MIXED-DOMAIN MULTI-SIMULATOR STATISTICAL DEVICE MODELING AND YIELD-DRIVEN DESIGN

J.W. Bandler, R.M. Biernacki and S.H. Chen

SOS-97-8-V

August 29, 1997

MIXED-DOMAIN MULTI-SIMULATOR STATISTICAL DEVICE MODELING AND YIELD-DRIVEN DESIGN

J.W. Bandler, R.M. Biernacki and S.H. Chen

Simulation Optimization Systems Research Laboratory
and Department of Electrical and Computer Engineering
McMaster University, Hamilton, Canada L8S 4L7

Email bandler@mcmaster.ca
URL http://www.sos.mcmaster.ca

presented at

1997 Gallium Arsenide Applications Symposium, Bologna, Italy, September 1997

Introduction

statistical modeling and design are indispensable for today’s microwave CAD, especially for MMIC design

taking into account manufacturing tolerances and model uncertainties

microwave designers are forced to use different CAD systems to address different aspects of their designs; tedious and time consuming because of incompatible user interfaces and data formats

public domain SPICE does not provide means for optimization; rigid structure of commercial versions of SPICE permits only limited optimization

intelligent computational interfaces are needed

we present a flexible approach to mixed-domain, multi-simulator statistical modeling and design

we integrate time-domain, frequency-domain and EM simulations into a versatile optimization environment

Outline

Datapipe - an open architecture interface to connect various CAD systems in a uniform and user-friendly manner

mixed-domain, multi-simulator approach to device modeling and yield-driven optimization

- theory
- simple example

integrating SPICE device models, Sonnet's ***em*** electromagnetic simulations and OSA's OSA90/hope circuit level optimization

- statistical modeling of GaAs MESFETs
- yield optimization

Space Mapping optimization to align mode-matching and finite element based electromagnetic simulations

Datapipe Technique for Optimization Interface

utilizes interprocess pipe communication

establishes high speed data connections between different processes

communication with each child

Datapipe protocol at the parent side

Datapipe server at the child side

a set of communication standards defines the sequence and meaning of the data fields exchanged between the parent and the child

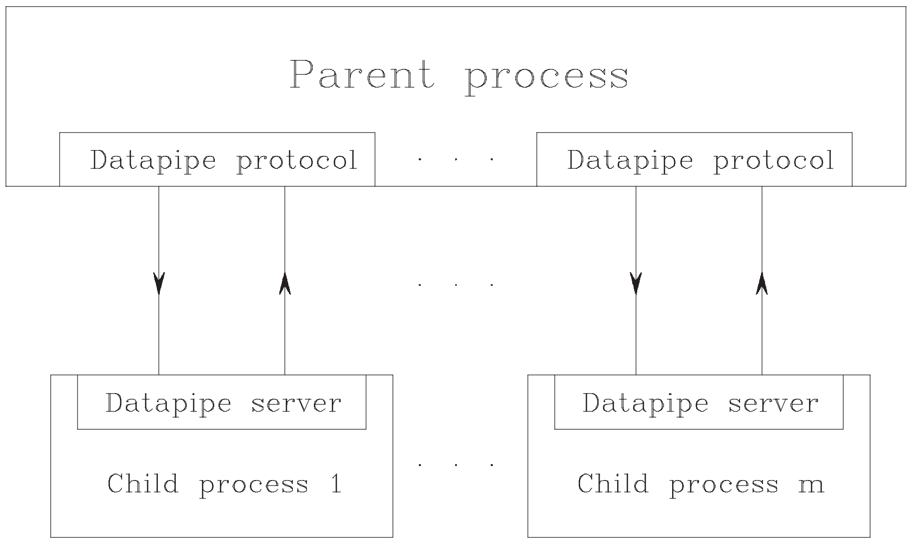
the parent and the child can be totally independent

source code does not need to be revealed (suitable for sensitive software)

no limit to the number of children connected with a single parent

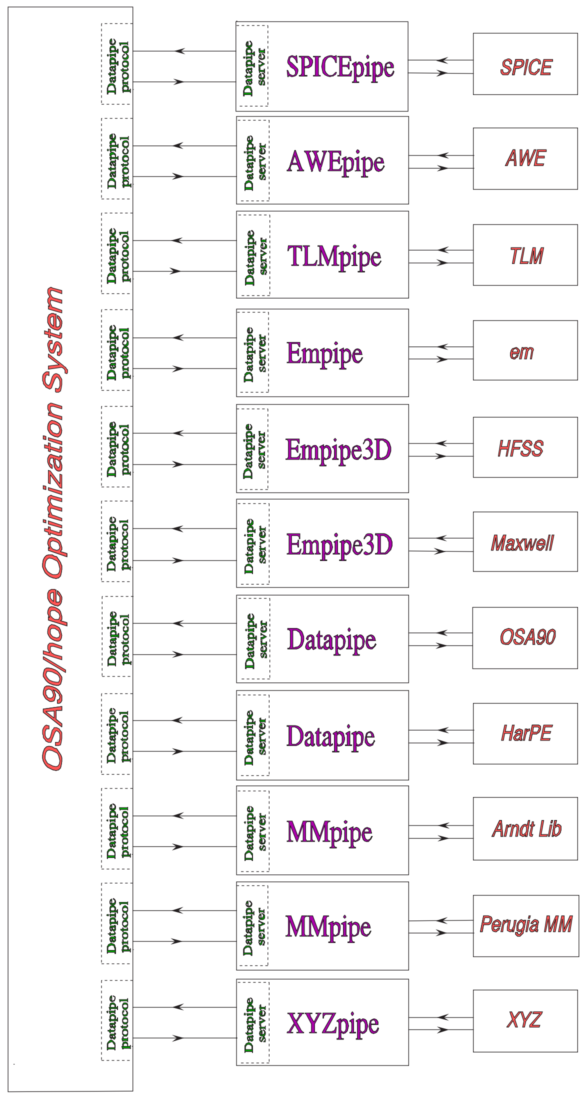
the parent and the children can communicate through a network of computers

