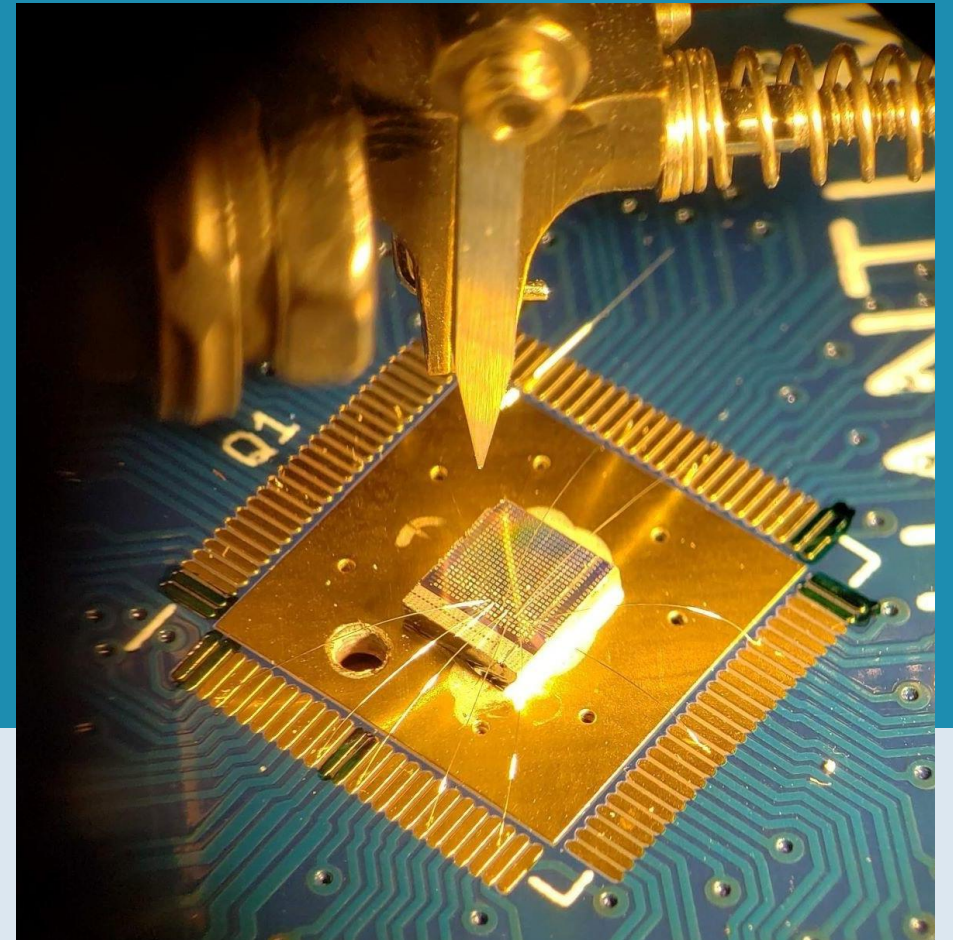


Design of Analog and Mixed-Signal Integrated Circuits B-KUL-H05E3A

Simulation Demo's STB, PSS+PNOISE, PAC

Alberto Gatti, Rico J. Maestro, Jun Feng, Shuangmu Li, Prayag J. Wakale
Prof. Filip Tavernier
Departement Elektrotechniek (ESAT)



Overview

- 🔧 These slides show **how to setup simulations** for
 1. Stability → STB simulation
 2. Noise → PSS + PNOISE simulation
 3. Accuracy → PAC simulation
- 🔧 We also provide some **demo's** so you can try for yourself
- 🔧 Note: we skip basic transient and AC simulations, these are **crucial** for stability and accuracy simulations though! See syllabus appendix, if needed

