



Elevate Circuit Analysis with ANSYS Nexxim Circuit

Bryce Goswick, Application Engineer

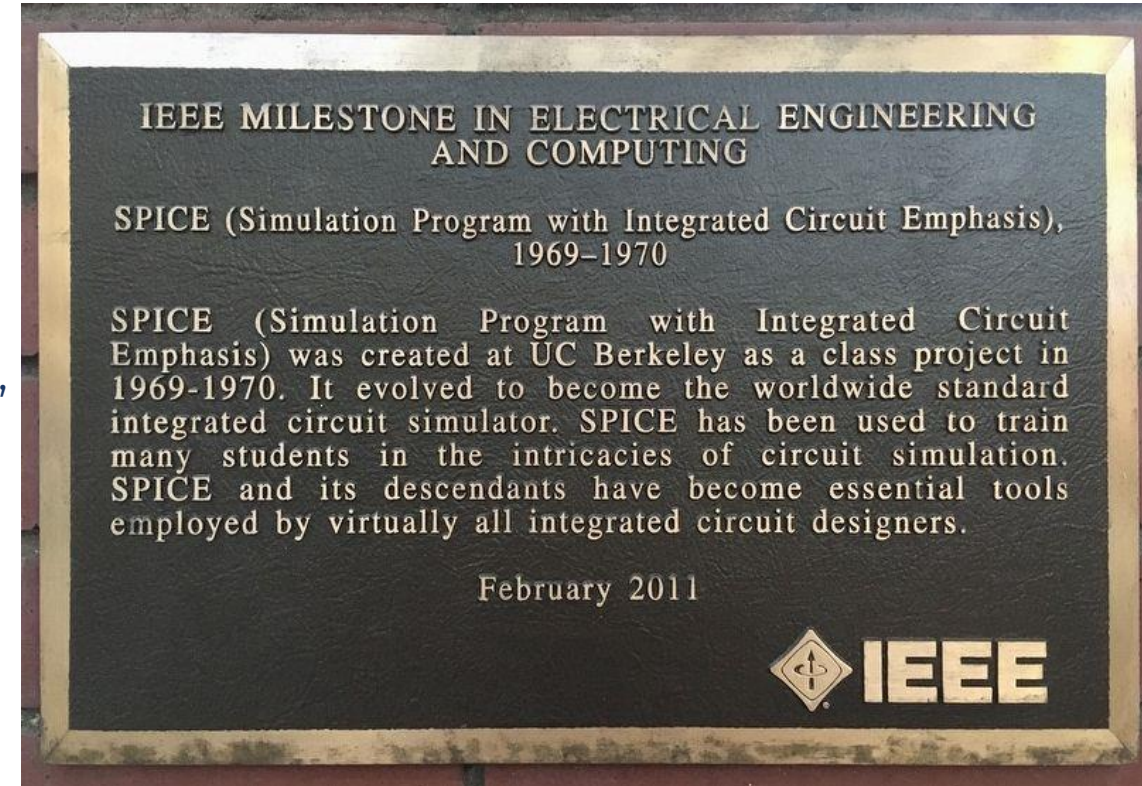
April 1, 2025

Agenda

Background
Project Overview
LTspice
Nexxim Circuit
Results
Comparison

Background

- Why use Schematic Tools?
 - Circuit Design
 - Operational Verification
 - Fast Structural Analyses
- SPICE was first circuit simulation tool
- Many different versions: ISPICE, LTSPICE, HSPICE, PSPICE, XSPICE, NGSPICE, etc.
- Virtually all Electrical Engineers have used SPICE



Advantages of Nexxim Circuit

Why Nexxim Circuit?