Datapipe Schematic



Datapipe interface between a parent process and a number of child processes

OSA90/hope Connections



Datapipe interfaces between OSA90/hope and several external simulators, including SPICE (public domain), AWE (*Carleton University*), TLM (*University of Victoria*), ***em*** (*Sonnet Software*), HFSS (*Ansoft, HP-EEsof*) and RWGMM (*Arndt, University of Bremen*)

Mixed-Domain Multi-Simulator Yield-Driven Design

n_o outcomes are used in yield optimization

for all outcomes ϕ^i the parent integrates the results returned from each child and performs the circuit-level simulation

$$\boldsymbol{R}_P(\phi^i) = \boldsymbol{R}_P(\phi^i, \boldsymbol{R}_{C_1}(\phi^i), \boldsymbol{R}_{C_2}(\phi^i), \ldots, \boldsymbol{R}_{C_m}(\phi^i))$$

where

$\boldsymbol{R}_P$ circuit-level responses simulated by the parent
$\boldsymbol{R}_{C_k}$ subcircuits simulated by the *k*th child

in general

$$\boldsymbol{R}_{C_k}(\phi^i) = \boldsymbol{R}_{C_k}(\boldsymbol{R}^t_{C_k}(\phi^i), \boldsymbol{R}^f_{C_k}(\phi^i), \boldsymbol{R}^e_{C_k}(\phi^i))$$

where

$\boldsymbol{R}^t_{C_k}$ time-domain responses

$\boldsymbol{R}^f_{C_k}$ frequency-domain responses

$\boldsymbol{R}^e_{C_k}$ EM responses

each child is usually devoted to only one particular type of simulation

Formulation for Yield-Driven Design

error functions

$$e_j(\phi^i) = R_{P_j}(\phi^i) - S_j \qquad \text{upper specifications}$$

$$e_j(\phi^i) = S_j - R_{P_j}(\phi^i) \qquad \text{lower specifications}$$

for all outcomes ϕ^i and all specifications S_j, $j = 1, 2, ..., n_s$

outcome ϕ^i is acceptable if all $e_j(\phi^i)$, $j = 1, 2, ..., n_s$, are nonpositive (all design specifications are satisfied)

design yield

the ratio of acceptable outcomes to the total number of outcomes considered

yield-driven design formulation

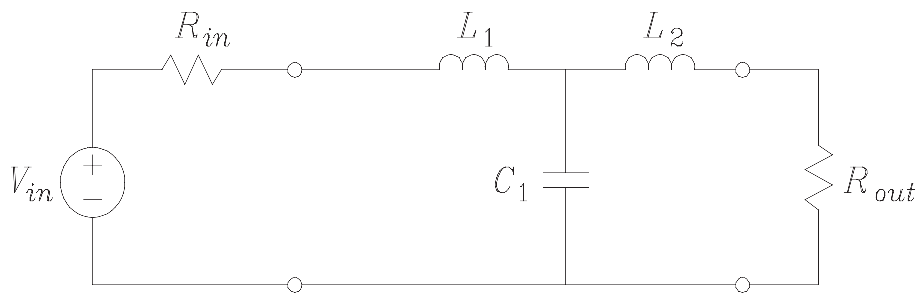
$$\underset{\phi^0}{\text{minimize}} \quad U(\phi^0) = \sum_{i=1}^{n_o} H[\alpha_i v(\phi^i)]$$

where

α_i	positive multipliers
$v(\phi^i)$	the generalized ℓ_p function
H	one-sided ℓ_1 or one-sided Huber

Mixed-Domain Multi-Simulator Yield-Driven Design: Example

a simple low-pass filter



specifications defined in the frequency domain

insertion loss ≤ 1.5 dB	for $0 < \omega < 1$
insertion loss ≥ 25 dB	for $\omega > 2.5$

and in the time domain

$0.45 \text{ V} \le V_{out} \le 0.55 \text{ V}$	for $3.5 \text{ s} < t < 20 \text{ s}$

design variables: L_1, L_2 and C_1 with a uniform distribution within a 10% tolerance