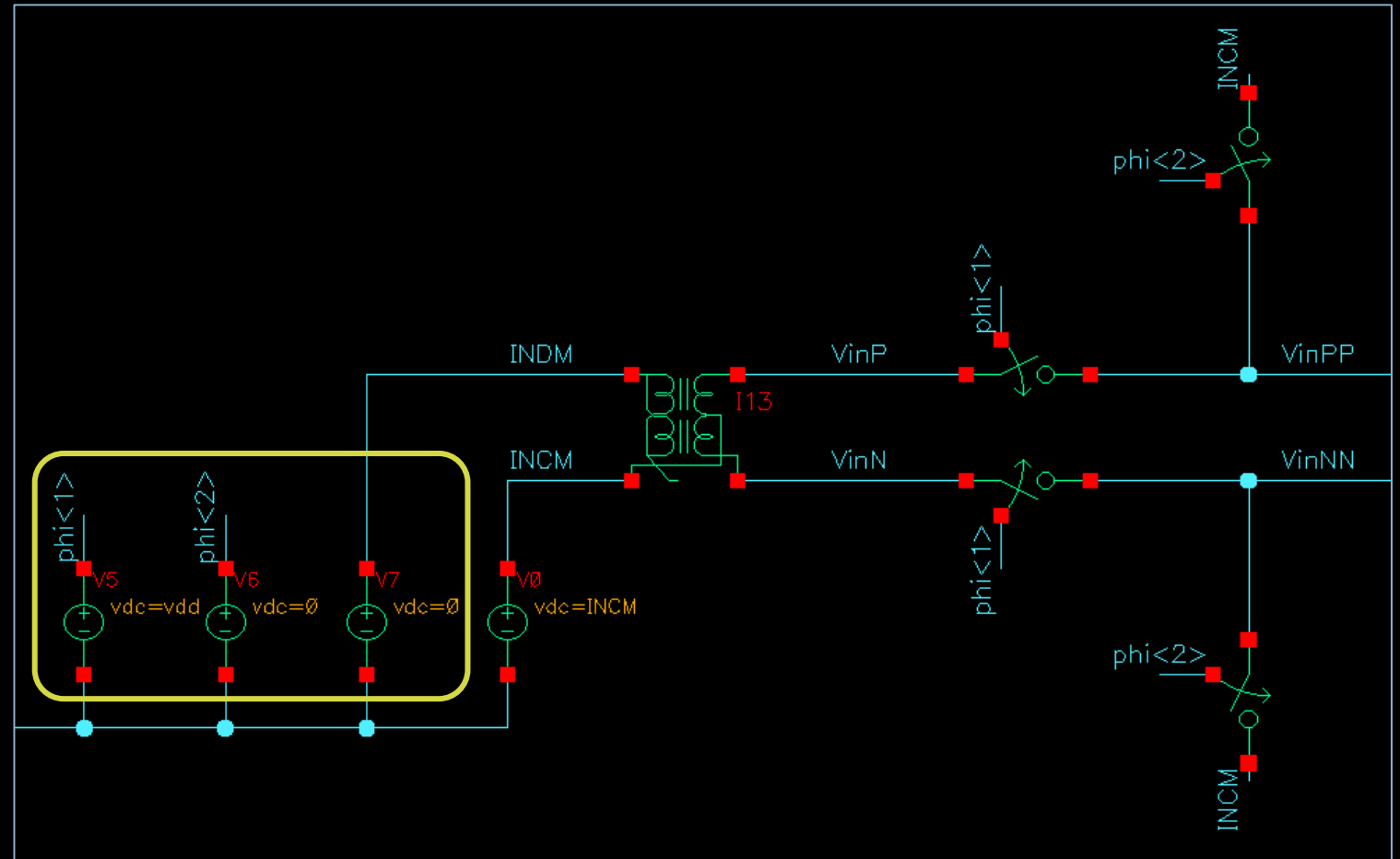
STABILITY LOOP GAIN SIMULATION SETUP AND DEMO

Stability simulation: what's needed?

- ⚙ An STB analysis assumes a single **operating point**
 - “Steady-state solution”
- ⚙ So, you must force your switch-cap system in a single state first
- ⚙ In this system, you should force it into the **sample** phase
 - Hardest to get stable (unity feedback)
 - Also, the other phase doesn't bias the input of the OTA in steady-state..

Modifying the testbench for STB

1. Copy the standard transient testbench
2. Modify to force the sampling phase
3. Make the differential input 0V too, since this is not an AC or transient simulation



diffstbprobe



You also need to insert a “diffstbprobe”

- Differential stability probe

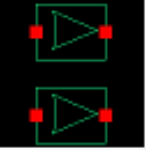


It must completely break your loop of interest



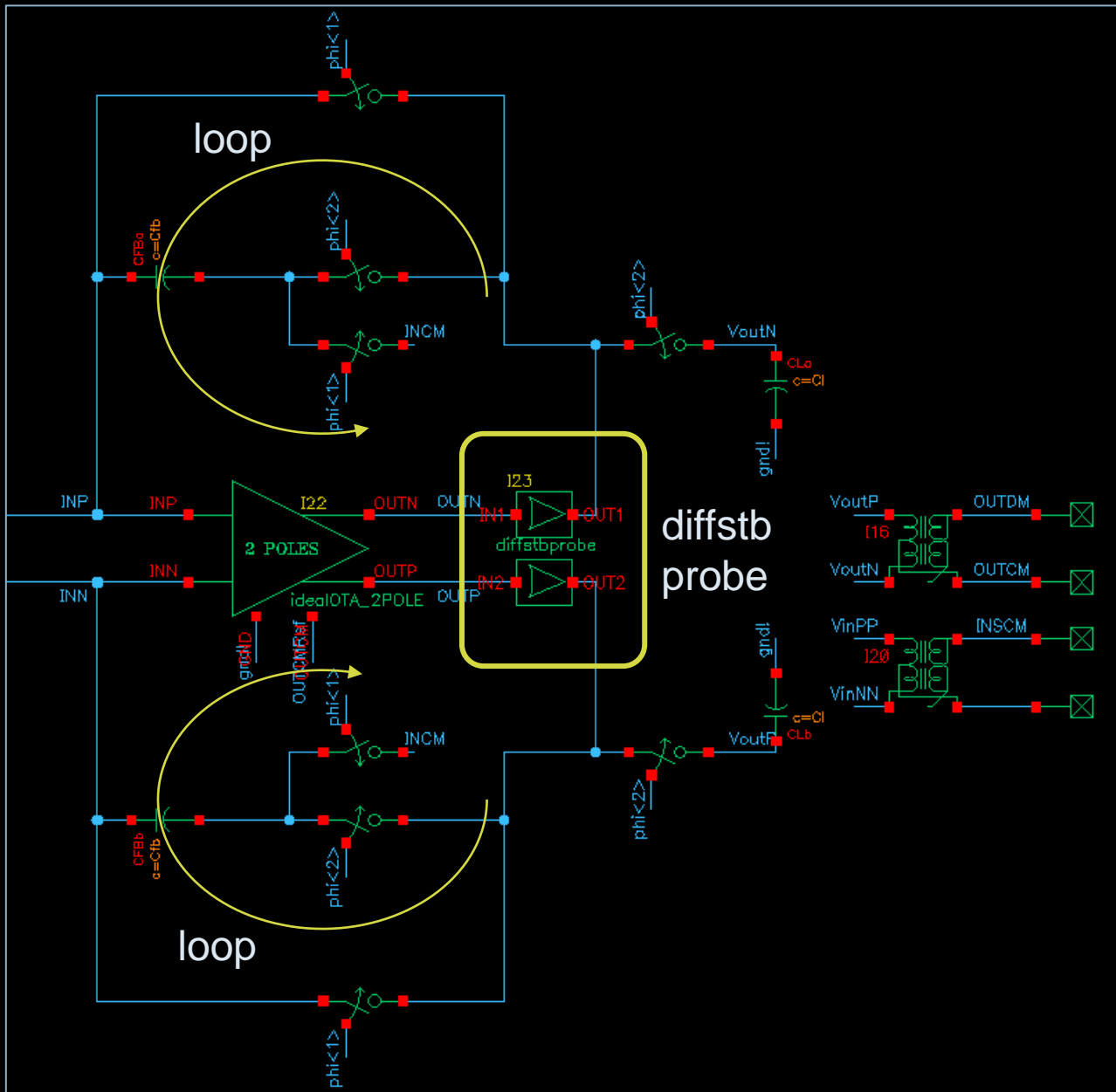
In this example, we have an ideal 2-pole OTA which doesn't have common-mode feedback. There's only one differential feedback loop.

