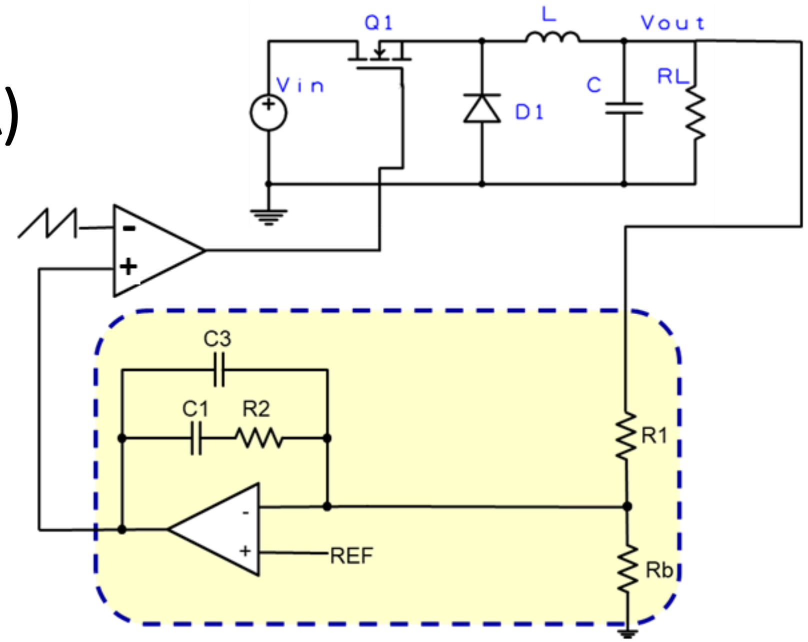
BIRICHA DIGITAL

Power Supply Compensator Design Using Operational Transconductance Amplifiers

Expert from Biricha Digital’s Power Supply Design Workshop

Register here for free resources and future workshops
www.biricha.com/omicron.html

Peak Current Mode Control Compensator Design (with an OTA)



- As you may already be aware, for current mode control we usually only need a Type II compensator as shown on the right
  - Type II design with a *standard op-amp* is well documented*
- However, sometimes PWM ICs contain an internal *"Operational Transconductance Amplifier"* (OTA) instead of the standard voltage amplifier
  - Advantages → usually cheaper
  - Disadvantage:
    - Can't easily make a Type III with an OTA
    - Isolated loop design process is not well defined or documented
- For completeness we will design our next converter using a transconductance amplifier
  - We will use the extremely popular LM515x range of controllers to show an example of compensator design using OTAs

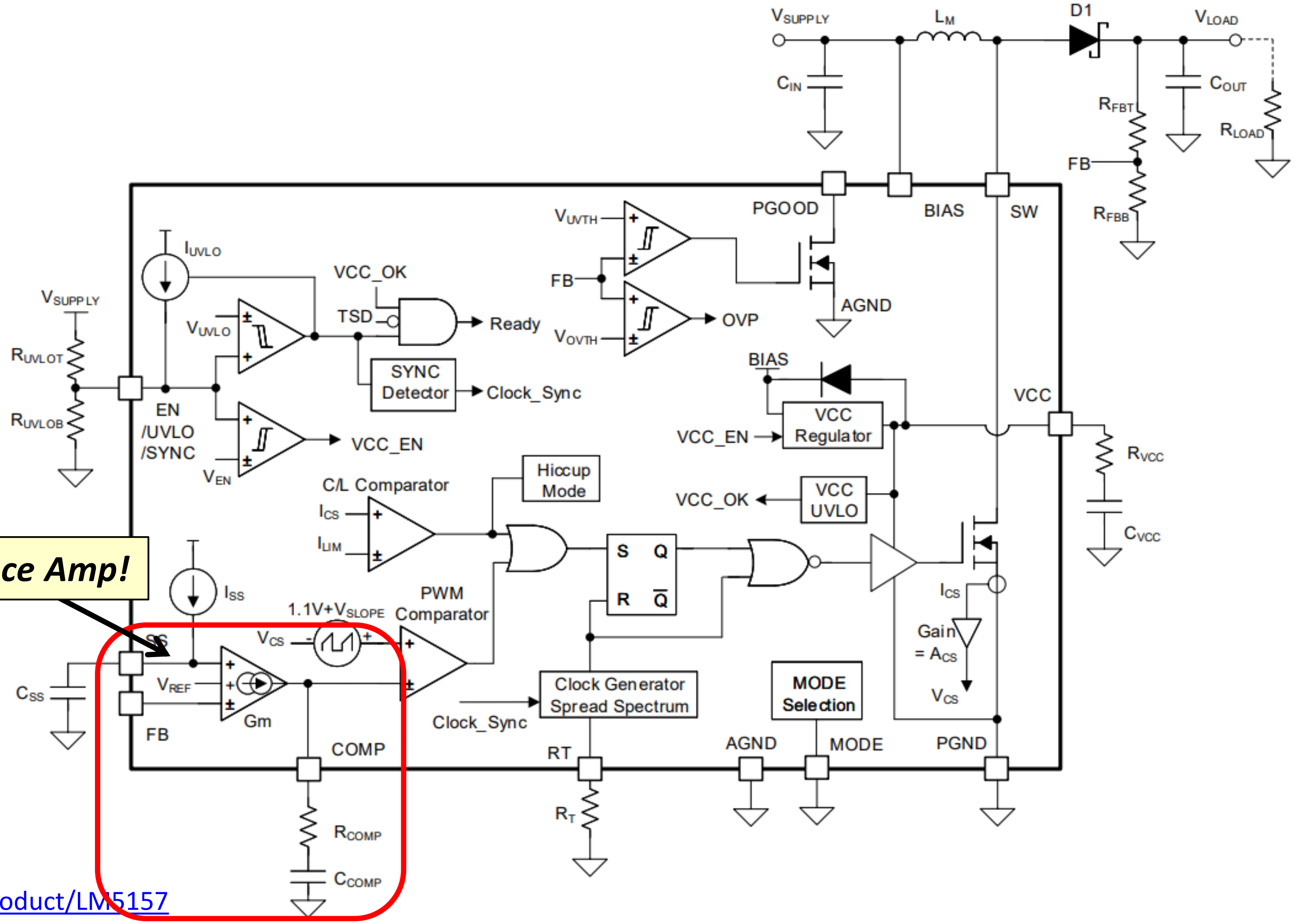
Before we start our design, let us review Transconductance Amps

* Please see Basso ISBN 978-0071823463
* Maniktala ISBN 978-0123865335

CCM Boost Converter Control Loop Design Example

- We will use a TI LM5157*:
  - Internal switch
  - Internal slope
    - Ramp size in datasheet
  - Internal current sense circuitry including signal amplification
    - Current Sense Gain from datasheet
  - A Transconductance Amplifier
    - VREF and Gm in datasheet

Transconductance Amp!



* https://www.ti.com/product/LM5157

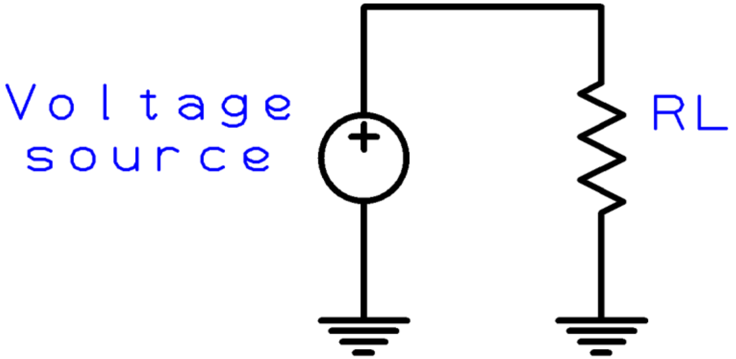
Voltage Sources, Current Sources and Transconductance Amplifiers

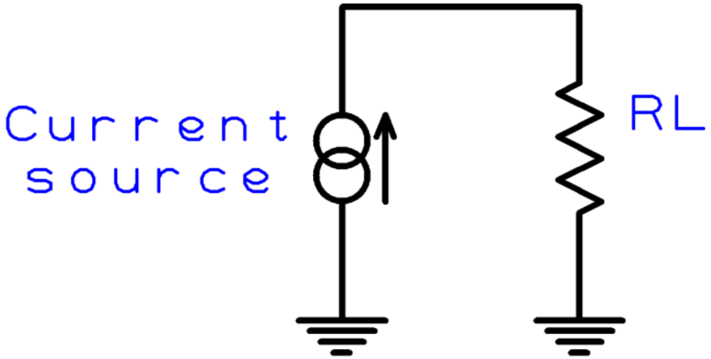
- Some of the very first topics that we study in electronic engineering are ideal voltages sources and op-amps
  - Typically only Voltage Operational Amplifiers i.e. standard op-amps
  - Typically voltages sources are covered in depth but current sources are usually only mentioned briefly
- Operational Transconductance Amplifiers (OTA) barely get a mention in most university courses
  - It is not a standard voltage Op-Amp *AND* it is a current source ☺
  - In many modern PWM ICs however, we will find only a OTA rather than a normal op-amp
- So we may have to design our compensator using an OTA
  - But most textbooks and App Notes only deal with normal op-amps
  - But a very nice detailed explanation can be found in Christophe Basso's book ISBN 978-1-60807-557-7

A Quick Review Voltage and Current Sources

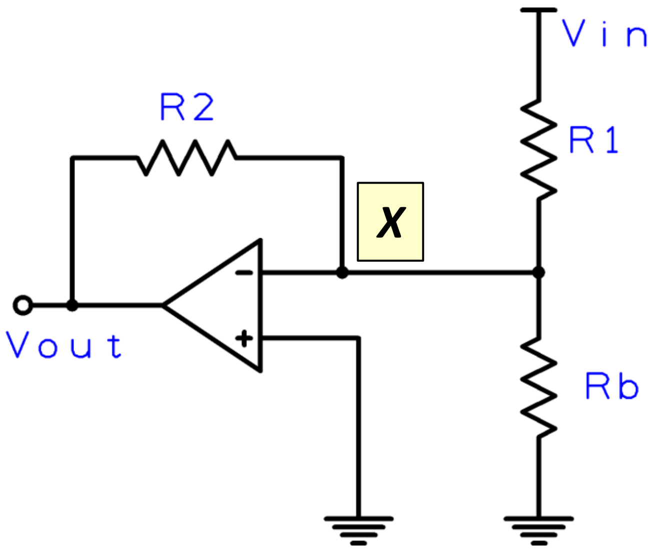
A Very Quick Review!