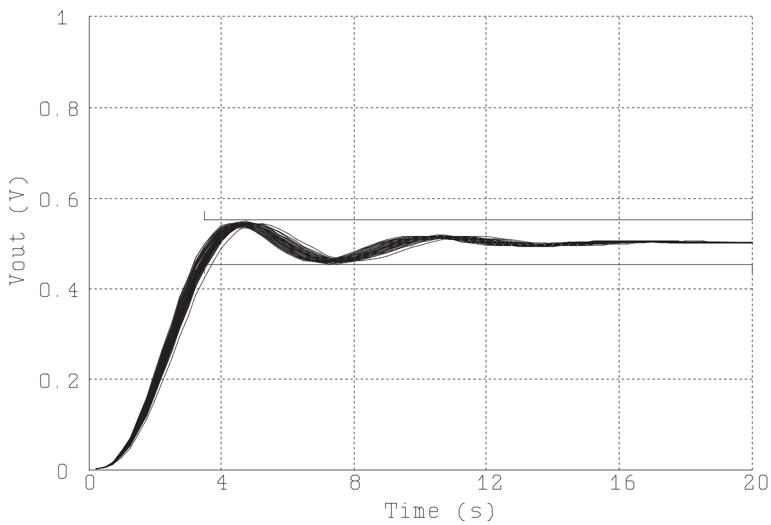
time-domain simulation performed by SPICE

frequency-domain simulation performed by OSA90/hope

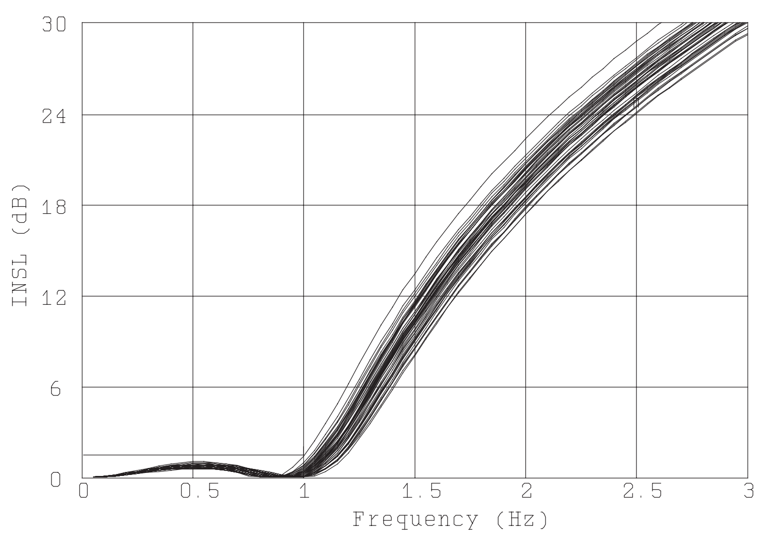
mixed-domain optimization performed by OSA90/hope nominal design followed by yield optimization

Statistical Responses of the Low-Pass Filter

time-domain Monte Carlo sweep



frequency-domain Monte Carlo sweep



yield is increased from 29% at the nominal design to 67% after optimization

Capturing SPICE Device Models

SPICE is invoked to simulate individual devices only

to obtain *S* parameters we invoke SPICE evaluation of node voltages

two SPICE simulations determine the parameters of a 2-port network

SPICE output returned to OSA90/hope may need to be postprocessed

the node voltages are converted to the *S* parameters using mathematical expressions in the OSA90/hope input file

Error Functions for Statistical Device Modeling with SPICE
(*Bandler et al., Int. J. MIMICAE, 1997*)

measurement data

$$\boldsymbol{S}^i = [S_1^i \; S_2^i \; \dots \; S_{n_i}^i]^T, \qquad i = 1, 2, \dots, n_d$$

n_d sets of data measured from n_d devices

n_i measured responses in the ith data set

the corresponding SPICE responses

$$\boldsymbol{R}_{SP}(\phi^i) = [R_{SP_1}(\phi^i) \; R_{SP_2}(\phi^i) \; \dots \; R_{SP_{n_i}}(\phi^i)]^T$$

where ϕ^i is the ith set of model parameters to be extracted

for each data set, the error vector is defined as

$$\boldsymbol{e}_{OS}(\phi^i) = [e_{OS_1}(\phi^i) \; e_{OS_2}(\phi^i) \; \dots \; e_{OS_{n_i}}(\phi^i)]^T$$

individual errors represent the equality constraints of the matching problem

$$e_{OS_j}(\phi^i) = R_{SP_j}(\phi^i) - S_j^i$$

Parameter Extraction for Individual Outcomes

$$\underset{\phi^i}{\text{minimize}} \quad U_{OS}(\phi^i)$$

U_{OS} is an objective function such as the ℓ_1, ℓ_2 or the Huber norm

Statistical Postprocessing

to derive the statistical model

for each device outcome the parameter extraction is driven by OSA90/hope’s optimizer with the device model captured from SPICE

repeated for each data set, it leads to a sample of individually extracted device models

the model statistics including the mean values, standard deviations and the correlation matrix are produced by postprocessing this sample of models