Specialized
Solvers

Signal
Integrity
Oriented

Wider Scope

Larger
Component
Library

ANSYS
Optimetrics

3D
Simulation
Integration

Nexxim Circuit is included in every ANSYS
Electronics package

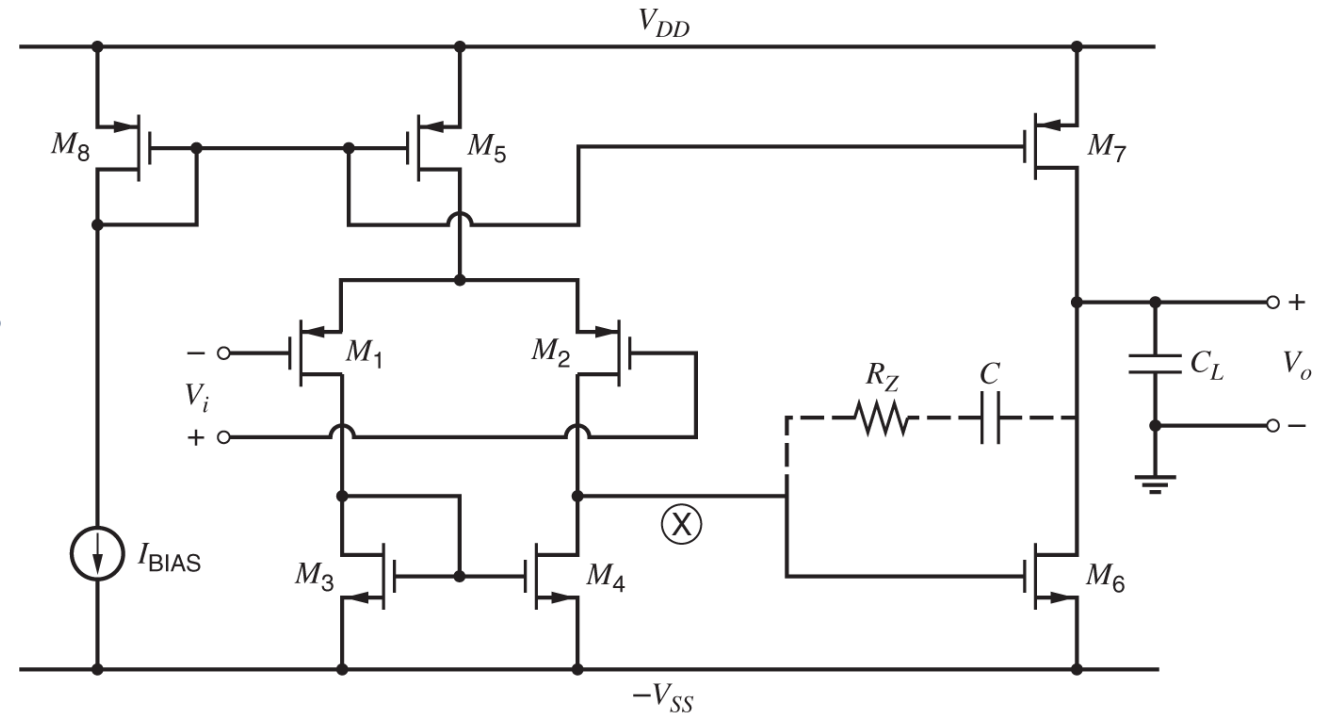
Project Overview

- Construct Two-Stage Operation Amplifier with the following specs:

- Gain @ 1kHz > 40dB
- Unity Gain Freq > 50kHz
- Phase Margin > 45°
- Gain Margin > 10dB
- Quiescent Current $\cong$ 100uA
- Output Capacitor = 5pF
- VDD = 5V
- Compensation Cap < 30pF
- Compensation Res < 1000k Ω
- MOSFET model = 0.18um CMOS

- Variables:

- W/L Values
- Compensation C/R



Procedure

LTspice:

- Create subcircuit topology
- Use .model to assign MOSFET model
- Assign W/L and C/R with .param definitions
- Use hierarchy to create subcircuit symbol
- Apply input and output components
- Configure analysis
- Choose nodes to plot

Nexxim:

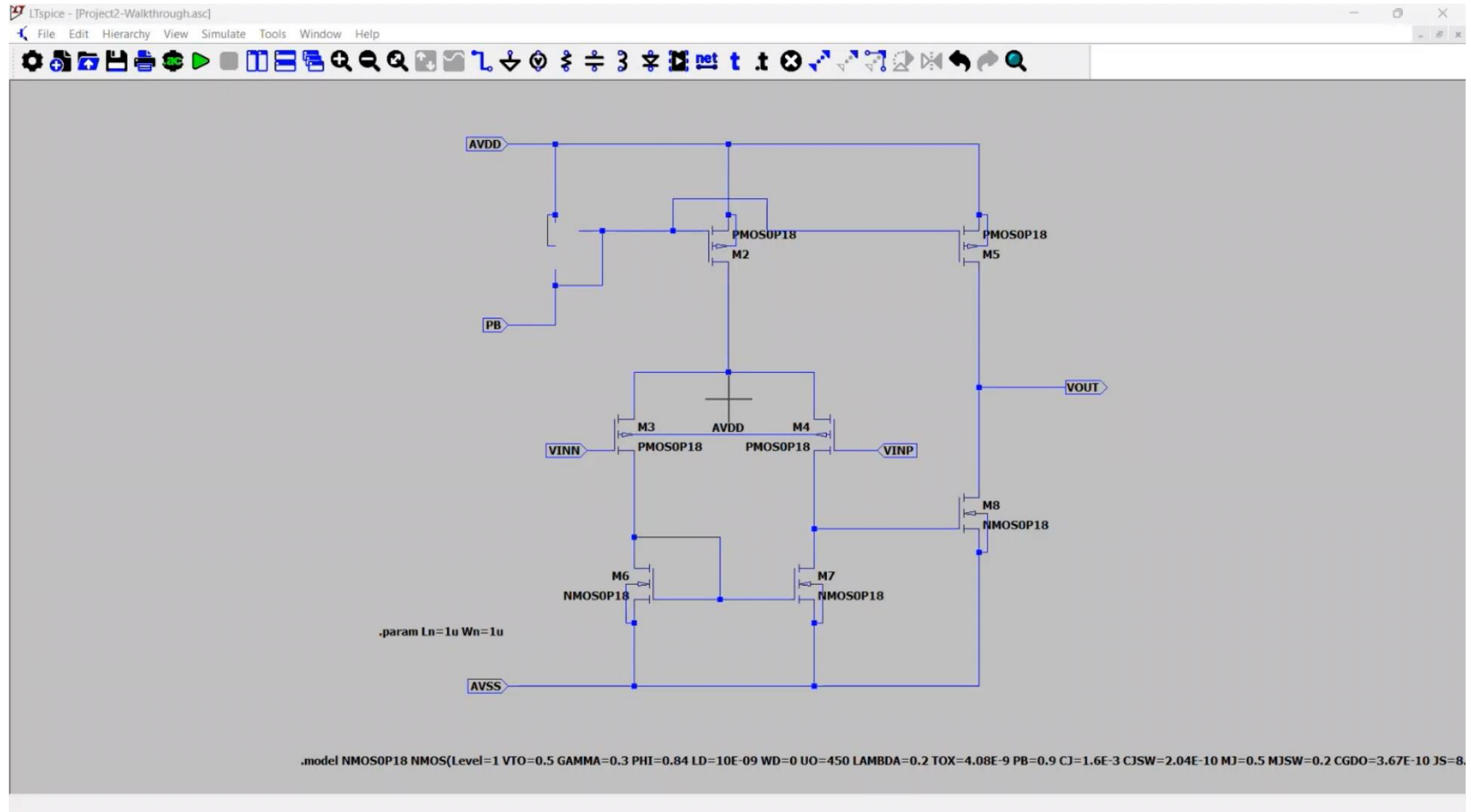
- Create subcircuit topology
- Use model blocks to assign MOSFET model
- Assign W/L and C/R with project variables
- Use hierarchy to create subcircuit symbol
- Apply input and output components
- Configure analysis and parametric sweep
- Choose nodes to plot
- (optional) Optimize for known specs