- An *ideal* voltage source provides whatever current that is necessary to keep the voltage constant across RL as RL changes
  - A 1V voltage source with RL = 1Ω would source 1A
  - The same voltage source with 1MΩ would source 1uA
  - The same voltage source with 1mΩ would source 1000A
- An *ideal* current source provides whatever potential difference that is required to keep the current through RL constant as RL changes
  - A 1A current source with RL = 1Ω would form a p.d. of 1V across RL
  - The same current source with 1MΩ would form a p.d. of 1MV!
  - The same current source with 1mΩ would form a p.d. of 1mV





A Quick Review of Standard Voltage Amplifier

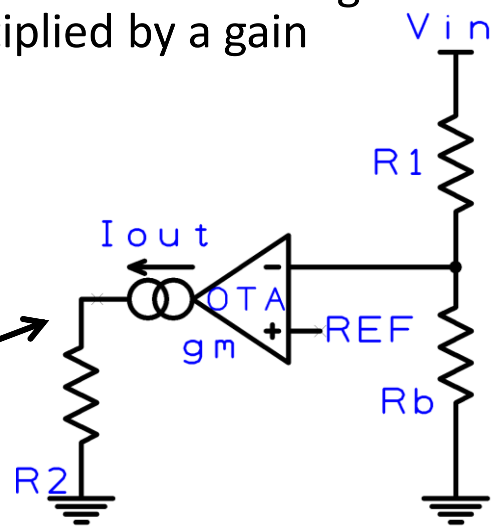


- The standard *ideal* op-amp is effectively a *Controlled Voltage Source*
  - Vin is a voltage source and Vout is also a voltage source
  - The current out of the output pin would be whatever is needed to make sure Vout stayed at the desired value → i.e. just like an ideal voltage source
- With an OTA, our output is a current (Iout) and not a voltage, so to make a voltage amplifier, we first need to convert the output current to a voltage
  - To make an OTA equivalent of a standard op-amp, all we have to do is to connect our feedback impedance to ground instead of point X of the standard op-amp
  - Current will now flow out of the Iout pin, flow through R2 and a voltage (Vout) will appear across R2
- Output of our OTA, is the potential difference between the non-inverting (Vin+) and the inverting (Vin-) pins of our Op-Amp multiplied by a gain called "transconductance" or gm

$$Iout = (V_{in+} - V_{in-}).gm$$

- Therefore, gm *WILL* appear in the transfer function
  - But thankfully it is always specified in the datasheet ☺

Output is a current so we convert to a voltage → Vout = Iout x R2

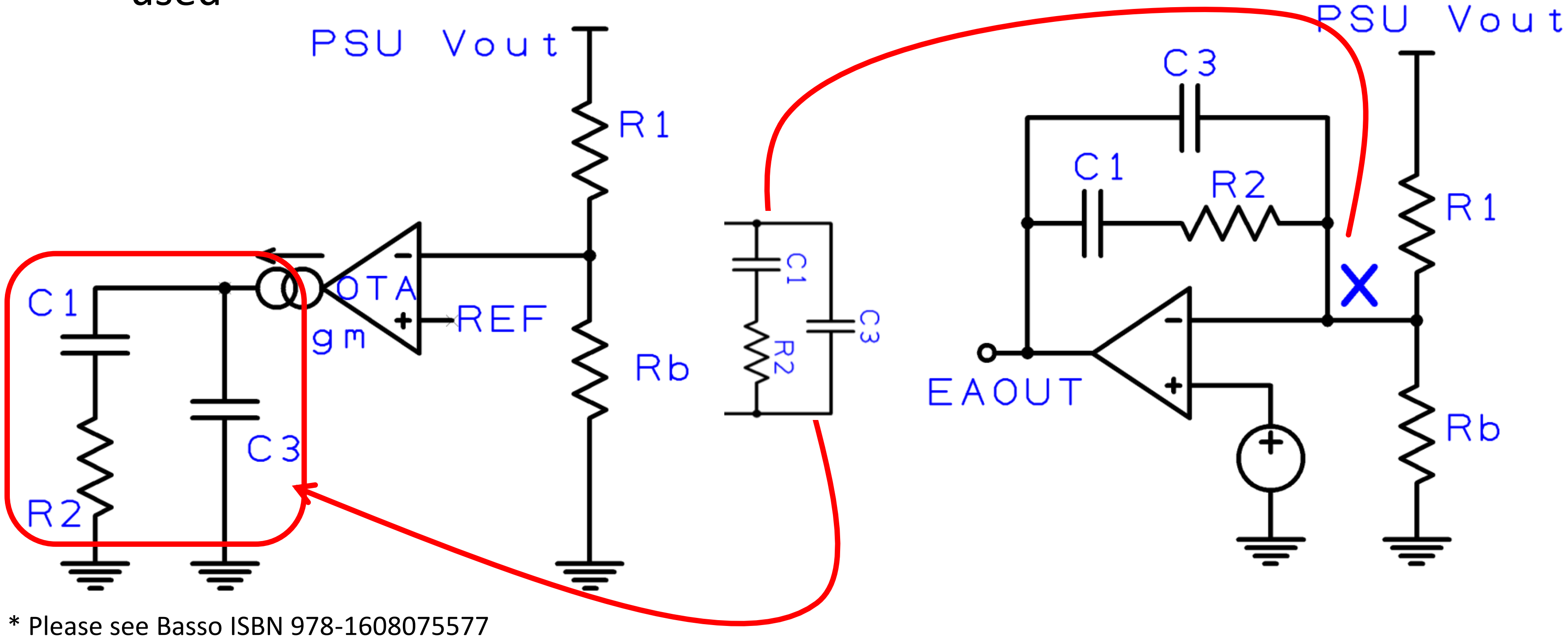


How to Make a Compensator with an OTA

- OTA Type II is quite easy, all you have to do is to disconnect the feedback components from point X (of a standard op-amp) and connect to ground
  - Current will flow from the OTA output, and the voltage that appears across our impedances will have the frequency response determined by C1, R2, C3 (… and R1 and Rb and gm ☺)
- Note that a practical Type III is not simple and therefore it is almost never used**

All we have to do now is to derive the transfer function of our Type II compensator

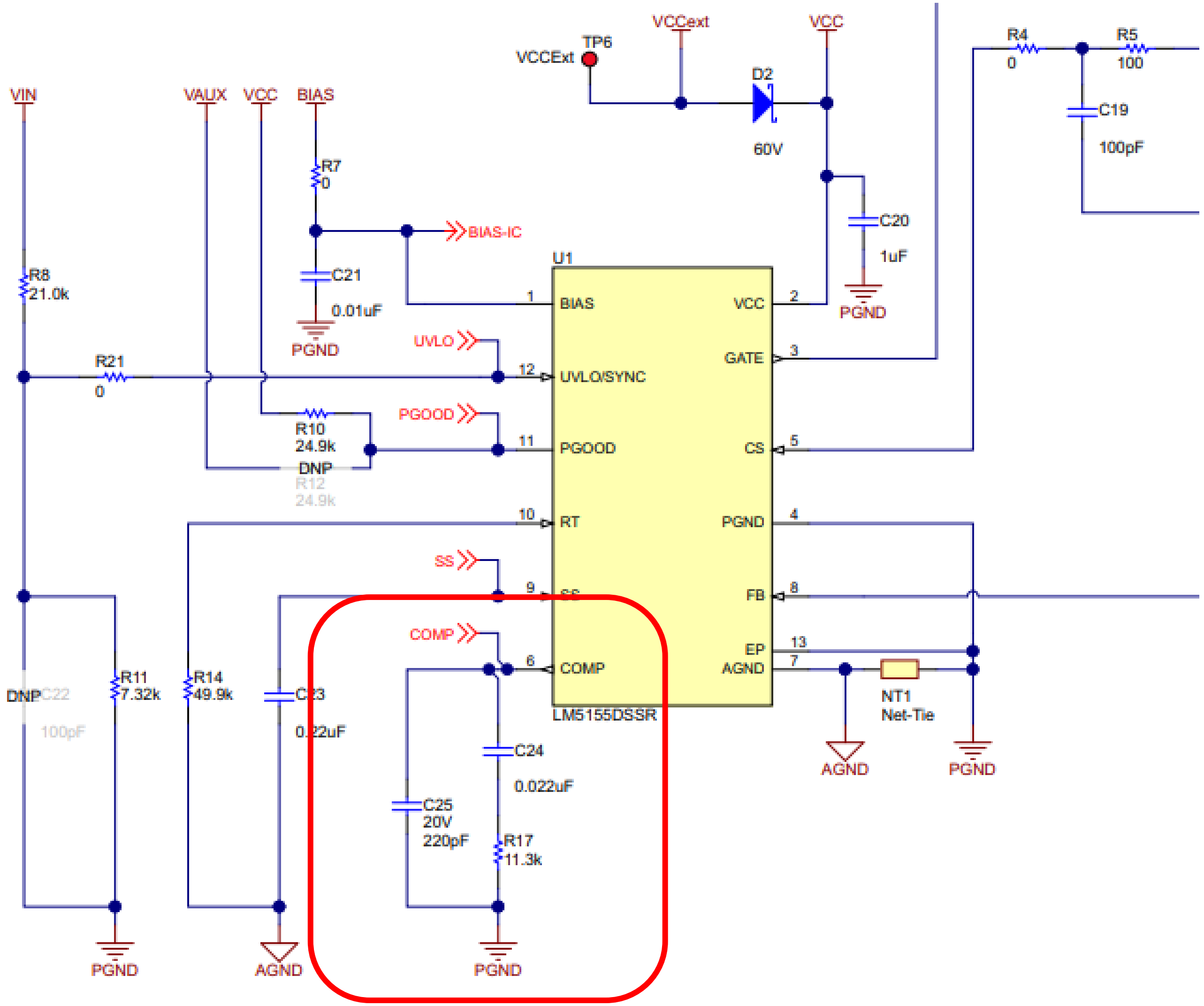
Luckily this has already been done* and it is already in Biricha WDS + various other PSU design tools



