Control Loop Design and Easy Verification Method
Control Loop Design and Easy Verification Method

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. It is when the manufacturing of a converter started after the design phase, that instability signs were sometime observed due to some lotto–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 margin 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.

![Feedback Loop Block Diagram](image_url)

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:

\[
\begin{align*}
\text{Error} &= \text{Reference} - K \times \text{Output} \\
\text{Output} &= C \times H \times \text{Error}
\end{align*}
\]

(eq. 1)

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

\[
\text{Output} = \frac{H \times C}{1 + H \times K \times C} \times \text{Reference}
\]

(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:

\[
\lim_{\omega \to 0} |C(\omega)| = \infty
\]

(eq. 3)

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

\[
\text{Output} = \frac{1}{R} \times \text{Reference}
\]

(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.

\[
\text{Measurement} = H \times K \times C \times \text{Error}
\]

(eq. 5)

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

\[
T = H \times K \times C
\]

(eq. 6)

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

\[
\text{Measure} = \frac{H \times K \times C}{1 + H \times K \times C} \times \text{Reference}
\]

(eq. 7)

This formula describes the system closed–loop equation that we can call $S$.

\[
S = \frac{H \times K \times C}{1 + H \times K \times C}
\]

(eq. 8)

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

\[
S = \frac{T}{T + T} \quad \text{and} \quad T = \frac{S}{1 - S}
\]

(eq. 9)
**Stability Criteria**

In the system equation (8), we can clearly 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 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](image)

To have a better robustness and ensure 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 $-180^\circ$ 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 $-180^\circ$.

This gives the following graphs to evaluate those criteria.

![Figure 3. Margins in a Nyquist Plot](image)

![Figure 4. Margins in Bode Plot](image)

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](image)

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.
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. Also 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.

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 $\varepsilon$ effects, we can solve the system and get the return equation as a function of either $\varepsilon$ or $Addition$.

We can easily get $Return$ over $\varepsilon$.

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

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

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

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
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 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.

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.

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 (−KC) 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:

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

We will simulate a basic buck dc–dc converter.

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.

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

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

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.

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

Sometime, ac simulations’ results use dB to display ratio, the impedance is given in \( \text{dB} \) instead of W.

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.
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](image)

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., 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](image)

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 ON Semiconductor
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 need to be “bigger” than the sine wave injected. Generally, $-1 \text{ V}$ 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 $2 \text{ V}$ above the output maximum value is a good practice.

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

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

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 $\text{Addition}$ and $\text{Return}$ exhibit similar amplitudes, then we are at the crossover frequency and the open loop gain is $1$ or $0 \text{ dB}$. 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 $180^\circ$ already in the measurement. That gives us directly the phase margin

2. When the $\text{Addition}$ and $\text{Return}$ 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 $\text{Addition over Return}$ (and not $\text{Return over Addition}$ as for the open–loop transfer function). At that point $\text{Return}$ should be lower than $\text{Addition}$. 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 $\text{Return over } \epsilon$ using (12). Them, calculate the open–loop using equation (9). The advantage of that close–loop measurement is: below the crossover frequency, $\text{Return}$ and $\epsilon$ 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 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](image1)

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](image2)

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](image3)

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](image4)

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

On the Black–Nichols graph below, 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.
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 form 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 ON Semiconductor) 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 margin has been shown.

More details can be also found in annexes.

ACKNOWLEDGMENT

I want to thank my Ph.D. advisor Mr. Zardini for teaching me power electronics and initiating me to loop measurement with the HP4194A network analyzer. It is a very nice and powerful tool.

I was lucky to have the freedom by my previous company manager Mr. Buhannic to investigate and to implement this loop measurement method in the company in conjunction with simulation tool.

One person gives me the chance to discuss and get deeper in all of those simulations’ aspects, review his books and chatting together time to time. Christophe, un grand merci. I want to thank also all the customers in Fairchild and ON I met who were having loop stability issues. We discussed and sometimes applied this method to their products. I had to explain that method while practicing at the same time. It gave me the background material to organize, synthetize and write this paper today.

My warm recognition goes to Fairchild and ON people. They gave me the opportunity to be here and make this seminar on this very practical topic. They also make this injection adder board becomes real.

AUTHOR

Didier Balocco, He received is Electronics Engineer diploma in the “École Nationale Supérieure d’Électronique et de RadioÉlectricité de Bordeaux”, France in 1992 and his Ph. D. degree in Power Electronics form the University of Bordeaux in 1997. In 1996, he joined AEG Power Solutions, formerly Alcatel Converters, as a research engineer for dc–dc and ac–dc converters design in a range of 1 W to 1 kW mainly for telecom equipment. He managed the research activities from 2000 to 2014. He published more than 10 papers on power electronics. He holds 1 patent. From 2011 to 2013, he worked 18 months on a 15 kW solar inverter module for a 150 kW cabinet in Dallas, Texas, USA. His main interests during that period were switching mode power supply, converter stability and modeling, high power factor rectifiers.