Statistical Modeling of a GaAs MESFET

measurement data for a sample of a GaAs MESFET

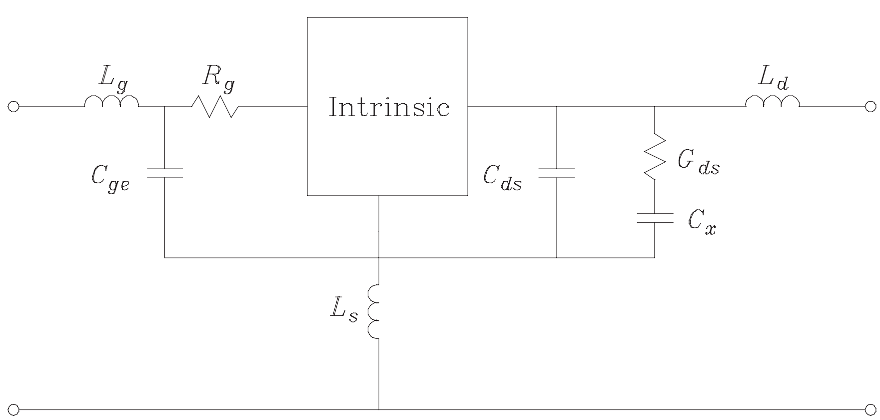
35 data sets (devices)

small-signal *S* parameters at frequencies from 1 to 21 GHz with a 2 GHz step

under two bias conditions

the wafer measurements aligned to consistent bias conditions

the equivalent circuit to model the GaAs MESFET



18 model parameters

Results of Statistical Modeling of a GaAs MESFET

MEAN VALUES AND STANDARD DEVIATIONS
FOR THE STATISTICAL SPICE MESFET MODEL

Parameter	Mean	Standard Deviation (%)
C_{gs} (pF)	0.4651	2.87
C_{gd} (pF)	0.0293	2.52
λ (1/V)	4.046×10^{-3}	9.75
V_{to} (V)	-2.4863	5.32
β (A/V^2)	0.0135	5.64
B (1/V)	2.3032×10^{-3}	9.44
α (1/V)	1.9413	7.61
R_d (Ω)	0.0111	8.35
R_s (Ω)	6.5941	5.15
PB (V)	0.6279	7.80
R_g (Ω)	3.7129	6.62
G_{ds} (1/Ω)	3.5593×10^{-3}	2.28
C_{ds} (pF)	0.0485	2.50
L_g (nH)	0.0306	7.97
L_d (nH)	0.0783	9.11
L_s (nH)	0.0344	3.40
C_{ge} (pF)	0.0379	9.96
C_x (pF)	20.0	-

C_{gs} through PB are the intrinsic SPICE MESFET parameters.
R_g through C_x are extrinsic parameters. C_x is non-statistical.

Verification of the Statistical Model

the complete statistics include the mean values, standard deviations, discrete density functions (DDF) and the correlation matrix

comparing the statistics of the model responses estimated by Monte Carlo simulation with those of the data

S parameters at 11 GHz

bias point: V_G = -0.5 V, V_D = 5 V

MEAN VALUES AND STANDARD DEVIATIONS OF
DATA AND SPICE MODEL RESPONSES

	Data		SPICE MODEL	
	Mean	Dev. (%)	Mean	Dev. (%)
$Re\{S_{11}\}$	-0.197	9.18	-0.192	12.5
$Im\{S_{11}\}$	-0.756	1.1	-0.747	1.07
$Re\{S_{12}\}$	0.0733	2.7	0.0770	3.1
$Im\{S_{12}\}$	0.0519	2.36	0.0527	4.89
$Re\{S_{21}\}$	-0.212	8.35	-0.432	15.2
$Im\{S_{21}\}$	1.78	1.22	1.736	8.71
$Re\{S_{22}\}$	0.440	1.43	0.434	3.33
$Im\{S_{22}\}$	-0.364	0.89	-0.364	0.96
I_d (A)	0.0401	8.16	0.0407	14.7

Verification of the Statistical Model (cont'd)

S parameters at 11 GHz

bias point: V_G = -0.7 V, V_D = 5 V

MEAN VALUES AND STANDARD DEVIATIONS OF
DATA AND SPICE MODEL RESPONSES

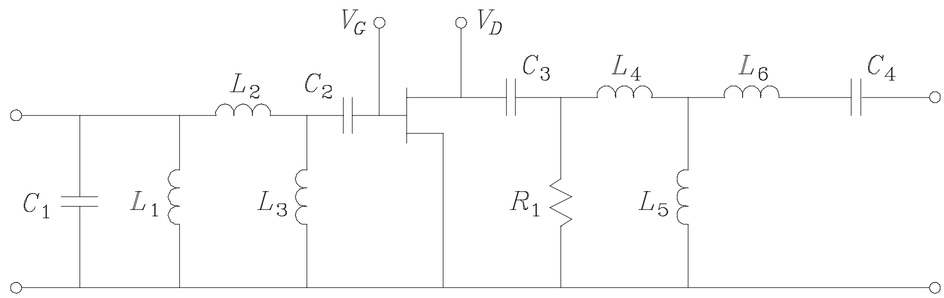
	Data		SPICE MODEL	
	Mean	Dev. (%)	Mean	Dev. (%)
$Re\{S_{11}\}$	-0.153	12.1	-0.170	13.7
$Im\{S_{11}\}$	-0.764	1.0	-0.760	1.01
$Re\{S_{12}\}$	0.0770	2.71	0.0784	2.93
$Im\{S_{12}\}$	0.0559	2.46	-0.054	4.68
$Re\{S_{21}\}$	-0.230	6.99	-0.433	15.3
$Im\{S_{21}\}$	1.687	1.67	1.650	9.22
$Re\{S_{22}\}$	0.439	1.44	0.442	3.27
$Im\{S_{22}\}$	-0.367	0.89	-0.366	0.97
I_d (A)	0.0332	9.51	0.0338	16.1

very good agreement between data and the model responses for the mean values

some discrepancies in standard deviations are likely due the inadequate statistical modeling capabilities of equivalent circuit models

