

Second, we can extract the *Measure* as a function of the *Reference*. Using (2), we get:

$$\text{Measure} = \frac{\text{H} \times \text{K} \times \text{C}}{1 + \text{H} \times \text{K} \times \text{C}} \times \text{Reference} \quad \text{(eq. 7)}$$

This formula describes the system *closed−loop* equation that we can call *S*.

$$\text{S} = \frac{\text{H} \times \text{K} \times \text{C}}{1 + \text{H} \times \text{K} \times \text{C}} \quad \text{(eq. 8)}$$

We can then combine these terms differently and obtain the closed−loop expression with the open−loop expression and vice−versa.

$$\text{S} = \frac{\text{T}}{1 + \text{T}} \text{ and } \text{T} = \frac{\text{S}}{1 - \text{S}} \quad \text{(eq. 9)}$$

Stability Criteria

In the system equation (8), we can clear see that S is undetermined if the denominator equals zero. This is one condition of existence for the system output value. In practice, if this condition is not satisfied, the system will be either become unstable (and oscillate) or saturate to one of the supply rails.

The minimum requirement to have a stable system is:

$$1 + H \times K \times C \neq 0 \leftrightarrow H \times K \times C \neq -1 \quad \text{(eq. 10)}$$

Nyquist [1] was the first to work on feedback loop stability with a graphical method to determine if a system is stable or not. The open−loop transfer function $T(j\omega)$ is plotted on the imaginary axis as a function of the pulsating frequency $2\pi f$. If this curve encircles the $(-1 + j0)$ point when frequency increases, the system (made by closing feedback loop) is unstable. If not, obviously, the system is stable [2].

This Nyquist criterion gives us the following diagrams for stable and unstable systems.

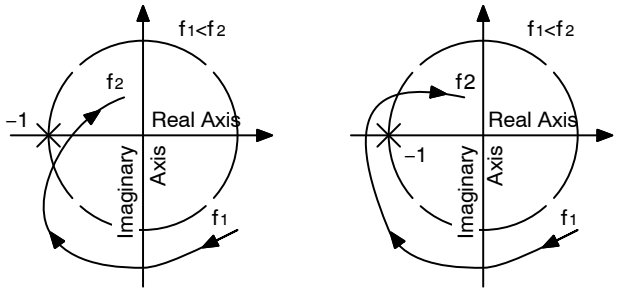


Figure 2. Nyquist Criterion

To have a better robustness and insure stability, two main criteria have been defined. They are based on open loop transfer function plots analysis. For the plots, we can either use Nyquist [3], Bode or Black−Nichols [4] plots.

The phase margin [5] is defined as the difference between the open−loop transfer function phase and −180° at the crossover frequency (i.e. when the open loop transfer function gain is *1* or *0 dB*).

The gain margin [5] is defined as the difference between the open−loop transfer function gain and the *0 dB* gain when the open−loop transfer function phase equals *−180°*.

This gives the following graphs to evaluate those criteria.

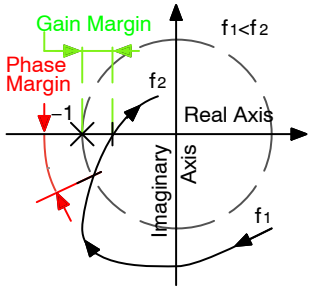


Figure 3. Margins in a Nyquist Plot

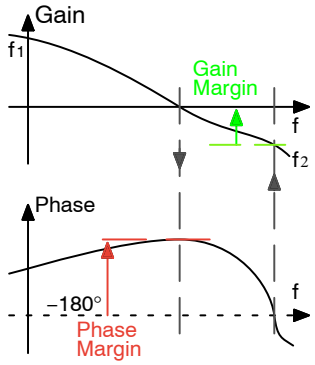


Figure 4. Margins in Bode Plot

Black−Nichols plot is the easiest way to measure phase and gain margins. At the beginning, having no link to frequency can disturb the user but it is a straightforward measurement as stability is concern.

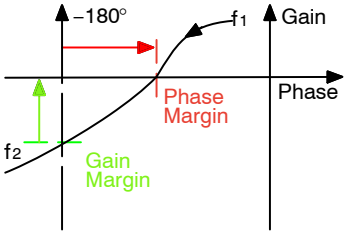


Figure 5. Margins in a Black−Nichols Plot

One other advantage of Black−Nichols plot is that if gain changes, it is only a shift up or down of the curve. This type of shift happens with opto−couplers when their current transfer ratio (CTR) moves with operating conditions.

So, it is simple to anticipate what will be the remaining margins as shown on the next figure.

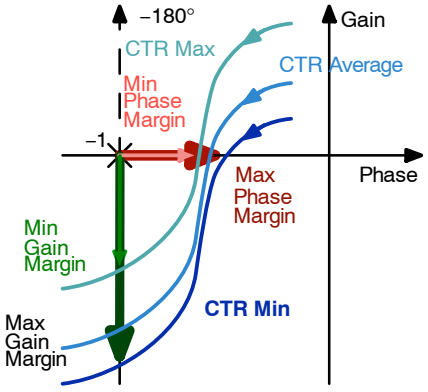


Figure 6. Gain and Phase Margins Change Versus Transfer Function Gain Variations

CHARACTERIZATION

Principle

When we do not have the literal math plant equation *H*, it is impossible to use feedback design method directly. We can or need to use simulations or measurements to get the plant (or system) transfer function.

The obvious way to do is to measure or simulate directly the plant to characterize it. In non−linear system, the plant equation varies with the continuous (or dc) operating point. During measurement, low−frequency effects, such as self−heating, can affect the operating point and modify the plant transfer function. In system with very high gain, the plant can even saturate. When that happens, the measurement is no longer done in a linear region and action is necessary to bring the system back into its linear mode.

The most commonly used method is to characterize the plant in closed−loop. The feedback will take care of setting up the operating point. During measurement, it will compensate low frequency drifting effects, like self−heating, and will keep the system in a linear region.

To extract the open−loop when the loop is closed, we inject (or add) a small signal in the feedback loop. The injected signal will perturb the system. By measuring the effects, we can get the open loop and/or the plant transfer function. We obtain the following setup.

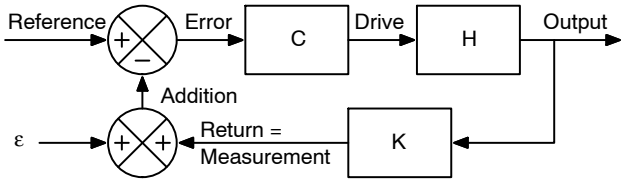


Figure 7. Measurement or Simulation Setup

On the figure above, ε represents the injected signal. We define the *Return* signal as the one that will be added to the injected signal. *Addition* is the result of that operation. *Return* can also be seen as the result of the injected signal perturbation. The equations of the system with the injection adder are given below:

$$\begin{cases} \text{Error} = \text{Reference} - \text{Addition} \\ \text{Drive} = C \times \text{Error} \\ \text{Output} = H \times \text{Drive} \\ \text{Measurement} = K \times \text{Output} \\ \text{Return} = \text{Measurement} \\ \text{Addition} = \varepsilon + \text{Return} \end{cases} \quad \text{(eq. 11)}$$

As we are looking to get the transfer function, we will use a sine wave signal as injected signal. By varying the frequency of the injected signal, we can determine the frequency response and draw a Bode plot of the transfer function.

As *Reference* is a dc signal and we focus on frequency response. It can be excluded from the picture because it does not contain frequency information.

Using those hypotheses, we can solve equations given in (11) to get the output equation. However, it is not the most interesting one. As we have "broken" the loop with the injection adder, we can focus on signal around it. As *Return* is, somehow, the result of the injection signal ε effects, we can solve the system and get the *Return* equation as a function of either ε or *Addition*.

We can easily get *Return* over ε.

$$\frac{\text{Return}}{\varepsilon} = \frac{-H \times K \times C}{1 + H \times K \times C} \quad \text{(eq. 12)}$$

But, the most interesting equation to get is the *Return* over *Addition.*

$$\frac{\text{Return}}{\text{Addition}} = -H \times K \times C \quad \text{(eq. 13)}$$

We can see that (13) and (12) are the opposite of respectively open−loop and closed−loop equations. This is because in the theory the comparator is out of the picture and serves as a "loop breaker". In the measurement or simulation setup, the injection adder serves the purpose of "breaking" the loop at that point and the comparator stays in.

Discussion

The injection adder can be placed anywhere in the loop, the equations (13) and (12) are still valid.

There are three easy ways to make a simple injection adder:

1. Inject the voltage in series with a floating source.
2. Like in a radio frequency amplifier in which a single coaxial cable transmits the signal and remotely delivers power, use a low and high pass filters to merge the injected ac signal and its dc content.
3. Use a real voltage adder.

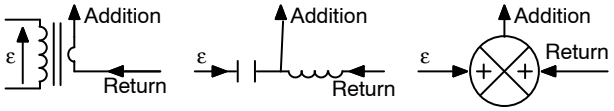


Figure 8. Injection Adders

For the first solution in a simulation files, we just insert a pure ac source in the wire where we want to inject. For the measurement in a real product, the most common way is to have a floating sine wave generator operated via a transformer to provide galvanic isolation.

For the second solution in a simulation files, we use very large component values to form a very low crossover frequency filter. A 1 kH inductor will block the ac content and be a short circuit for dc content. On the opposite, a 1 kF capacitor to will block the dc content and be a short circuit for the ac content even at a very low frequency. For measurements, as injection signal frequency is in the range of tens of Hz up to hundreds of kHz, it is difficult to get components large enough and this option suits simulation purposes only.

For the third solution, adder blocks are commonly available in simulators libraries. For measurement, we can use an operational amplifier and some resistors to build one featuring a unity gain.

Practical Aspect for Measurement

First, if we want to measure the system, we need to inject the signal without affecting the operating environment. At the point where we break the loop, we have to take care of impedance matching.

This statement implies that the input impedance of the injection adder matches the one at the break point while the output impedance of the injection adder is the same as the one at the break point.

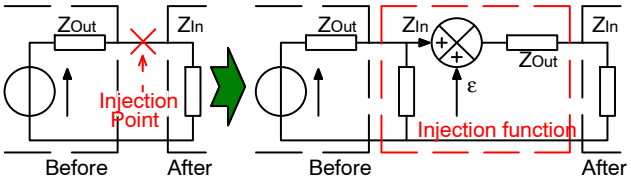


Figure 9. Impedance Matching Principle

To simplify these conditions, we can choose a break point where the input impedance is infinite or where the output impedance is equal to zero.

In theory, ideal operational amplifier has no input impedance (it is an open input) and no output impedance.

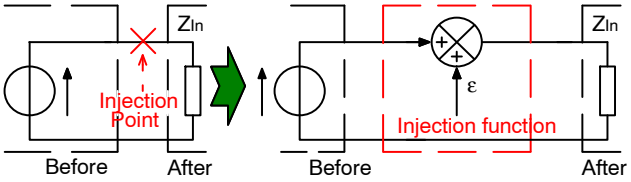


Figure 10. Operational Amplifier Impedance Matching

This condition is satisfied if we have an operational amplifier in the loop. Indeed, the operational amplifier output impedance is nearly zero (or below 1 Ω). Generally, the operational amplifier output load is in the range of kΩ.

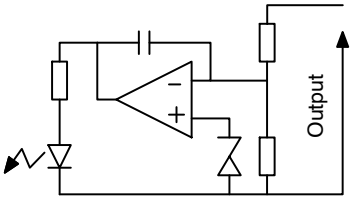


Figure 11. Breaking the Loop at the Operational Amplifier

As in many cases, the operational amplifier output is connected inside the PWM device, it is impossible to break the loop at that point. One solution consists of breaking the loop at the output. Generally, the output impedance is very low and the measurement network input impedance is high. So, we can use that point because it almost like a voltage source. This will only introduce a small error proportional to the output over input impedances ratio.

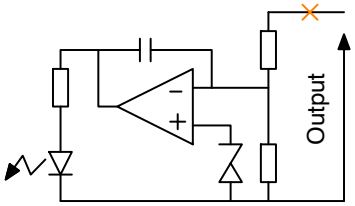


Figure 12. Breaking the Loop at the Output

In simulation, we have to care also of impedance matching but, generally, the amplifier output is available.

Second, to ensure adequate measurement conditions, we must ensure the system remains in the linear region. If the injection signal is a sine wave, all other signals (and in particular *Addition* and *Return*) should also be sine waves. So, we need to use a small value for ε.

However, if modulation amplitudes are too small, you won't be able to properly observe key signals as they will be drowned in the environment. Measurements are impossible in that case. The modulation level should thus be of sufficient amplitude to stay above the noise floor.

To ease the measurement process, we can use some of the functions offered by modern oscilloscopes such as averaging, synchronous detection or advanced triggering.

In simulation, as the program linearizes the circuit before running the small−signal analysis, the amplitude doesn't matter and results are not depending on the injection signal amplitude.

In any case, measurement or simulation should never depend on injection adder in the loop, the injection adder setup and injected signal amplitude. If dynamic results vary too much as modulation amplitude changes, it is an indication that the system does not operate in its linear zone and the setup may need an adjustment.

