Computational Electromagnetics
Research Laboratory
University of Victoria, Canada

Faustus
Scientific Corporation

# A Multi-Level Modeling Environment for High-Frequency/High-Speed Structures and Materials

Wolfgang J. R. Hoefer

Professor
University of Victoria

President and CEO
Faustus Scientific Corporation

IEEE Lecture, University of Toronto, 28 January 2004

# Contact Information

Dept. of El. & Comp. Engineering
P.O. Box 3055
University of Victoria,
Victoria, BC, V8W 3P6
Ph. (250) 721-6030 F: -6230
whoefer@ece.uvic.ca

Faustus Scientific Corporation
1256 Beach Drive
Victoria, BC, V8S 2N3
Ph. (250) 596-6554 F: -6557
hoefer@faustcorp.com

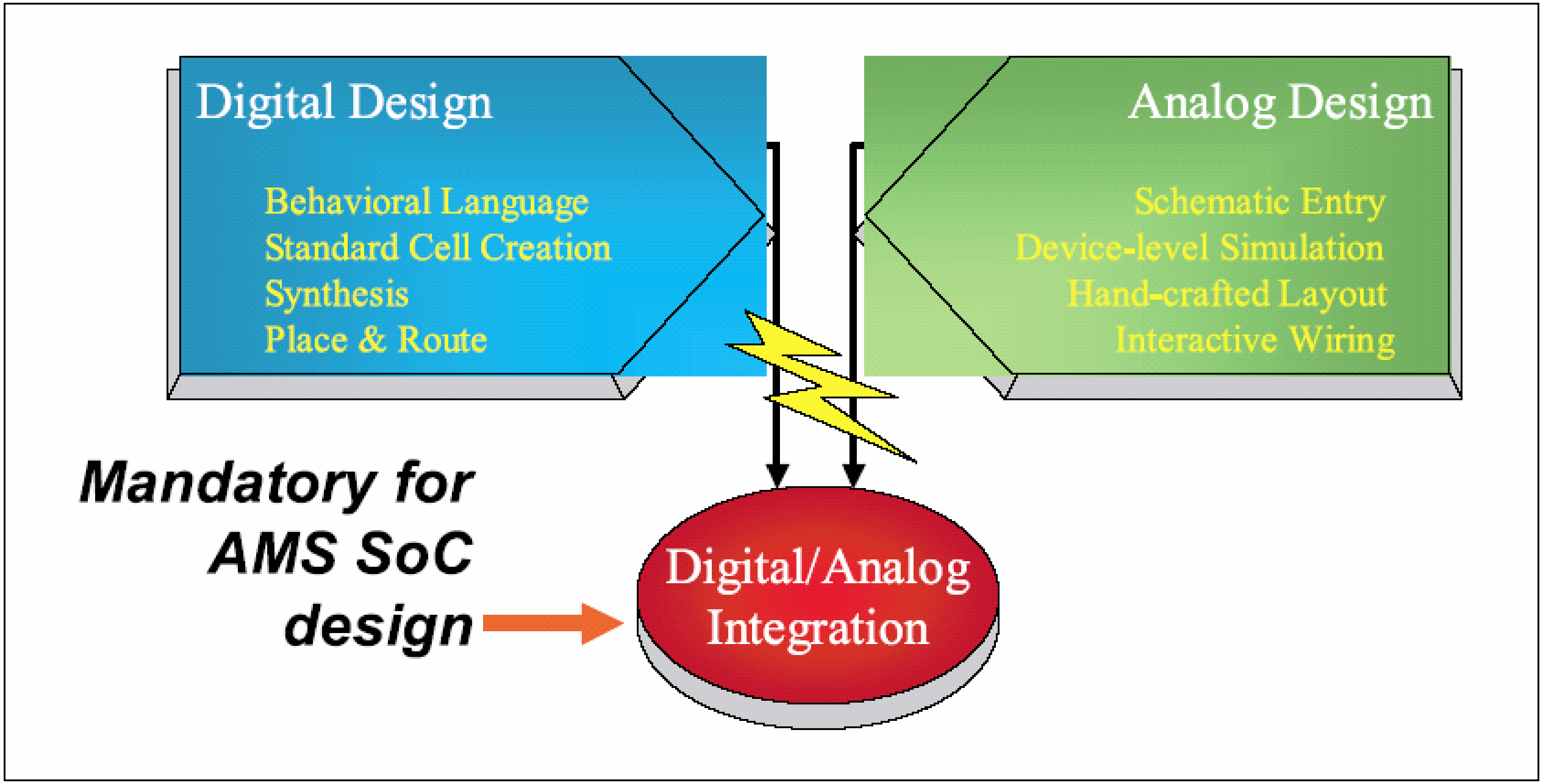
# Outline

- Introduction and Context
- Design and Verification of Complex Systems
- Modeling Requirements
- Modeling Features already developed
- Modeling Features to be developed
- Modeling Examples and Demonstrations