LTspice Circuit

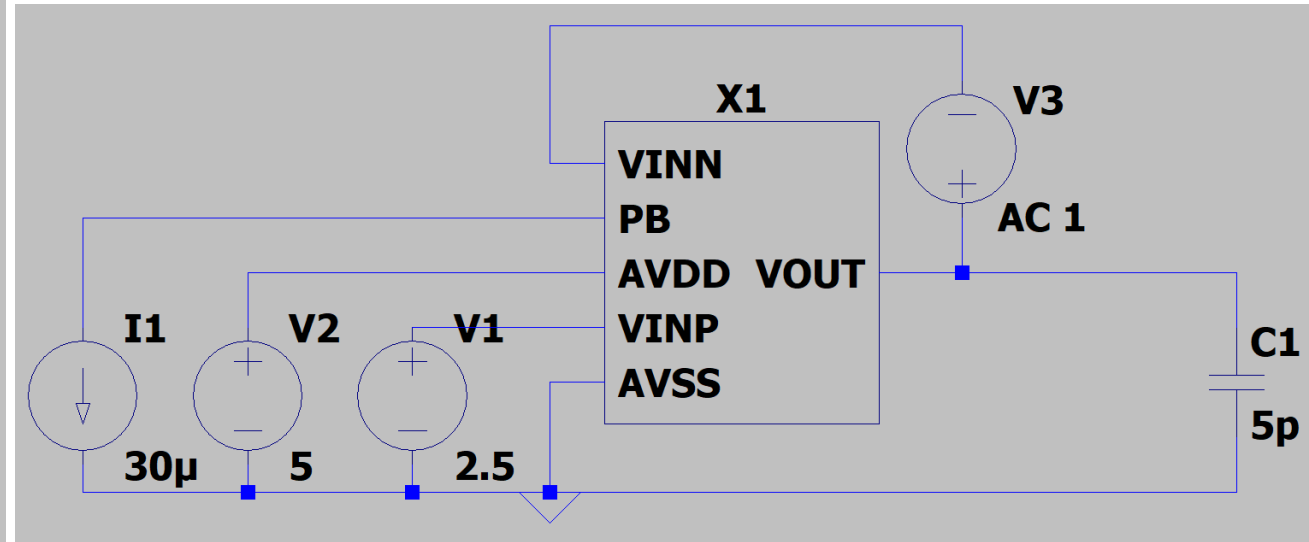
- Walkthrough of setup:
- Defining Models
- Defining Parameters
- Settings up Analyses
- Post Processing



Defining Models and Parameters



LTspice



Nexxim Circuit

- Walkthrough of setup:
- Defining Model Blocks
- Defining Project Parameters
- Settings up Analyses
- Optimization and Parameterization
- Post Processing



Defining Models and Parameters

The screenshot displays a circuit design software interface for a CMOS Two-Stage Op Amp. The main workspace shows a schematic diagram with various components and their connections. The interface includes a Project Manager on the left, a Properties panel at the bottom left, a Component Libraries panel on the right, and a Message Manager at the bottom.

Project Manager: Shows the project structure, including "Project2*", "Circuit1*", and various sub-components like Data, Excitations, Ports, Analysis, SoD Config, Optimetrics, Results, and Definitions.

Properties Panel: Contains a table with columns: Name, Value, Unit, and Evaluated V... The table is currently empty. Below the table are sections for "General" (Run ER..., Use Dy..., Update ...) and "Solver" (Use Sp..., Transfor..., Use Loc...).