More Results

Depending on the available signals, it is also possible to plot (by measurement or simulation) other transfer functions. The only condition is that the injection adder is outside of the transfer function we want to plot.

For example, observing the ratio *Output* over *Drive* will give you the plant transfer function *H*. This is obvious and the most difficult point here is to have access the *Drive* signal.

If the injection adder is connected to the *Output* (and *Return* = *Output* in this case), if we plot *Drive*/*Addition*, we get the opposite of the compensator multiply by the scaling factor (−*K.C*) because the comparator lies inside the path.

SIMULATIONS

Requirements

The goal of this paper is not to develop simulation models but only showing how to use them. (Many books and papers are available on that topic. For references, see Christophe Basso ones [6].) The PWM models or average small−signal models can be found on dedicated web sites like Christophe Basso personal web site [7]. Some programs like SIMetrix/SIMPLIS [8] can deliver the dynamic response of a switching circuit without going through the average model step.

The following figure shows an example of the PWM switch model for voltage mode control:

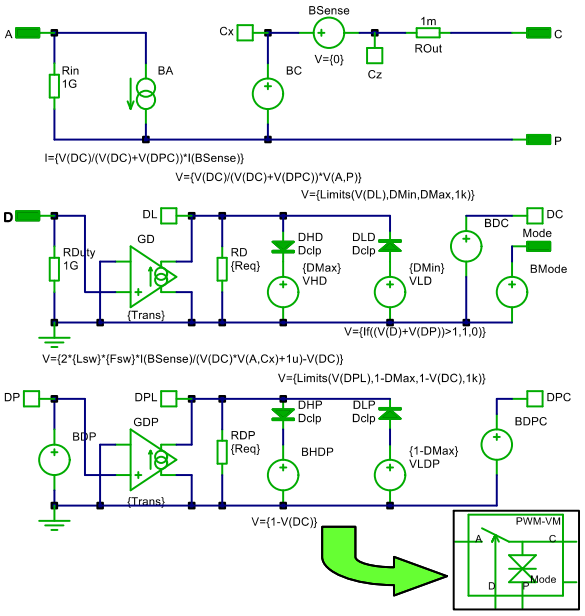


Figure 13. Voltage Mode version of the PWM switch average model

All those models are delivered in a library or pieces of schematics that should be included to use them.

Here, we assume models are available and ready to use for simulations.

Simulation Example

We will simulate a basic buck dc−dc converter.

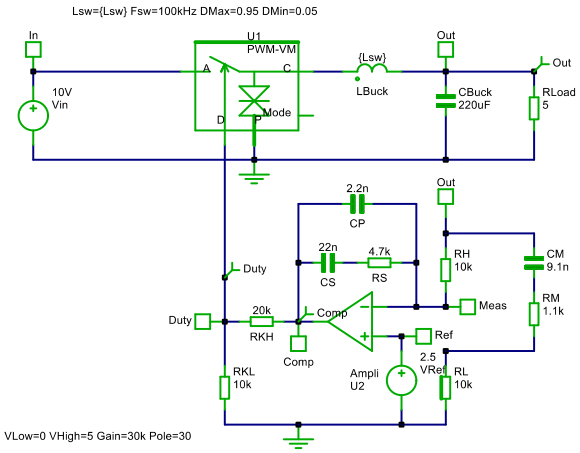


Figure 14. Buck Converter Schematic

If we simulate the schematic as it is, the loop is completely closed and we cannot plot the open−loop transfer function. There is also no ac stimulus to run a frequency sweep analysis. We will only obtain dc values.

As seen in the previous chapter, we will use a pure ac source as a stimulus and an L−C injection adder.

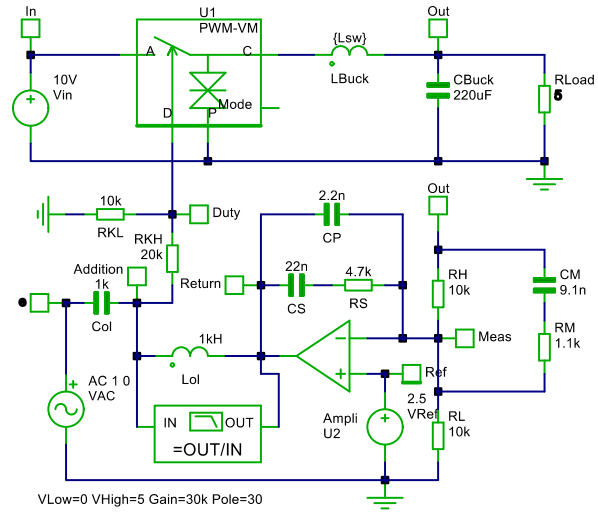


Figure 15. Simulation Schematic with Stimulus Source and Injection Adder

If we plot *–Return/Addition* we have the open−loop transfer function.

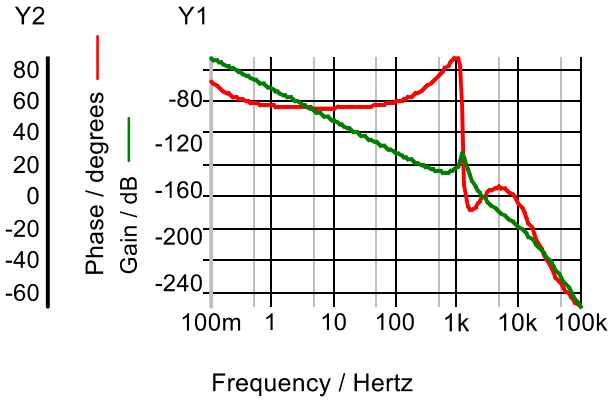


Figure 16. Open−loop Simulation Results

As discussed in the chapter Discussion. , we can also plot other transfer functions like that of the plant for instance.

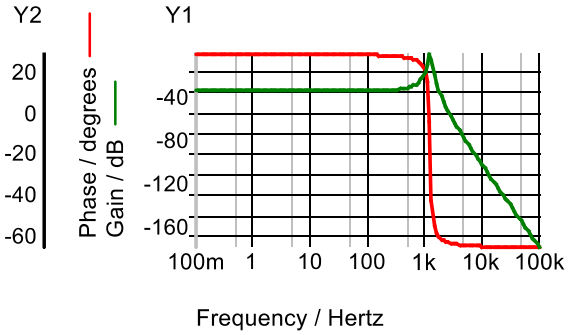


Figure 17. Buck Stage Transfer Function

More Results from “Same” Simulation Schematic

For simplicity and have a better focus on the way to implement simulations using the same schematic or buck converter, the schematic has been reorganized using hierarchical blocks. At the top level, we kept only interesting signals like: input, output, drive and reference as a minimum.

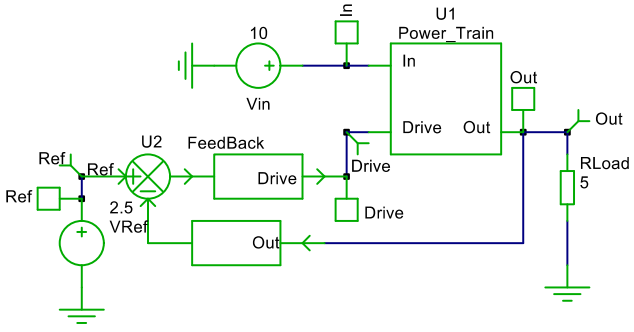


Figure 18. New Hierarchical Simulation Schematic

If we inject the sine wave at the input (instead of injecting in the loop), by measuring the effect on the output, we can determine the input to output rejection ratio. Here is the setup.

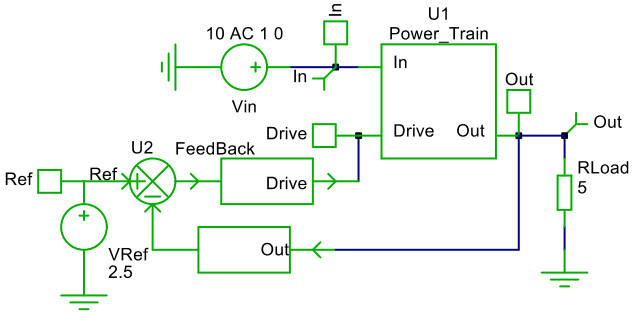


Figure 19. Input to Output Rejection Ratio Simulation Setup

If we plot output voltage over input voltage, we obtain the rejection ratio.

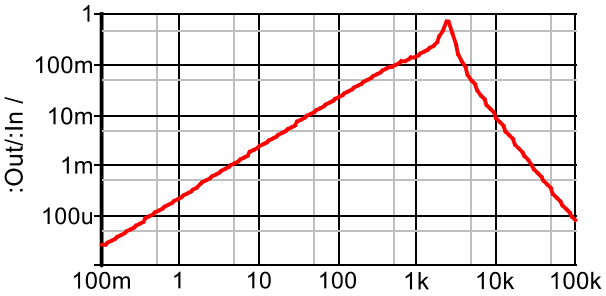


Figure 20. Input Rejection Ratio Simulation Results

By injecting the signal as a current on the input or the output, we can simulate respectively the input or output impedance. This impedance is given by the ratio of the voltage and current.

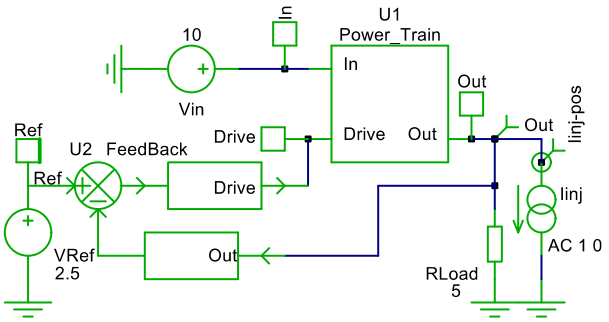


Figure 21. Output Impedance Simulation Setup

If we plot output voltage over output current, we get the rejection ratio.

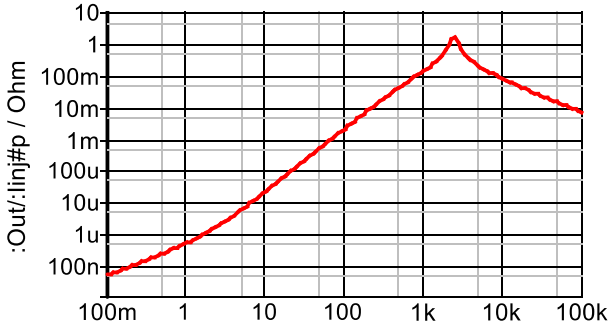


Figure 22. Output Impedance Simulation Result

Sometime, ac simulations' results use *dB* to display ratio, the impedance is given in *dB* instead of Ω.

Instead of running ac simulation, we can use the same schematic (with the average PWM model) to simulate the average response of the system to a transient condition like input voltage transient, output load step and reference voltage tracking.

Here is an example of average results for a reference tracking simulation.

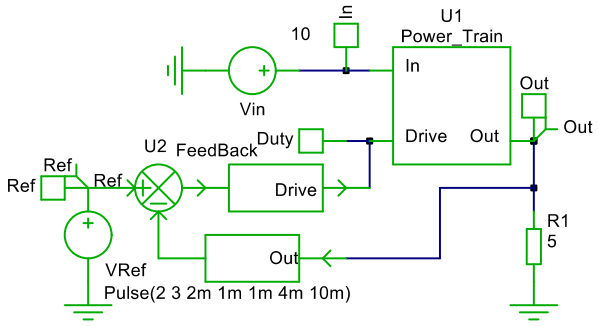


Figure 23. Reference Tracking Simulation Setup

We apply a square waveform as reference for the buck system. This setup can represent a LED dc−dc converter featuring a square PWM dimming for example.

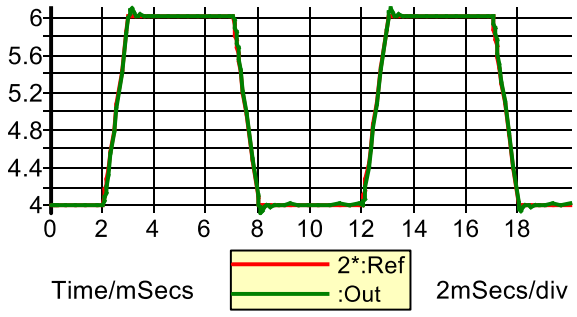


Figure 24. Reference Tracking Simulation Results

As you can note on those previous examples, we have used the same basic simulation schematic. We have just changed either the simulation mode ac or transient and applied different stimuli (a voltage, ac current, square waveform, ...) on selected schematic nodes (Input, Output, Reference, ...) depending on the transfer function we wanted.

See Annex I for details about average PWM model schematic, simulations schematics and simulations setups.

MEASUREMENT METHOD

Measurement Setup

As presented in chapter III. Characterization, we need to break the loop and use one injection adder. To generate the stimulus, we can use a signal generator. To measure the *Addition* and *Return* signals, we can use an oscilloscope. Both measurement tools are embedded in a network analyzer.