Library	analogLib	▼	Browse
Cell	diffstbprobe	▼	
View	symbol	▼	
Names			



Modifying the testbench for STB

- 🔍 Easiest location here:
output of the OTA
- 🔍 In the future you will
have multiple feedback
loops, resulting in
multiple options



Simulation setup

1. Launch new) ADE Explorer view
2. Add analysis: choose “stb” and pick a frequency sweep (to get bode plot of loop gain)
3. “Probe Instance” = the diffstbprobe you inserted
4. Local Ground Name = /gnd!
5. Finish setting up design variables → run!

The screenshot shows a simulation setup dialog box with the following sections and controls:

- Sweep Range:** Includes radio buttons for **Start-Stop** (selected) and **Center-Span**. The **Start** field is set to **1** and the **Stop** field is set to **100G**.
- Sweep Type:** Includes a dropdown menu set to **Logarithmic** and radio buttons for **Points Per Decade** (selected, set to **20**) and **Number of Steps**.
- Add Specific Points:** A checkbox that is currently unchecked.
- Add Points By File:** A checkbox that is currently unchecked.
- Probe Instance/Terminal:** A text field containing **/I23** and a **Select** button.
- Local Ground Name:** A text field containing **/gnd!** and a **Select** button.
- Mode Type:** Includes radio buttons for **common** and **differential** (selected).
- Enabled:** A checkbox with a checkmark.
- Buttons:** Includes an **Options...** button and a row of buttons at the bottom: **OK** (highlighted in red), **Cancel**, **Defaults**, **Apply**, and **Help**.

Plotting STB's result

1. In ADE: Results > Direct Plot > Main Form ..
2. You can plot either Loop Gain bode plot, or just print the Stability Summary
3. Check “Add To Outputs” to push the expression to ADE, so you can re-use 😊
 - You could do the same for Phase Margin or Gain Margin options of the Direct Plot Form

The screenshot shows the 'Direct Plot Form' dialog box with the following settings and highlighted elements:













- Plotting Mode:** Append (dropdown menu)
- Analysis:** stb (radio button)
- Function:** Loop Gain (radio button, highlighted with a red arrow)
- Modifier:** Magnitude and Phase (radio button, highlighted with a red arrow)
- Magnitude Modifier:** dB20 (radio button, highlighted with a red arrow)
- Add To Outputs:** Checked (checkbox, highlighted with a red arrow)
- Plot:** Button (highlighted with a red arrow)

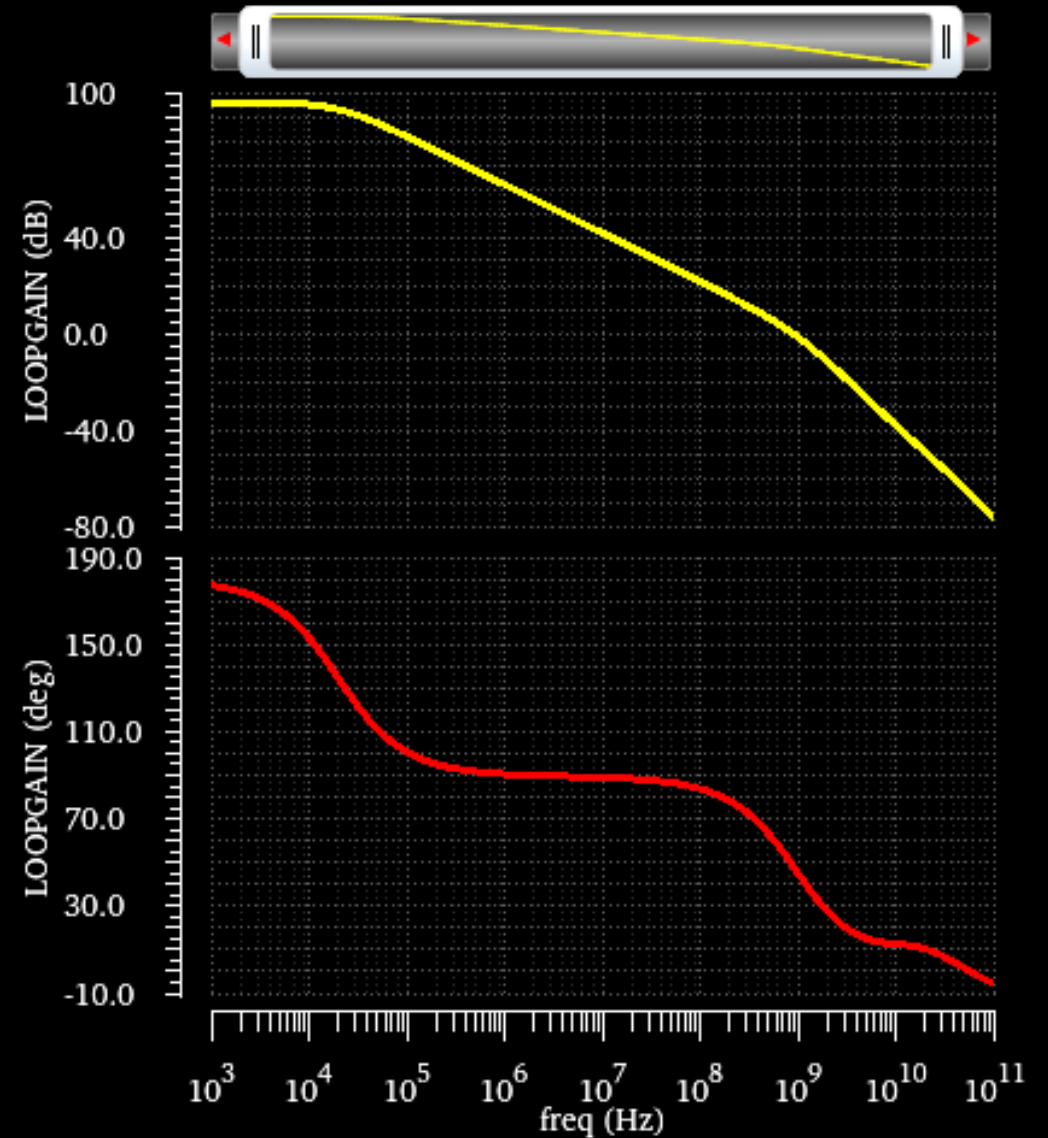
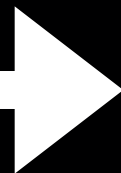
At the bottom, there is a text prompt: '> Press plot button on this form...' and two buttons: 'Close' and 'Help'.