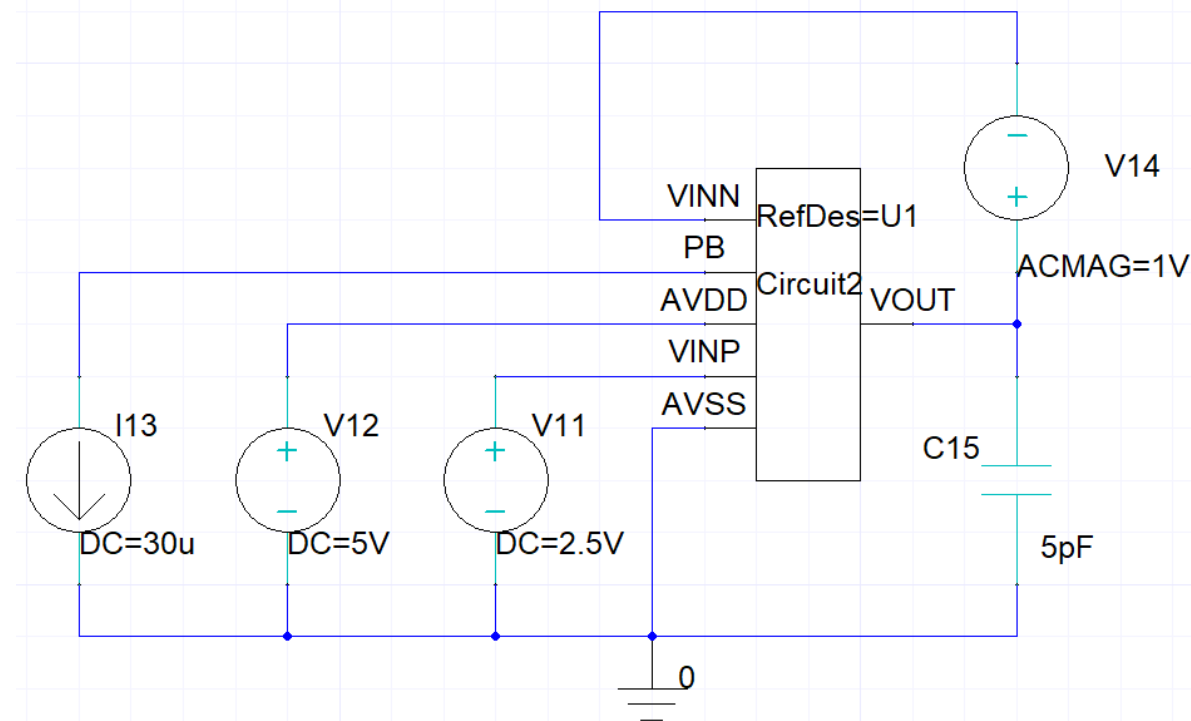
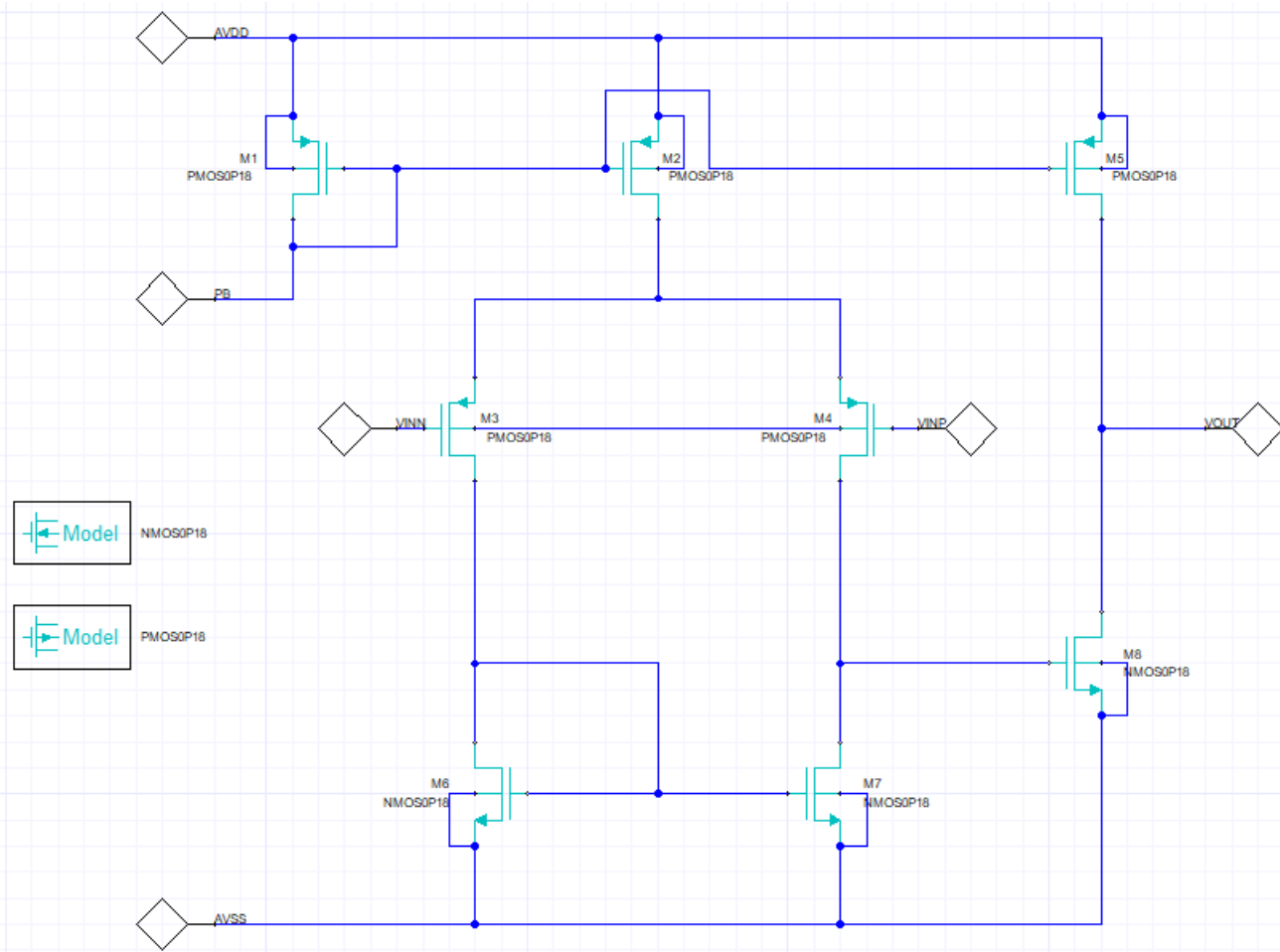
Component Libraries: Displays a list of components, including "Level01_NMOS", "Level01_PMOS", and "Level01_PMOS_4pin". A search bar is present at the top of the panel.

Schematic Diagram: The central workspace shows a detailed circuit diagram. It includes a PMOS network (M9, M10, M11, M7) and an NMOS network (M4, M3, M5). The circuit is powered by AVDD and AVSS, and has input ports VIN1 and VIN2, and an output port VOUT. The schematic is labeled "Page1" at the bottom.