- Summary

# Introduction & Context

- Efficient design of complex systems, in particular mixed signal systems, require a multi-level approach;
- The system level design requires a modeling architecture in which functional blocks are described by their behavior (Mixed-Signal Hardware Description Languages or MS-HDLs) (Verilog- or VHDL-MLDS , Matlab, Simulink, etc);
- Behavioral requirements of blocks determine physical design specifications;
- System verification requires accurate extraction of the real physical behavior of functional block design to
  - test them at the system level
  - evaluate their impact on overall system behavior.

# Digital & Analog Integration



**Digital Design**

Behavioral Language
Standard Cell Creation
Synthesis
Place & Route

**Analog Design**

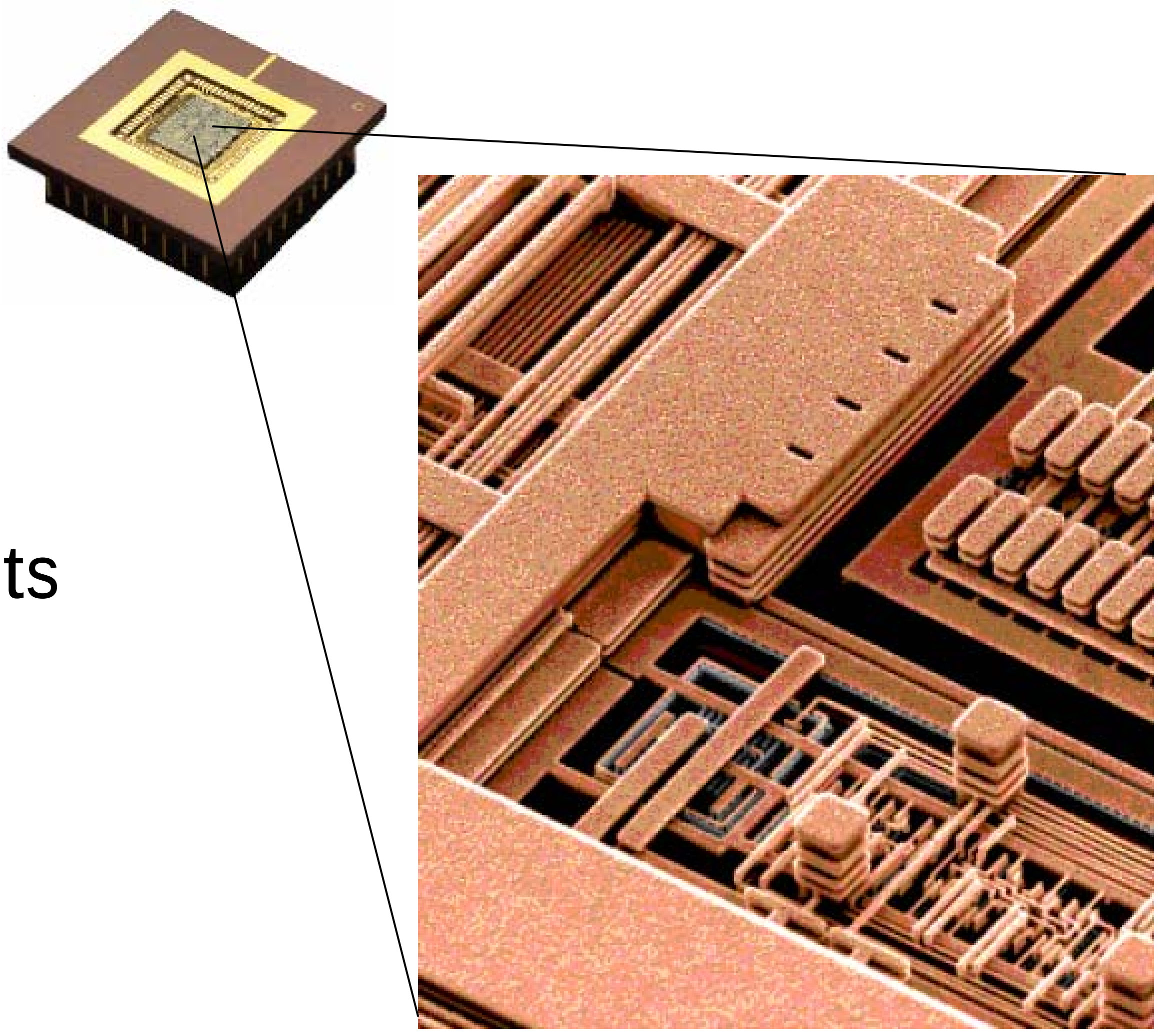
Schematic Entry
Device-level Simulation
Hand-crafted Layout
Interactive Wiring