Demo 1: “demo_SCAMPLIFIER_STB_TB”

- We've modified the testbench schematic as described
- We've replaced the idealOTA by a 2-pole idealOTA
 - It has an extra gm-C stage, where $gm = 1$ and $C = CP2$ (variable)
 - So, an extra pole will be placed at roughly $1/(2\pi \cdot CP2)$
- The maestro view is already setup with design variables and a second pole that is close to the dominant unity-gain frequency
 - What is the phase margin?
 - Could you place the same OTA in a transient to see the predicted instability?

Demo 1 example

	Cfb	0.2p
	Cin	2p
	Cl	2p
	gm_OTA	25m
	ro_OTA	2.5M
	C_OTA	1p
	F_POLE2	1G
	CP2	$1/(6.28 \cdot F_POLE2)$
	INCM	0.5
	OUTCMRef	0.5
	ron	5
	vdd	1.1



PSS + PNOISE SIMULATION SETUP AND DEMO

PSS simulation and setup (1)

- ⚙️ Periodic steady state analysis
- ⚙️ It computes the **steady states vs. time** in your circuit within one period (normal AC sim = just one steady state)
- ⚙️ To keep simulation time low, we **disable the input sine wave** in the schematic → the “period” will be limited by the clock
 - Trade-off for accuracy, generally OK for this kind of design in DAMSIC
 - You can set the input sine wave frequency to 0 Hz, Cadence treats it as DC source then

PSS setup (2)

1. Add PSS analysis
2. Shooting engine
3. Beat Frequency:
 - It's the period in which PSS perform analysis
 - Set equal to clock frequency variable
 - Or: Auto Calculate (will be same result)

The screenshot shows the 'Analysis' section of a circuit simulator's setup window. The 'Analysis' section contains a grid of radio buttons for selecting the analysis type. The 'PSS' (Periodic Steady State) option is selected. Below this, the 'Engine' section shows 'Shooting' selected over 'Harmonic Balance'. The 'Fundamental Tones' section contains a table with one row: #1, Name: 1/(1/F_CLK), Expr: 1M, Value: Large, Signal: V1, SrcId: V1. Below the table are buttons for 'Clear/Add', 'Delete', and 'Update From Hierarchy'. At the bottom, the 'Beat Frequency' option is selected, and the 'Auto Calculate' checkbox is checked.

Analysis

☐ tran ☐ dc ☐ ac ☐ noise
☐ xf ☐ sens ☐ dcmatch ☐ acmatch
☐ stb ☐ pz ☐ lf ☐ sp
☐ envlp ☒ pss ☐ pac ☐ pstb
☐ pnoise ☐ pxf ☐ psp ☐ qpss
☐ qpac ☐ qpnoise ☐ qpxf ☐ qpsp
☐ hb ☐ hbac ☐ hbstb ☐ hbnoise
☐ hbsp ☐ hbxf

Periodic Steady State Analysis

Engine ☒ Shooting ☐ Harmonic Balance

Fundamental Tones

#	Name	Expr	Value	Signal	SrcId
1	1/(1/F_CLK	1M	Large	V1	V1

F_CLK-0) 1M Large V1

Clear/Add Delete Update From Hierarchy

☒ Beat Frequency ☐ Beat Period VAR("F_CLK") Auto Calculate ☒

PSS setup (3)

1. Make number of harmonics into 100's (trade-off with accuracy)
 - It sets the f_{MAX} considered by PSS
 - Above a certain amount, it will make no difference anymore (no extra noise)
 - Usually, 200 is good enough
2. Transient: decide automatically
 - Helps PSS to converge

The screenshot shows the PSS setup dialog box with the following settings:

- Output harmonics:** Number of harmonics is set to 200.
- Accuracy Defaults (errpreset):** conservative, moderate, and liberal options are all unchecked.
- Transient-Aided Options:**
 - Run transient? is set to **Decide automatically** (radio button selected).
 - Detect Steady State is checked.
 - Stop Time (tstab) is set to an empty field.
 - Save Initial Transient Results (saveinit) is set to no.
- Dynamic Parameter:** unchecked.
- Oscillator:** unchecked.
- Sweep:** unchecked.
- New Initial Value For Each Point (restart):** no and yes options are both unchecked.
- Loadpull:** unchecked.
- Enabled:** checked.
- Options...** button is present at the bottom right.

PNOISE simulation



Periodic noise analysis



It uses the steady-states computed by PSS to find either:

- A time-average noise spectrum at the output, OR
- Noise spectrum at the output at specific times



We want the latter (i.e. after amplification phase)



We integrate the noise spectrum to obtain the RMS noise voltage

PNOISE setup (1)

1. PSS Beat Frequency is obtained from PSS sim. setup of the same ADE test
2. Set frequency sweep range (for the noise PSD) **up to $f_s/2$!**
 - In this example, $500K = 1M/2$
 - Tip: you could also write in the Stop field $VAR("F_CLK")/2$, if you have such a variable F_CLK

The screenshot shows the 'Analysis' tab of a circuit simulator's setup window. Under the 'Analysis' section, the 'pnoise' option is selected with a red dot. Below this, the 'Periodic Noise Analysis' section is visible. It includes a 'PSS Beat Frequency (Hz)' field set to '1M'. A 'Multiple pnoise' checkbox is present and unchecked. The 'Sweeptype' is set to 'default' with a dropdown arrow, and a note states 'Sweep is currently absolute'. The 'Output Frequency Sweep Range (Hz)' is configured with a 'Start-Stop' dropdown, 'Start' field set to '1', and 'Stop' field set to '500K'. The 'Sweep Type' is set to 'Logarithmic' with a dropdown arrow. Below this, there are two radio buttons: 'Points Per Decade' (selected with a red dot) and 'Number of Steps', with a field set to '20'. At the bottom, there are two unchecked checkboxes: 'Add Specific Points' and 'Add Points By File'.