* Please see Basso ISBN 978-1608075577
* Maniktala ISBN 978-0123865335

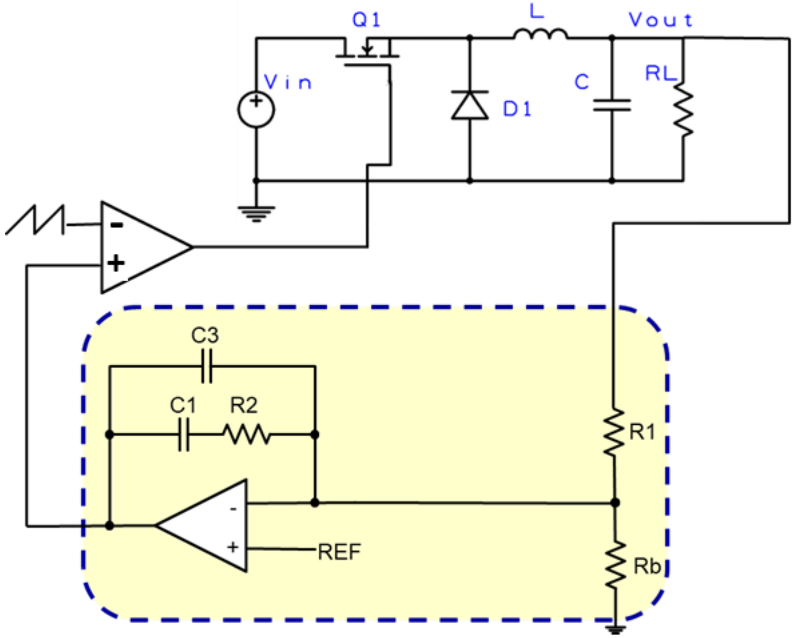
LM5155 Boost EVM OTA Example*

Very easy to say of your chip has an OTA
If compensation components are tied to ground then it is an OTA



* https://www.ti.com/lit/ug/snvu608/snvu608.pdf?ts=1713127283620&ref_url=https%253A%252F%252Fwww.ti.com%252Ftool%252FLM5155EVM-BST

Type II Compensator with a Standard Voltage Amplifier



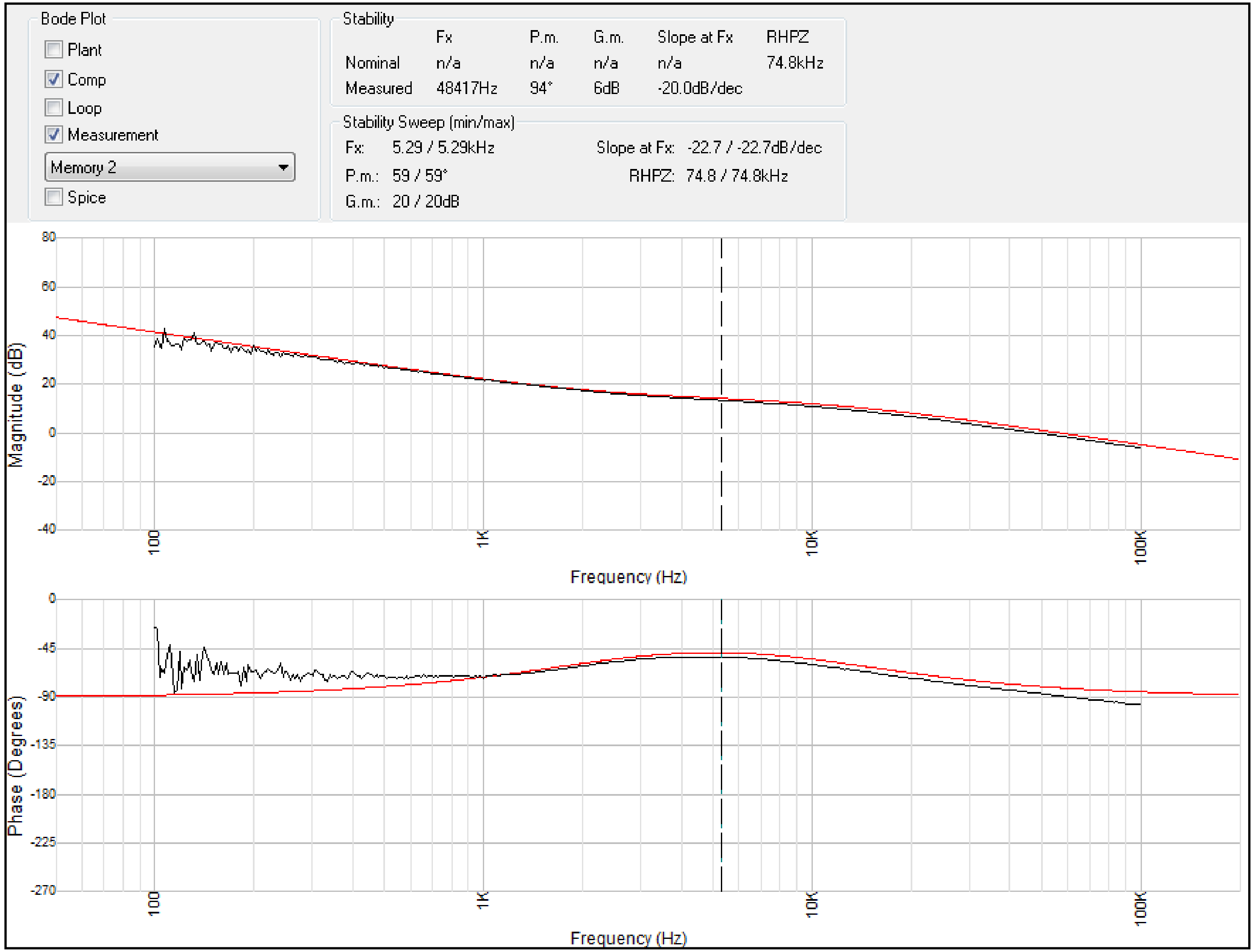
- Type II
  - 2 poles & 1 zero
  - Most commonly used for current mode control (Flybacks/Boost/SEPIC etc.)
  - Not suitable for voltage mode Buck or Forward converters (need Type III)
  - Transfer function:

$$H_c(s) = \left(\frac{\omega_{p0}}{s}\right) \cdot \frac{\left(\frac{s}{\omega_{z1}} + 1\right)}{\left(\frac{s}{\omega_{p2}} + 1\right)}$$

  - Typically start by positioning our poles and zeros → Day 1 of our workshop or WDS
  - Then we select R1 & Rb (more on this later) and then we can calculate everything else
  - ωp0 = frequency of the pole @ origin, ωp0 frequency of the 1st pole, ωz1 = frequency of the 1st zero in rad/s for Hz just divide by 2π

$$\omega_{z1} = \frac{1}{R_2\, C_1}\; ; \omega_{p0} = \frac{1}{R_1\left(C_1 + C_3\right)}\; ; \omega_{p2} = \frac{\left(C_1 + C_3\right)}{R_2\, C_1\, C_3}$$

Type II Compensator with Standard Voltage Error Amplifier

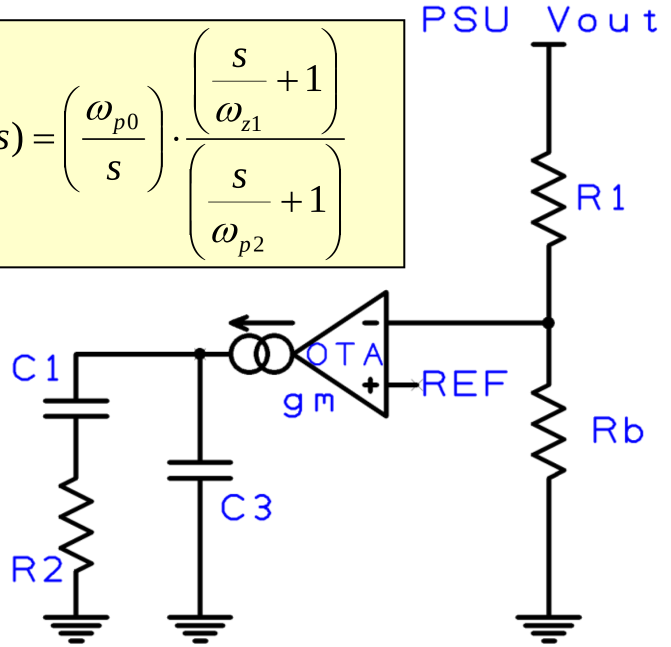


- Red line: WDS simulation
- Black line: real measurement with Bode100

Type II Compensator with an OTA

- Transfer function is exactly the same as the standard op-amp Type II
  - But the equations for pole zero placement are different as shown below
  - fp0, fp1, fz1 etc. the pole and zero positions in Hz
- First we will use Biricha's WDS Power Supply Design Software to place our poles and zeros
  - We cover the principles of how this is done in our workshop
  - After selecting the position of our poles and zeros we can select the component values using the following equations*
- gm is defined in the datasheet and typically displayed in mA/V, uA/V, uSiemens (uS) or in old devices in Mhos
- Typically start by selecting R1 & Rb such that the current through them is >100uA
- Then we can calculate everything else because we have 3 equations & 3 unknowns ☺

$$H_c(s) = \left(\frac{\omega_{p0}}{s}\right) \cdot \frac{\left(\frac{s}{\omega_{z1}} + 1\right)}{\left(\frac{s}{\omega_{p2}} + 1\right)}$$



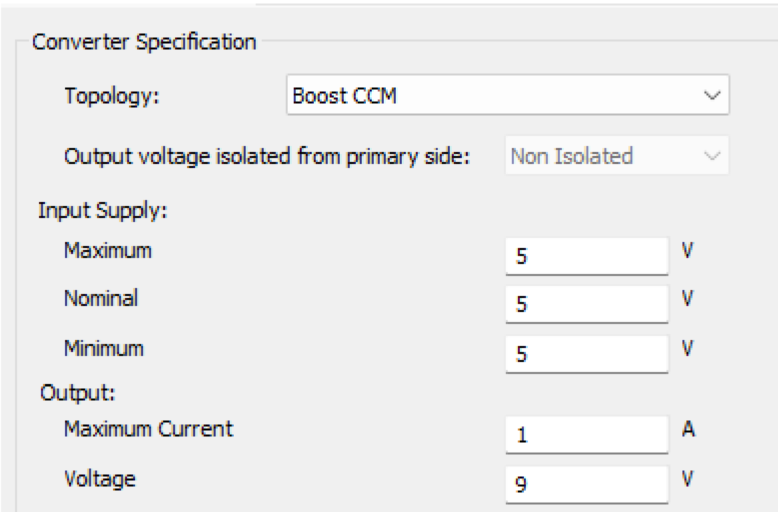
$$f_{z1} = \frac{1}{2\pi\, R_2\, C_1} \; ; \; f_{p0} = \frac{g_m R_b}{2\pi\, (R_1 + R_b)\, (C_1 + C_3)} \; ; \; f_{p2} = \frac{(C_1 + C_3)}{2\pi\, R_2\, C_1\, C_3}$$