**Digital/Analog Integration**

***Mandatory for AMS SoC design***

Historically, tools and methodologies for design of digital and analog circuits have been very different. Design is done in isolation, patched together at layout and tested after manufacturing. (From H.Y. Chang, Mixed signal — spearheading the need for improved design methodologies, RF Design, Oct. 2002)

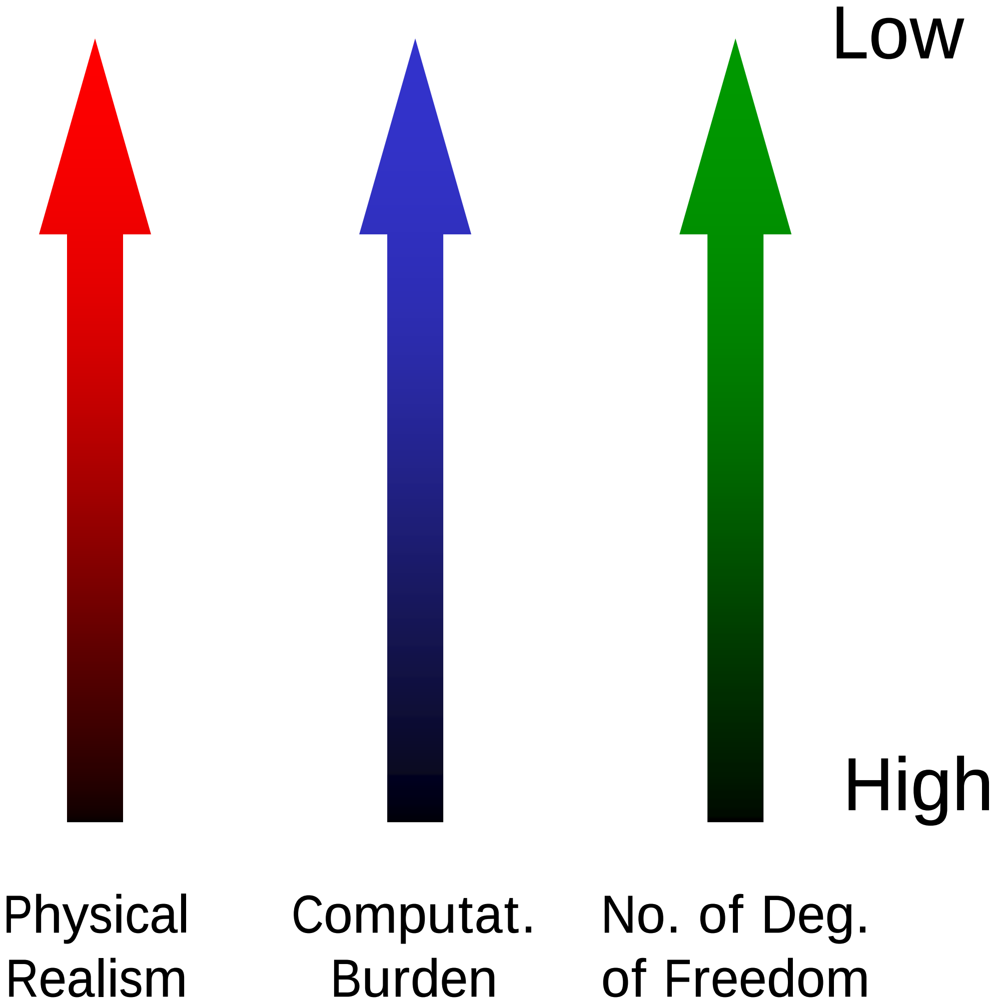
# Mixed Signal Chips



- Logic Gates
- Devices
- Interconnects
- Thermal Effects
- Packaging Effects
- Signal Integrity
- EMC, Crosstalk
- Power Supply
- Reliability
- etc……

# Modeling Levels

- System Level
  (Behavioral)
- Network Level
- Circuit Level
- Field Level
  (Physical)


Low

High

Physical Realism

Computat. Burden

No. of Deg. of Freedom

# Top-Down Design

- The system architect divides a complex system into functional blocks;
- Each block is described in behavioral terms and modeled at a high level of abstraction to design system functionality;
- The desired behavior of each functional block serves as the specifications for its respective physical design.
- Individual blocks are designed at the circuit and field levels to meet these specifications

# Bottom-up Verification

- Model and simulate a block as implemented, at the field or circuit level, including thermal effects;
- Include possible parasitic interactions with other blocks (EMC, package effects, etc.);
- Extract a reduced-order model that reflects the actual behavior of the block and its interactions;
- Introduce this realistic behavioral model into the system architecture model.

# Our Research Objectives

- Create a flexible multi-level modeling environment at the field, network, circuit, and behavioral levels;