PNOISE setup (2)

1. Method = fullspectrum
 - To make sure you fold all the noise in
 - Keep maximum sidebands empty
2. Noise Type = sampled (jitter)
 - We need PSD at specific time!

Sidebands

Method ☐ default ☒ fullspectrum

Maximum sideband

Calculates noise contributions up to the frequency determined by PSS time point resolution

Noise Type Sample Ratio

#	Event	Trig	TrigVal	Targ	TargVal	TD
1	phase			V2P-V2N		

Add Change Delete Enabled ☒

Timing Event ☐ Edge Crossing ☐ Edge Delay ☒ Sampled Phase

Sampled Phase: strobed noise analysis

Measurement

Positive Output Node

Negative Output Node

Samples Per Period

Add Specific Points ☒

PNOISE setup (3)

1. You will need to set sampling events:
 - Timing Event = Sampled Phase
 - Then select and positive and negative output node
 - Don't use a balun to measure differential noise (buggy)! Select the two diff. nodes of interest
 - Select no. samples per period and/or specific point in time (of the PSS result) where you want the PSD from (re-run later if the timing is off)
2. Then press “Add”! The event must be added to the list now
3. If you ever change the settings (e.g. samples per period), you **must** press “Change”!

The screenshot shows the PNOISE setup interface. The 'Sidebands' section at the top has a 'Method' dropdown set to 'default' and a 'Maximum sideband' input field. Below it, a note states: 'Calculates noise contributions up to the frequency determined by PSS time point resolution'. The 'Noise Type' section below has a dropdown set to 'sampled(jitter)' and a 'Sample Ratio' input field. A table lists sampling events:

#	Event	Trig	TrigVal	Targ	TargVal	TD
1	phase			V2P - V2N		

Below the table are 'Add', 'Change', and 'Delete' buttons. To the right is an 'Enabled' checkbox which is checked. The 'Timing Event' section has three radio buttons: 'Edge Crossing', 'Edge Delay', and 'Sampled Phase' (which is selected). Below this is the text 'Sampled Phase: strobed noise analysis'. The 'Measurement' section has a 'voltage' dropdown, 'Positive Output Node' set to '/V2P' (with a 'Select' button), and 'Negative Output Node' set to '/V2N' (with a 'Select' button). 'Samples Per Period' is set to 6. At the bottom, 'Add Specific Points' is checked with a checkbox, followed by an empty input field.

Plotting PSS results

- As usual, in ADE use Results > Direct Plot > Main Form ..
- PSS form: plotting a voltage vs. time (in a single period) is usually what you do
- Check “Add To Outputs” to add the expression to ADE for future simulations 😊

The screenshot shows the 'Plotting Mode' dialog box in ADE. At the top, 'Append' is selected in the dropdown. The 'Analysis' section has 'pss' selected with a red arrow. The 'Function' section has 'Voltage' selected with a red arrow. The 'Sweep' section has 'time' selected with a red arrow. The 'Scalar Expression' section has 'Add To Outputs' checked with a red arrow. A callout box points to the 'Net' dropdown with the text 'Can also select a differential net if you want!'. At the bottom, there is a button '> Select Net on schematic...'. The 'Value At (s)' field is empty.

Plotting PNOISE results

Noise Spectrum

Analysis

☐ pss ☒ pnoise sampled

Measurement Selection: pm0-(VoutP VoutN)-Phase

Function

☒ Output Noise ☐ Integ Output Noise

Select: Total Noise

Sweep

☐ time ☒ spectrum

Modifier

☒ V/sqrt(Hz) ☐ V**2/Hz ☐ dBV/Hz

Add To Outputs ☒ Plot

> Press plot button on this form...

You set this up in PNOISE earlier

Total noise, or “noise per instance”, useful for design

Sweep spectrum, you’ll get the PSDs at different time moments

Integrated Noise

Analysis

☐ pss ☒ pnoise sampled

Measurement Selection: pm0-(VoutP VoutN)-Phase

Function

☐ Output Noise ☒ Integ Output Noise

Select: Total Noise

Modifier

☒ Magnitude(V) ☐ dB20

Start Frequency (Hz):

Stop Frequency (Hz):

Add To Outputs ☒

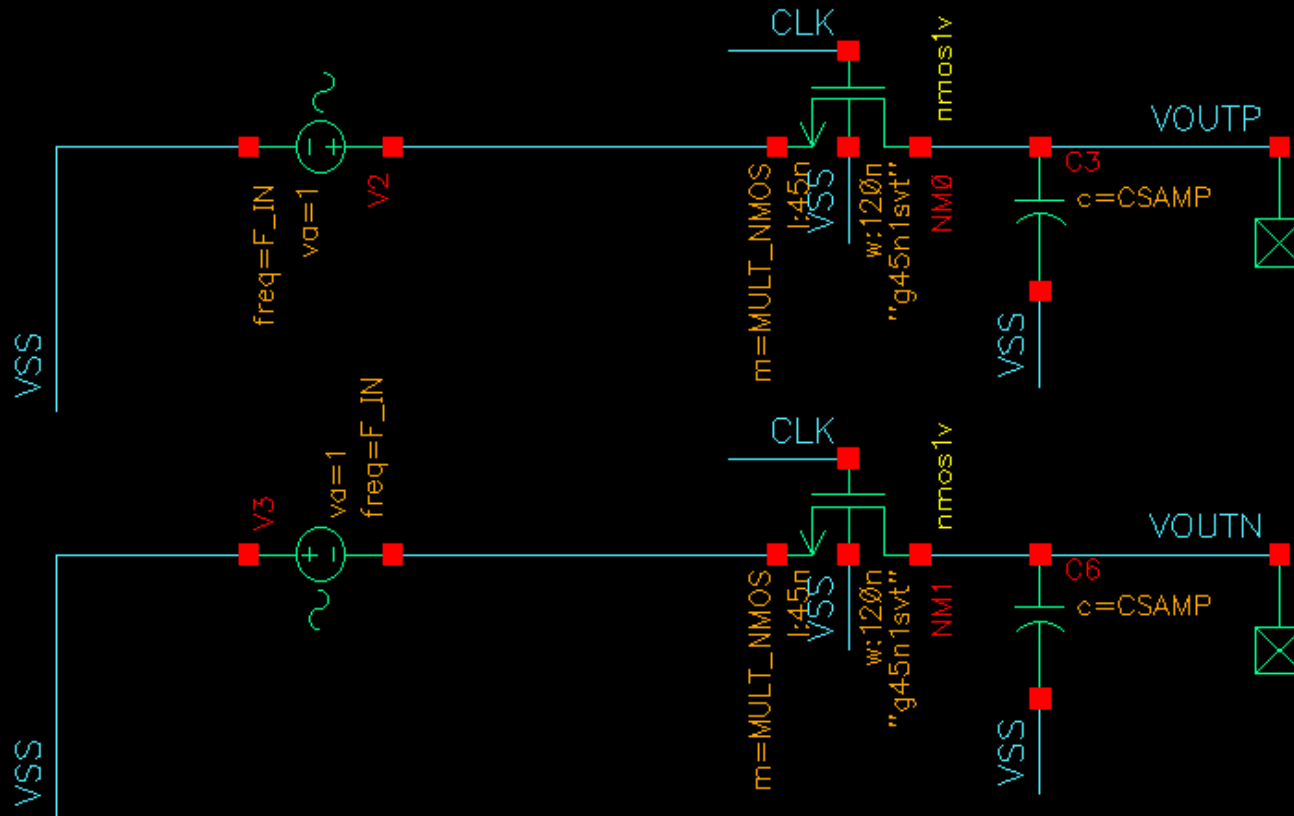
> Provide a numeric value for "Start Frequency (Hz)" on this form.