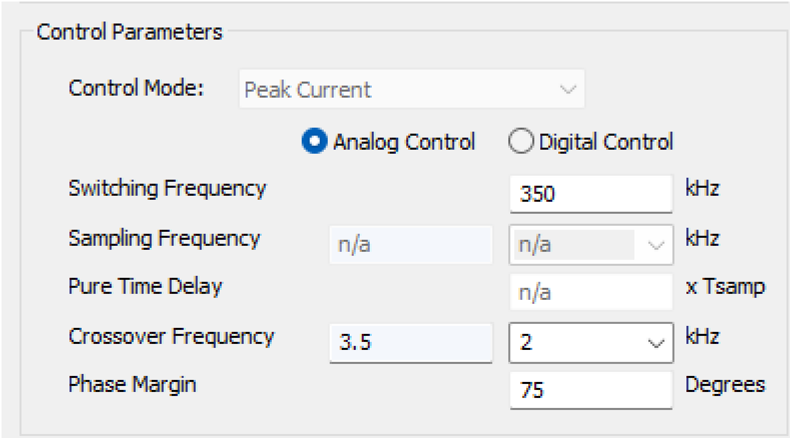
Design example coming up shortly

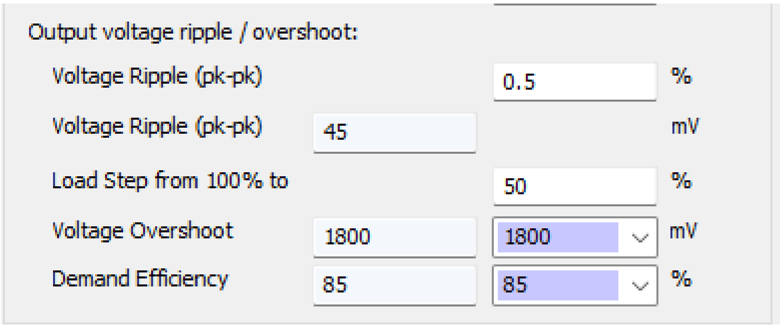
* All terms denoted by ω in the transfer function are in rad/s → for Hz, just divide by 2π

CCM Boost Converter Control Loop Real Design Example

- Let us design a CCM Boost with the following specification
  - From specifications tab





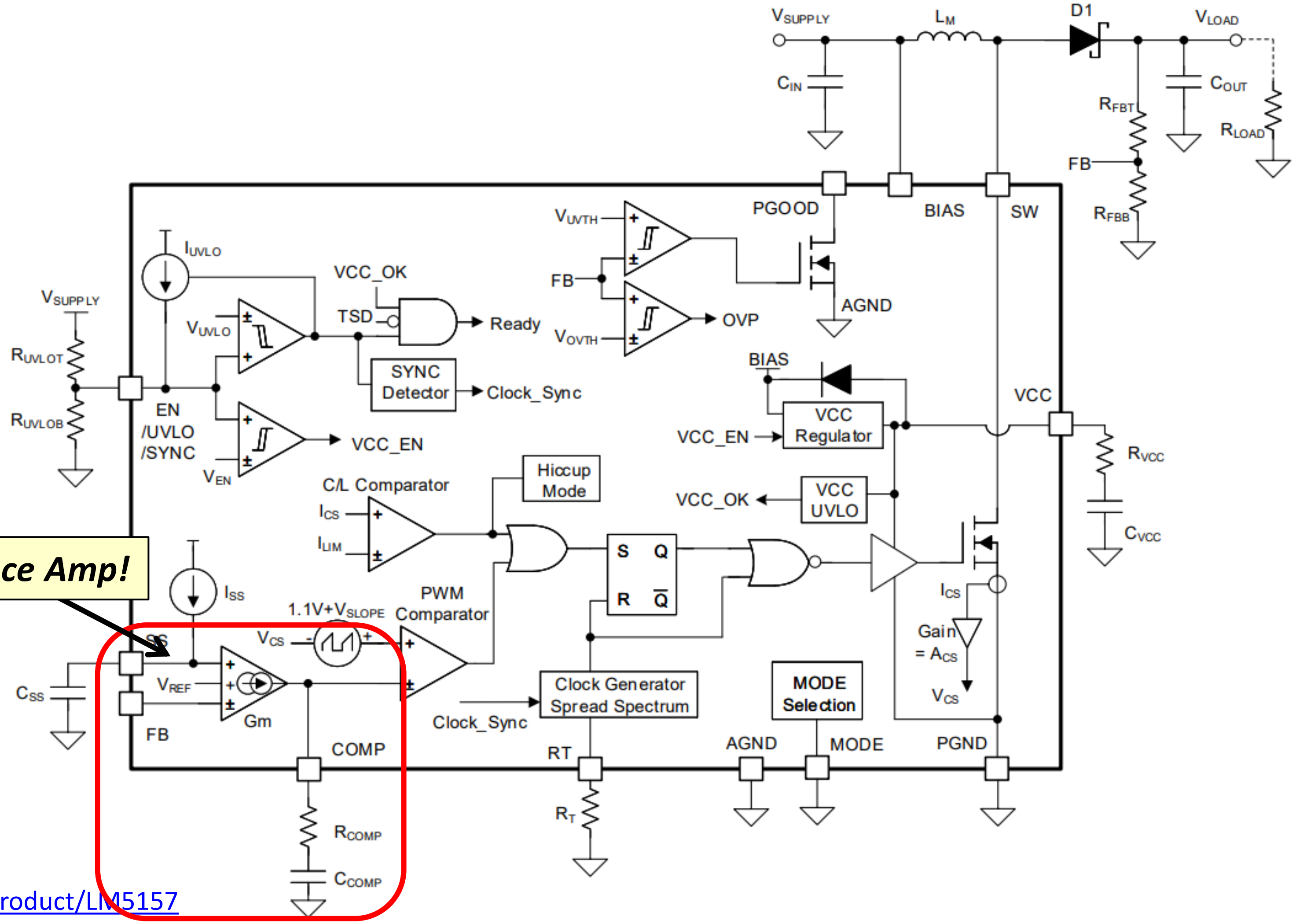


- Under CCM conditions Boost will have a RHPZ issue
  - We must use current mode
    - In fact under DCM we usually use Imode also mainly so that we can limit the peak of the inductor current
  - Under CCM we will have a subharmonic oscillation issue
    - We will use a PWM IC with internal slope compensation

CCM Boost Converter Control Loop Real Design Example

- We will use a TI LM5157*:
  - Internal switch
  - Internal slope
    - Ramp size in datasheet
  - Internal current sense circuitry inducing signal amplification
    - Current Sense Gain from datasheet
  - A Transconductance Amplifier
    - VREF and Gm in datasheet

Transconductance Amp!

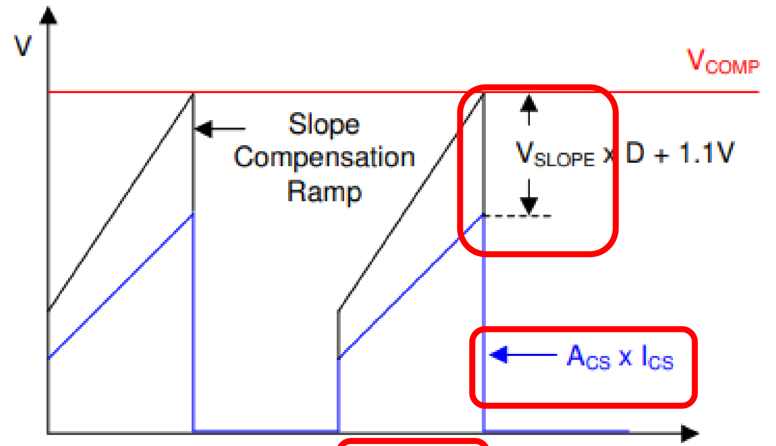


* https://www.ti.com/product/LM5157

BIRICHA DIGITAL

CCM Boost Converter Control Loop Real Design Example

- First we need to extract the data that we/WDS needs from the datasheet*
  - Slope → the chip adds a fixed slope of 500mV without synchronization
  - Op-Amp Reference Voltage (VREF)
  - Current Gain Ri (note: Acs is directly ∝ Ri)
    - In WDS Ri is in V/A
  - Op-Amp Transconductance (Gm)
    - Gm is in uA/V or uSiemens or uMho



$$V_{SLOPE} = 500mV \times \frac{f_{RT}}{f_{SYNC}}$$

- f_{SYNC} is f_{RT} if clock synchronization is not used

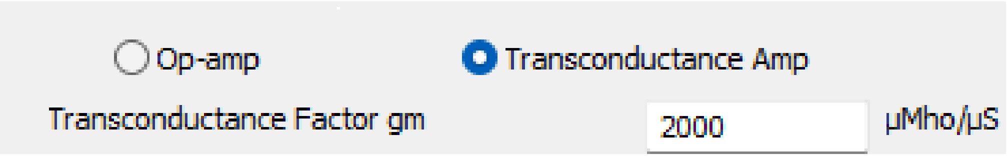
ERROR AMPLIFIER						
V_{REF}	FB reference		0.99	1	1.01	V
Gm	Transconductance			2		mA/V
	COMP sourcing current	V_{COMP} = 1.2 V	180			µA
	COMP clamp voltage	COMP rising (V_{UVLO} = 2.0 V)	2.5	2.8		V
	COMP clamp voltage	COMP falling		1	1.1	V
A_{CS}	$\Delta V_{COMP} / \Delta I_{SW}$			0.095		

Important: make sure the units for Ri and Gm are in the right dimensions
Acs is given in V/A as Rsns is internal so this is all we need and Ri = Acs
When Rsns is external, Acs it is often given in V/V and we have to multiply by Rsns or Rsns/N