- Describe different parts of a complex structure by the most appropriate model of lowest order;
- Maintain a two-way correspondence between the behavior and the physics throughout the modeling hierarchy;
- Include coupled thermal, mechanical and mass transfer processes.

# Our Aims and Goals

- Minimize the number of unknowns (degrees of freedom) at each modeling level.
  - Key Concept: Model Order Reduction
- Combine and integrate all modeling levels.
  - Key Concept: Hybridization
- Preserve a two-way correspondence across all modeling levels
  - Key Concepts: Model Extraction and Structure Synthesis

# Simulation Environment

- Lumped element and circuit embedding
- Sub-Cell modeling and embedding
- Time Domain Diakoptics (Convolution)
- Equivalent circuit extraction from field models
- Neural network design and training
- Integration of em/thermal/mt processes
- Fast optimization using space mapping
- Field synthesis by monochromatic injection

# Some Selected Examples

- Time Domain Diakoptics (Convolution)
- SPICE models embedded into a field model
- Field model embedded into a SPICE circuit
- Circuit elements embedded into a field simulator (metamaterials)
- Integration of field and thermal simulation

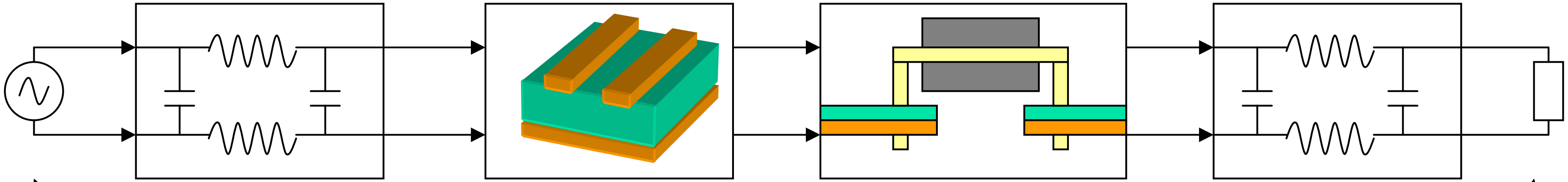
# TLM Modeling Environment

- TLM is a discrete network model of the 3D Maxwell Equations;
- It thus represents a high-order network model that can be operated in both time and frequency domains;
- TD analysis of a TLM network involves a sequence of impulse scattering and transmission (connection) operations;
- We use TLM exclusively in time domain.