To get RMS value

Up to $f_s/2$!

You’ll get RMS noise vs. time

Demo 2: “demo_TH_NOISE_TB”



- 🔧 Differential T/H with transistors →
- 🔧 Total sampled differential noise power should be $2kT/C$

Demo 2: Setup

- 🔧 We've set the sampling cap to 1 pF on each side
 - NMOS multiplier = 1, $f_{\text{CLK}} = 10 \text{ MHz} \Leftrightarrow$ shouldn't matter much
 - Expected single-ended RMS noise voltage is $\text{sqrt}(kT/C) = 64 \text{ uV}_{\text{RMS}}$
 - Expected differential RMS noise voltage is $\text{sqrt}(2kT/C) = 90.5 \text{ uV}_{\text{RMS}}$
- 🔧 PSS, PNOISE are setup properly
- 🔧 Setup and results are on next slides

Demo 2: Setup

Periodic Steady State Analysis

Engine ☒ Shooting ☐ Harmonic Balance

Fundamental Tones

#	Name	Expr	Value	Signal	SrcId
---	------	------	-------	--------	-------

Clear/Add Delete Update From Hierarchy

☒ Beat Frequency ☐ Beat Period 10M Auto Calculate ☐

Output harmonics
Number of harmonics 200

Accuracy Defaults (errpreset)
☐ conservative ☐ moderate ☐ liberal

Transient-Aided Options
Run transient? ☐ Yes ☐ No ☒ Decide automatically

Periodic Noise Analysis

PSS Beat Frequency (Hz) 10M

Multiple noise ☐

Sweeptype default Sweep is currently absolute

Output Frequency Sweep Range (Hz)
Start-Stop Start 1 Stop 5M

Sweep Type Automatic

Add Specific Points ☐

Add Points By File ☐

Sidebands

Method ☐ default ☒ fullspectrum

Maximum sideband

Calculates noise contributions up to the frequency determined by PSS time point resolution

Noise Type sampled(jitter) Sample Ratio

#	Event	Trig	TrigVal	Targ	TargVal	TD
1	phase			VOUTP-VOU		
2	phase			VOUTP-gnd		

Add Change Delete Enabled ☒

Timing Event ☐ Edge Crossing ☐ Edge Delay ☒ Sampled Phase

Sampled Phase: strobed noise analysis

Measurement
voltage Positive Output Node /VOUTP Select
Negative Output Node /gnd! Select