Message Manager: Shows a message from "Global - Messages" at the bottom of the interface.

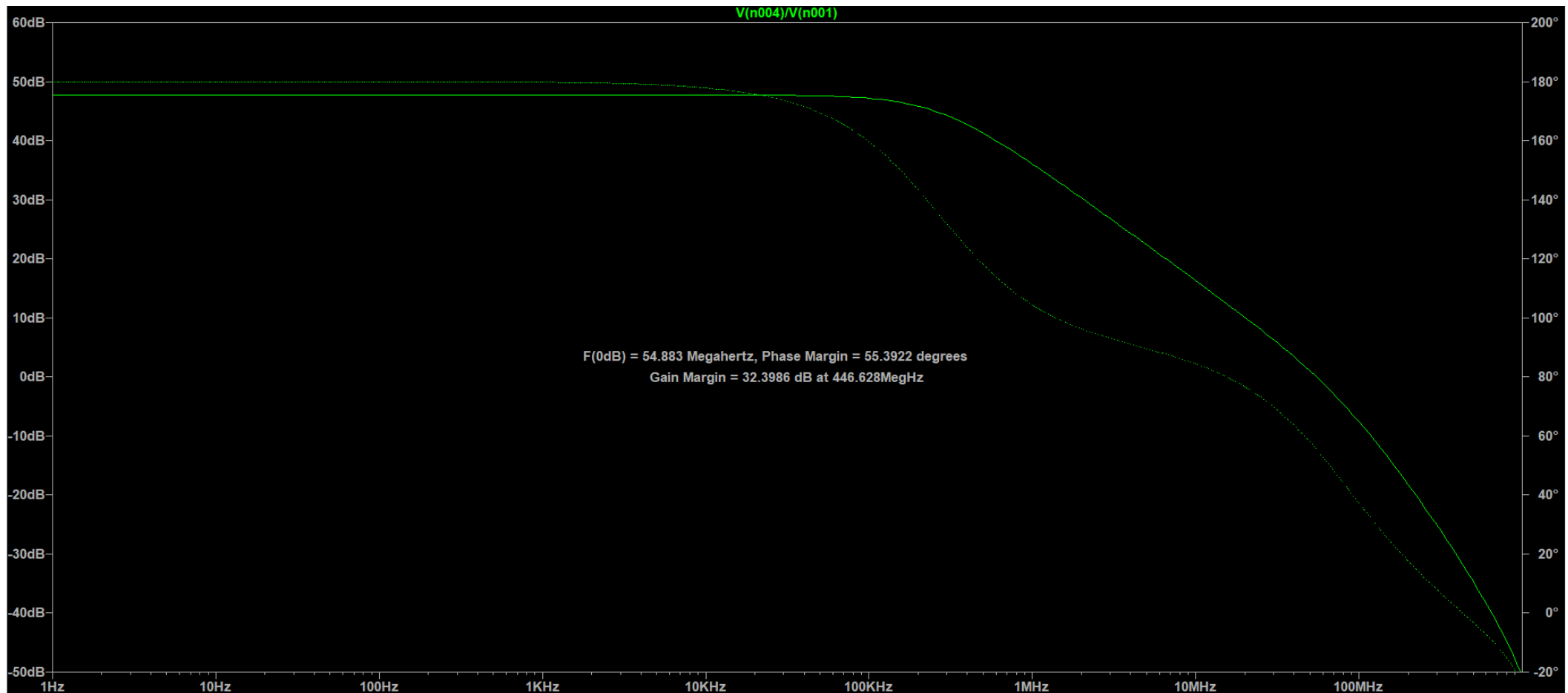
Status Bar: At the bottom, it indicates "Number of selected items: 0 X: 7066mil Y: 2052mil".

Nexxim Circuit



Results

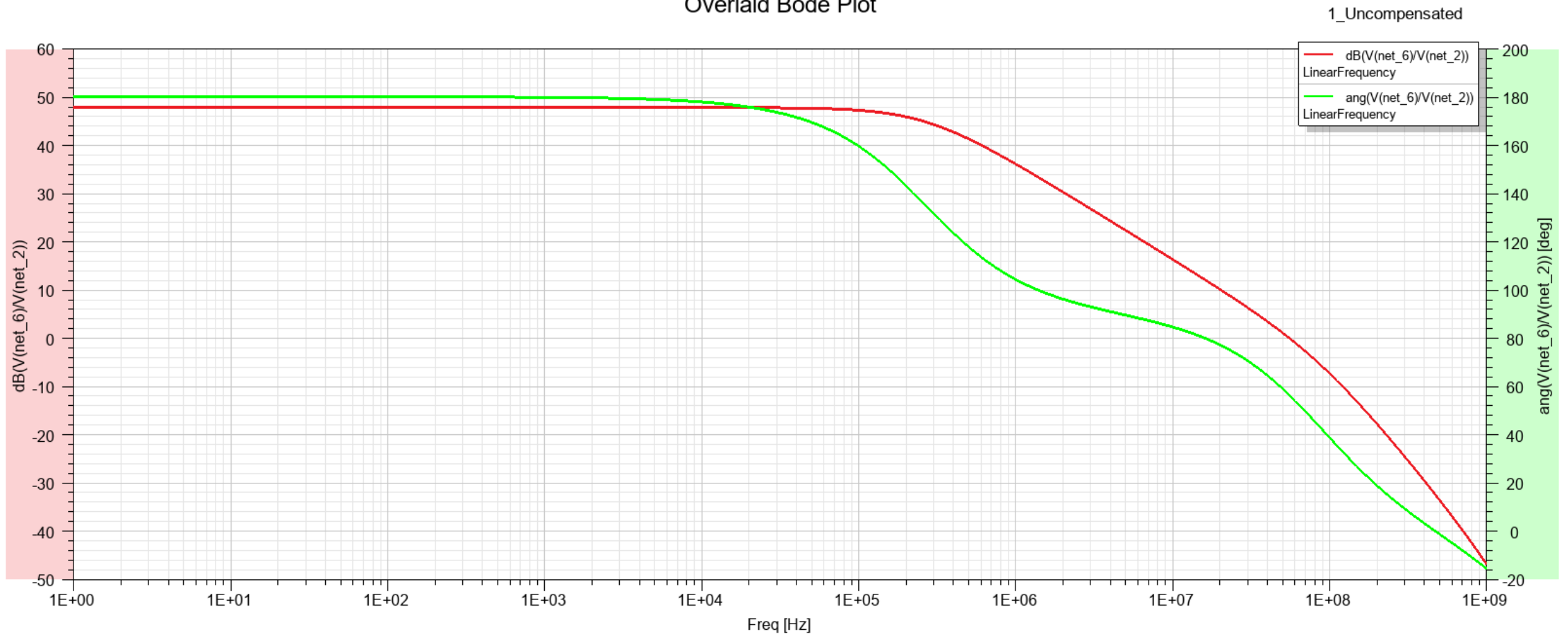
LTspice Uncompensated:



Results

Nexxim Circuit Uncompensated:

Overlaid Bode Plot



Results

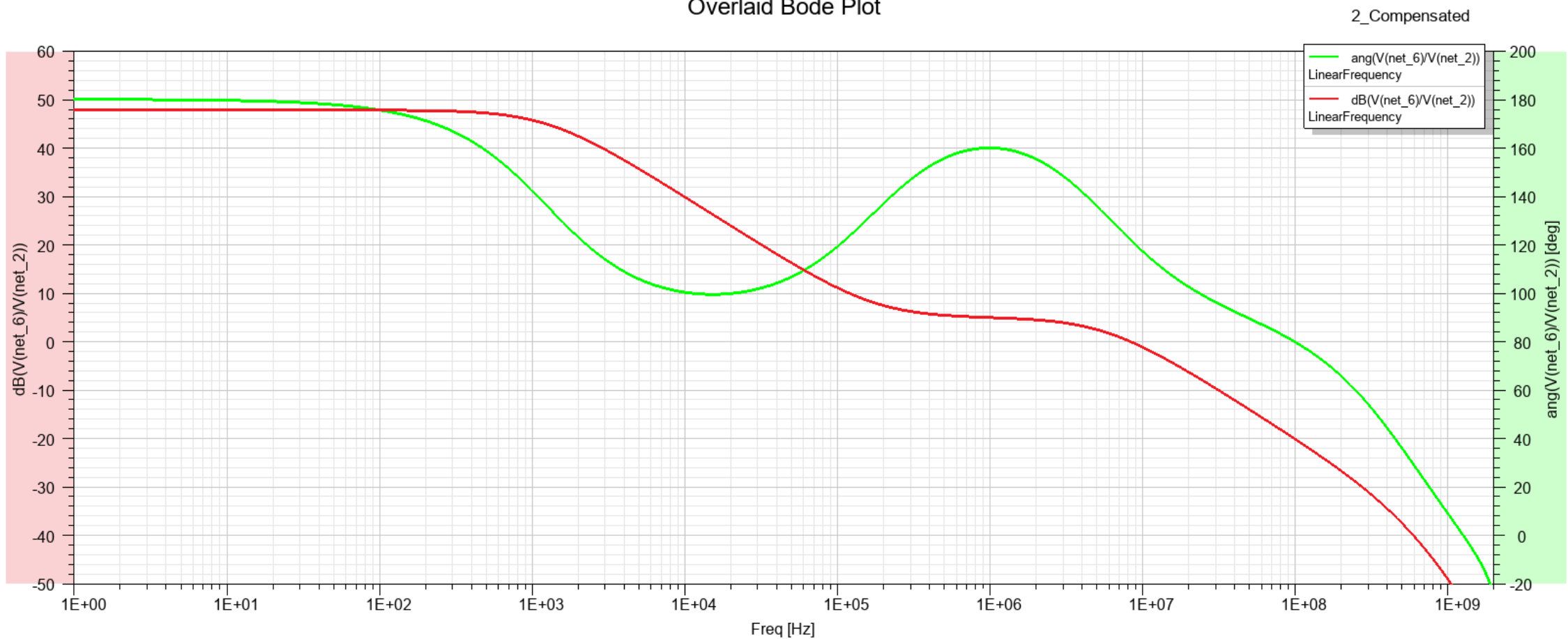
LTspice Compensated:



Results

Nexxim Circuit Compensated:

Overlaid Bode Plot



Optimetrics

- Multiple Optimization Schemes
- Define Optimization Functions and Goals
- Assign Importance with Weight
- Minimized Dimensions and Compensation R and C

Screening(Search-based)
Multi-Objective Genetic Algorithm(Random-search)
Nonlinear Programming by Quadratic Lagrangian(Gradient)
Mixed-Integer Sequential Quadratic Programming(Gradient and Discrete)
Adaptive Multiple-Objective(Random Search)
Adaptive Single-Objective(Gradient)
Merit-based Sequential Quadratic Programming(Gradient)
MATLAB
----- Legacy -----
Sequential Nonlinear Programming(Gradient)
Sequential Mixed Integer NonLinear Programming(Gradient and Discrete)
Quasi Newton(Gradient)
Pattern Search(Search-based)
Genetic Algorithm(Random search)

Setup Optimization

Goals | Variables | General | Options

Optimizer: Multi-Objective Genetic Algorithm(Random-search) Setup...