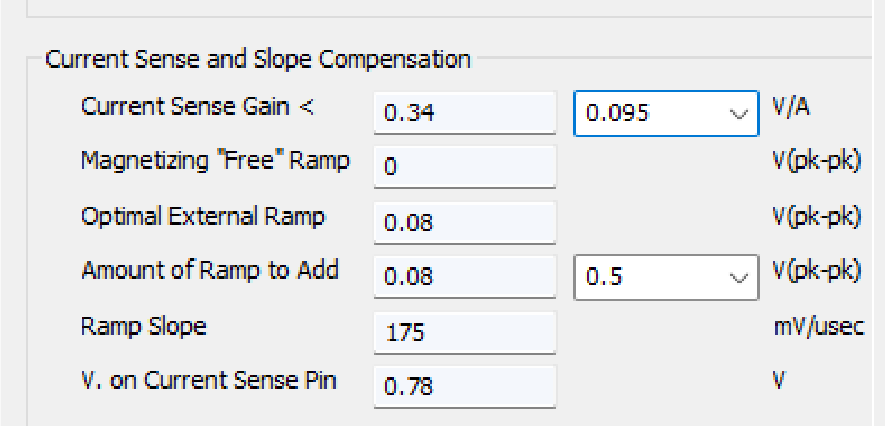
* https://www.ti.com/product/LM5157

CCM Boost Converter Control Loop Real Design Example

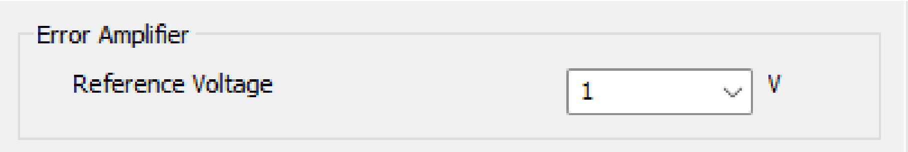
- Gm is given as 2mA/V in the datasheet, but WDS expects it in uA/V ∴
  - In Controller Design tab



- Acs is 0.095V/A from the datasheet in *V/A*, therefore Ri = Acs = 0.095 V/A
  - In Controller Design Tab

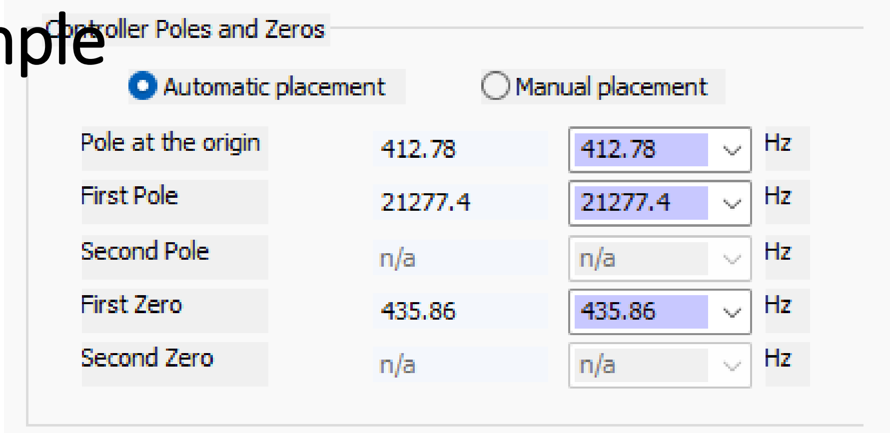


- From the datasheet we know that our OTA's VREF = 1V
  - In Analog (Non-Isolated tab)



- Please note that:
  - Acs is already in V/A as in LM5157, our current sense resistor Rsns is internal
  - If Rsns is external (e.g. TPS40210) then Acs is often given as the gain of the internal current sense amplifier in V/V or with no units
  - Then we must multiply by Acs by external Rsns or Rsns/N → more on this later
  - A/V, Siemens (S) and Mho are all the same units

CCM Boost Converter Control Loop Real Design Example

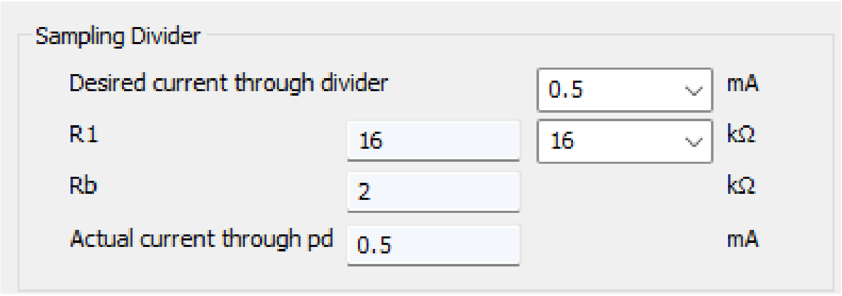


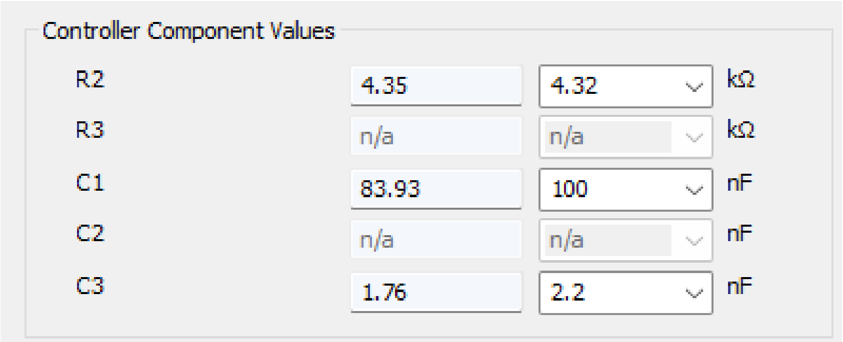
- WDS can now calculate our poles and zeros (on the right)
  - In Analog (Controller Design tab)
  - Or you can place them yourselves by hand or with other tools
- You or WDS can now calculate our Rs and Cs based on

$$f_{z1} = \frac{1}{2\pi\, R_2\, C_1}\ ;\ f_{p0} = \frac{g_m R_b}{2\pi\,(R_1 + R_b)\,(C_1 + C_3)}\ ;\ f_{p2} = \frac{(C_1 + C_3)}{2\pi\, R_2\, C_1\, C_3}$$

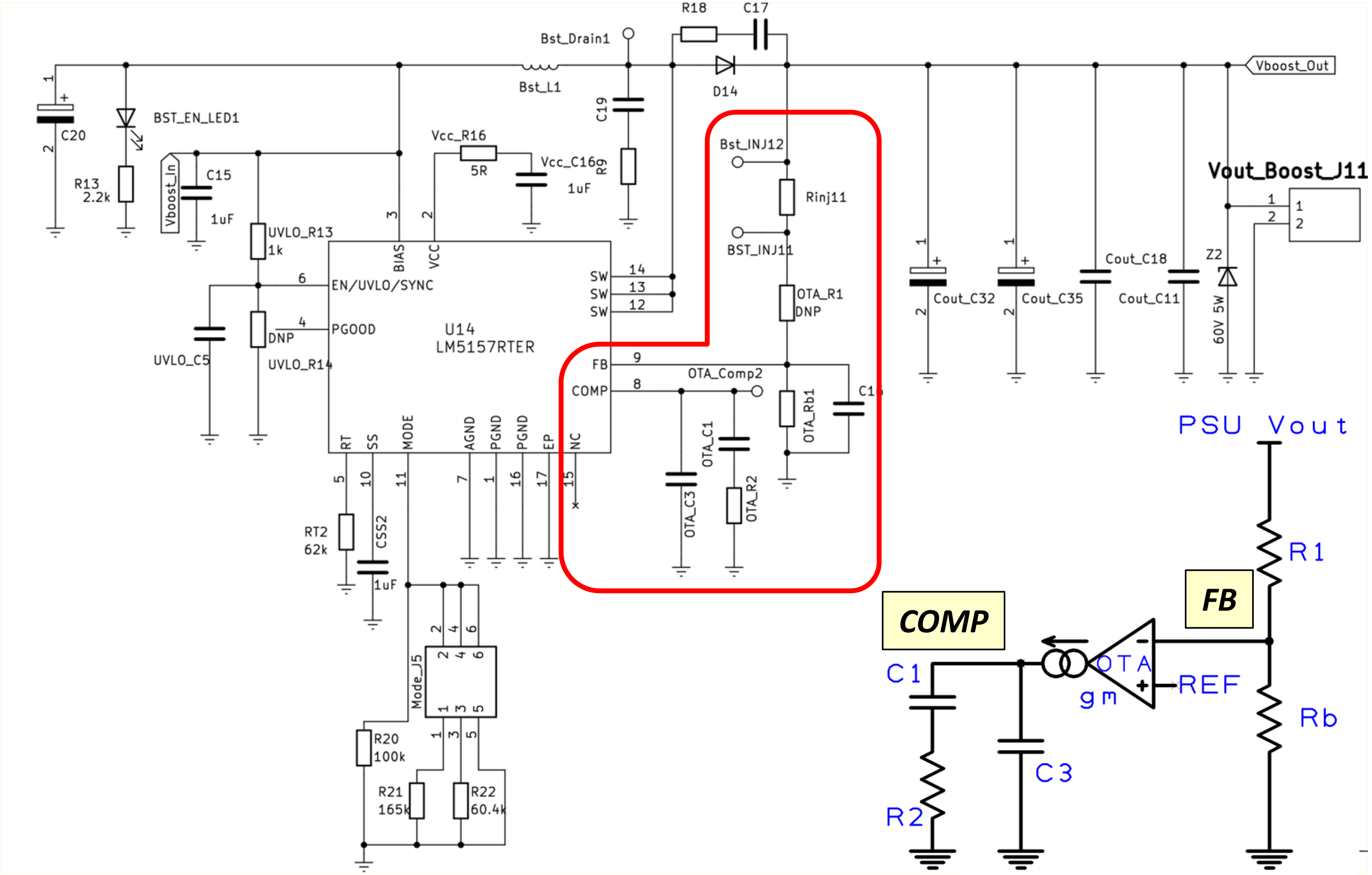
Our Design is Complete! ☺

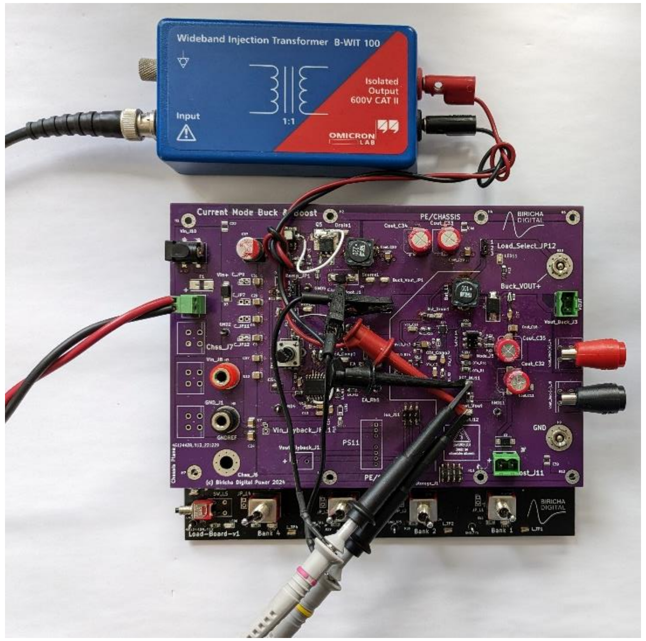
- Typically we specify how much current we wish to flow down R1 & Rb and then everything else is worked out by WDS
  - Try to keep the current >100uA for better radiated immunity in the EMC test chamber