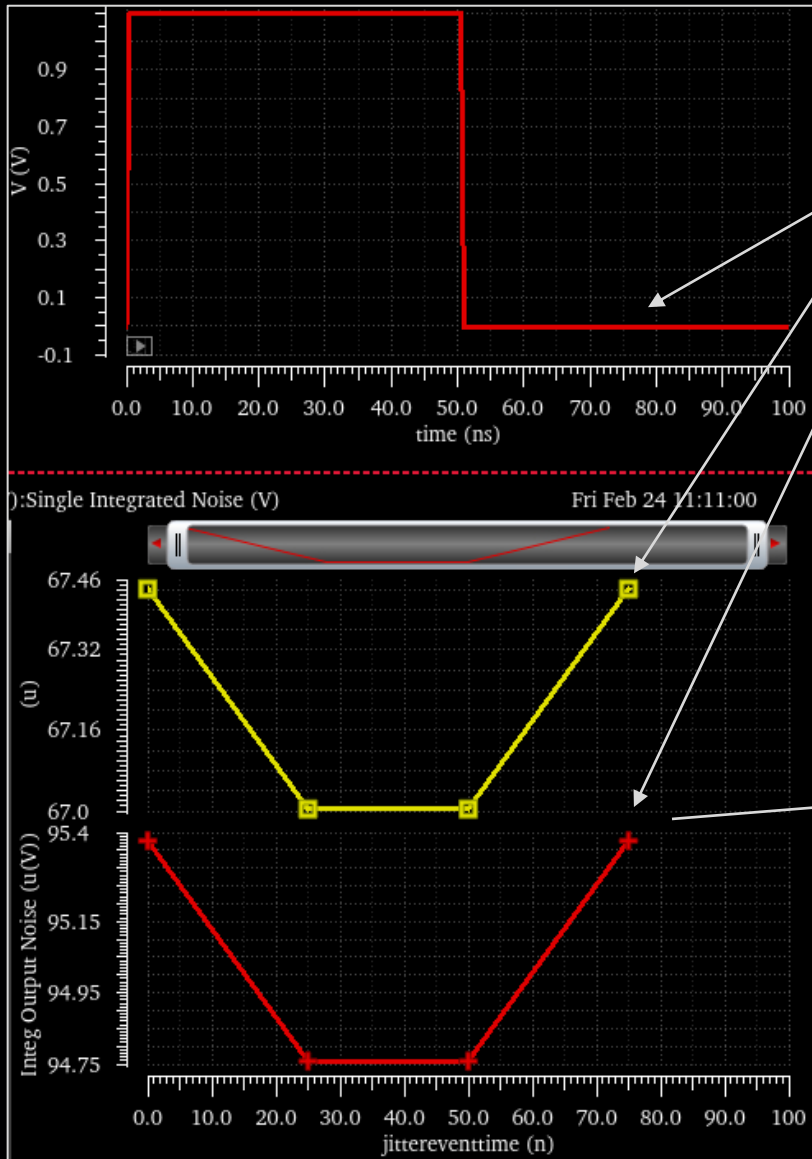
Samples Per Period 4

Add Specific Points ☐

Differential and single-ended measurements

Demo 2: Results

Close enough! kT/C ignores $1/f$ 😊
And normally you'd overdesign

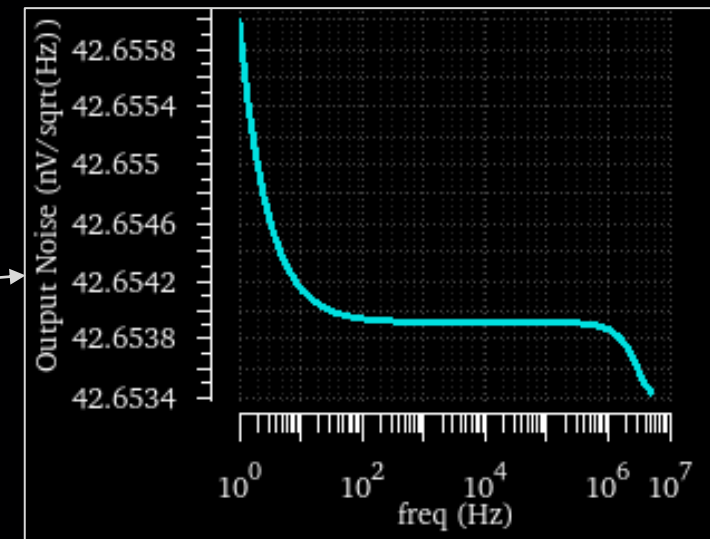


Check when
clock = low

Single-ended:
 $\sim 67.5 \mu V_{RMS}$

Differential:
 $\sim 95.4 \mu V_{RMS}$

Noise Spectrum



Demo 3: “demo_SCAmplifier_PNOISE_TB”

- 🔧 To showcase how you would simulate noise in your DAMSIC project. Setup is very similar to demo 2!
- 🔧 Important notes (see ADE):
 - Made $f_{in} = 0$, $V_{in} = 0$ (via design variables)
 - Output baluns are disable in the schematic (else PNOISE becomes buggy)
 - Switching frequency is 70 MHz, so PNOISE and integral up to 35 MHz

Demo 3: setup

Design Variables		
	alpha	0.01
	Cfb	0.2p
	Cin	2p
	Cl	3p
	fin	0
	fsw	70M
	gm_OTA	20m
	C_OTA	0.5p
	ro_OTA	1M
	INCM	0.5
	OUTCM...	0.5
	ron	5
	vdd	1.1
	Vin	0

Periodic Noise Analysis

PSS Beat Frequency (Hz)

Multiple pnoise ☐

Sweeptype default Sweep is currently absolute

Output Frequency Sweep Range (Hz)