Yield-Driven Design of an Amplifier

a small-signal amplifier



design specifications (for frequencies from 8 to 12 GHz)

> $7.25 \text{ dB} < |S_{21}| < 8.75 \text{ dB}$
> $|S_{11}| < 0.5$
> $|S_{22}| < 0.5$

design variables

> the matching circuit elements L_1, L_2, L_3, L_4, L_5, L_6, C_1, C_2, C_3, C_4 and R_1

28 statistical parameters

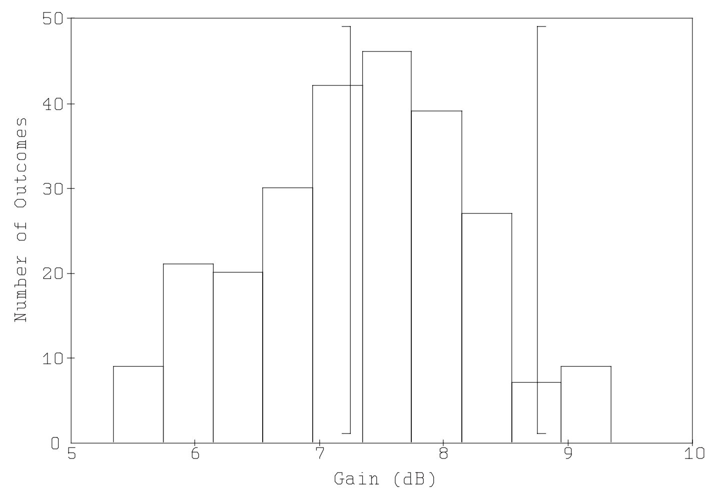
uniform distribution within a 5% tolerance

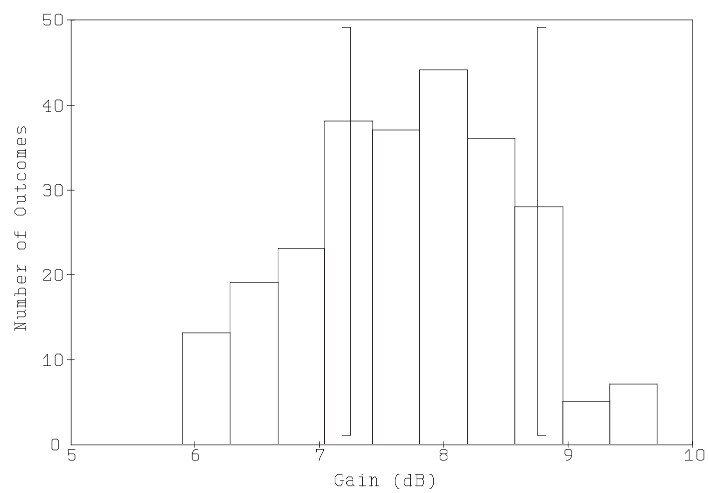
the MESFET is simulated in SPICE

SPICE results are returned to OSA90/hope through Spicepipe for circuit-level simulation and optimization

Histograms Before and After Yield Optimization

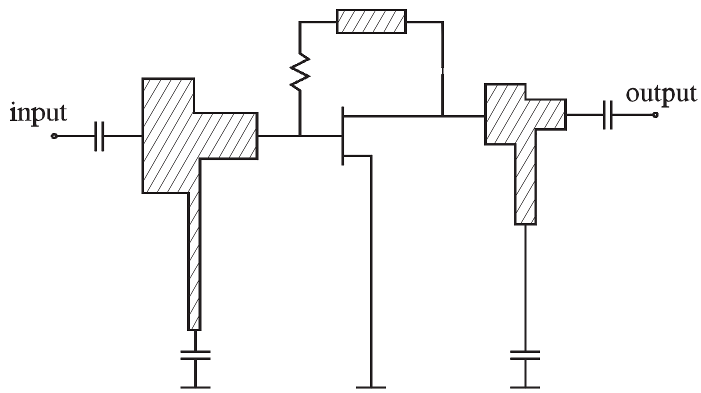
$|S_{21}|$ at 12 GHz



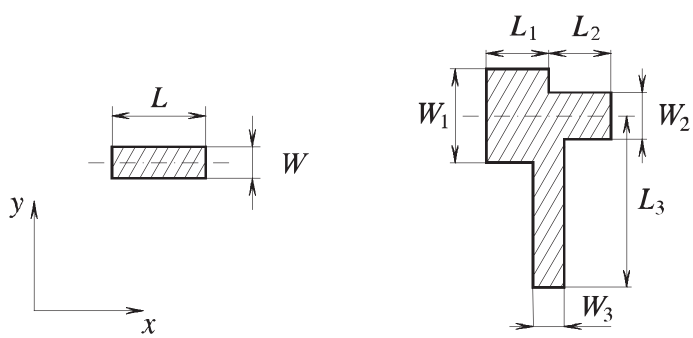


yield is increased from 16% at the nominal design to 52% after optimization

A Broadband Small-Signal Amplifier



parameters of the feedback microstrip line and the microstrip *T*-structures



design specifications

$$7\text{ dB} \le |S_{21}| \le 8\text{ dB} \quad \text{for} \quad 6\text{ GHz} \le f \le 18\text{ GHz}$$