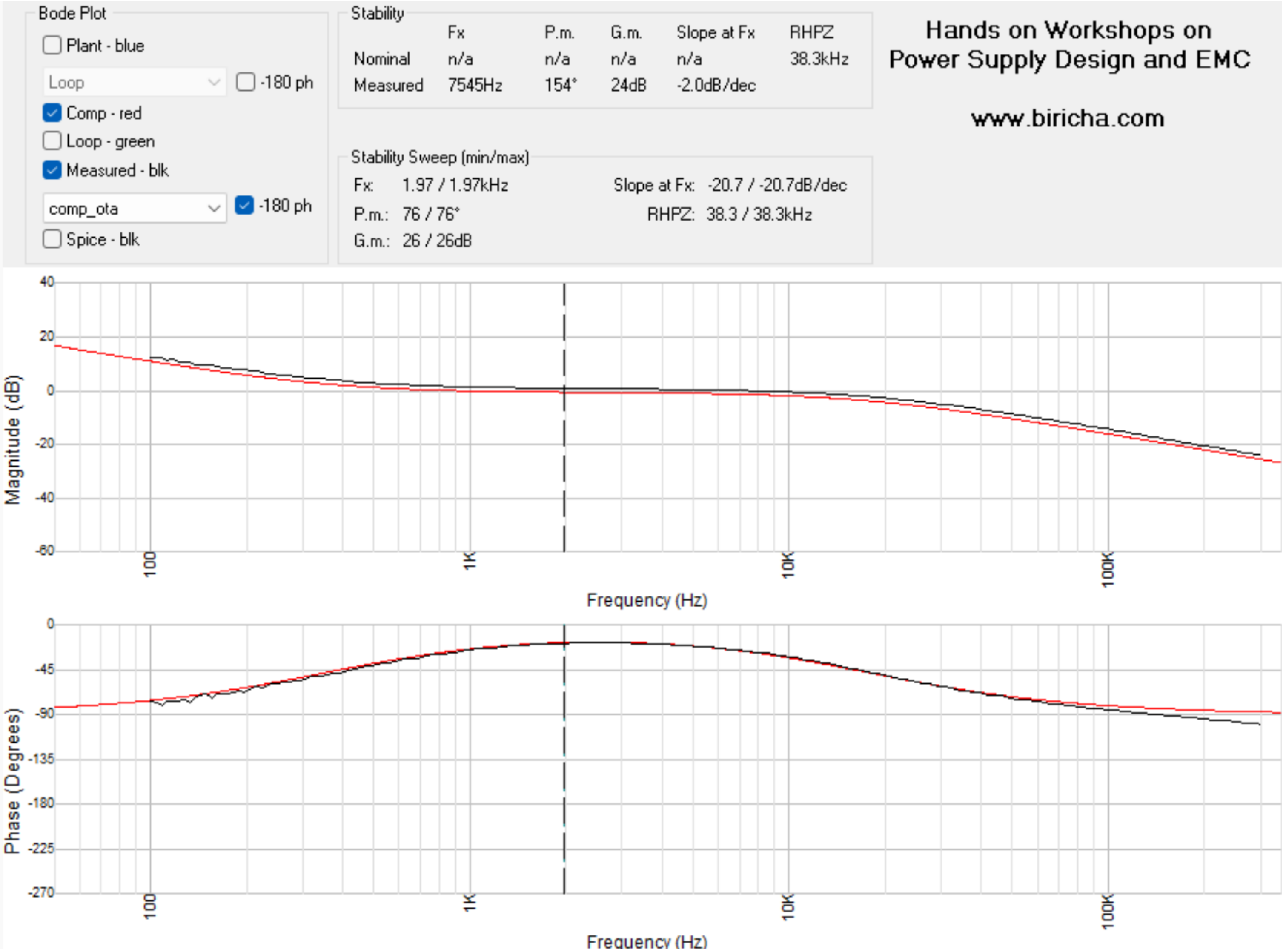


CCM Boost Converter Control Loop Real Design Example





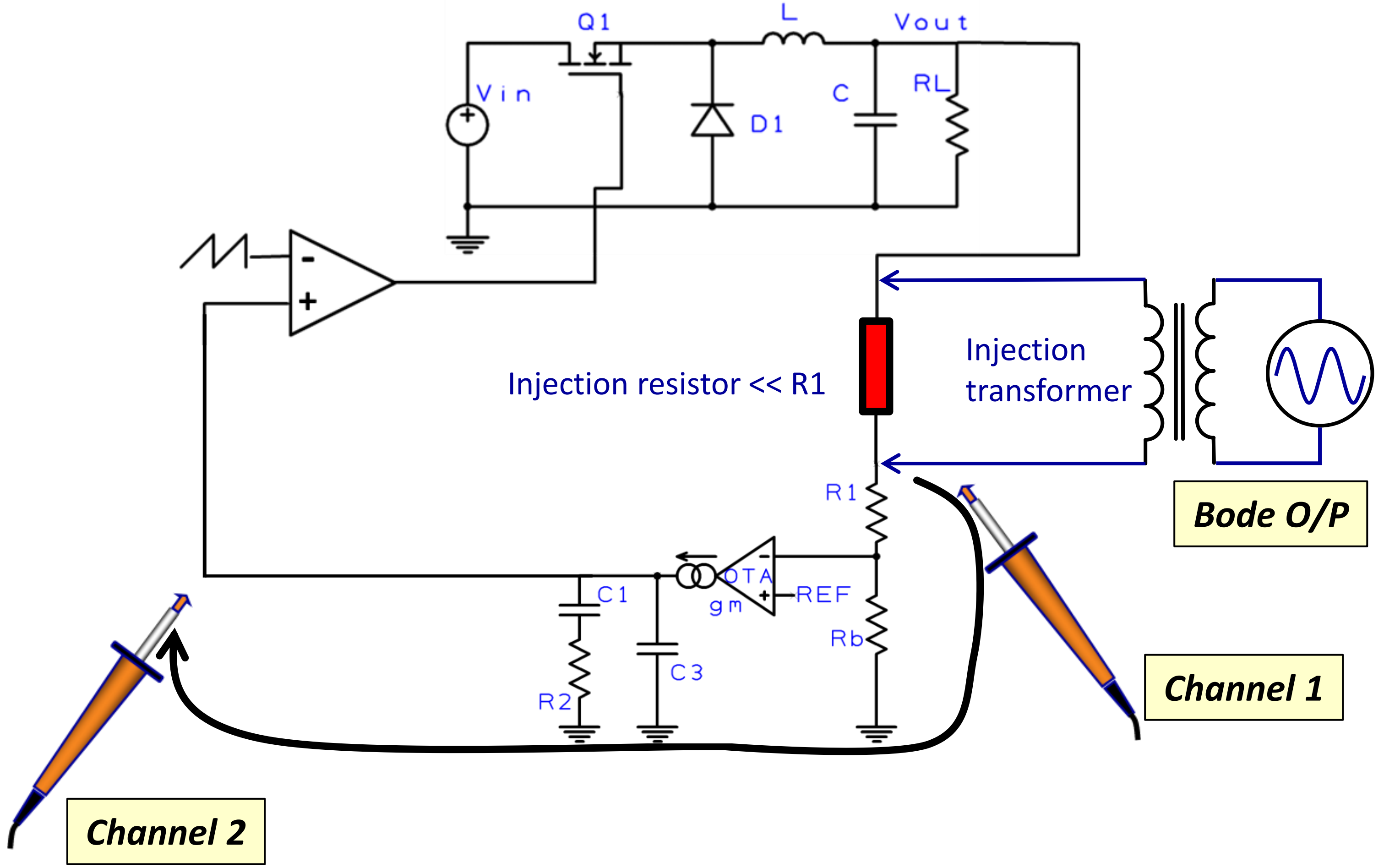
OTA Based Boost Compensator Measurement



- Red line: WDS simulation
- Black line: real measurement with Bode100

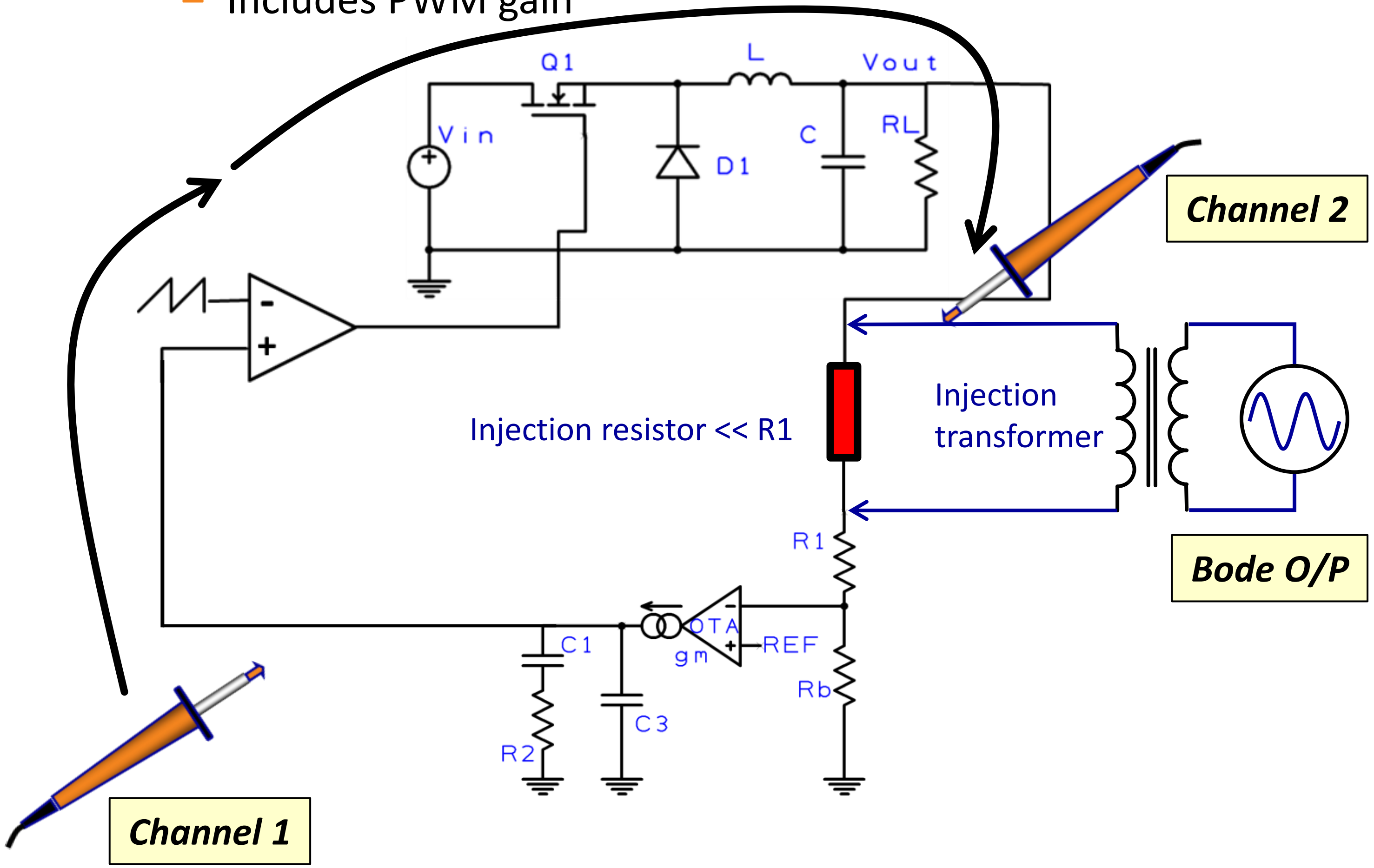
OTA Based Boost Compensator Measurement