Estimated Variations: 1100

Cost Function:

	Calc. Solution	Calculation	Calc. Range	Condition	Goal	Weight
	LinearFrequency	Gainat1kHz	Freq(1Hz:1GHz)	>=	[40]	[10]
	LinearFrequency	UnityGainFreq	Freq(1Hz:1GHz)	>=	[50000]	[10]
	LinearFrequency	PhaseMargin	Freq(1Hz:1GHz)	>=	[45]	[10]
	LinearFrequency	gainmargin	Freq(1Hz:1GHz)	>=	[10]	[10]
	LinearFrequency	Minimization	Freq(1Hz:1GHz)	<=	[5]	[1]

Setup Calculations... Delete

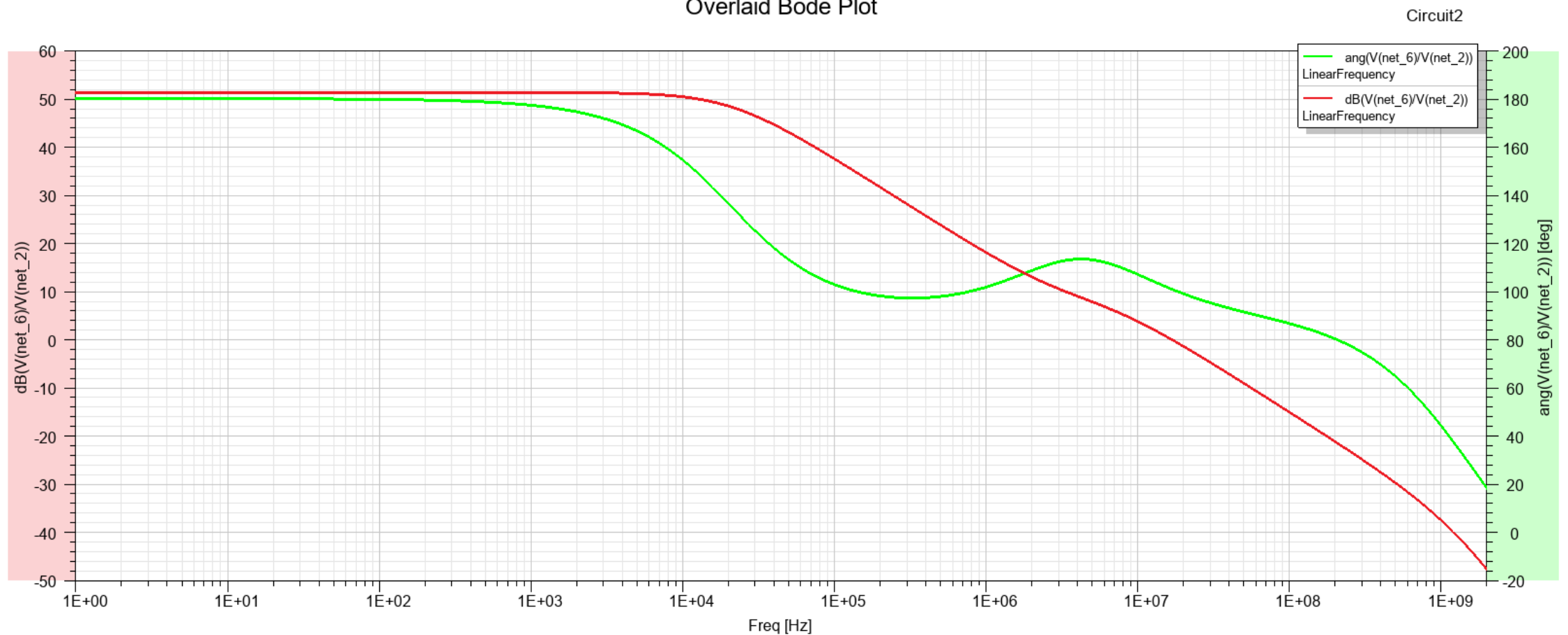
Acceptable Cost: 0 Noise: 0.0001 ☐ Stop on satisfy Acceptable Cost ☐ Show Advanced Options

Edit Variables HPC and Analysis Options... OK Cancel

Results

Optimized Nexxim Circuit:

Overlaid Bode Plot



Comparison

- LTspice and Nexxim agree
- Optimetrics found an optimal setup

	LTspice	Nexxim Circuit	Nexxim Optimetrics
Gain @1kHz	45.73 dB	45.73 dB	51.27 dB
Unity Gain Frequency	7.06 MHz	7.06 MHz	14.91 MHz
Gain Margin	55.49 dB	53.91 dB	56.66 dB
Phase Margin	121.75 °	121.77 °	100.96 °

Initial:

Variable	Value
Wp	0.5u
Wn	1u
Lp	1u
Ln	1u
Cc	20p
Rc	50k

Optimized:

Variable	Value
Wp	0.54u
Wn	0.61u
Lp	0.51u
Ln	0.52u
Cc	1.15p
Rc	52.92k

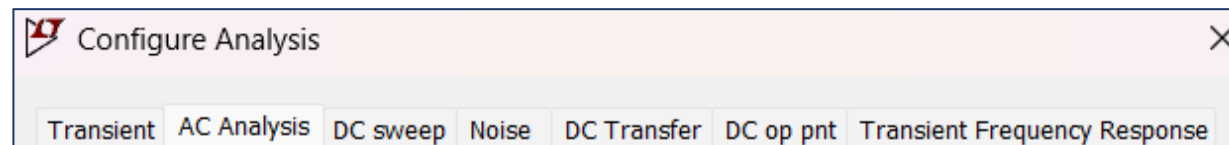
Comparison

- SPICE:
 - Transient and Frequency Domain
 - DC sweep
 - Basic Circuit Analysis
- Nexxim Circuit:
 - Transient and Frequency Domain
 - DC Analysis
 - Signal Integrity Analysis
 - Resonant Analysis
 - Structure Blocks (trace models, via models, etc.)
 - Cosimulation with FEM analysis
 - Parameterization/Optimization of Circuits

