DRD TECHNOLOGY

Elevate Circuit Analysis with ANSYS Nexxim Circuit

Bryce Goswick, Application Engineer

April 1, 2025

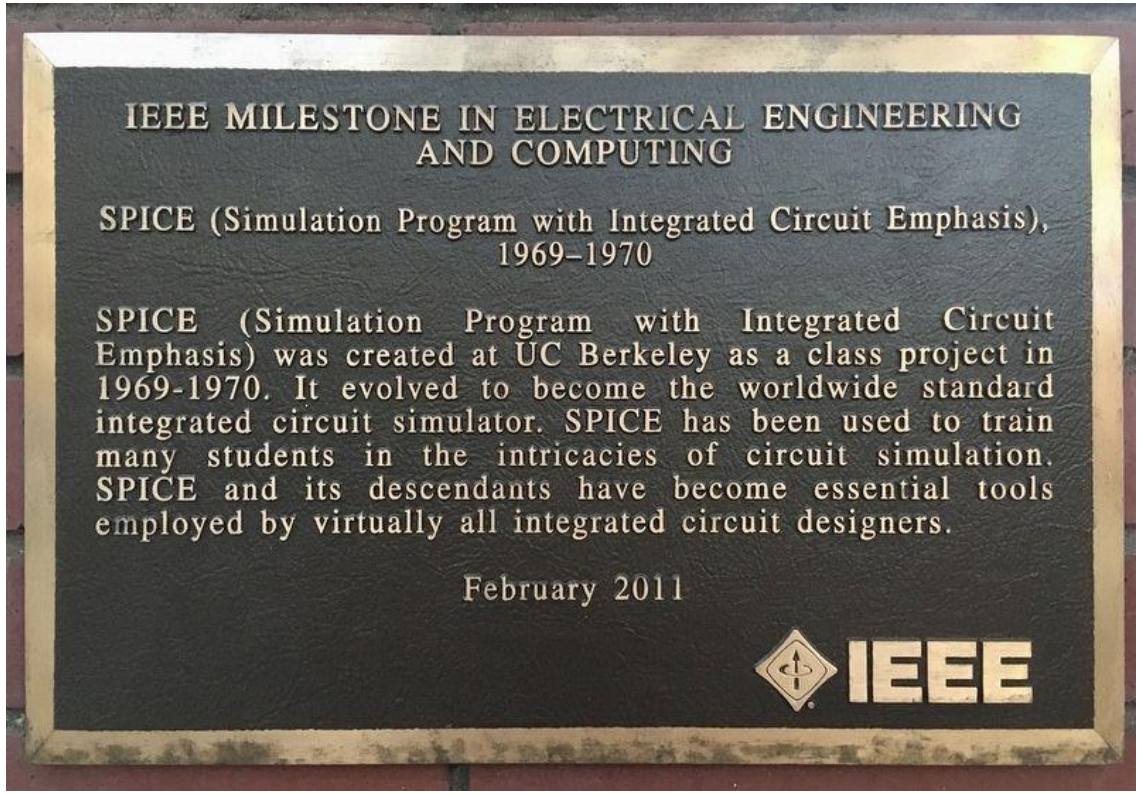
DRD

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

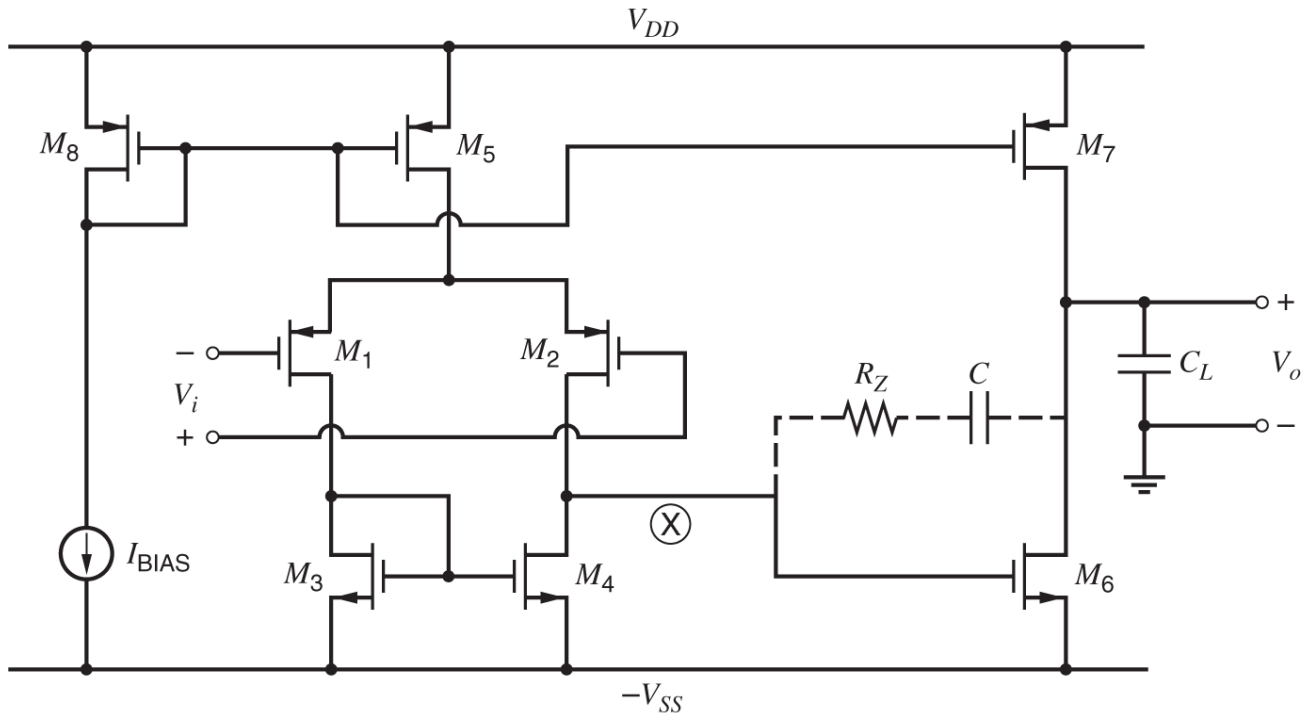
Why Nexxim Circuit?