For the injection adder, the transformer remains the easiest one to use and it can be placed anywhere (as far as we respect impedance matching condition). Its major drawbacks are the fact that transformer are very non−linear components and can have a limited bandwidth. We have to pay a particular attention to stay in the linear region.

Otherwise, we can use an adder made with an operational amplifier. To have higher input impedance, we can add a follower at the input of the adder but that is not really needed if we use kΩ resistances and a nearly zero output impedance node.

To prevent the dc level from being pushed inside the wave generator, we inserted a buffer for the injected sine wave between the wave generator and the adder input.

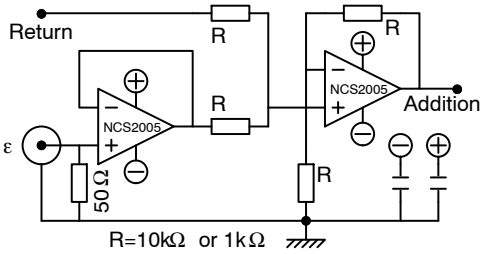


Figure 25. Injection Adder Made with an Amplifier

The main advantages of using an operational amplifier are:

1. Large bandwidth can be obtained with a high bandwidth amplifier like the new *NCS2005* (8 MHz, rail to rail) from **onsemi**.
2. Very good linearity and low distortion even with large voltage (*NCS2005* maximum supply voltage 32 V and 2.2 V as a minimum, 2.8 V/μs slew rate).
3. If the adder is not "perfect" (for the frequency range we use it) and introduces a phase shift, as it is basically a low pass filter, it always increases the open loop transfer function phase. So, when the measurement setup will be removed, the phase margin will be the same or little bit higher. With a transformer, it depends on transformer parasitic components. This is an advantage for the operational amplifier injection adder. Nevertheless, using the *NCS2005*, the bandwidth is high enough to not influence measurements up to *hundred of kHz.*

We need however to respect some specific points:

1. The injection adder could not be placed anywhere. It has to be placed in the low power path of the loop.
2. The amplifier needs a supply (best are two supplies, negative and positive).
3. The amplifier ground should be connected to the system ground for measurement.

To avoid distortion (i.e. be in the linear region) and be above the noise floor, we can set the injected sine wave amplitude between *20 mV rms* as a minimum and *100 mV peak to peak* as a maximum.

The minimum averaging time should be higher than *16*. We can also limit the oscilloscope bandwidth to *20 MHz*. On an analog oscilloscope; we can use the brightness to do the averaging.

It is a good practice to synchronize the oscilloscope trigger with the synchronization signal provided by the wave generator. This will help for averaging and makes it more efficient. With a network analyzer, this is automatically internally done.

The negative supply needs to be “bigger” than the sine wave injected. Generally, *−1 V* is more than enough. For the positive supply, it should be higher than the dc level at the breaking loop point. If the output is used as breaking loop point, a *1 or 2 V* above the output maximum value is a good practice.

Oscilloscope probes should be compensated. In a lowfrequency system, like an ac−dc power supply, probes do not significantly influence results as in most cases, the loop crossover frequency is around 1 kHz. However, keep in mind that this is not always the case and some high−speed dc−dc converters can exhibit 0−dB crossover points at frequencies above 100 kHz.

Finally, with the injection adder, we should connect all grounds together (oscilloscope ground, wave generator ground, supply voltages ground and the system ground).

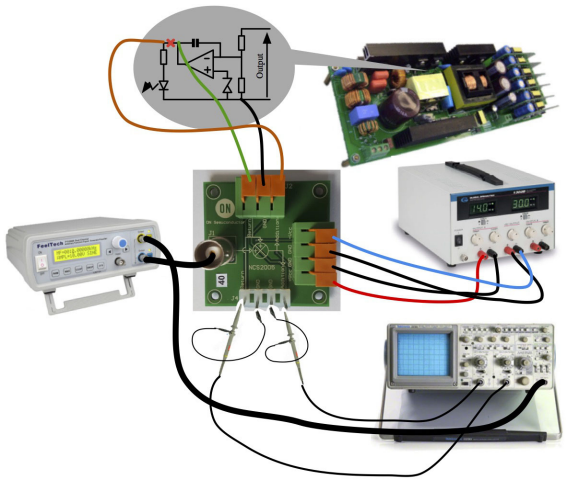


Figure 26. Measurement Setup or Video Tutorial [10]

See Annex III for a bigger view.

Measurement Key Points

This can sound disturbing, but loop measurement can only be undertaken with a stable converter! Indeed, if the system is unstable, an ac signal will disturb our measurement. Thus, before measuring the loop, we can set a very low bandwidth compensator (with a large integrator capacitor, for example) to ensure stability. We can also increase the soft−start time to avoid too large overshoot that could trip the overvoltage protection.

Due to the sampling effects of the PWM operation (or Frequency Modulator in the case of LLC resonant converter), the maximum measurement frequency should always be lower than half of the switching frequency.

When all is installed and running, we can sweep the frequency to measure the loop magnitude and phase response versus frequency and obtain a plot of the desired open−loop transfer function.

For a fast approach, for example to verify simulation results, mainly phase and gain margin criteria, we can sweep the frequency and measure at two points only:

1. When *Addition* and *Return* exhibit similar amplitudes, then we are at the crossover frequency and the open loop gain is *1* or *0 dB*. At that point, the phase shift between the two signals is directly the phase margin. Indeed, as we measure the opposite of the open loop transfer function, there is a phase shift of *180°* already in the measurement. That gives us directly the phase margin.
2. When the *Addition* and *Return* are in phase (and not out of phase due to the opposite open−loop measurement), we can measure directly the phase margin by measuring the ratio *Addition over Return* (and not *Return over Addition* as for the open−loop transfer function). At that point *Return* should be lower than *Addition*. We are above the crossover frequency.

When an automatic equipment, like a network analyzer, is used, it better to measure the close−loop by measuring *Return* over ε using (12). Them, calculate the open−loop using equation (9). The advantage of that close−loop measurement is: below the crossover frequency, *Return* and ε signals have the same amplitude. There is less noise on both signal and only a small phase shift between them. Finally, the accuracy is better.

As discussed in chapter III. D. More Results. , we can also measure directly other transfer functions like the plant if we want to optimize the loop response for instance by fine−tuning the compensator.

Never forget to connect all grounds together. Otherwise, we will measure the impedance between grounds. In this case, the measurement looks like a capacitor impedance curve.

LOOP OPTIMIZATION

In this chapter, we will focus on a method based on measurement obtained with the method explained in the previous chapter.

Method

We can use a mathematical program to manipulate the data obtained by measurement.

If the measure is the open−loop transfer function (or the opposite), we can compute the plant transfer function by removing the compensator (and scaling) used to measure.

We can design a new compensator using methods explained in books like those of Christophe Basso [6] or placing poles and zeros manually. We obtain a new compensator (and scaling) transfer function.

With that function, we will multiply the Laplace expression with that of the plant transfer function (or we sum up dB plots) to obtain the new open loop transfer function and extract the new stability criteria (gain and phase margins).

Example

Here is an open−loop measurement.

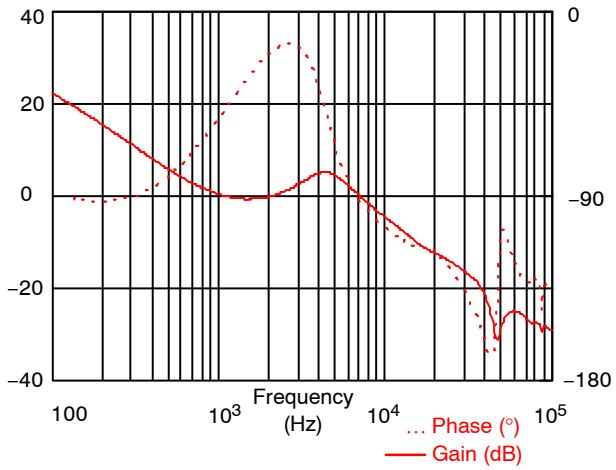


Figure 27. Open Loop Measurement Results

This measurement has been done with a very low bandwidth feedback loop to insure product stability. It was measured using the network analyzer HP4194A.

We evaluate the plant transfer function by removing the compensator and the scaling factor used for the measurement.

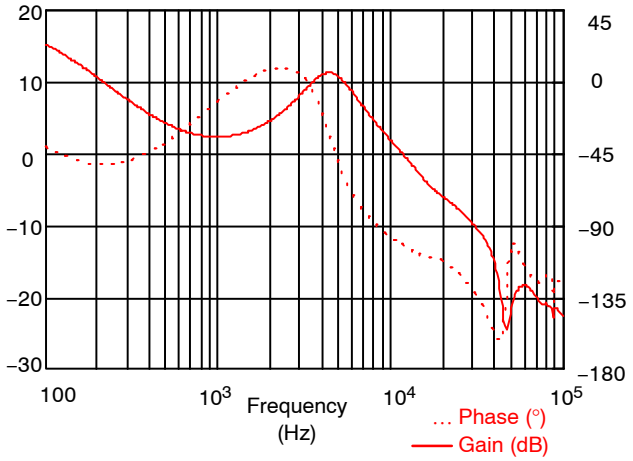


Figure 28. Plant Transfer Function

We design a new compensator to meet the stability criteria we want.

Here is the comparison between both (measured and simulated) compensators (with similar scaling factor).

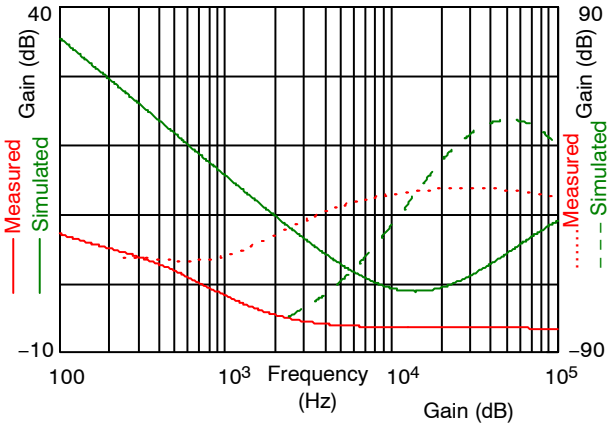


Figure 29. Compensator Design Results

We apply the new compensator (and scaling factor) to the plant to obtain the new open−loop transfer function to verify the stability criteria.

On the following picture, we can compare both (measured and simulated) open−loop responses.

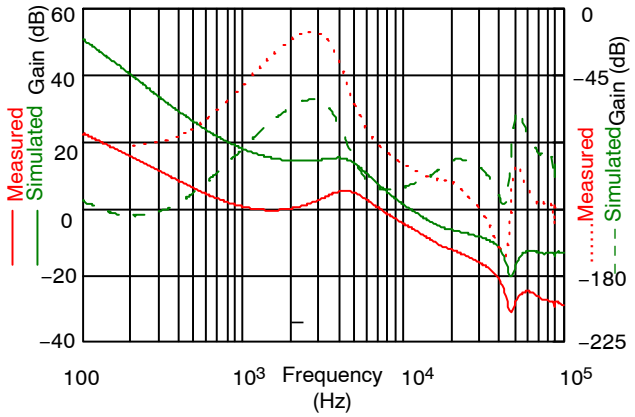


Figure 30. Open Loop before and After Optimization

The new phase margin is slightly higher than 60°. We optimized the open−loop phase margin and increased the system stability.

On the following Black−Nichols graph, if the plant gain varies between + *or* −*10 dB*, we can easily see that the phase margin will be higher than 45°. This provides a good robustness.

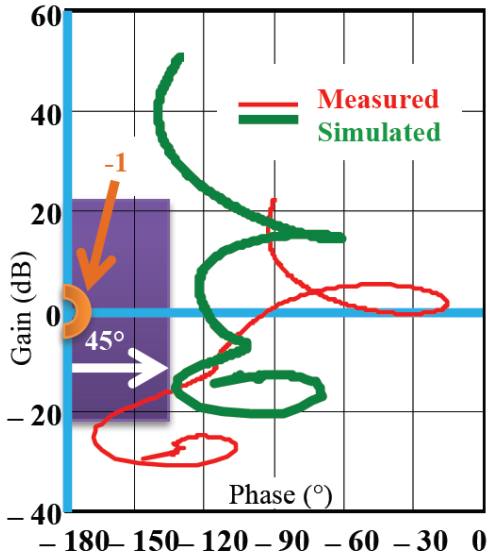


Figure 31. Open Loop Optimized in Black−Nichols

On example of such calculus using Mathcad® [9] are given in Annex II.

SIMULATION VERSUS MEASUREMENTS

Simulations and Measurements are most of the time shown as two independent ways to obtain the same results. Fans of both methods are fighting each other's sometime. However, measurement is the only referee for verifying simulations accuracy or simulation models.

Simulations and measurements can be used together and bring a more complete picture of the system. Simulations are used during design phase. They can display many parameters and help designers to understand the system before prototyping. They can be helpful to design the system and tune it.

Measurements are made on a real system. The beauty of them is to integrate all parasitic effects (or components). They give a complete picture of the system (including non−linear effects that are difficult to model). We have learned a lot by measuring loops, analyzing them and then comparing them with simulations' results.