Nexxim:

DC Analysis
Linear Network Analysis
Oscillator Resonant Frequency Search
Oscillator Analysis (1-Tone)
Oscillator Analysis (N-Tone)
Transient Analysis
TV Noise Analysis
Envelope Analysis
Periodic Transfer Function (PXF)
VerifEye (Statistical Eye) Analysis
Quick Eye Analysis
AMI Analysis
Harmonic Balance (1-Tone)
Harmonic Balance (N-Tone)
System Frequency Domain Analysis

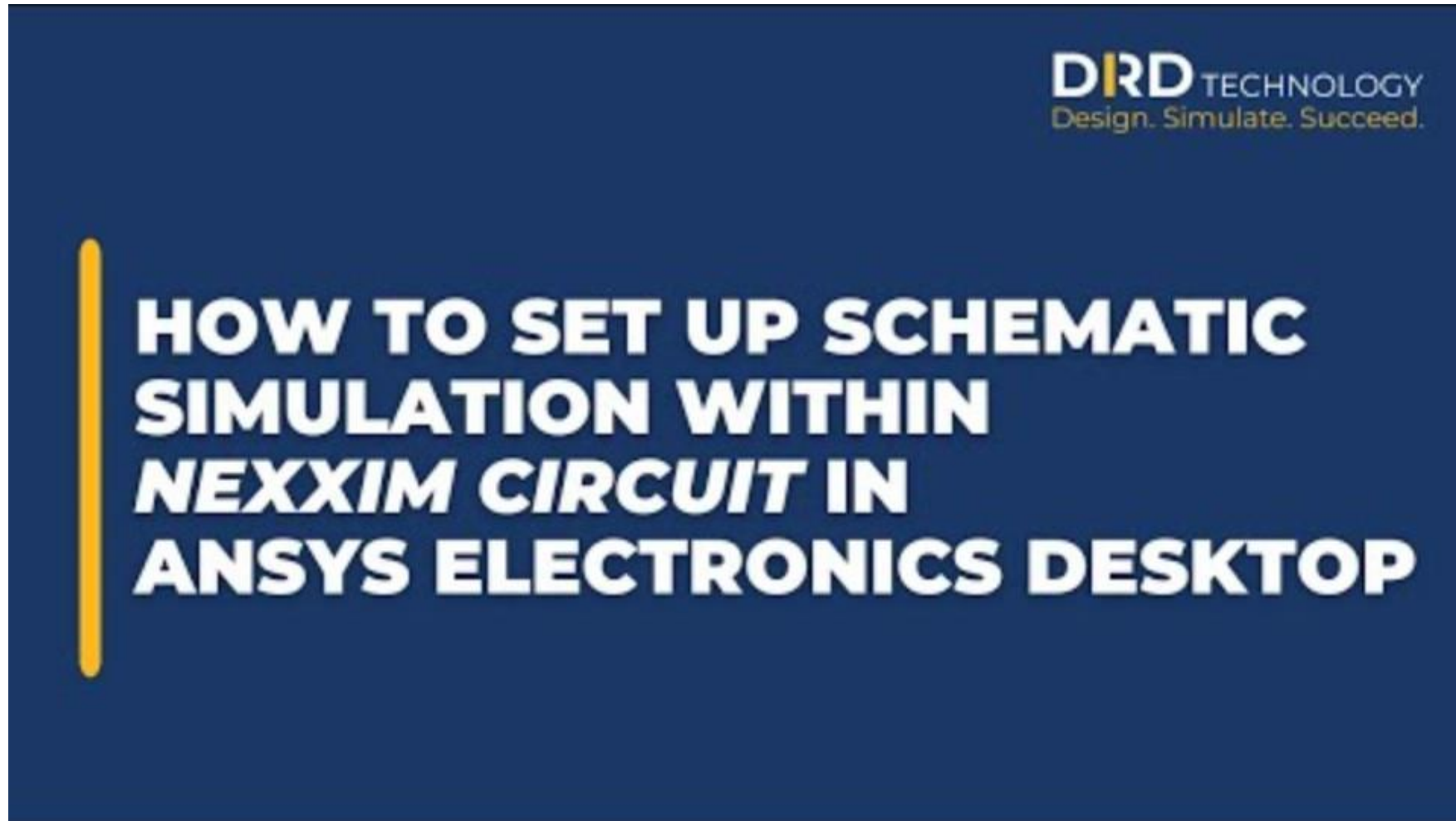
LTspice:



Conclusion

- Nexxim Circuit has all the capabilities of LTspice plus more:
 - Additional Analyses options
 - Parameterization of subcircuits
 - Optimization of designs
- Applications of Nexxim Circuit:
 - Signal Integrity
 - Bit-Error-Rate
 - Transmitter/Receiver Circuit Link Performance
 - Resonant Search
 - Time-Varying Noise Analysis
 - Circuit Matching with Smith Tool
 - FEM Co-simulation
 - Parametrization of Components
- Nexxim Circuit is included in every Ansys Electronics Package

How to Simulate within Nexxim Circuit



Questions?



Website: drd.com



Support: support@drd.com



YouTube: youtube.com/@drdtechnology



More: [How to Simulate Within Nexxim Circuit](#)