- Measuring the compensator's response
  - Measures the frequency response of the compensator

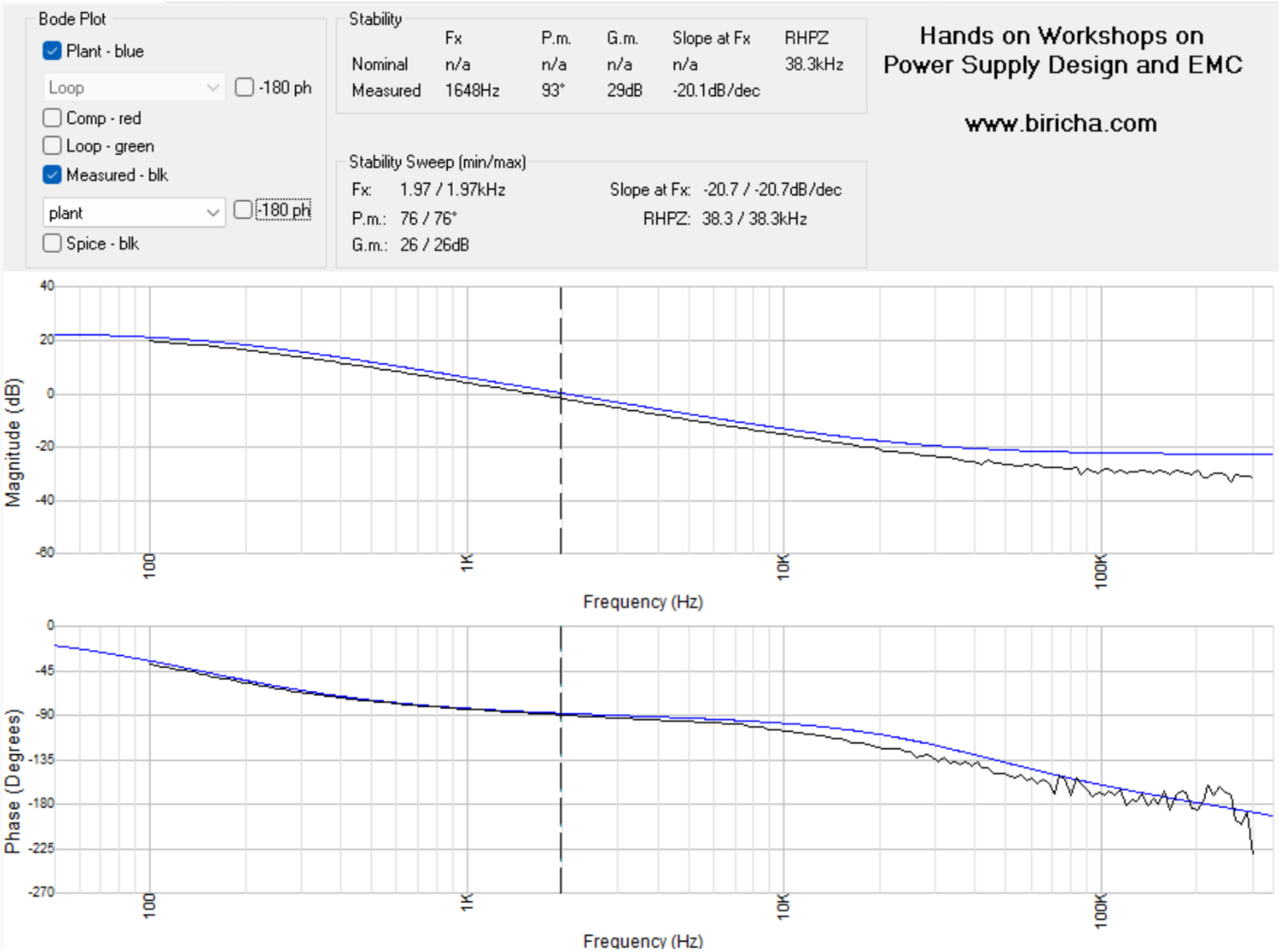


OTA Based Boost Plant Measurement

- Measure the plant's response:
  - Also known as the control to output transfer function
  - Includes PWM gain



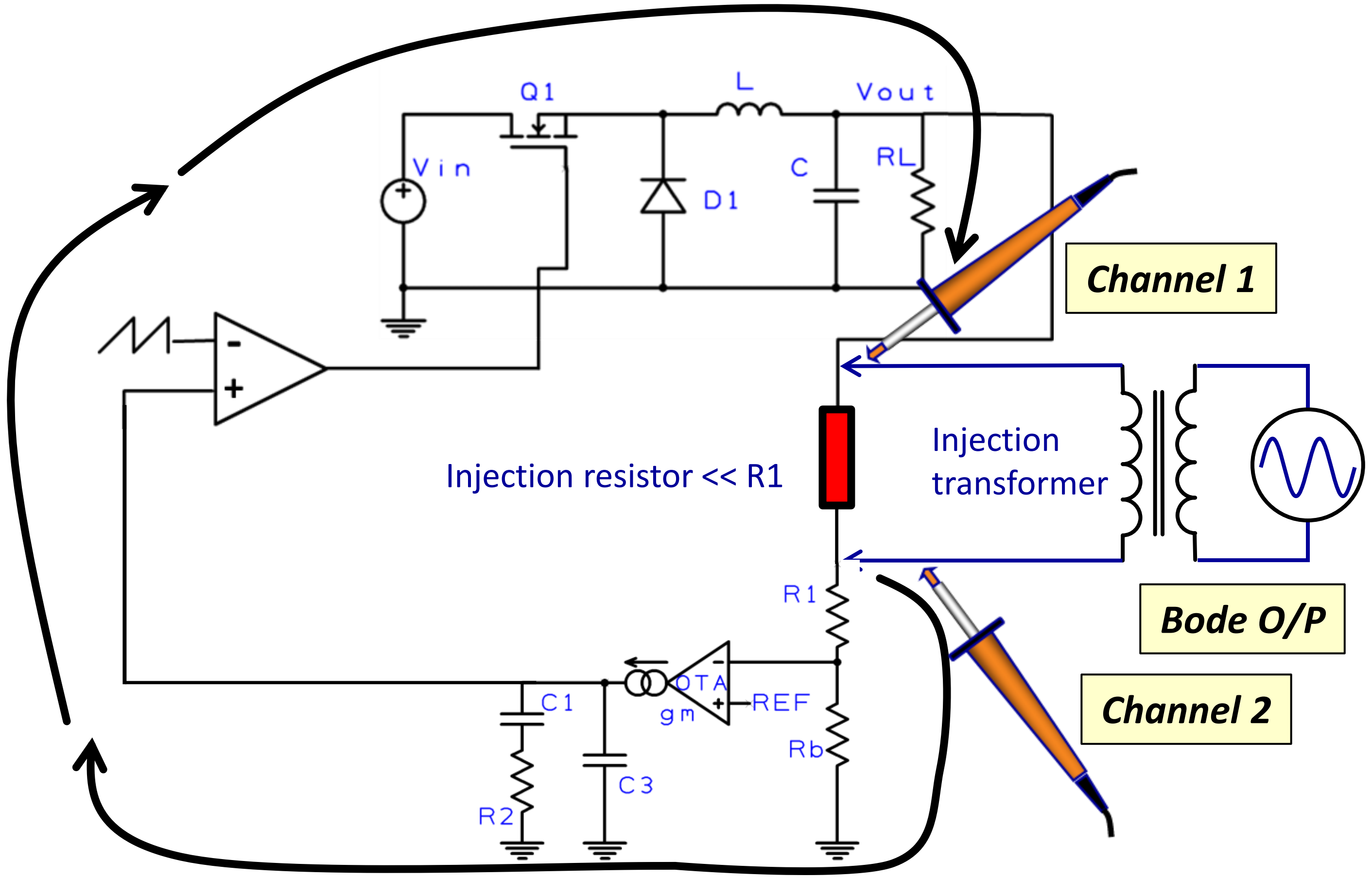
OTA Based Boost Boost’s Plant Measurement