# Signal Integrity Simulation

Courtesy of Dan Gorcea, Nortel Networks

**Generator IC** **Propagation medium** **Receiver IC**



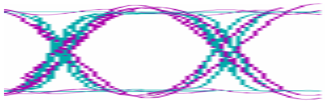
Spice models with package parasitics

Layout

Connectors, etc.

Spice models with package parasitics

Input test signal



Output time domain or eye diagram

- Analyzing only separate blocks is not enough
- The whole chain must be simulated to properly capture interactions between blocks

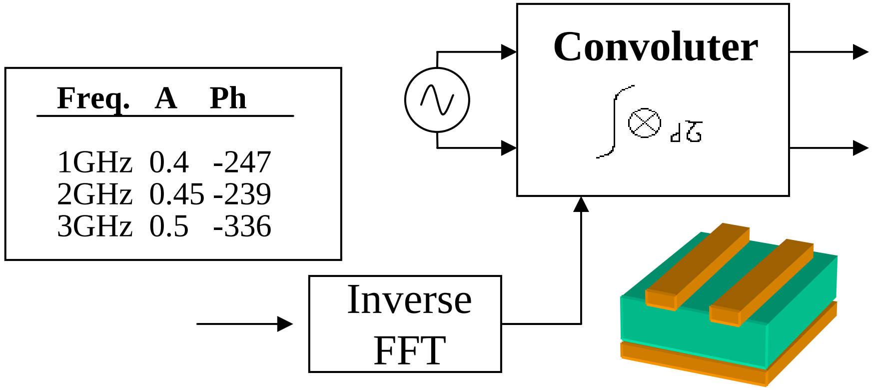
# Issues with FD Solvers

Courtesy of Dan Gorcea, Nortel Networks

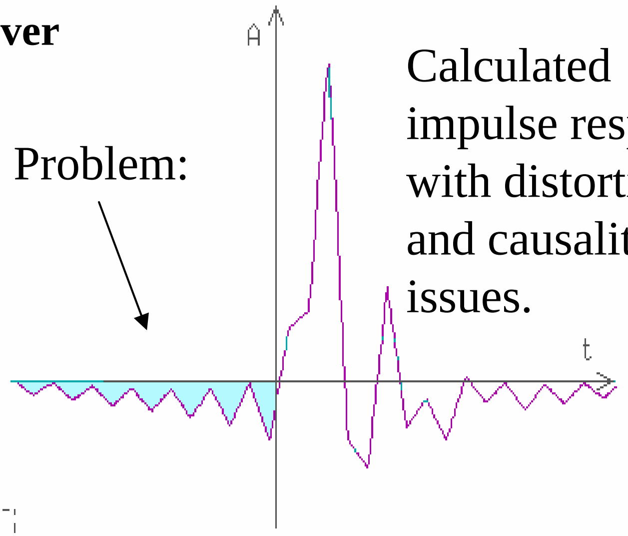
- Problem spots inside a structure cannot be identified from its time-harmonic solution;
- Extra work is involved in processing/modifying data in order to make it compatible with time domain analysis;
- The impulse response of a structure calculated from limited information from frequency domain is distorted and non-causal.
- Higher modes of propagation must be specified from the start. Results are obtained separately for each mode;
- One cannot probe electric field values arbitrarily inside the structure without creating electric ports;
- Multiple simulation passes are needed, one for each frequency.