Specialized Solvers	Signal Integrity Oriented	Wider Scope	Larger Component Library	ANSYS Optimetrics	3D Simulation Integration

Nexxim Circuit is included in <u>every</u> 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 ≅ 100uA
  - Output Capacitor = 5pF
  - VDD = 5V
  - Compensation Cap < 30pF
  - Compensation Res < 1000kΩ
  - 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

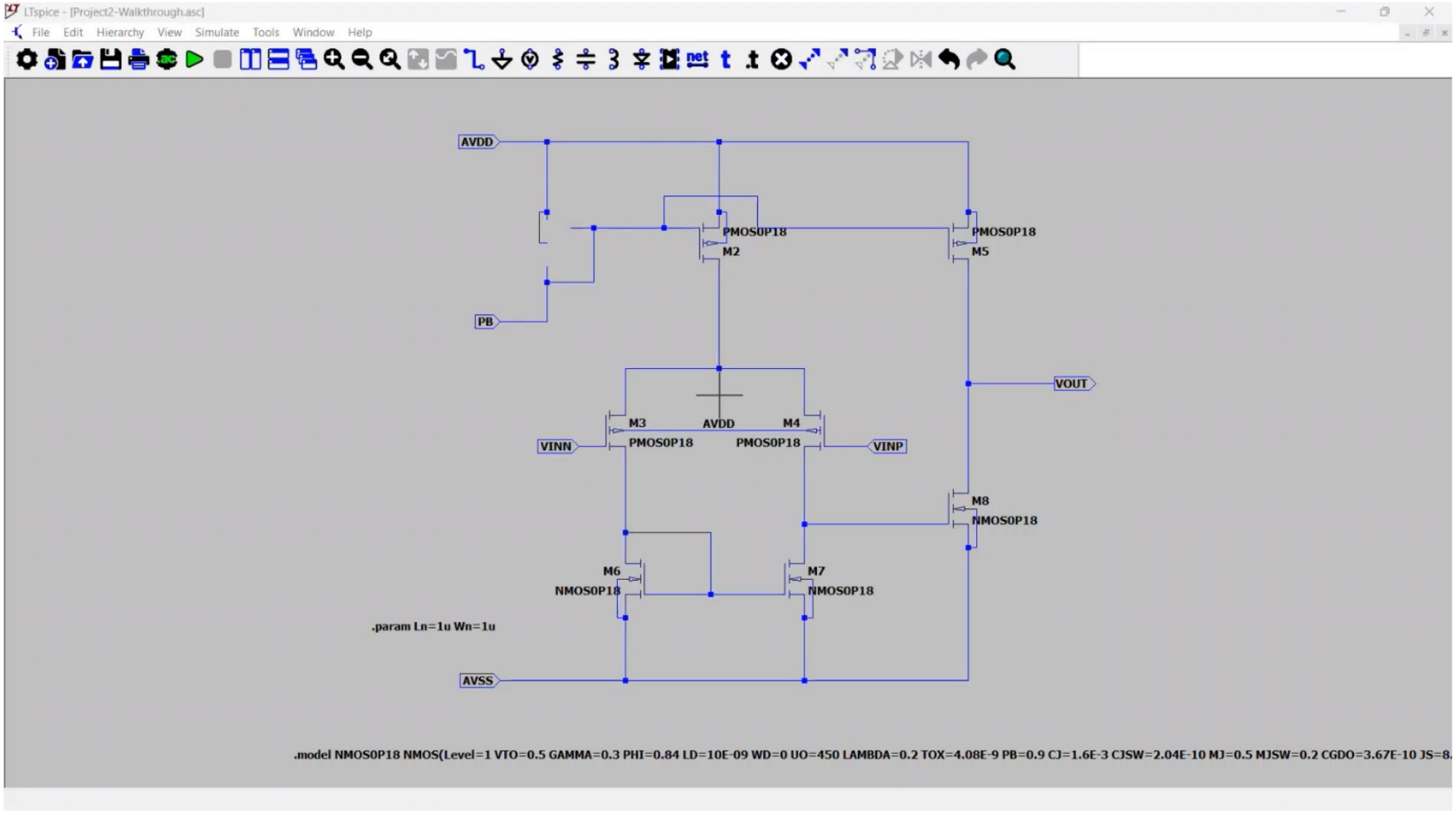
Nexxim:

- 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

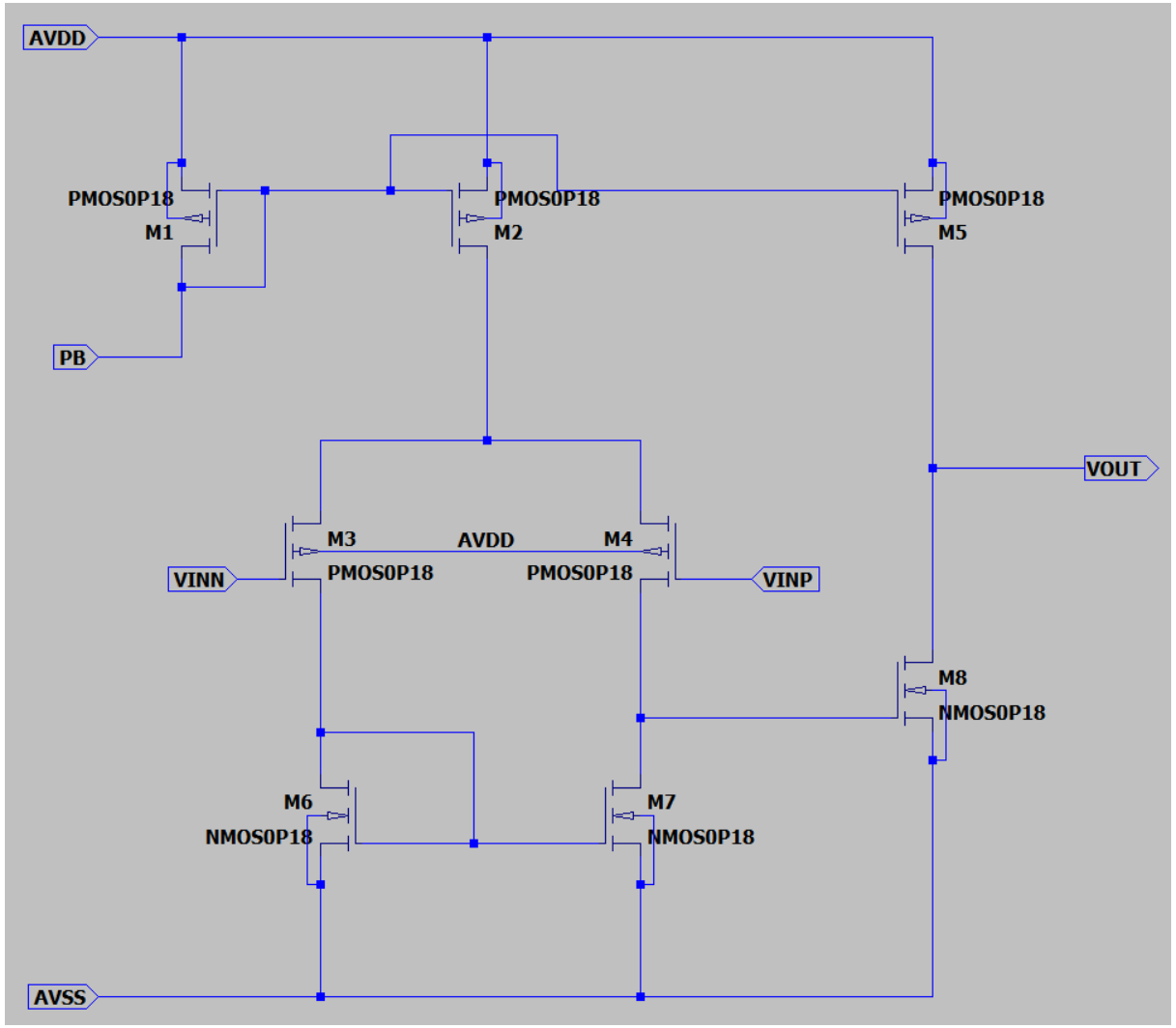
LTspice Circuit

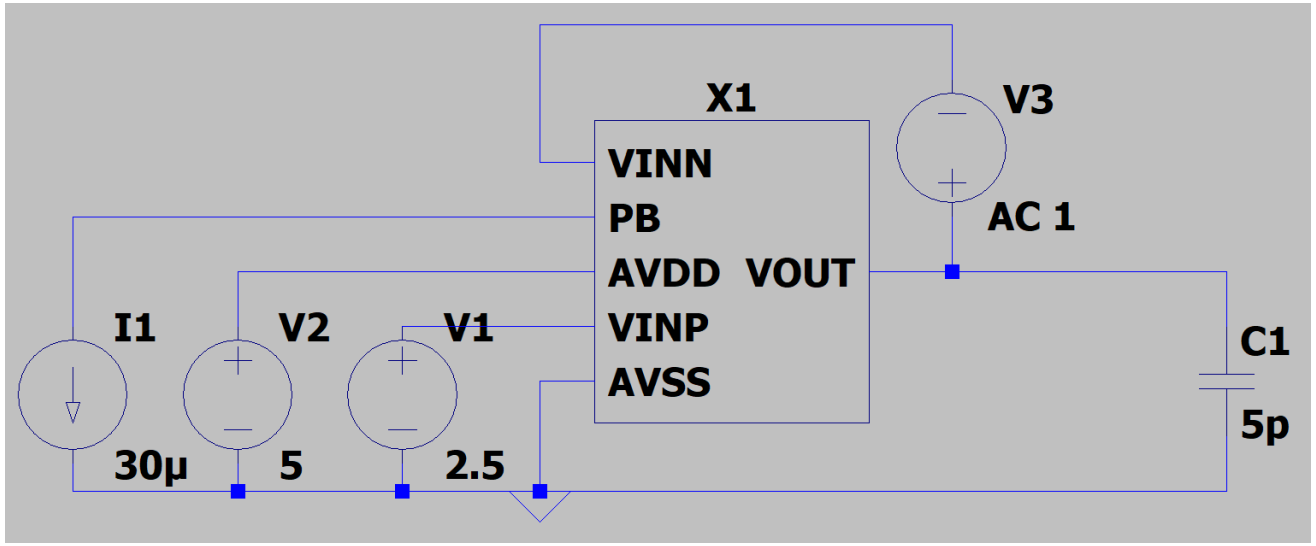
- Walkthrough of setup:
- Defining Models
- Defining Parameters
- Settings up Analyses
- Post Processing