Start-Stop Start Stop

Sweep Type

Logarithmic ☒ Points Per Decade ☐ Number of Steps

Add Specific Points ☐

Add Points By File ☐

Noise Type sampled(jitter) Sample Ratio

#	Event	Trig	TrigVal	Targ	TargVal	TD
1	phase			VoutP - Vou		

Add Change Delete Enabled ☒

Timing Event ☐ Edge Crossing ☐ Edge Delay ☒ Sampled Phase

Sampled Phase: strobed noise analysis

Measurement

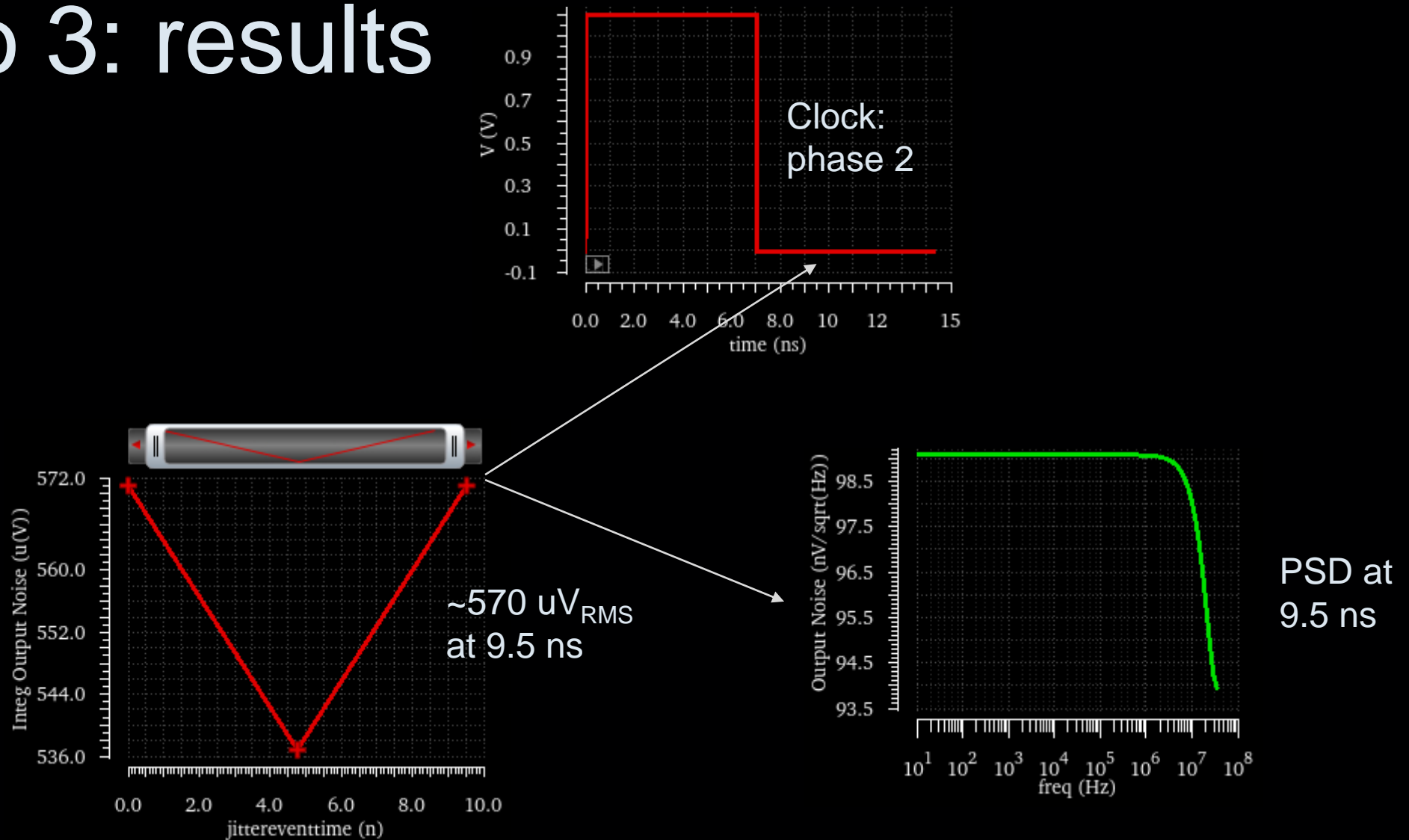
voltage Positive Output Node Select

Negative Output Node Select

Samples Per Period

Add Specific Points ☐

Demo 3: results



PSS + PAC SIMULATION SETUP AND DEMO

Static settling error \Leftrightarrow PAC

- 🔌 PAC = **periodic AC simulation**
- 🔌 It's a way to find your switched-cap amplifier's AC response
- 🔌 PAC uses PSS to see the average voltage gain at the output 😊
 - remember, normal AC sim. considers only one steady state
- 🔌 ➔ you can check the DC-gain of the switched-cap. \Leftrightarrow **static settling error!**
 - **PAC = alternative to long-duration transient simulation**

Demo 4: “demo_SCAmplifier_PAC_TB”

 Almost same setup as demo 3, now PSS + PAC

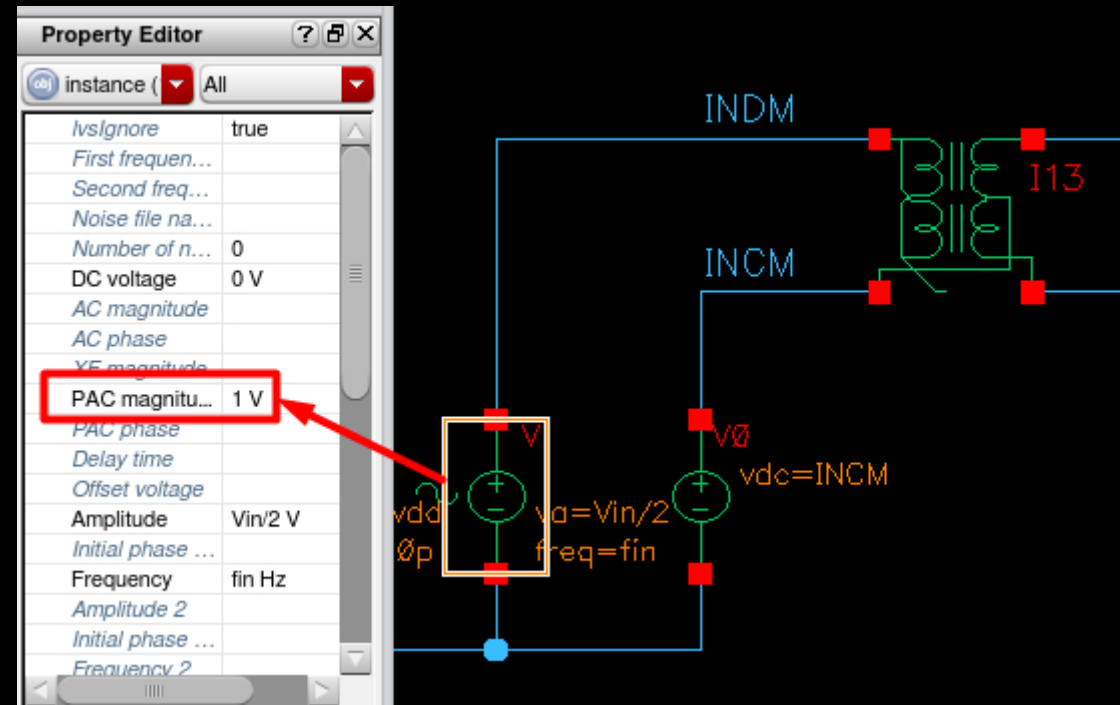
 Schematic:

- Input source must have a **PACmagnitude** now (equal to 1)

 ADE:

- **Lower switching frequency** to ~100k, you are interested in low-f effects
- Enable PAC simulation (see setup on next slide) → **sidebands = 0!**

Demo 4: schematic setup



PAC setup + plotting (Direct Plot Form)

Periodic AC Analysis

PSS Beat Frequency (Hz)

Sweeptype Sweep is currently absolute

Input Frequency Sweep Range (Hz)

Start Stop

Sweep Type

☒ Points Per Decade ☐ Number of Steps

Add Specific Points ☐

Add Points By File ☐

Sidebands

When using shooting engine, default value is 7.

Specialized Analyses

Again up to $f_s/2$

Must be 0!

Plotting Mode

Analysis

☐ pss ☒ pac

Function

☒ Voltage ☐ Voltage Gain ☐ Current ☐ IPN Curves

Select

Sweep

☒ spectrum ☐ sideband

Modifier

☒ Magnitude ☐ Phase ☐ dB20 ☐ Real ☐ Imaginary

Add To Outputs ☒

freqaxis = absout

> Select Positive Net on schematic...

Demo 4: results