design variables

gate *T*-structure: W_{g1}, L_{g1}, W_{g2}, L_{g2}

drain *T*-structure: W_{d1}, L_{d1}, W_{d2}, L_{d2}

Combined *em*/SPICE Yield-Driven Design
(*Bandler et al., Int. J. MIMICAE, 1997*)

the MESFET simulated by SPICE

microstrip components accurately simulated by Sonnet's ***em***

the line and the *T*-structure primitives of the Empipe library are invoked

circuit-level simulation and optimization carried out by OSA90/hope

uniform distribution within a 0.5 mil tolerance for all geometrical parameters

yield at the nominal minimax solution is 43%

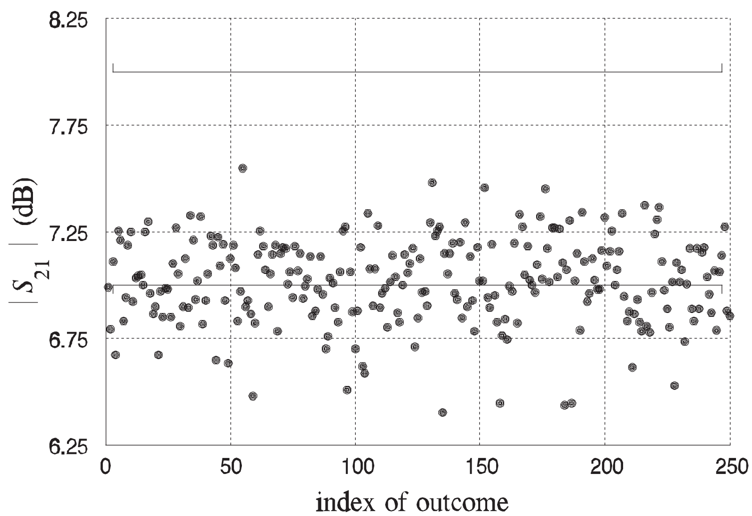
yield is increased to 74% after yield optimization

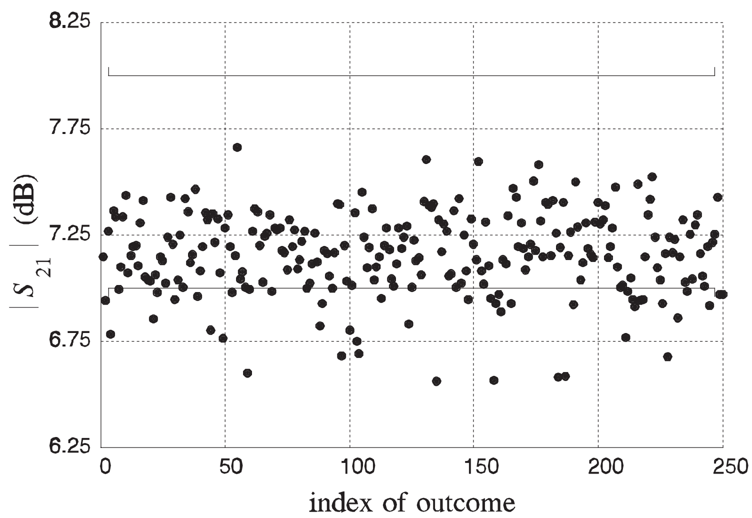
50 outcomes used for yield optimization

Run Charts Before and After Yield Optimization

$|S_{21}|$ at 18 GHz

250 outcomes





clearly, many more outcomes meet the specification after yield optimization

Space Mapping Optimization

to avoid direct optimization of computationally intensive models

the multi-simulator approach is particularly relevant and suitable for Space Mapping

automatic alignment of two distinct models of different accuracy and computational efficiency

such models would normally be facilitated by two disjoint simulators

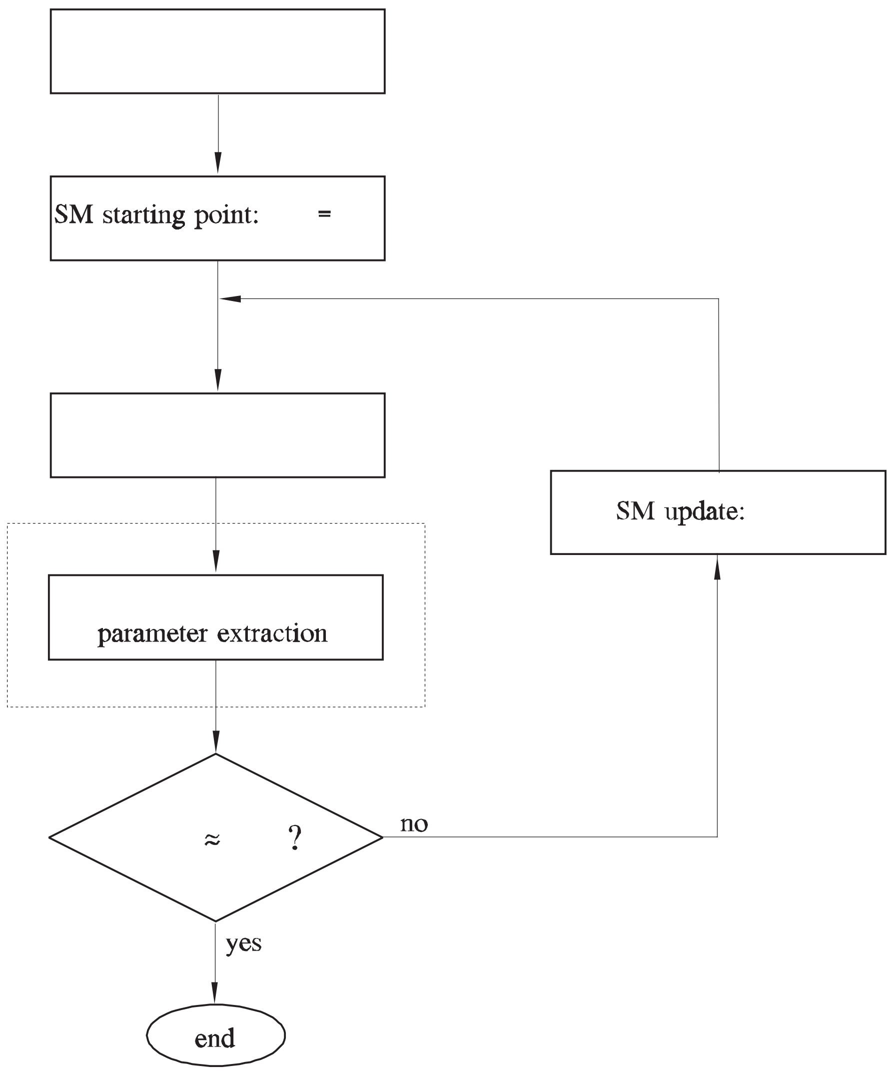
two different EM simulators are used here