# FD Simulation and Causality

**S parameter table obtained with Frequency Domain Solver**



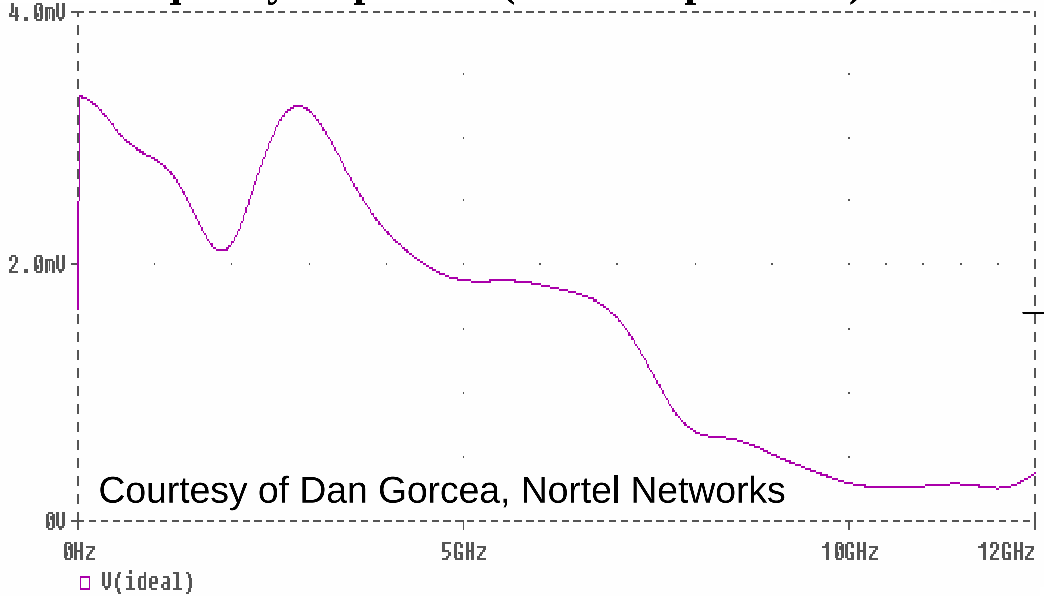
| Freq. | A | Ph |
|---|---|---|
| 1GHz | 0.4 | -247 |
| 2GHz | 0.45 | -239 |
| 3GHz | 0.5 | -336 |



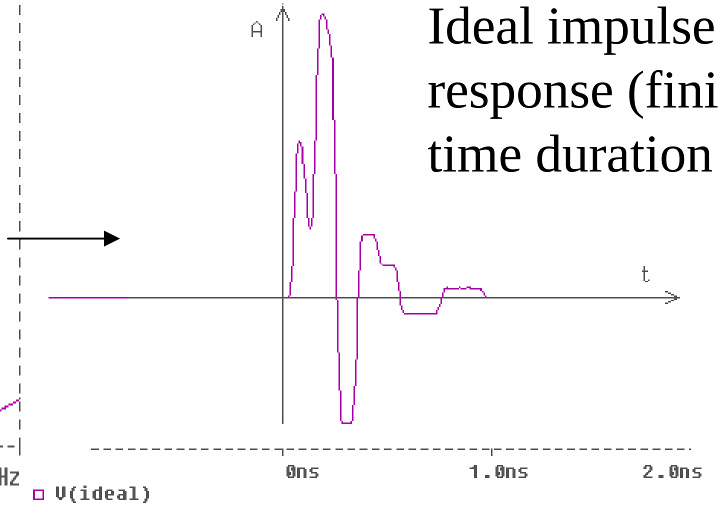
Problem:

Calculated impulse response with distortion and causality issues.