Of course, measurements can be and should be used to validate simulations' results. Furthermore, it can also be of invaluable help to fine−tune the system at the end by accounting for real environment effects (like a very specific load for instance).

Both simulations and measurements are complementary. They are used in different steps during the full design process, but they can also be use together. Using both and not relying on only one of them will make your design more robust.

CONCLUSION

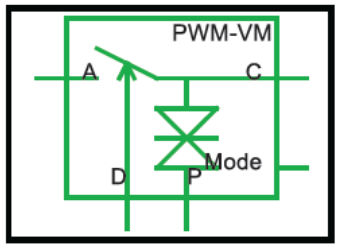
Simulation setup has been explained. A simple example has been used to demonstrate how to obtain an open−loop transfer function. The phase and gain margins can be extracted from the simulation results as shown. More simulation results (like input rejection ratio, impedances, transient responses) can be also obtained by re−using the same schematic with slightly small changes.

A new method to measure loop based on a high−bandwidth operational amplifier (like the new *NCS2005* from **onsemi**) has been explained. After measuring the loop, a simple method to optimize the system stability was demonstrated on a real example. How to obtain rapidly phase and gain margins have been shown.

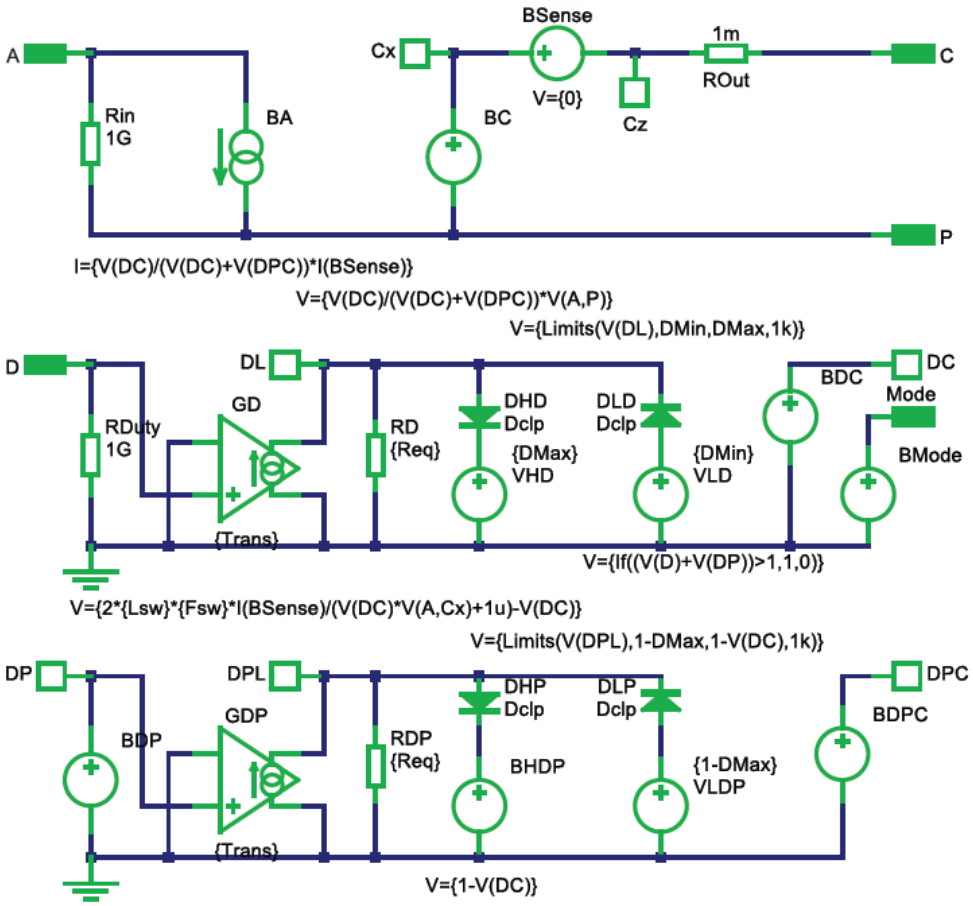
More details can be also found in annexes.

ANNEX I : SIMetrix SIMULATION SETUPS AND RESULTS

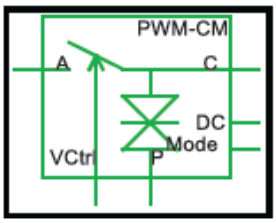
Here is the Voltage Mode average PWM Model schematic:



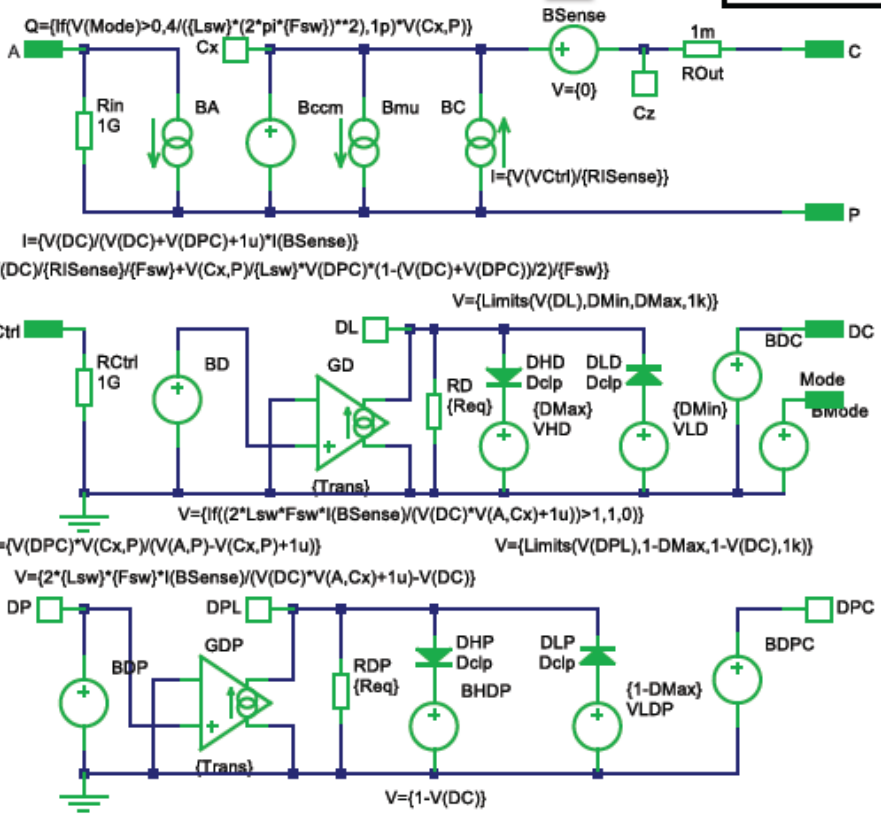
.Model Dclp D n=0.01 rs=100m
.Param Trans=1m Req=1k



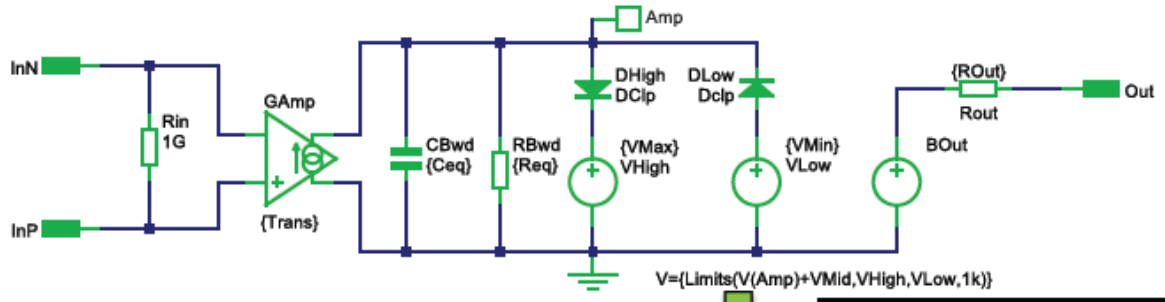
Here is the Current Mode average PWM Model schematic:



.Model Dclp D n=0.01 rs=100m
.Param Trans=1m Req=1k

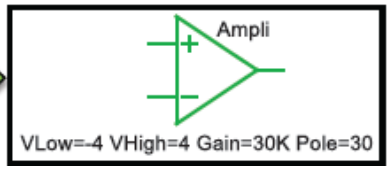


Here is the simple amplifier Model schematic:

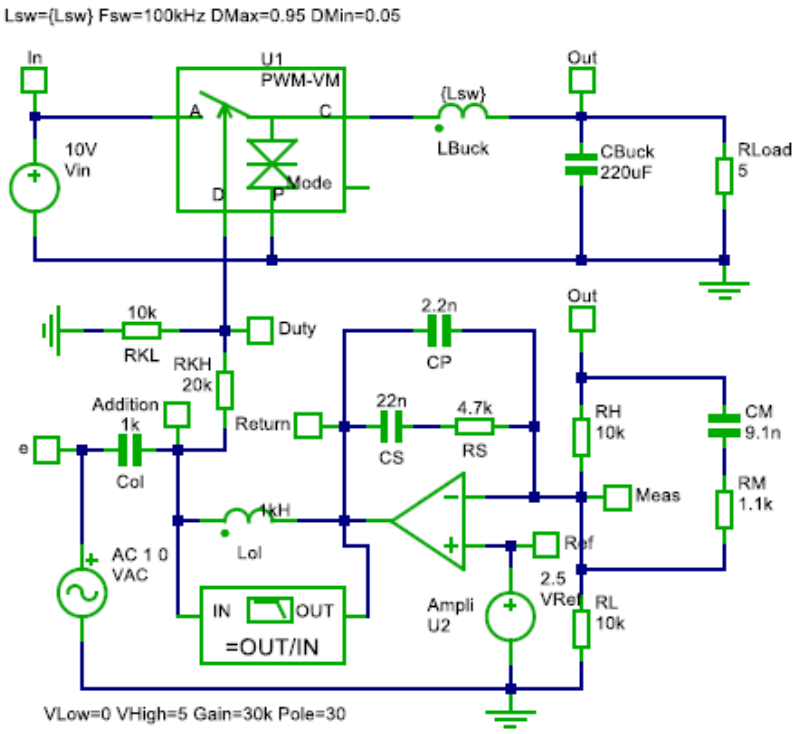


.Model Dclp D n=0.01 rs=100m
.Param VMid={(VHigh+VLow)/2}
.Param VMin={VLow-VMid}
.Param VMax={VHigh-VMid}

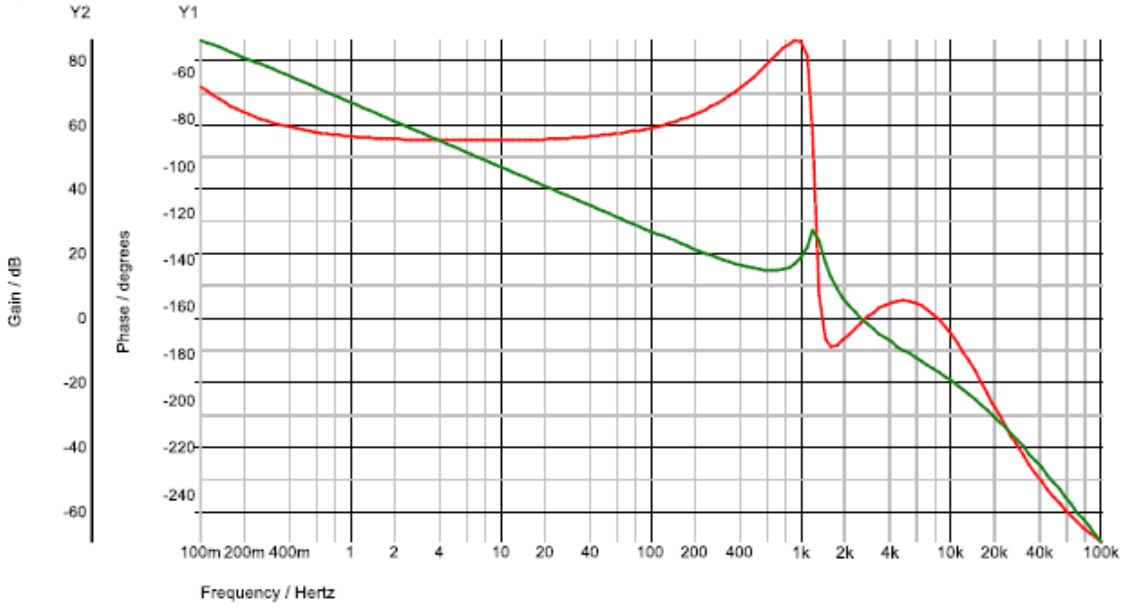
.Param Trans=1m ROut=10
.Param Req={Gain/Trans}
.Param Ceq={1/(2*PI*Req*Pole)}



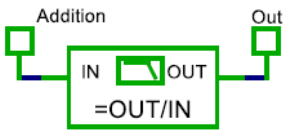
Buck stage with Voltage Mode PWM for open-loop simulation:

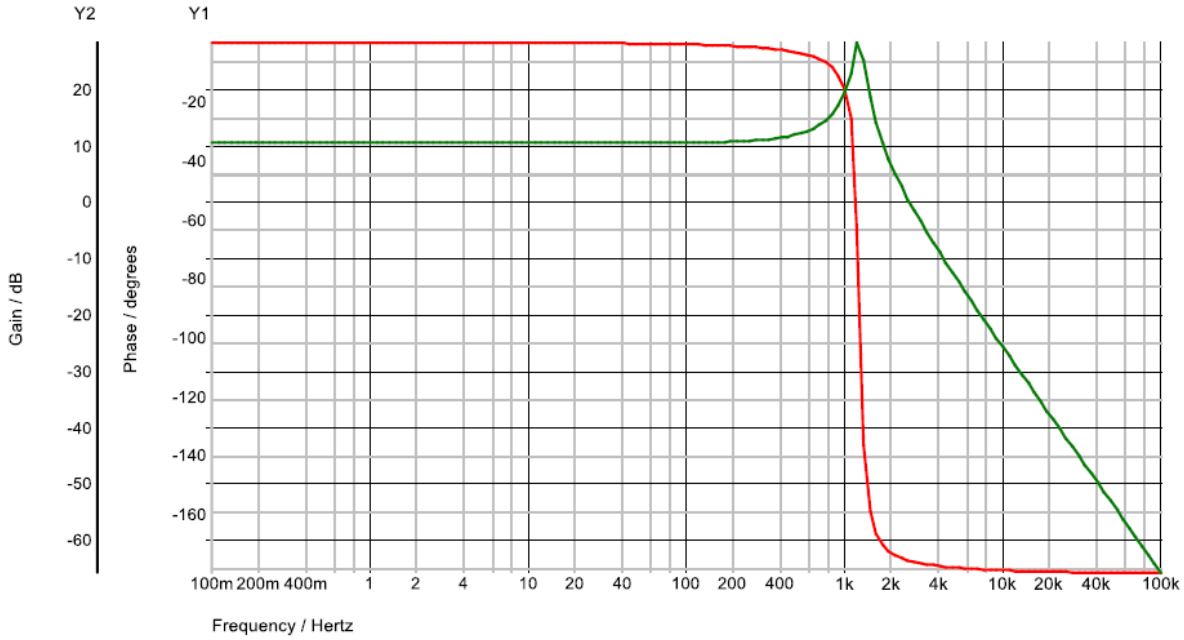


Open-loop results:

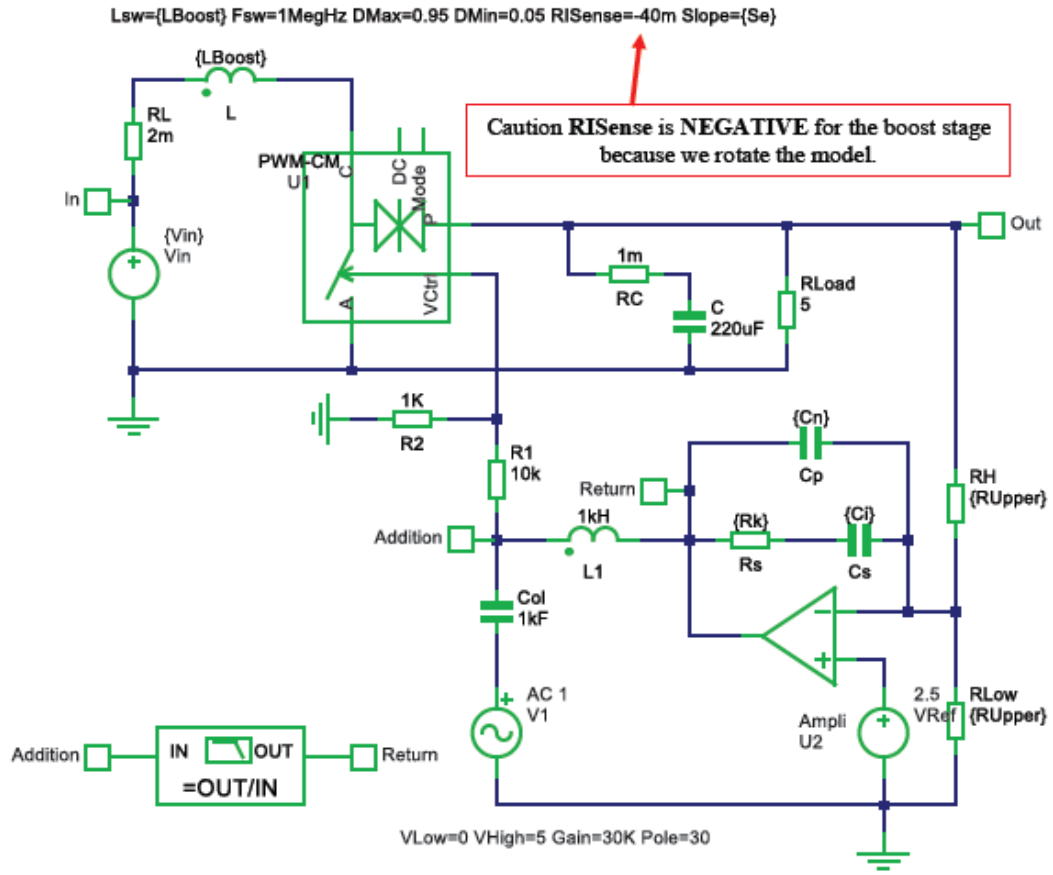


Changing the simulation Bode plot setup on the previous schematic:



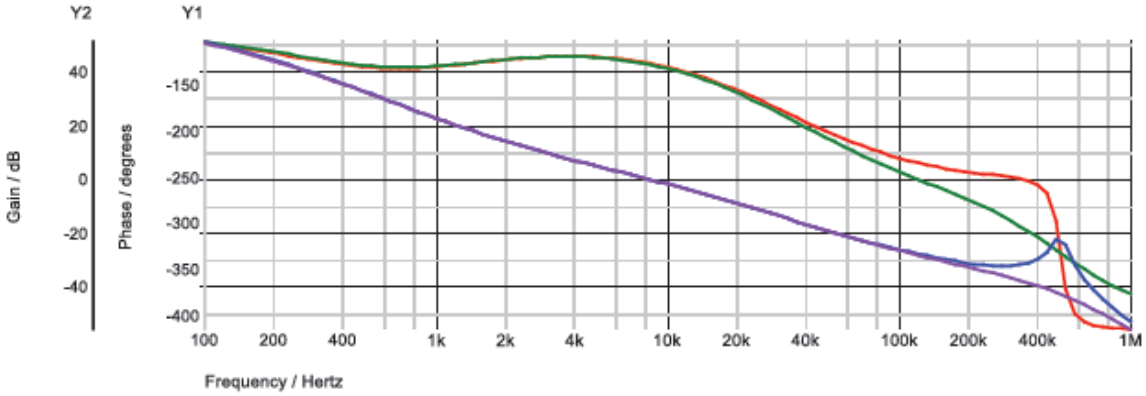


Boost stage with Current Mode PWM for open-loop simulation:



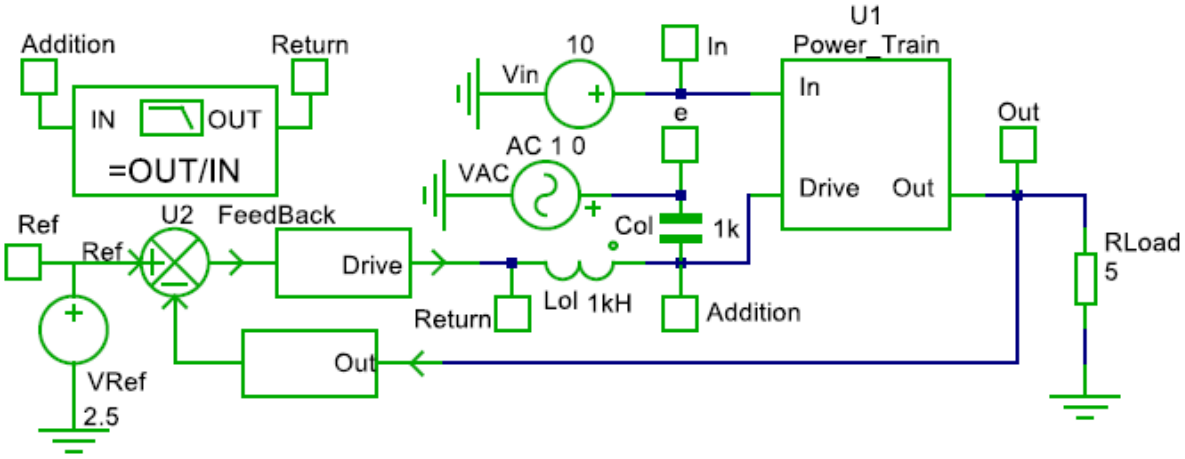
* Boost parameters
.Param LBoost=5uH
.Param Se=0
.Param Vln=2.7V
*Initial values
.Param RUpper=10k fc=8k
.Param pm=60 Gfc=-20 pfc=-101
.AC DEC 25 100 1Meg SWEEP PARAM=Se LIST 0 16.4k

* Feedback parameters
.Param G={10**(-Gfc/20)}
.Param Boost={pm-pfc-90}
.Param K={tan((Boost/2+45)*pi/180)}
.Param Cn={1/(2*pi*fc*G*k*RUpper)}
.Param Ci={Cn*(K**2-1)}
.Param Rk={K/(2*pi*fc*Ci)}

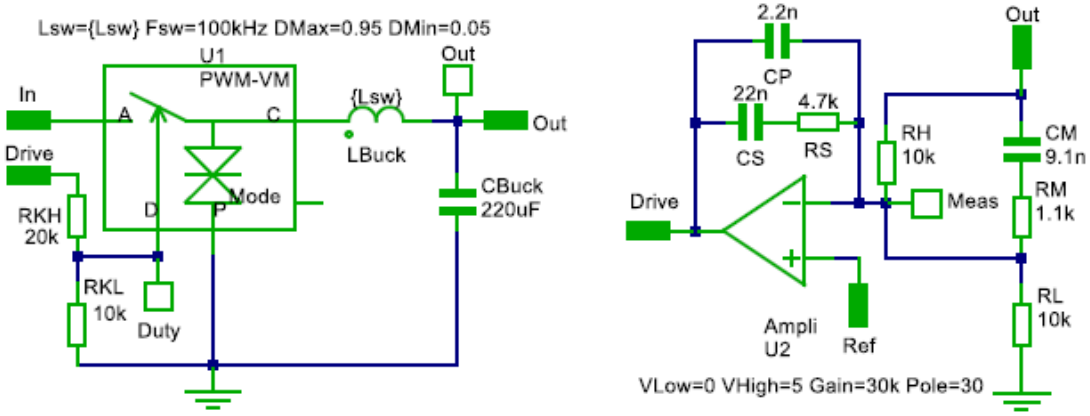


We can see the picky resonance link to the lack of slope compensation (when Se=0) in the gain curve and -180° phase shift.

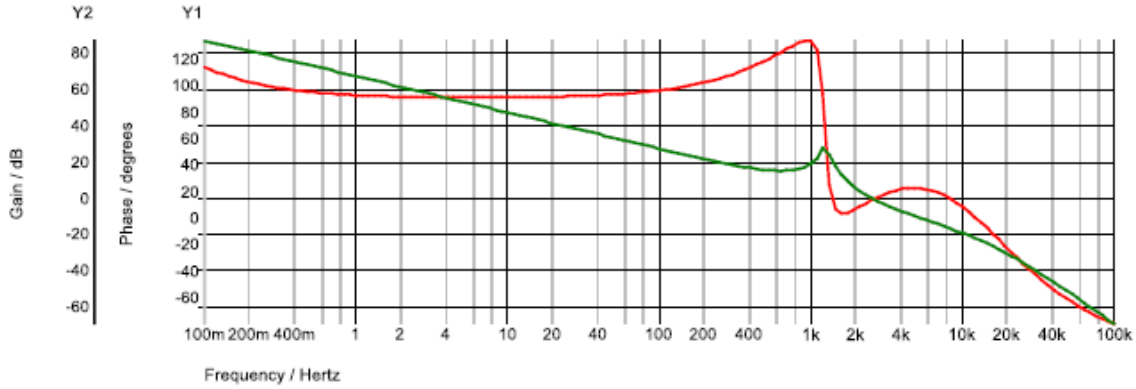
The buck converter block diagram:



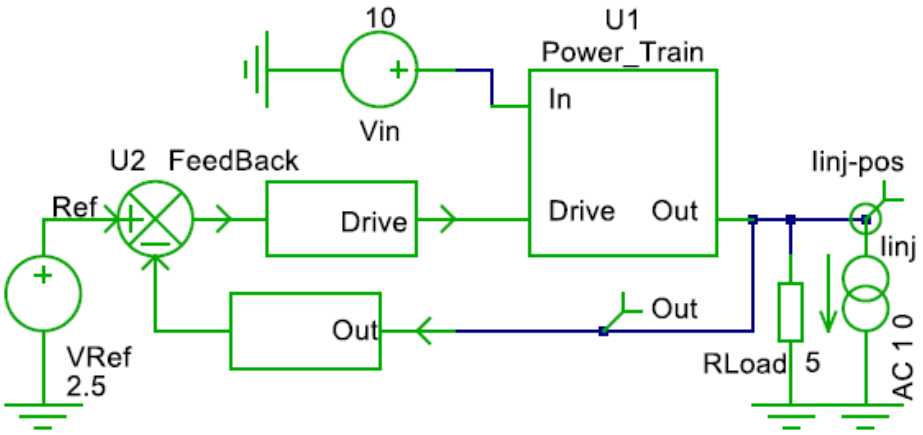
Here is the power plant and the feedback schematic:

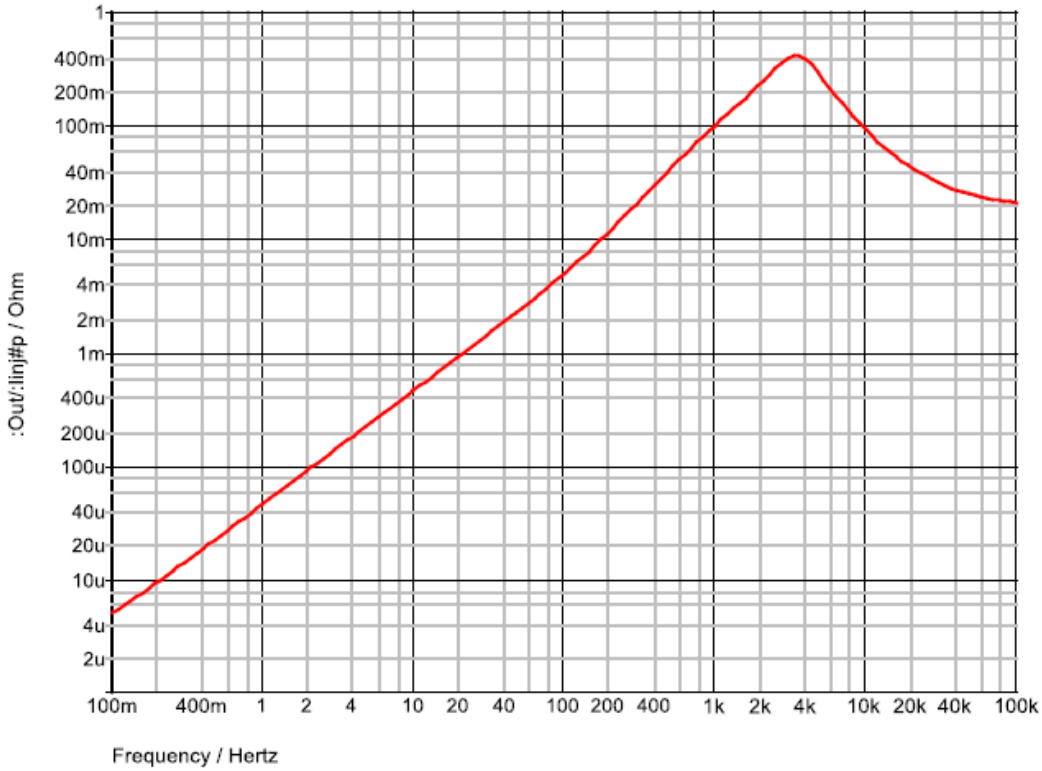


If we break the loop in **Drive**, we obtain the same results:

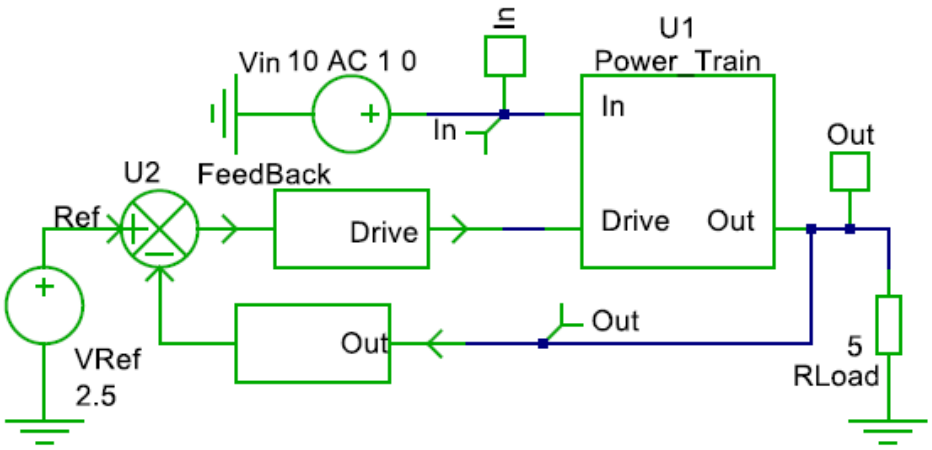


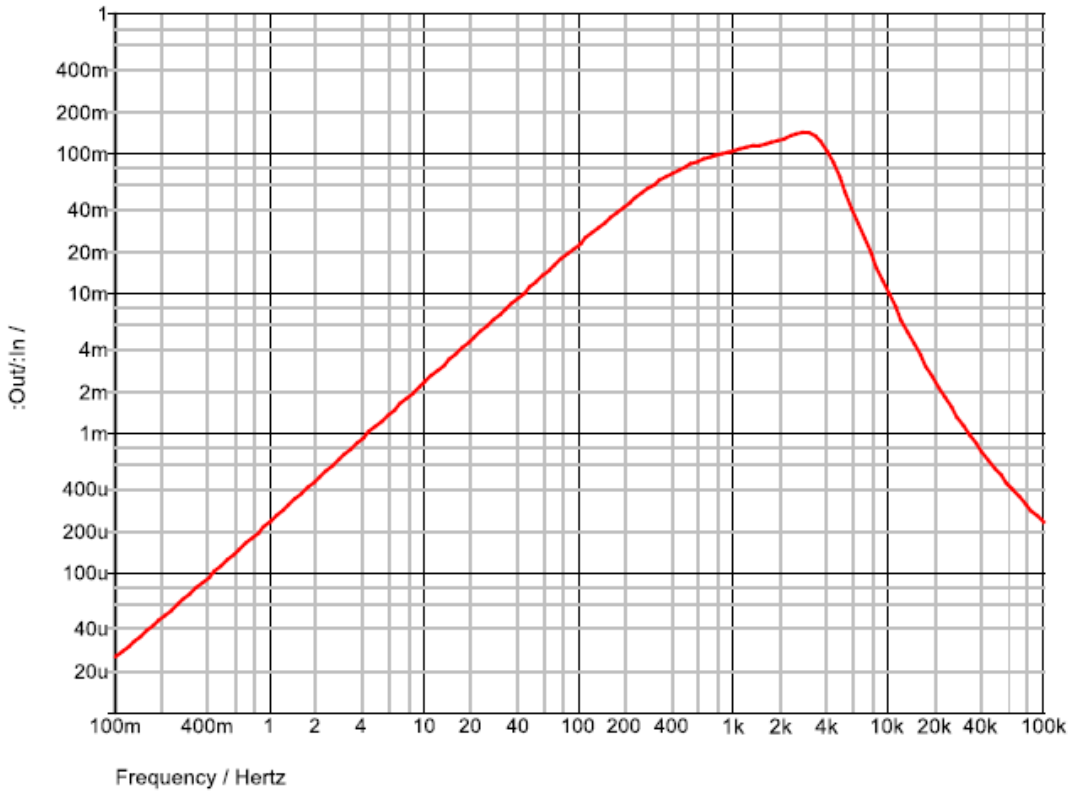
Here is the setup and results for **Output Impedance** analysis:



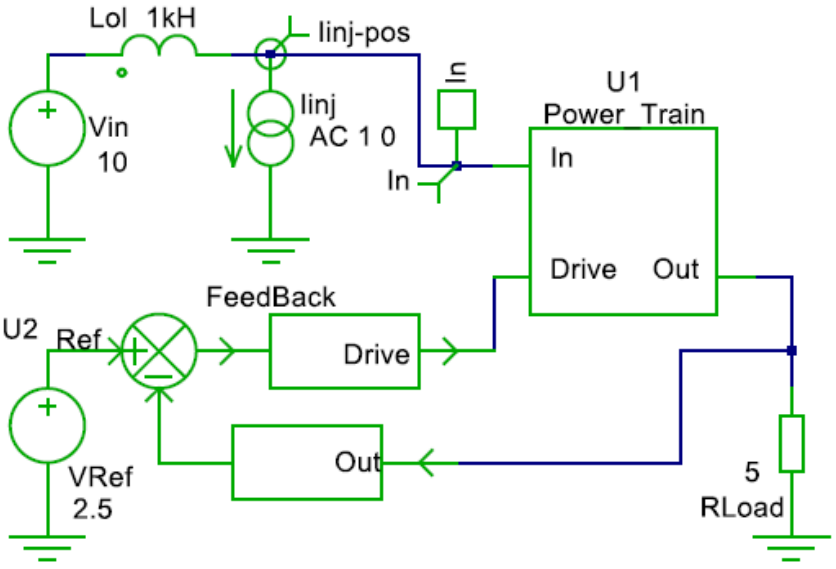


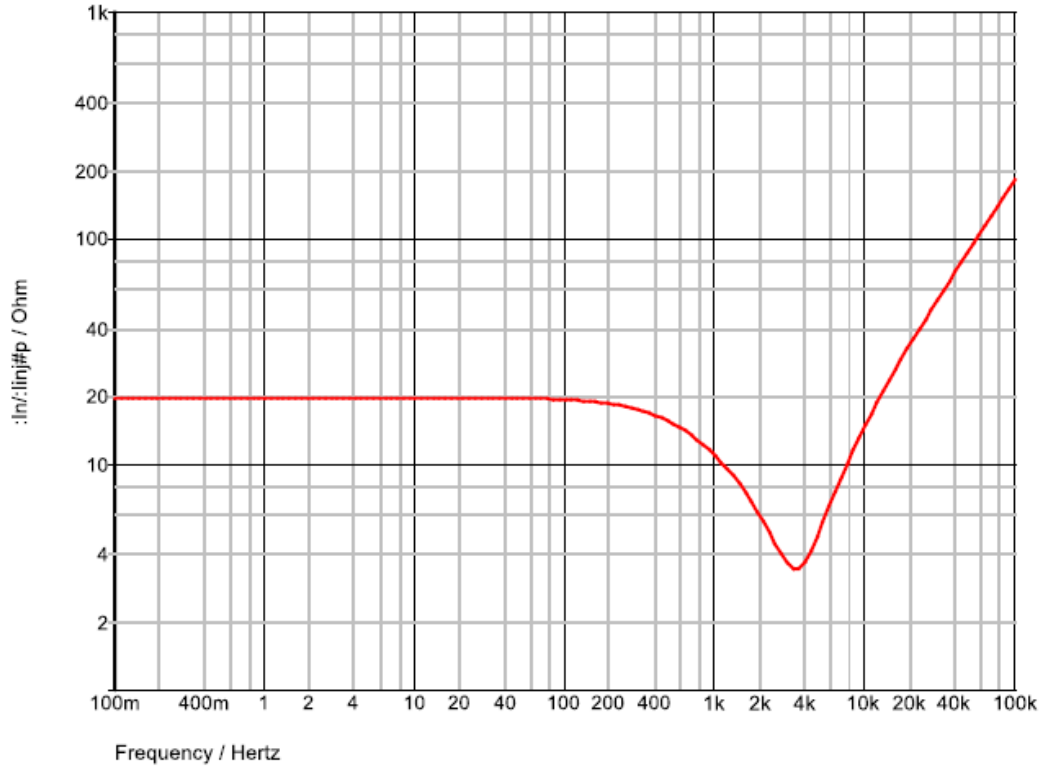
Here is the setup and results for **Input Rejection Ratio** analysis:



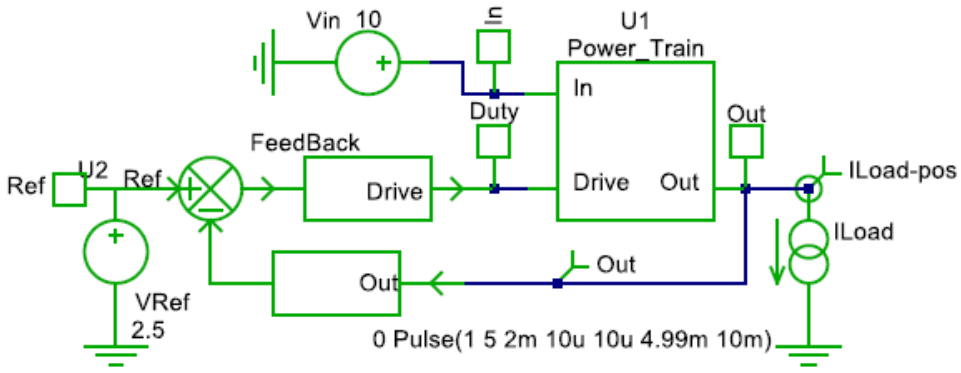


Here is the setup and results for **Input Impedance** analysis:

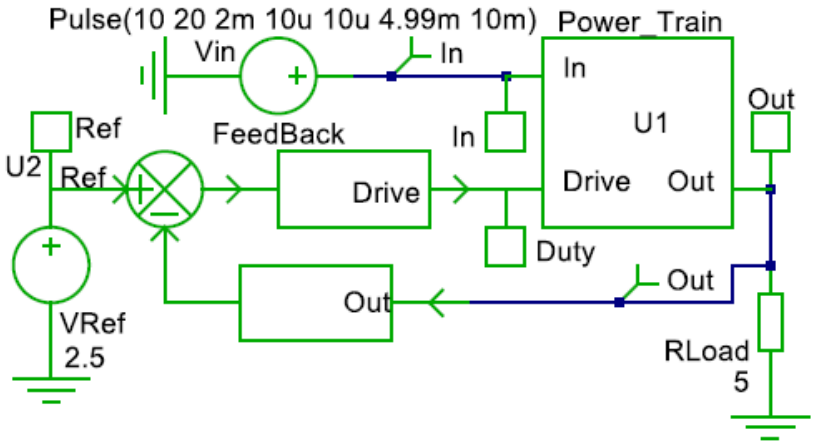




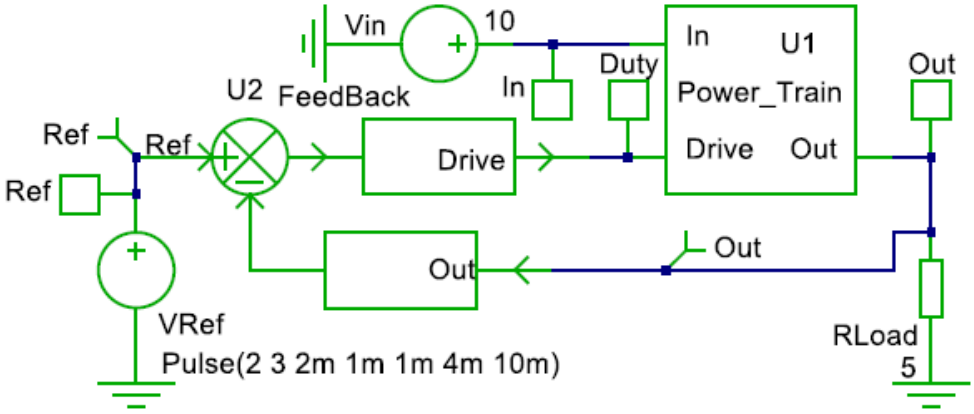
Here is the setup for **Load Transient** analysis:



Here is the setup for **Input Transient** analysis:



Here is the setup for **Reference Transient** analysis:



Here is the example in Mathcad® for the loop optimization:

Loop Optimisation

$dB := 1$

Measurement Circuit:

$RHm := 19.75 \cdot K\Omega$

$RBm := 10 \cdot K\Omega$

$CAm := 100 \cdot nF$

$REm := 10 \cdot K\Omega$

$RSm := 4.7 \cdot K\Omega$

$CSm := 22 \cdot nF$

Simulation Circuit:

$RHs := RHm$

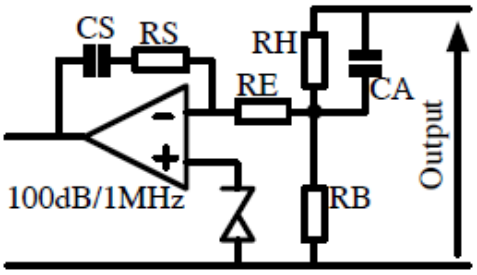
$RBs := RBm$

$CAs := 0.68 \cdot nF$

$REs := 2.2 \cdot K\Omega$

$RSs := 10 \cdot K\Omega$

$CSs := 1 \cdot nF$



Amplifier Parameters:

$A0 := 100 \cdot dB$

$fT := 1 \cdot MHz$

Common Variables or Functions:

$$fC := \frac{fT}{10^{\frac{A0}{20}}}$$

$fC = 10 \cdot Hz$

$$A(f) := \frac{10^{\frac{A0}{20}}}{1 + i \cdot \frac{f}{fC}}$$

$G_dB(X) := 20 \cdot \log(|X|) \cdot dB$

$$P_°(X) := \frac{\arg(X)}{deg}$$

Loading Measurement:

Row 0 = Fréeuency
Row 1 = Gain (dB)
Row 2 = Phase (°)

$Measure := READPRN("Mes1.prn")$

$NbV := rows(Measure)$

$NbV = 401$

$k := 0, 1 .. NbV - 1$

$f := Measure^{\langle 0 \rangle} \cdot Hz$

$GainV := Measure^{\langle 1 \rangle}$

$PhaseV := Measure^{\langle 2 \rangle} - 180$

$$TM := \left[10^{\frac{GainV}{20}} \cdot e^{i \cdot (PhaseV - 180) \cdot deg} \right]$$

Compensator Transfet Function:

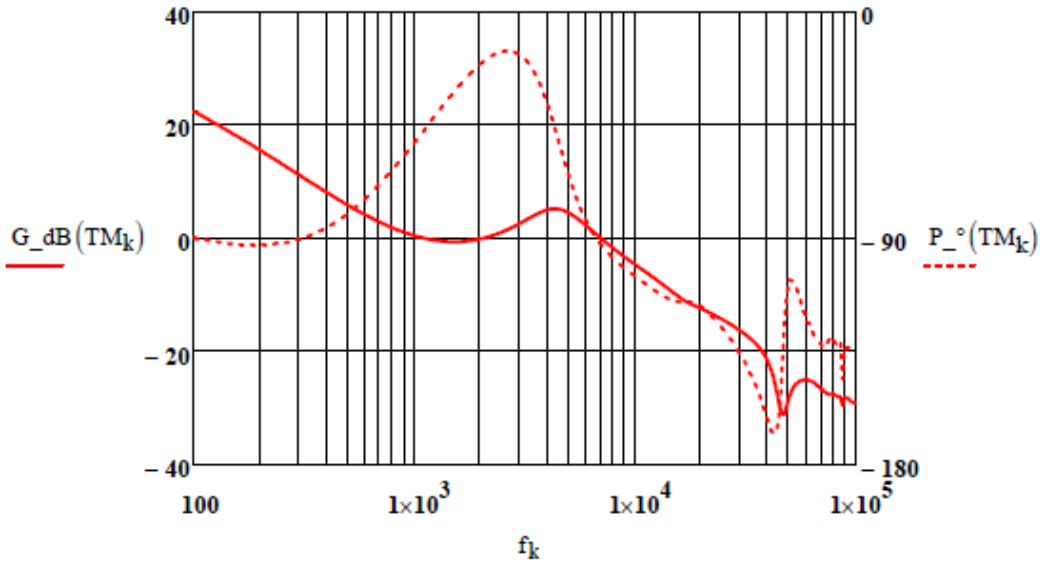
$$\text{Comp}(f,RH,RB,CA,RE,RS,CS) := \left| \begin{array}{l} \omega \leftarrow 2\cdot\pi\cdot f \\ Z2 \leftarrow RS + \dfrac{1}{i\cdot CS\cdot\omega} \\ ZH \leftarrow \dfrac{RH}{1 + i\cdot RH\cdot CA\cdot\omega} \\ KT \leftarrow \dfrac{RB}{RB + ZH} \\ ZT \leftarrow KT\cdot ZH \\ Z1 \leftarrow RE + ZT \\ \dfrac{-A(f)\cdot Z2}{A(f)\cdot Z1 + Z1 + Z2}\cdot KT \end{array} \right.$$

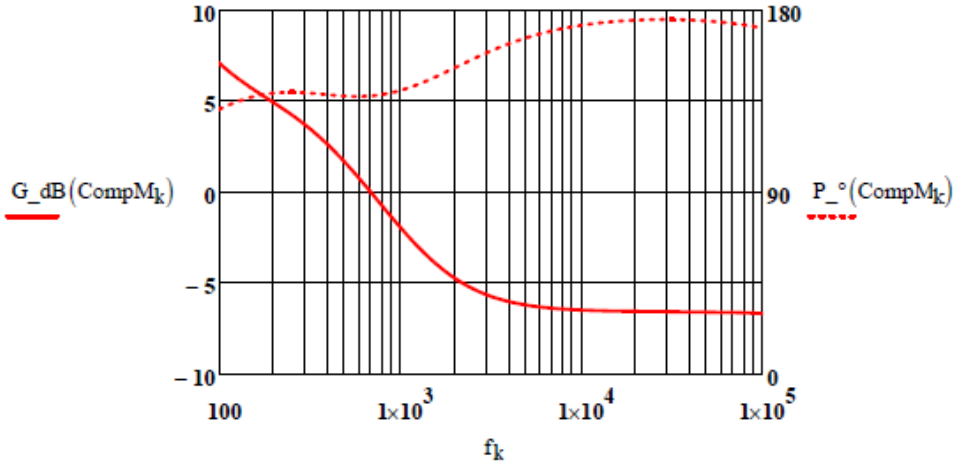
$$CompM_k := \text{Comp}(f_k, RHm, RBm, CAm, REm, RSm, CSm)$$

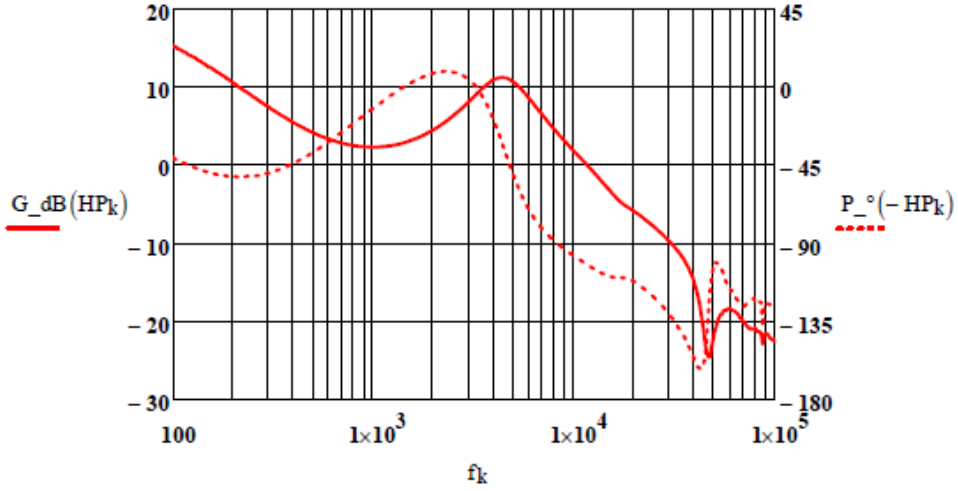
$$CompS_k := \text{Comp}(f_k, RHs, RBs, CAs, REs, RSs, CSs)$$

$$TS := \overrightarrow{\left(TM\cdot\frac{CompS}{CompM}\right)} \qquad HP := \overrightarrow{\left(\frac{TM}{CompM}\right)}$$

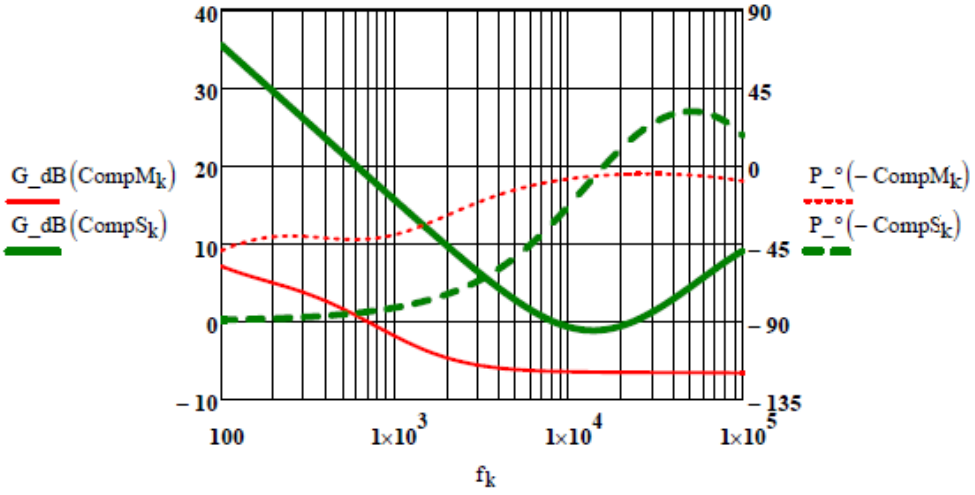
Measurement Graphs:

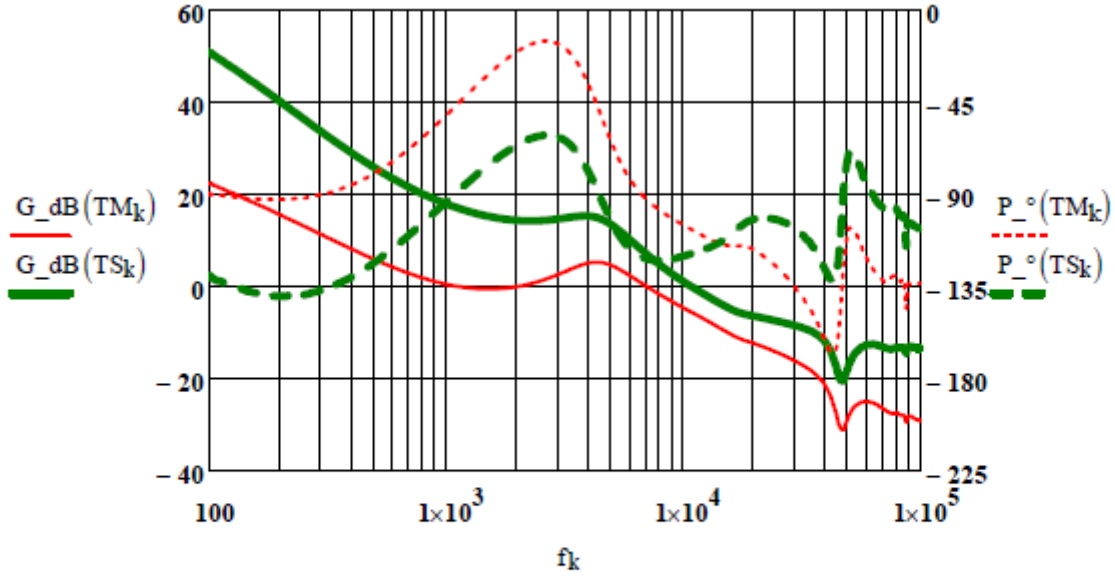


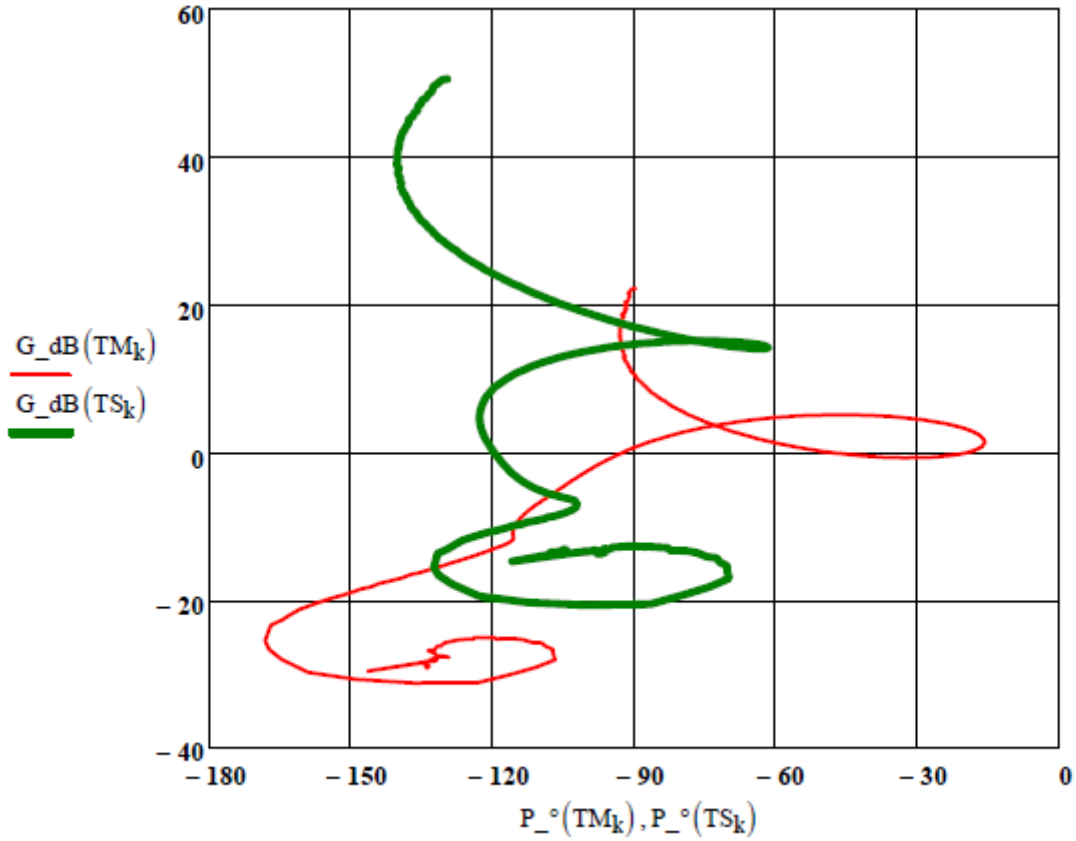




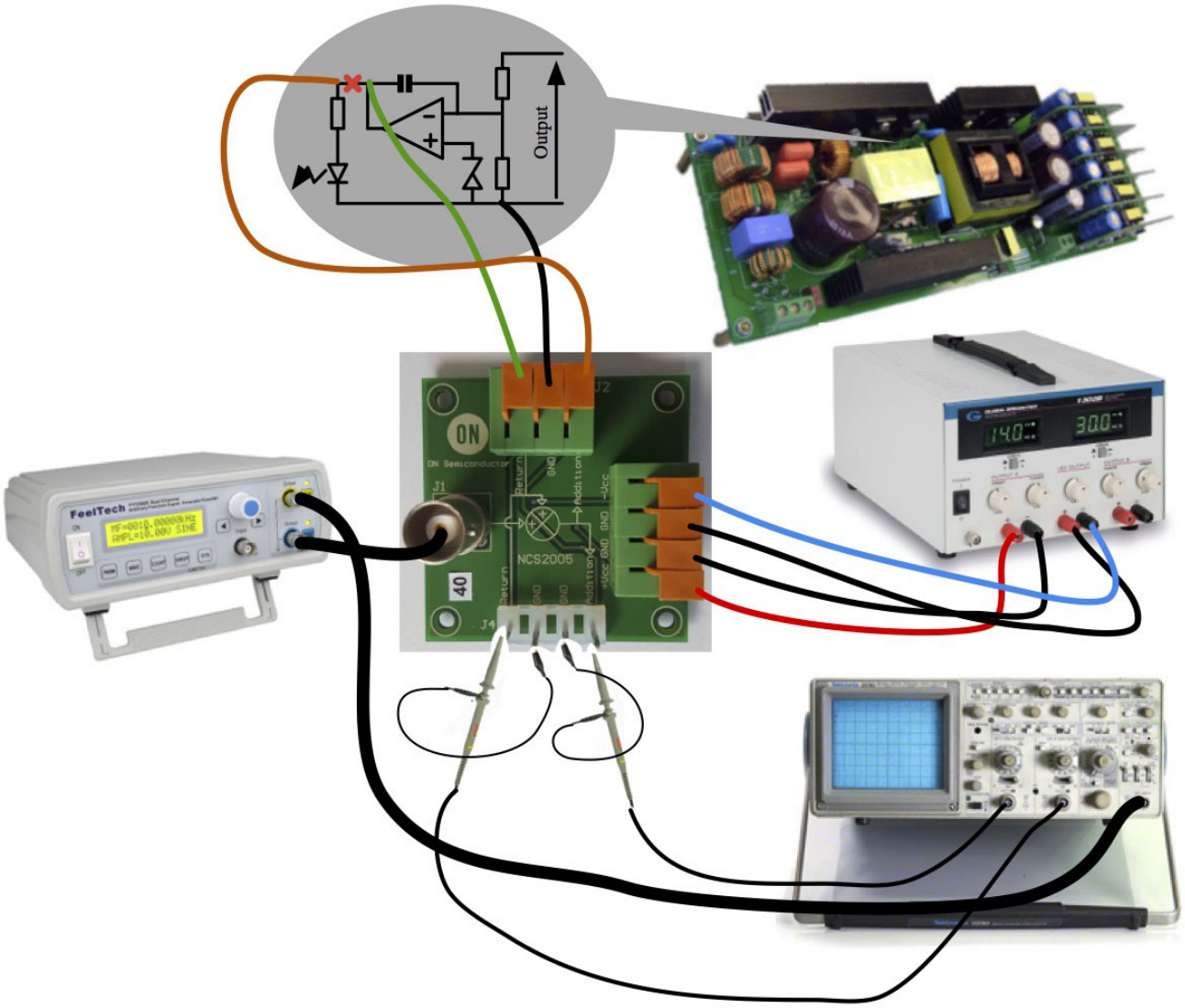
Simulation or optimisation results:




60
40
20
0
– 20
– 40
0
– 45
– 90
– 135
– 180
– 225
100
1×10^3
1×10^4
1×10^5
fk
G_dB(TMk)
G_dB(TSk)
P_°(TMk)
P_°(TSk)


60
40
20
0
– 20
– 40
– 180
– 150
– 120
– 90
– 60
– 30
0
P_°(TMk), P_°(TSk)
G_dB(TMk)
G_dB(TSk)

ANNEX III : MEASUREMENT SETUP



Video Tutorial:

"Simple control loop measurement and verification of the FAN65005−GEVB evaluation board, using an NCS2005 from **onsemi**."
http://www.onsemi.com/PowerSolutions/supportVideo.do?docId=1169064

REFERENCES

[1] Nyquist, H. (January 1932). “Regeneration Theory”. Bell System Tech. J. USA: American Tel. & Tel. 11 (1): 126–147.See link: http://ieeexplore.ieee.org/xpl/articleDetails.jsp?arnumber=6773211

[2] WikiBook: https://en.wikibooks.org/wiki/Control_Systems/Nyquist_Stability_Criteria#Nyquist_Criteria

[3] https://en.wikipedia.org/wiki/Nyquist_plot

[4] https://en.wikipedia.org/wiki/Nichols_plot

[5] https://en.wikipedia.org/wiki/Bode_plot#Gain_margin_and_phase_margin

[6] Christophe Basso books: “Switch Mode Power Supplies: SPICE Simulations and Practical Designs.” McGraw−Hill and “Designing Control Loops for Linear and Switching Power Supplies: A tutorial Guide.” Artech House Publishers

[7] http://cbasso.pagesperso−orange.fr/Spice.htm

[8] https://www.simetrix.co.uk/index.html

[9] Mathcad® is a software from PTC Inc., 140 Kendrick Street, Needham, MA 02494 U.S.A.

[10] Video tutorial: “Simple control loop measurement and verification of the FAN65005−GEVB evaluation board, using an NCS2005 from **onsemi**.” http://www.onsemi.com/PowerSolutions/supportVideo.do?docId=1169064

All brand names and product names appearing in this document are registered trademarks or trademarks of their respective holders.

onsemi, ONSEMI , and other names, marks, and brands are registered and/or common law trademarks of Semiconductor Components Industries, LLC dba “**onsemi**” or its affiliates and/or subsidiaries in the United States and/or other countries. **onsemi** owns the rights to a number of patents, trademarks, copyrights, trade secrets, and other intellectual property. A listing of **onsemi**’s product/patent coverage may be accessed at www.onsemi.com/site/pdf/Patent−Marking.pdf. **onsemi** reserves the right to make changes at any time to any products or information herein, without notice. The information herein is provided “as−is” and **onsemi** makes no warranty, representation or guarantee regarding the accuracy of the information, product features, availability, functionality, or suitability of its products for any particular purpose, nor does **onsemi** assume any liability arising out of the application or use of any product or circuit, and specifically disclaims any and all liability, including without limitation special, consequential or incidental damages. Buyer is responsible for its products and applications using **onsemi** products, including compliance with all laws, regulations and safety requirements or standards, regardless of any support or applications information provided by **onsemi**. “Typical” parameters which may be provided in **onsemi** data sheets and/or specifications can and do vary in different applications and actual performance may vary over time. All operating parameters, including “Typicals” must be validated for each customer application by customer’s technical experts. **onsemi** does not convey any license under any of its intellectual property rights nor the rights of others. **onsemi** products are not designed, intended, or authorized for use as a critical component in life support systems or any FDA Class 3 medical devices or medical devices with a same or similar classification in a foreign jurisdiction or any devices intended for implantation in the human body. Should Buyer purchase or use **onsemi** products for any such unintended or unauthorized application, Buyer shall indemnify and hold **onsemi** and its officers, employees, subsidiaries, affiliates, and distributors harmless against all claims, costs, damages, and expenses, and reasonable attorney fees arising out of, directly or indirectly, any claim of personal injury or death associated with such unintended or unauthorized use, even if such claim alleges that **onsemi** was negligent regarding the design or manufacture of the part. **onsemi** is an Equal Opportunity/Affirmative Action Employer. This literature is subject to all applicable copyright laws and is not for resale in any manner.

PUBLICATION ORDERING INFORMATION

LITERATURE FULFILLMENT:
Email Requests to: orderlit@onsemi.com

onsemi Website: www.onsemi.com

TECHNICAL SUPPORT
North American Technical Support:
Voice Mail: 1 800−282−9855 Toll Free USA/Canada
Phone: 011 421 33 790 2910

Europe, Middle East and Africa Technical Support:
Phone: 00421 33 790 2910
For additional information, please contact your local Sales Representative

◊