Defining Models and Parameters



LTspice

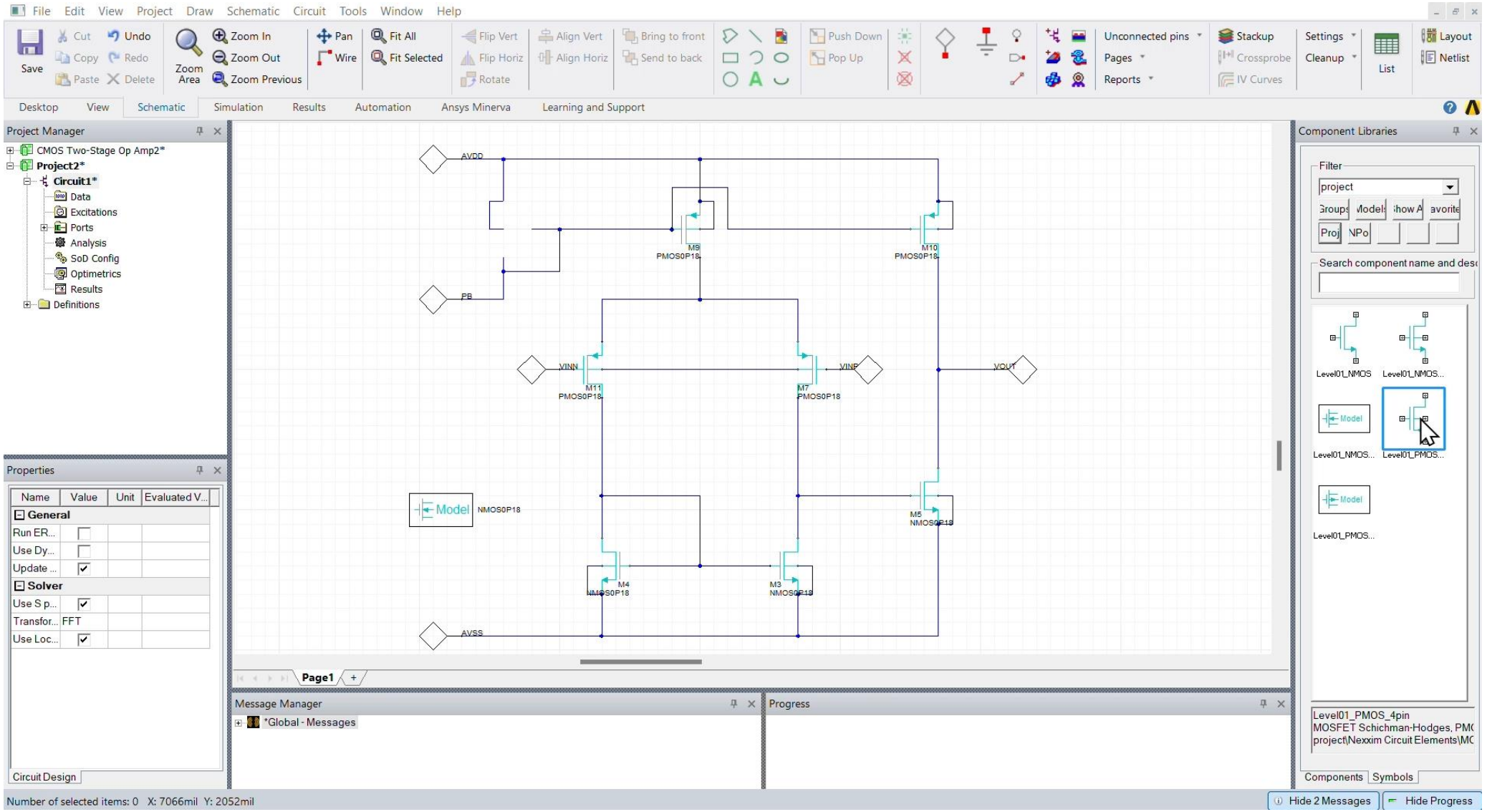

AVDD
PMOS0P18
M1
PMOS0P18
M2
PMOS0P18
M5
PB
VOUT
M3
AVDD
M4
VINN
PMOS0P18
PMOS0P18
VINP
M8
NMOS0P18
M6
NMOS0P18
M7
NMOS0P18
AVSS