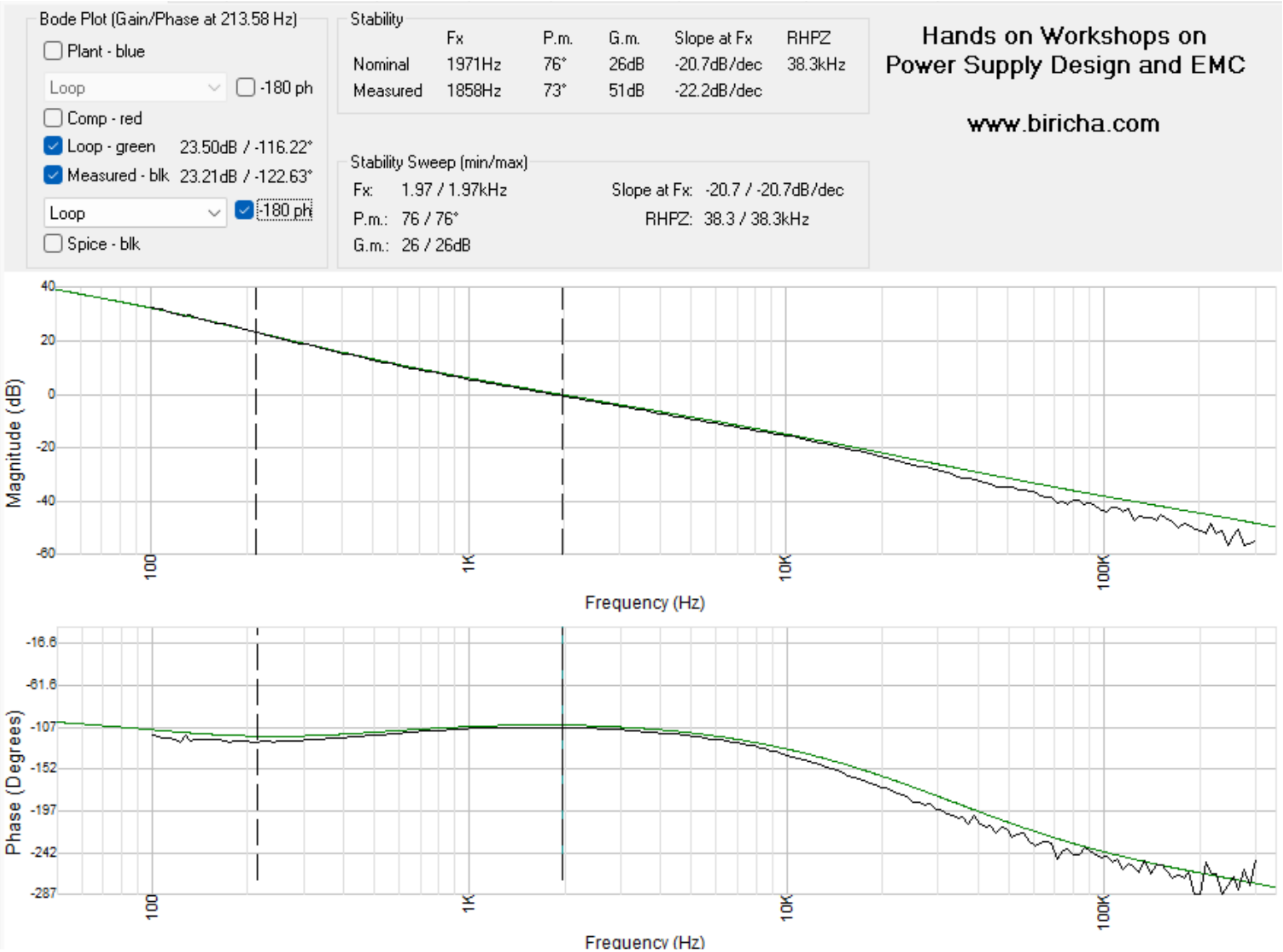
- Blue line: WDS simulation
- Black line: real measurement with Bode100

OTA Based Boost Loop Measurement

- Measuring the loop response
  - Measures the frequency response of the compensator, PWM and plant combined

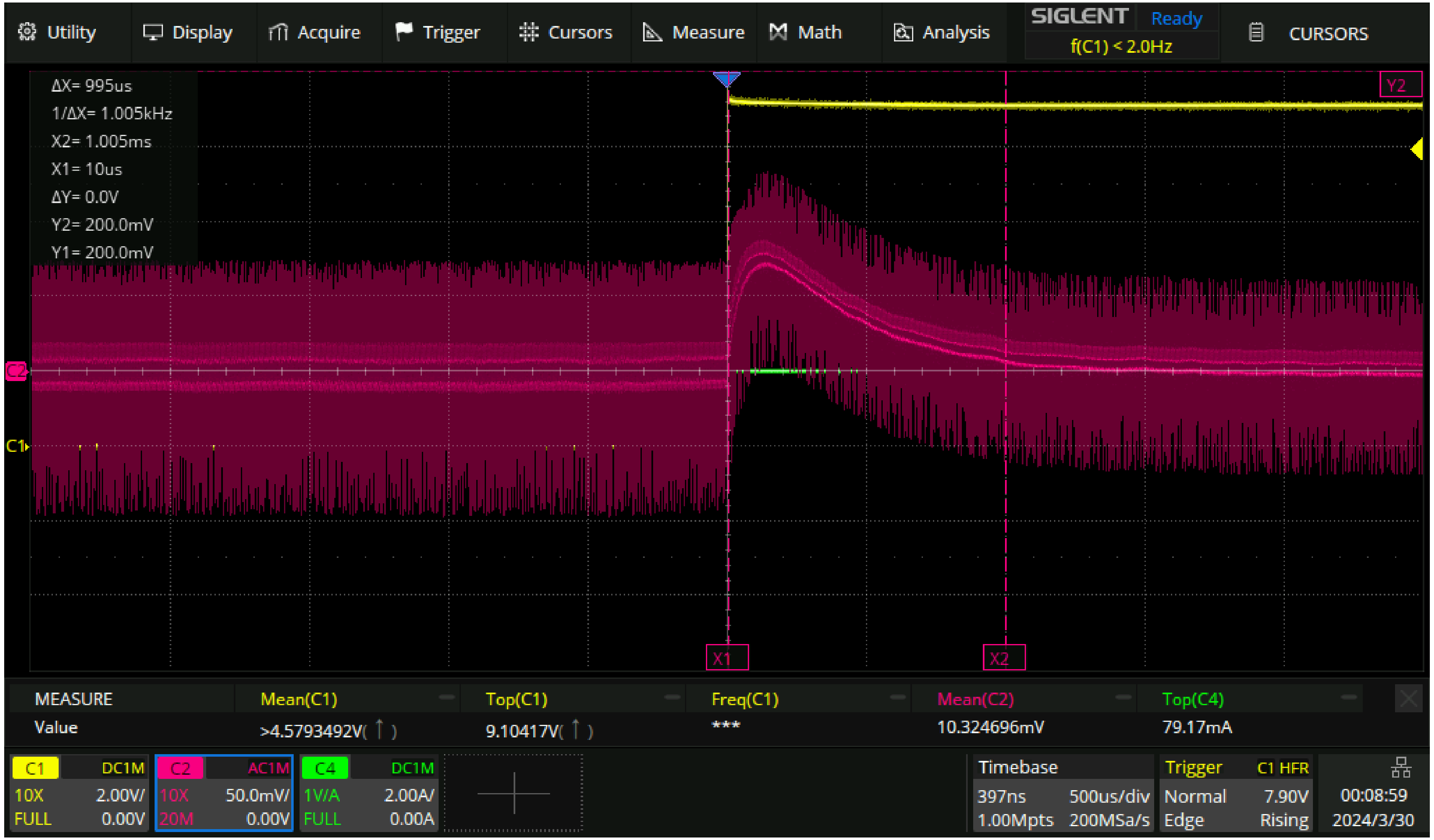


OTA Based Boost Loop Measurement



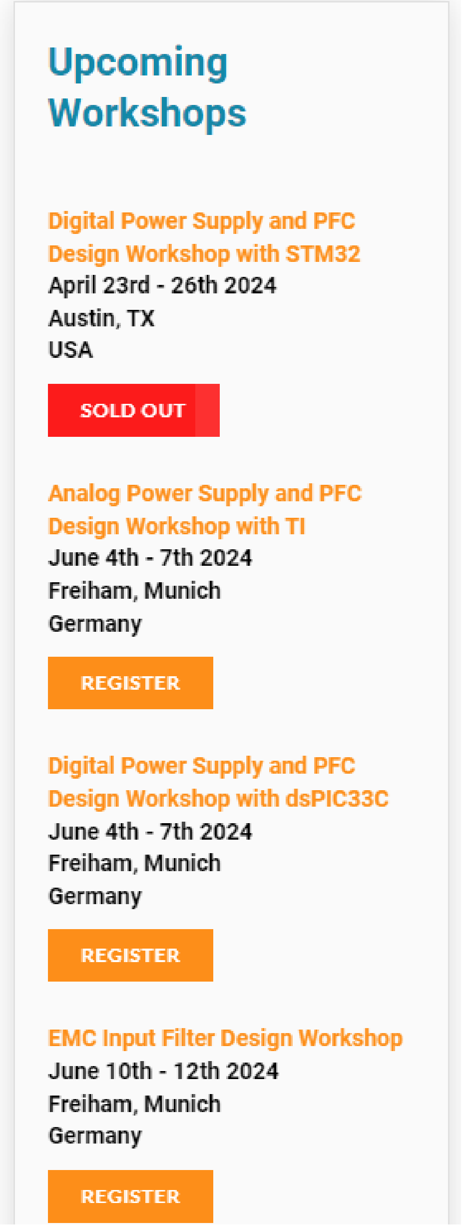
- Green line: WDS simulation
- Black line: real measurement with Bode100

OTA Based Boost Step Load (50% → 100%)



Biricha Workshops

- June 2024 Freiham Germany
  - Analog PSU Design
  - Digital PSU Design
  - EMC Filter Design
- New online workshops Q4 2024
- Register here to receive updates + free WDS:
  - www.biricha.com/omicron.html


Upcoming Workshops

Digital Power Supply and PFC Design Workshop with STM32
April 23rd - 26th 2024
Austin, TX
USA
SOLD OUT

Analog Power Supply and PFC Design Workshop with TI
June 4th - 7th 2024
Freiham, Munich
Germany
REGISTER

Digital Power Supply and PFC Design Workshop with dsPIC33C
June 4th - 7th 2024
Freiham, Munich
Germany
REGISTER

EMC Input Filter Design Workshop
June 10th - 12th 2024
Freiham, Munich
Germany
REGISTER