He joined Fairchild in August 2014 as a Field Application Engineer supporting South of France, Spain and Portugal. In 2016, ON acquired Fairchild. He is now Senior FAE for power application in France, Spain and Portugal.
ANNEX I : SIMetrix SIMULATION SETUPS AND RESULTS

Here is the Voltage Mode average PWM Model schematic:

.Model Dclip D n=0.01 rs=100m
.Param Trans=1m Req=1k
Here is the Current Mode average PWM Model schematic:

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

![PWM Model Schematic]

Here is the simple amplifier Model schematic:

```
.Model Dcpl D n=0.01 rs=100m
.Param VMid=(VHigh+VLow)/2
.Param Req=(Gain/Trans)
.Param VMin=(VLow-VMid)
.Param VMax=(VHigh-VMid)
```

![Amplifier Model Schematic]
Buck stage with Voltage Mode PWM for open-loop simulation:

Load = \{1\text{A}\}\text{ at } 100kHz, \text{ DMax = 0.95, DMn = 0.05}

Open-loop results:

Gain / dB

Phase / degrees

Frequency / Hz
Changing the simulation Bode plot setup on the previous schematic:
Boost stage with Current Mode PWM for open-loop simulation:

\[ L_{sw} = \text{[Boost]} \quad f_{sw} = 1 \text{MegHz} \quad D_{max} = 0.95 \quad D_{min} = 0.05 \quad R_{sense} = 40 \text{m} \quad \text{Slope} = (Se) \]

Caution: \( R_{sense} \) is NEGATIVE for the boost stage because we rotate the model.

* Boost parameters

- \( \text{Param LBoost} = 5 \mu\text{H} \)
- \( \text{Param S} = 0 \)
- \( \text{Param V} = 2.7 \text{V} \)
- Initial values
  - \( \text{Param RUpper} = 10 \text{k} \quad f_c = 8 \text{k} \)
  - \( \text{Param pm} = 60 \quad f_{pc} = -20 \quad f_{pc} = -101 \)

* Feedback parameters

- \( \text{Param G} = (10^{(\text{G}_{pc}/20))} \)
- \( \text{Param Boost} = (\text{pm}-\text{pfo}-90) \)
- \( \text{Param K} = \tan((\text{Boost} + 45)\pi/180) \)
- \( \text{Param Cn} = (1/(2(\pi)^2 f_c G_k \cdot R_{Upper})) \)
- \( \text{Param Ci} = (C_n (K^2 - 1)) \)
- \( \text{Param Rk} = (K/(2(\pi)^2 f_c C_i)) \)

AC DEC 25 100 1Meg SWEEP PARAM=Se LIST 0 16.4k

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:

[Diagram of a circuit with labels such as U1, Power Train, Drive, Out, Vin, VRef, linj-pos, linj, RLoad, AC1.0, and a graph showing Out/sin(lip) vs. Frequency.]
Here is the setup and results for Input Rejection Ratio analysis:

![Circuit Diagram]

![Graph]

Frequency / Hertz
Here is the setup and results for Input Impedance analysis:
Here is the setup for Load Transient analysis:

![Load Transient Diagram]

Here is the setup for Input Transient analysis:

![Input Transient Diagram]

Here is the setup for Reference Transient analysis:

![Reference Transient Diagram]
ANNEX II : MATHCAD® CALCULUS EXAMPLE

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

**Loop Optimisation**

**Measurement Circuit:**
- RHm := 19.75 KΩ
- RBm := 10 KΩ
- CAM := 100 nF
- REM := 10 KΩ
- RMS := 4.7 KΩ
- CSm := 22 nF

**Simulation Circuit:**
- RHs := RHm
- RBs := RBm
- CAS := 0.68 nF
- REs := 2.2 KΩ
- RSS := 10 KΩ
- CSs := 1 nF

**Amplifier Parameters:**
- A0 := 100 dB
- fT := 1 MHz

**Common Variables or Functions:**
- \( fC := \frac{fT}{10^{A0/20}} \)
- \( fC = 10 \text{ Hz} \)
- \( \frac{A0}{10^{20}} \)
- \( G_{dB}(X) := 20 \log(|X|) \cdot dB \)
- \( P_{\phi}(X) := \frac{\arg(X)}{\text{deg}} \)

**Loading Measurement:**
- Row 0 = Frequency
- Row 1 = Gain (dB)
- Row 2 = Phase (°)

Measure := READPRN("Mes1.prn")

NbV := rows(Measure)

k := 0, 1 .. NbV - 1

f := Measure(0).Hz

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

\[
\text{Comp}(f, RH, RB, CA, RE, RS, CS) :=
\begin{align*}
\omega & \leftrightarrow 2 \pi f \\
Z2 & \leftrightarrow RS + \frac{1}{iCS\omega} \\
ZH & \leftrightarrow \frac{RH}{1 + iRH\cdot CA\cdot \omega} \\
KT & \leftrightarrow \frac{RB}{RB + ZH} \\
ZT & \leftrightarrow KT \cdot ZH \\
Z1 & \leftrightarrow RE + ZT \\
& \frac{-A(f) \cdot Z2}{A(f) \cdot Z1 + Z1 + Z2} \cdot KT
\end{align*}
\]

\[
\text{CompM}_k := \text{Comp}(f_k, RH_{Im}, RB_{Im}, CA_{Im}, RE_{Im}, RS_{Im}, CS_{Im})
\]

\[
\text{CompS}_k := \text{Comp}(f_k, RH_{s}, RB_{s}, CA_{s}, RE_{s}, RS_{s}, CS_{s})
\]

\[
\text{TS} := \left( \frac{\text{TM}}{\text{CompM}} \right)
\]

\[
\text{HP} := \left( \frac{\text{TM}}{\text{CompM}} \right)
\]

**Measurement Graphs:**

![Graph](image-url)
REFERENCES


[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.