X1
VINN
PB
AVDD
VOUT
VINP
AVSS
V3
AC 1
I1
30µ
V2
5
V1
2.5
C1
5p

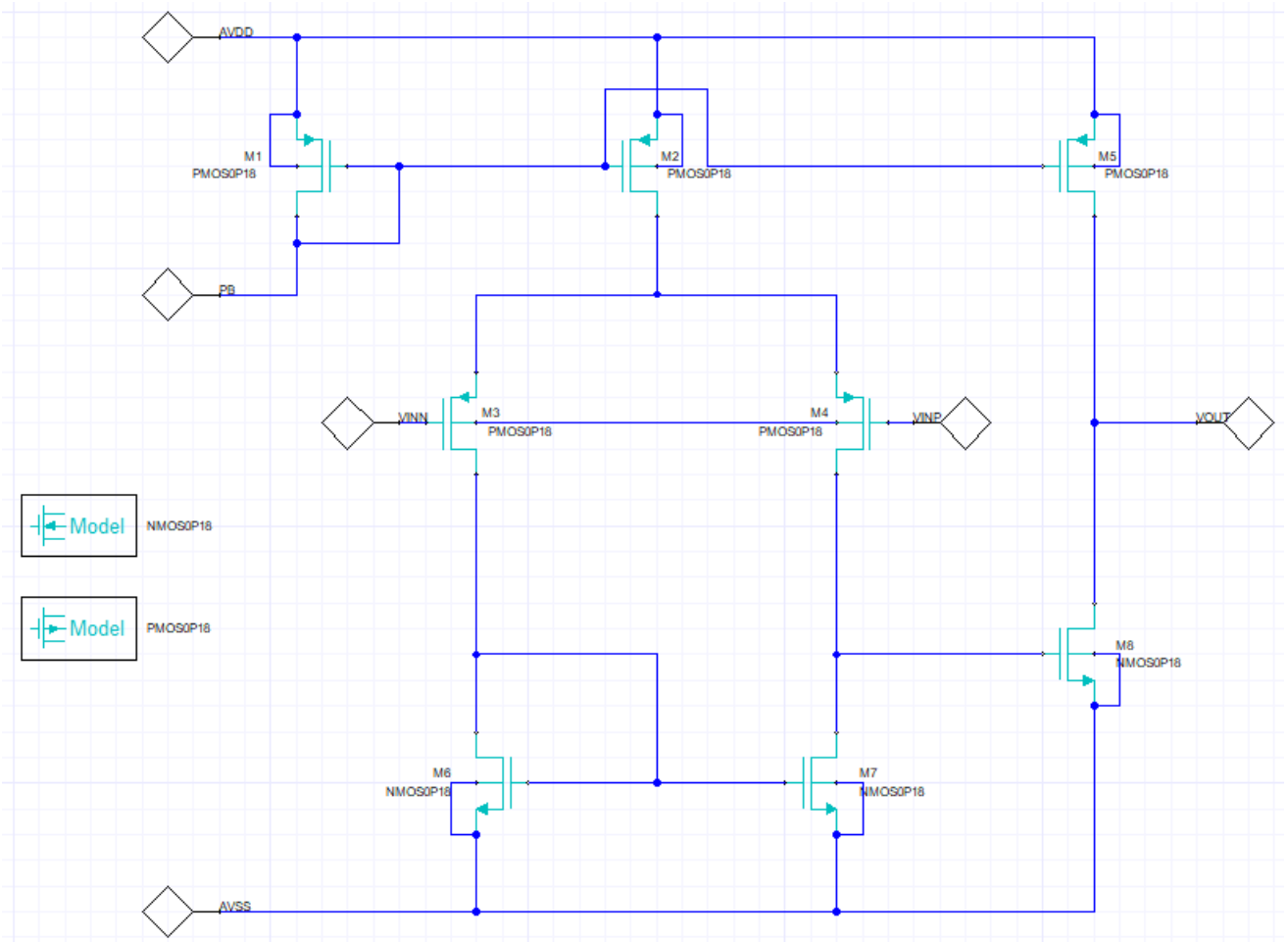
Nexxim Circuit

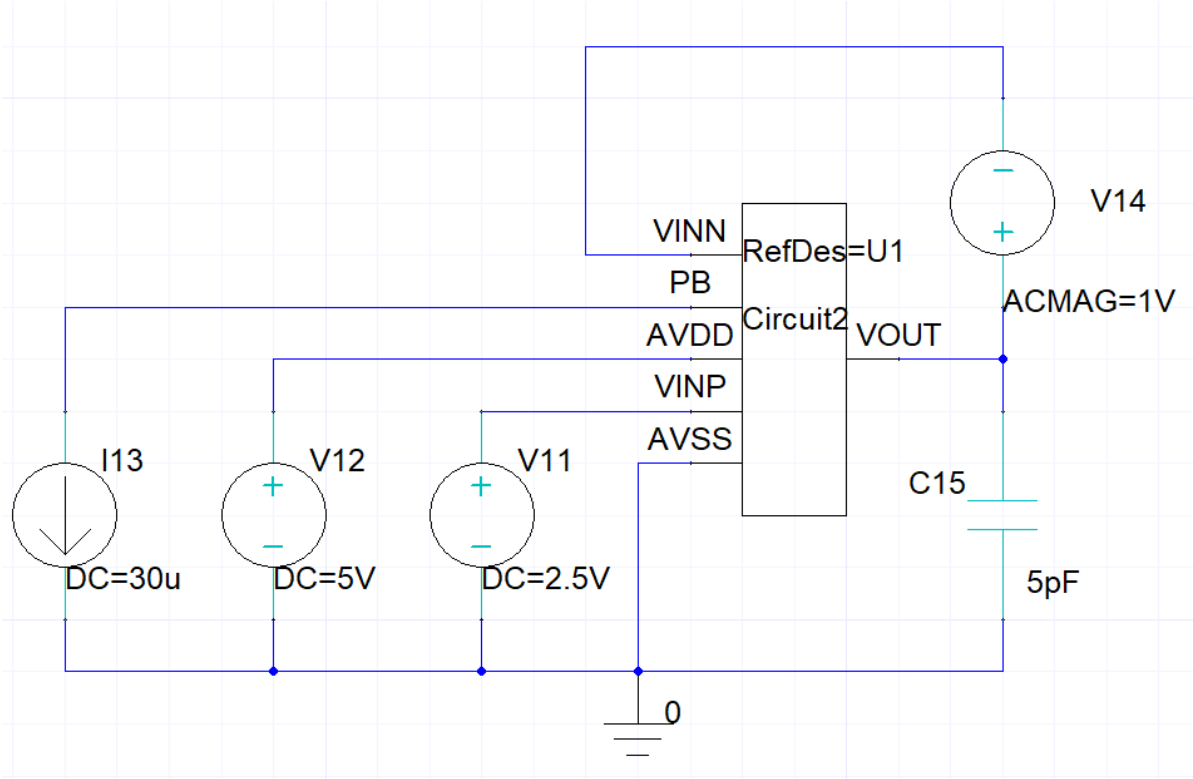
- Walkthrough of setup:
- Defining Model Blocks
- Defining Project Parameters
- Settings up Analyses
- Optimization and Parameterization
- Post Processing