EM space or "fine" model - 3D FEM-based field simulator Maxwell Eminence (*Ansoft Corp.*)

optimization space (OS) or "coarse" model - the RWGMM library of waveguide mode-matching (MM) models connected by network theory (*Fritz Arndt*)

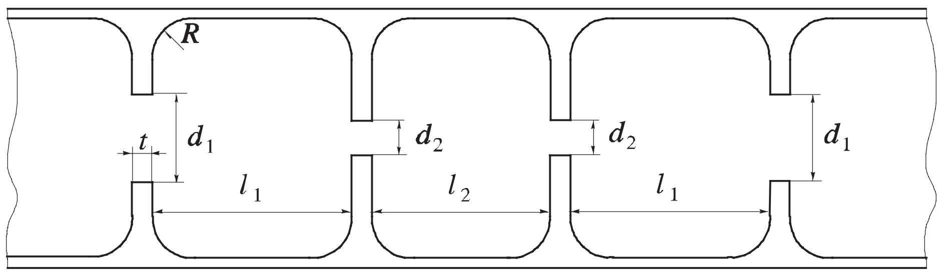
Space Mapping Using MM/Network Theory and FEM
(*Bandler et al., 1997 Int. Microwave Symp.*)

flow diagram of Space Mapping concurrently exploiting the hybrid MM/network theory and FEM simulation techniques



two-level Datapipe architecture

Optimization of an H-Plane Resonator Filter



the waveguide cross-section: 15.8 × 7.9 mm

iris and corner radius: t = 0.4 mm, R = 1 mm

design variables

d_1, d_2, l_1 and l_2

design specifications

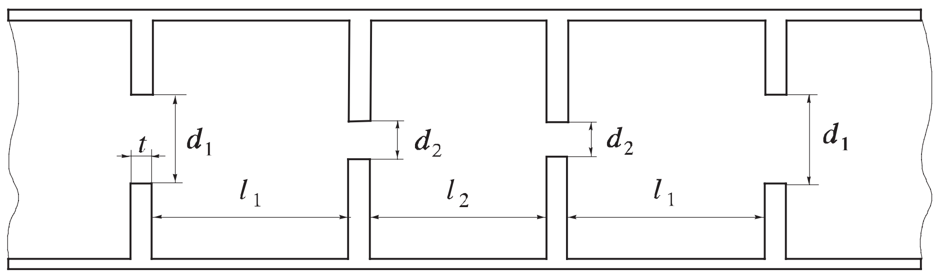
$|S_{21}|$ < -35 dB for $13.5 \le f \le 13.6$ GHz
$|S_{11}|$ < -20 dB for $14.0 \le f \le 14.2$ GHz
$|S_{21}|$ < -35 dB for $14.6 \le f \le 14.8$ GHz