Defining Models and Parameters



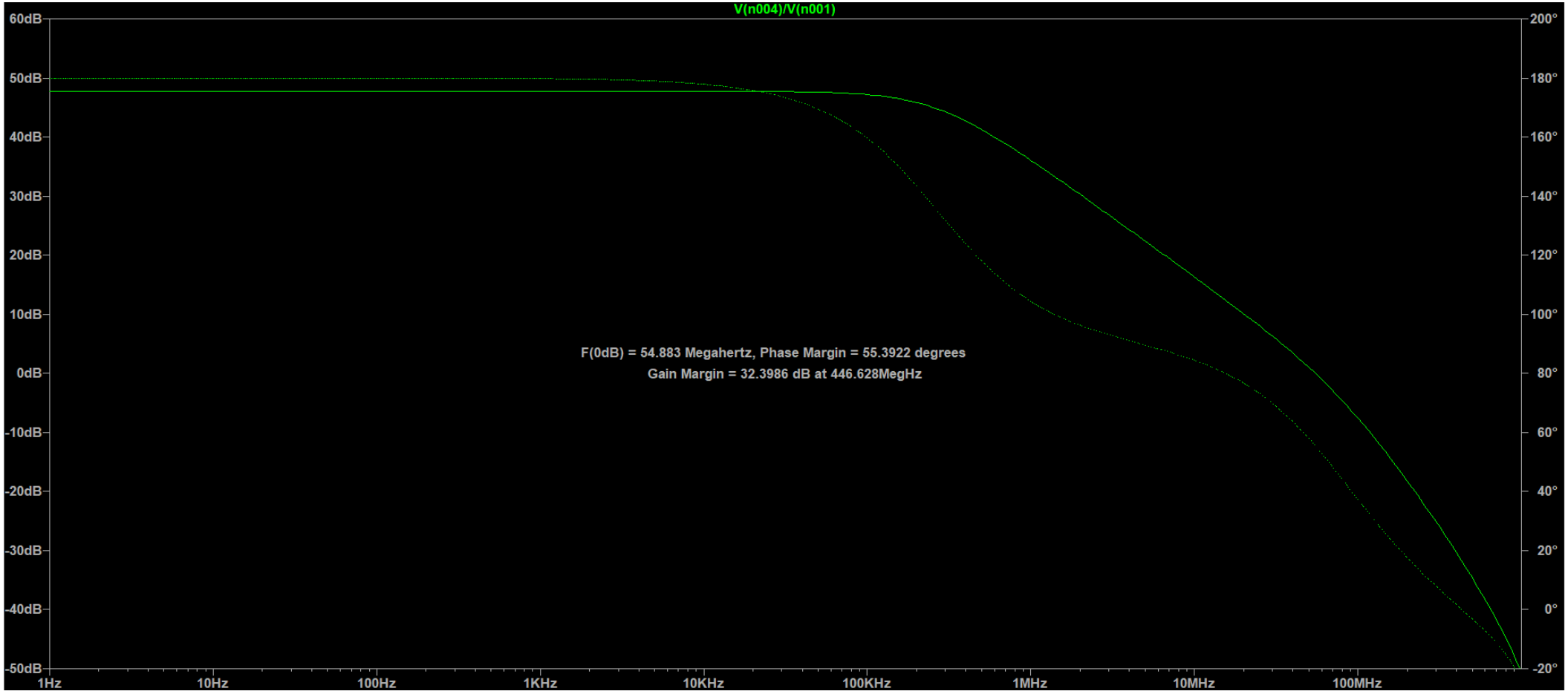
Nexxim Circuit





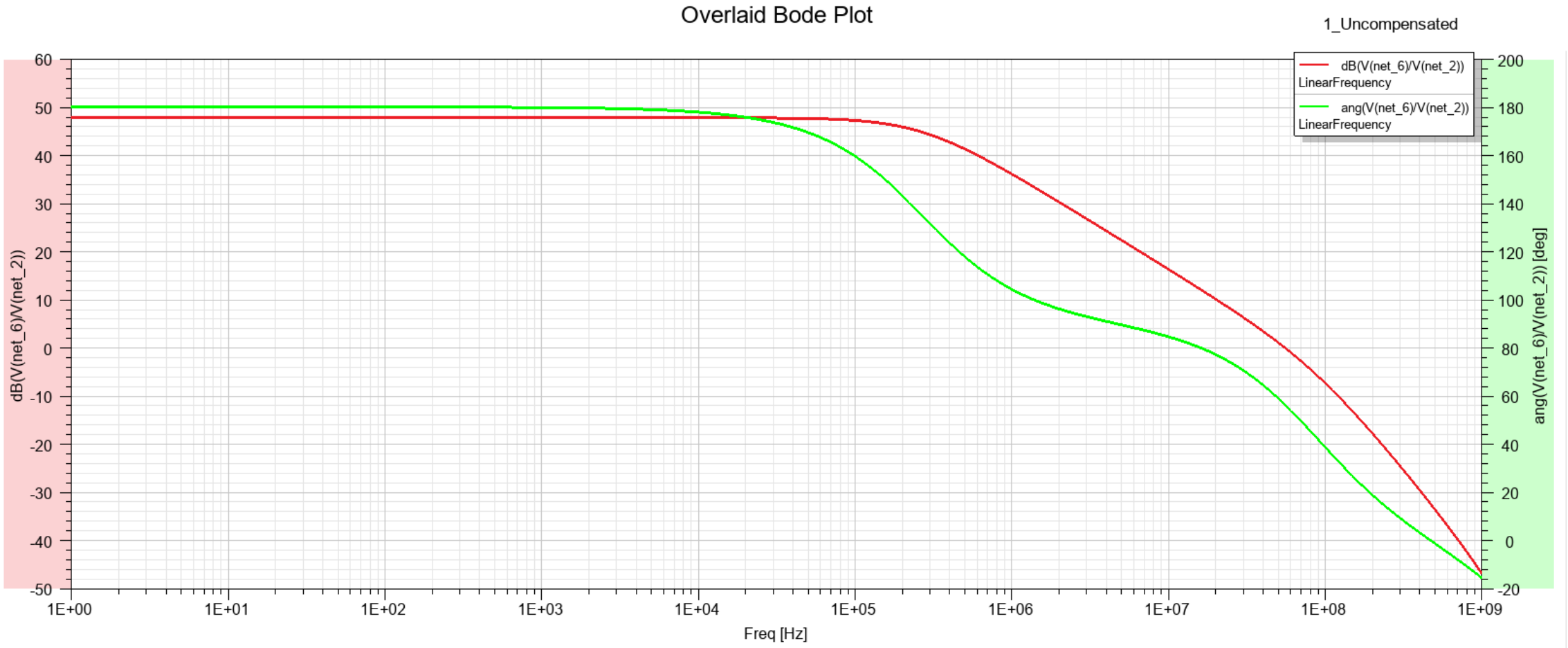
Results

LTspice Uncompensated:



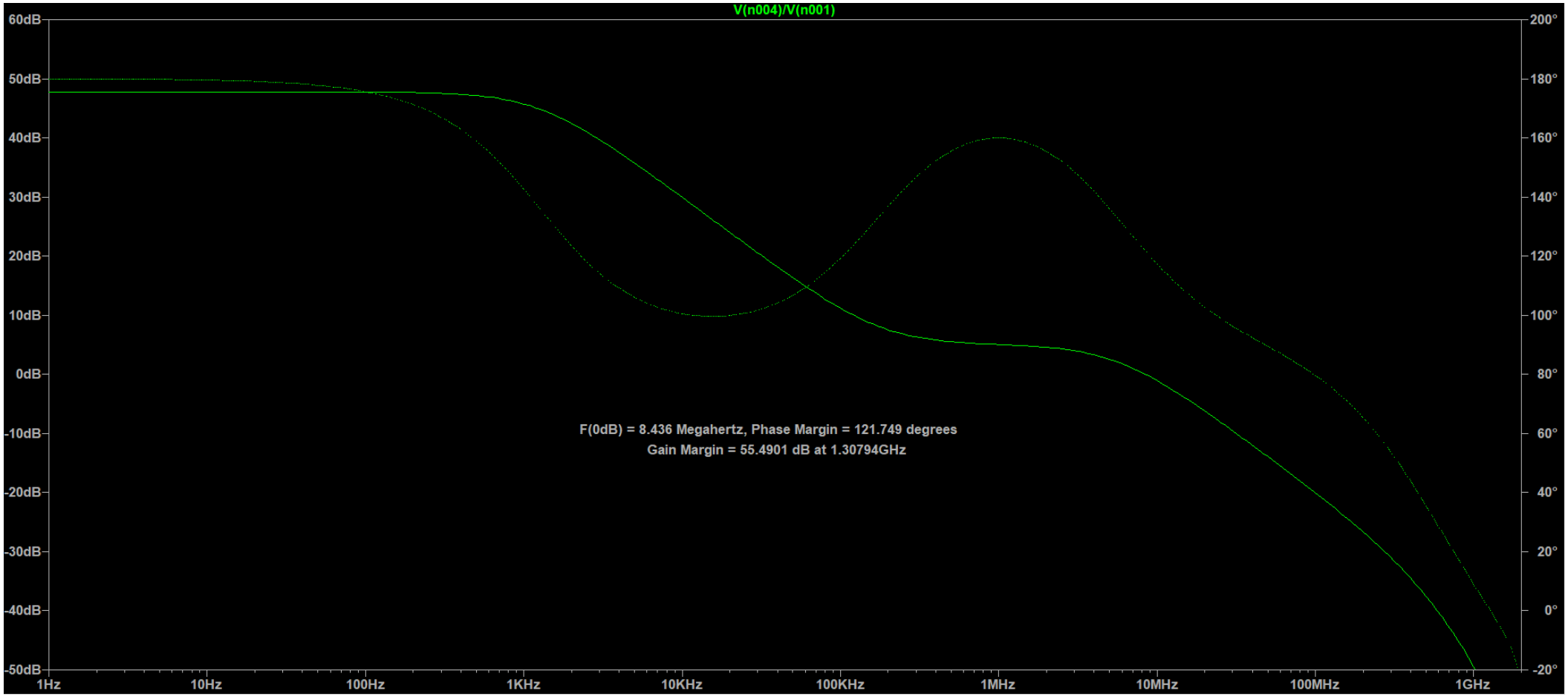
Results

Nexxim Circuit Uncompensated:



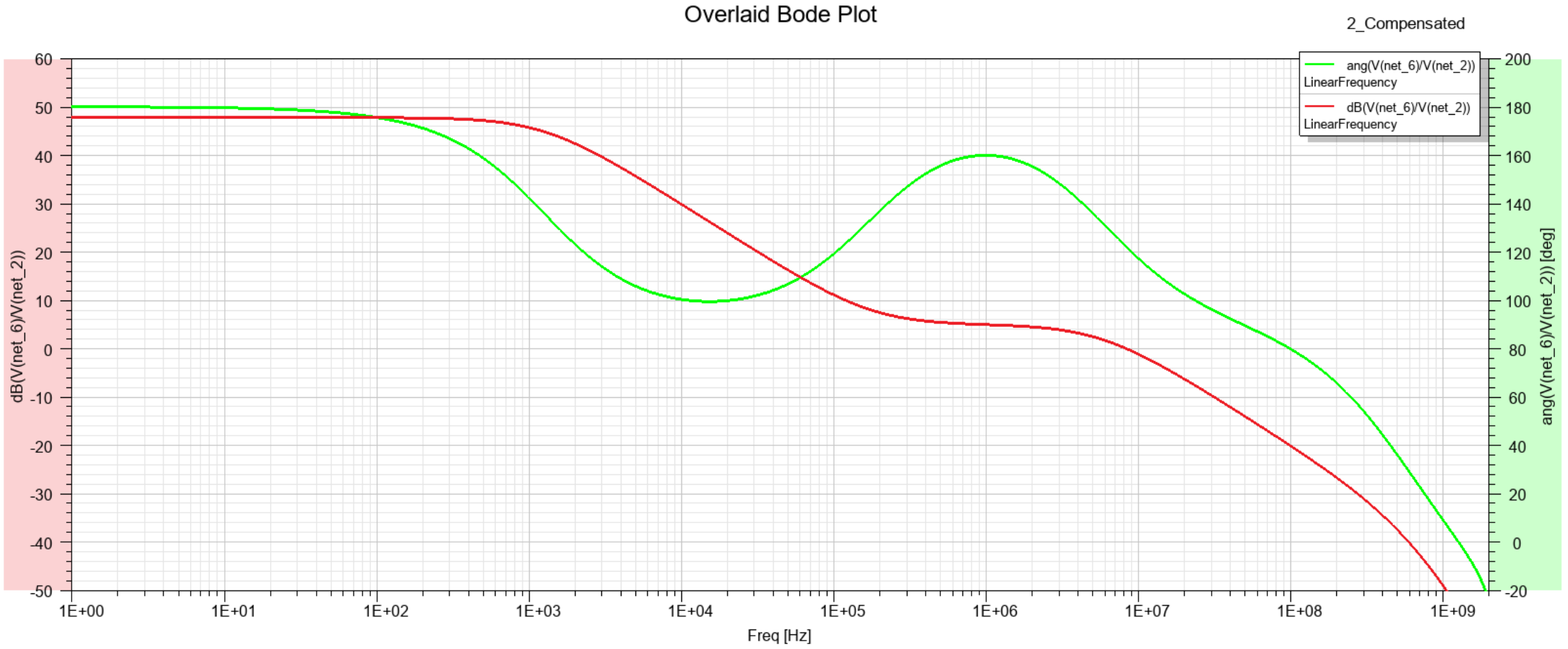
Results

LTspice Compensated:



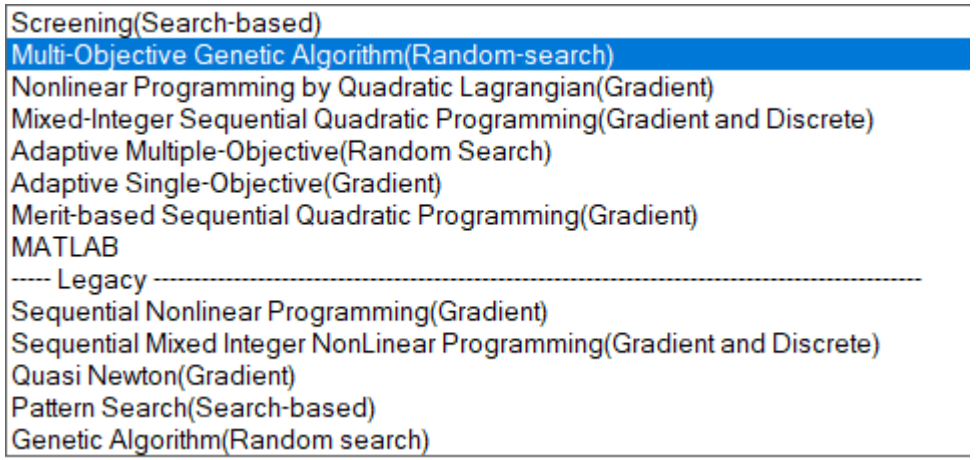
Results

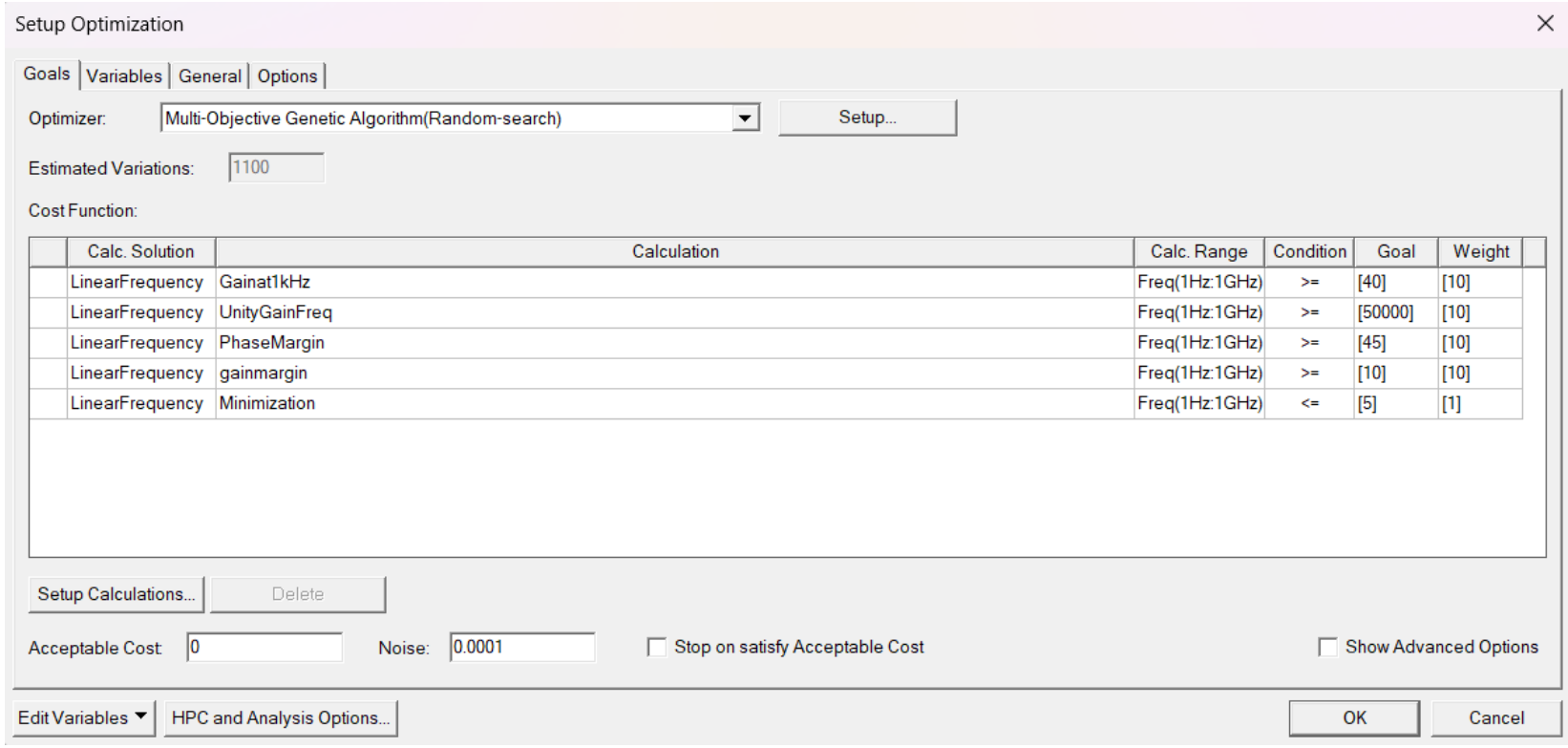
Nexxim Circuit Compensated:



Optimetrics

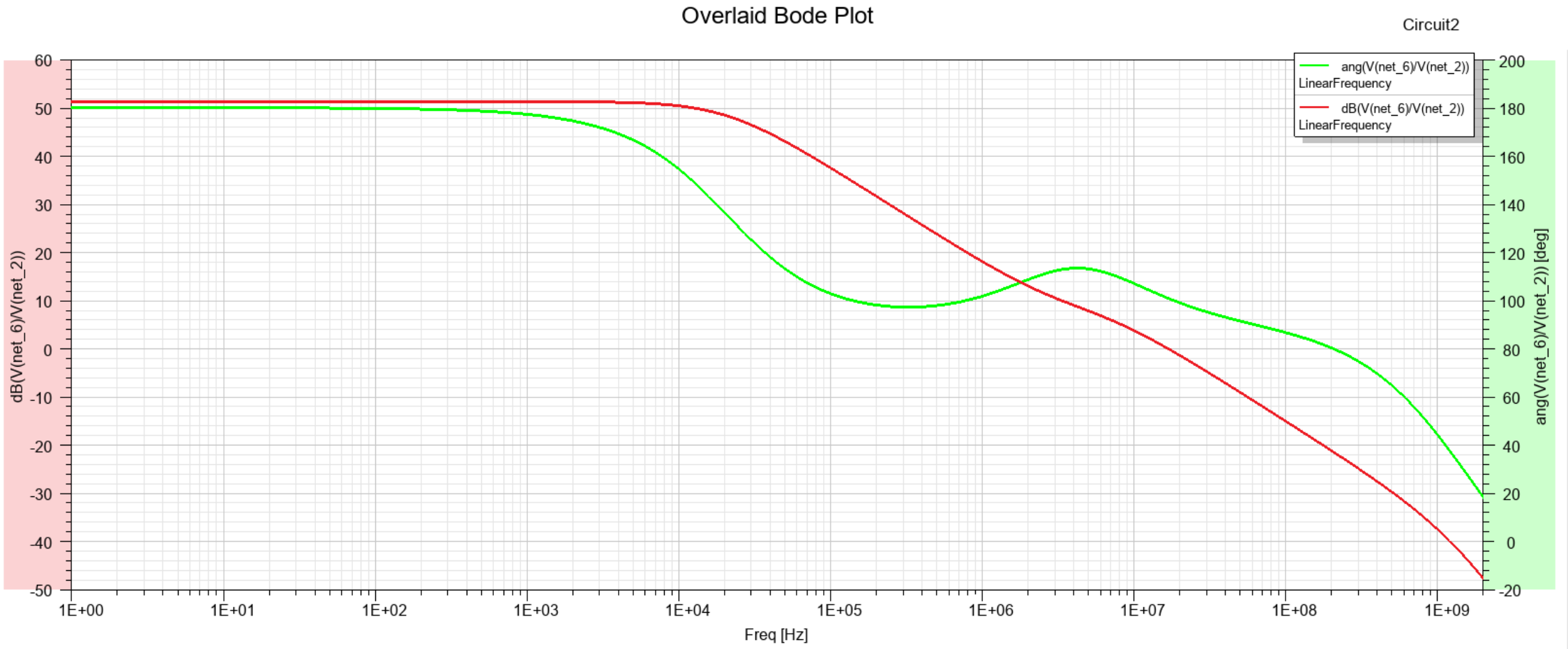
- Multiple Optimization Schemes
- Define Optimization Functions and Goals
- Assign Importance with Weight
- Minimized Dimensions and Compensation R and C





Results

Optimized Nexxim Circuit:



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:

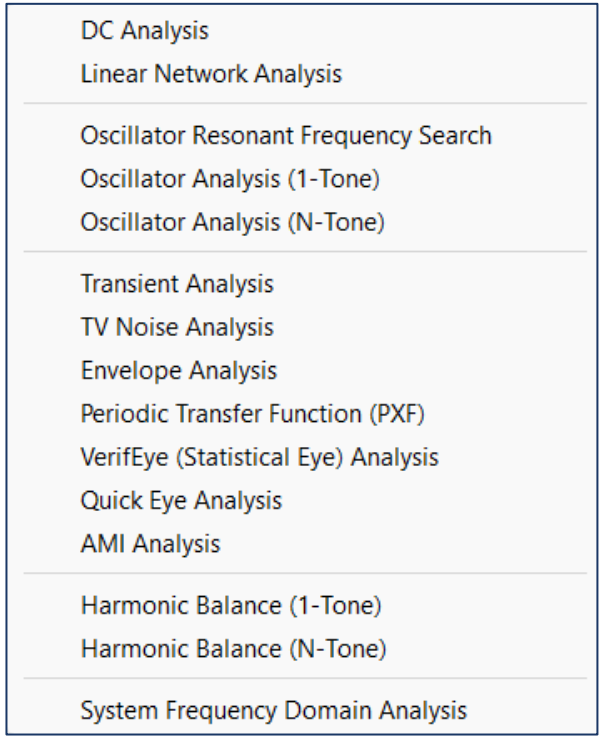
Variable	Value
Wp	0.54u
Wn	0.61u
Lp	0.51u
Ln	0.52u
Cc	1.15p
Rc	52.92k

DRD

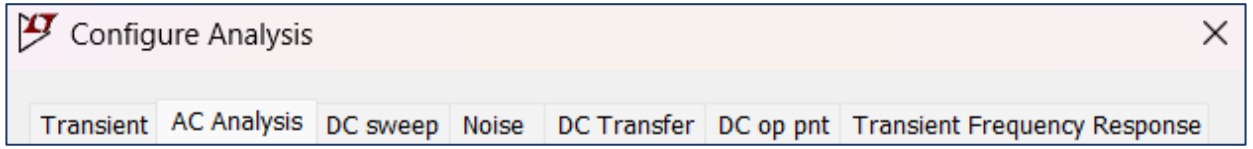
Comparison

- 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:



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