FEM analysis - fine (or EM) model for Space Mapping

capable of analyzing arbitrary shapes

computationally very intensive

Coarse Model for Space Mapping Optimization



OS model (coarse model) for Space Mapping

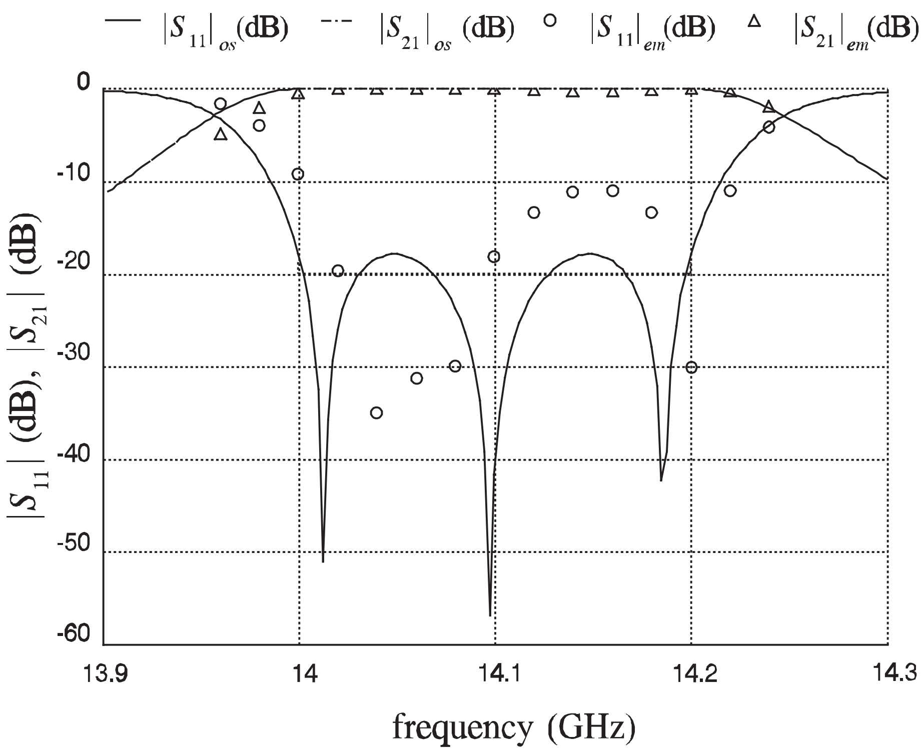
sharp corners

hybrid MM/network theory simulation

- computationally efficient
- accurately treats a variety of predefined geometries
- ideally suited for modeling complex waveguide structures decomposable into available library building blocks

minimax optimization of the OS model gives the starting point for Space Mapping

Responses at the Starting Point



focus on the passband: 13.96 to 14.24 GHz

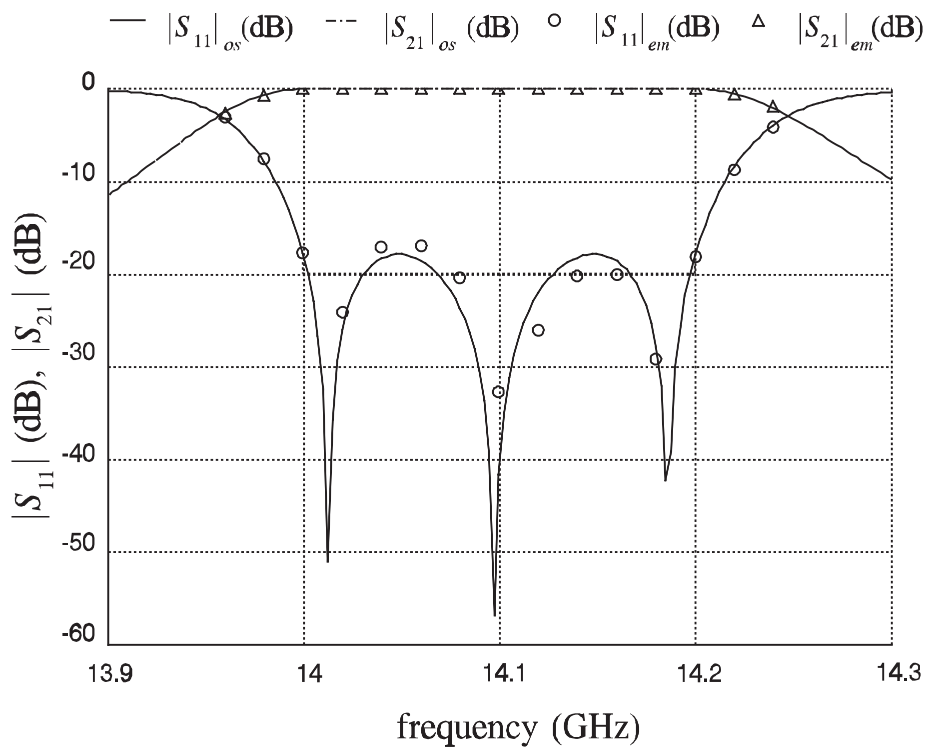
RWGMM (curves) and Maxwell Eminence (points)

discrepancy is evident

d_1 = 6.04541, d_2 = 3.21811, l_1 = 13.0688 and l_2 = 13.8841

the minimax solution in the OS space, $\boldsymbol{x}_{os}^*$, yields the target response for Space Mapping

SM Optimized FEM Responses



only 4 Maxwell Eminence simulations

RWGMM (curves) and Maxwell Eminence (points)

very good match

$d_1 = 6.17557$, $d_2 = 3.29058$, $l_1 = 13.0282$ and $l_2 = 13.8841$

direct optimization using Empipe3D confirms that the Space Mapping solution is indeed optimal

Conclusions

based on the Datapipe open architecture, intelligent computational interfaces combine and enhance the features of otherwise disjoint simulators

time-domain, frequency-domain and EM simulations are integrated for efficient statistical modeling and design

mixed-domain specifications illustrated by design of a simple low-pass filter with specifications in both the time and frequency domains

statistical device modeling: SPICE models captured by OSA90/hope’s optimizers

design of a broadband amplifier with microstrip components: the MESFET is simulated by SPICE and the microstrip components are analyzed by ***em***

further advantages of the multi-simulator approach are exemplified by Space Mapping optimization with two different EM simulators: mode-matching and finite element