**Ideal frequency response ( infinite spectrum )**



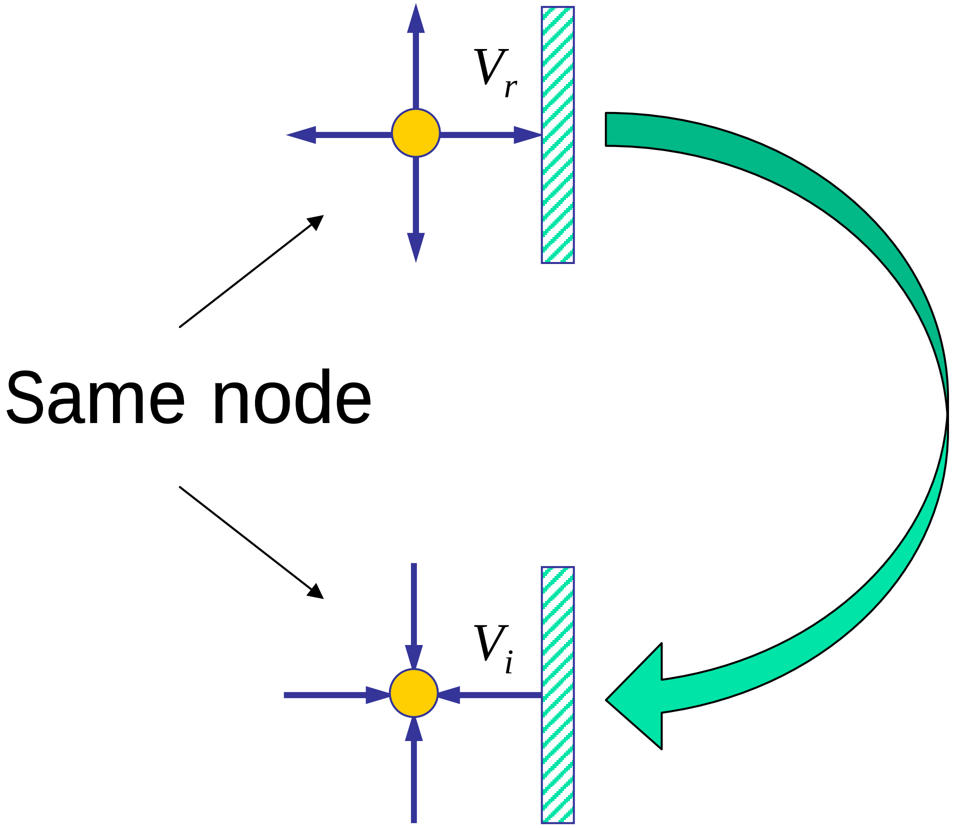
Courtesy of Dan Gorcea, Nortel Networks



Ideal impulse response (finite time duration )

# TLM Modeling Environment

## TLM boundary operation framework



Same node

$$V^i = \Gamma_{impulse} \times V^r$$

$$V^i = h(k) * V^r(k)$$

$$\vdots$$

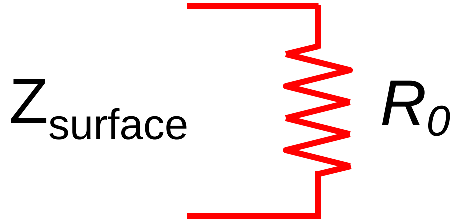
$$V^i = SPICE\left(V^r(k)\right)$$

$$V^i = \mathrm{Thermal}\left(V^r(k)\right)$$

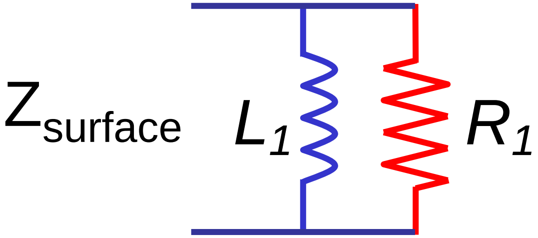
# Dispersive Boundaries

- Dispersive boundaries have reflection coefficients that depend on frequency and angle of incidence (Complex $\Gamma$);
- Examples are: Lossy conductors (skin effect), general absorbing boundaries, frequency dispersive surfaces, interfaces between subdomains, etc.;
- Dispersion is not applied to the individual TLM impulses, but results in a sequence of reflected impulses in response to each individual incident impulse.

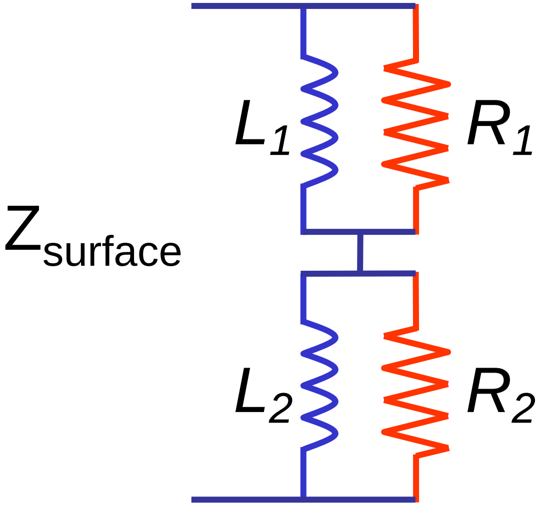
# Skin Effect



Zero order model of lossy boundary:
The surface impedance is approximated by a resistance $R_0$ .



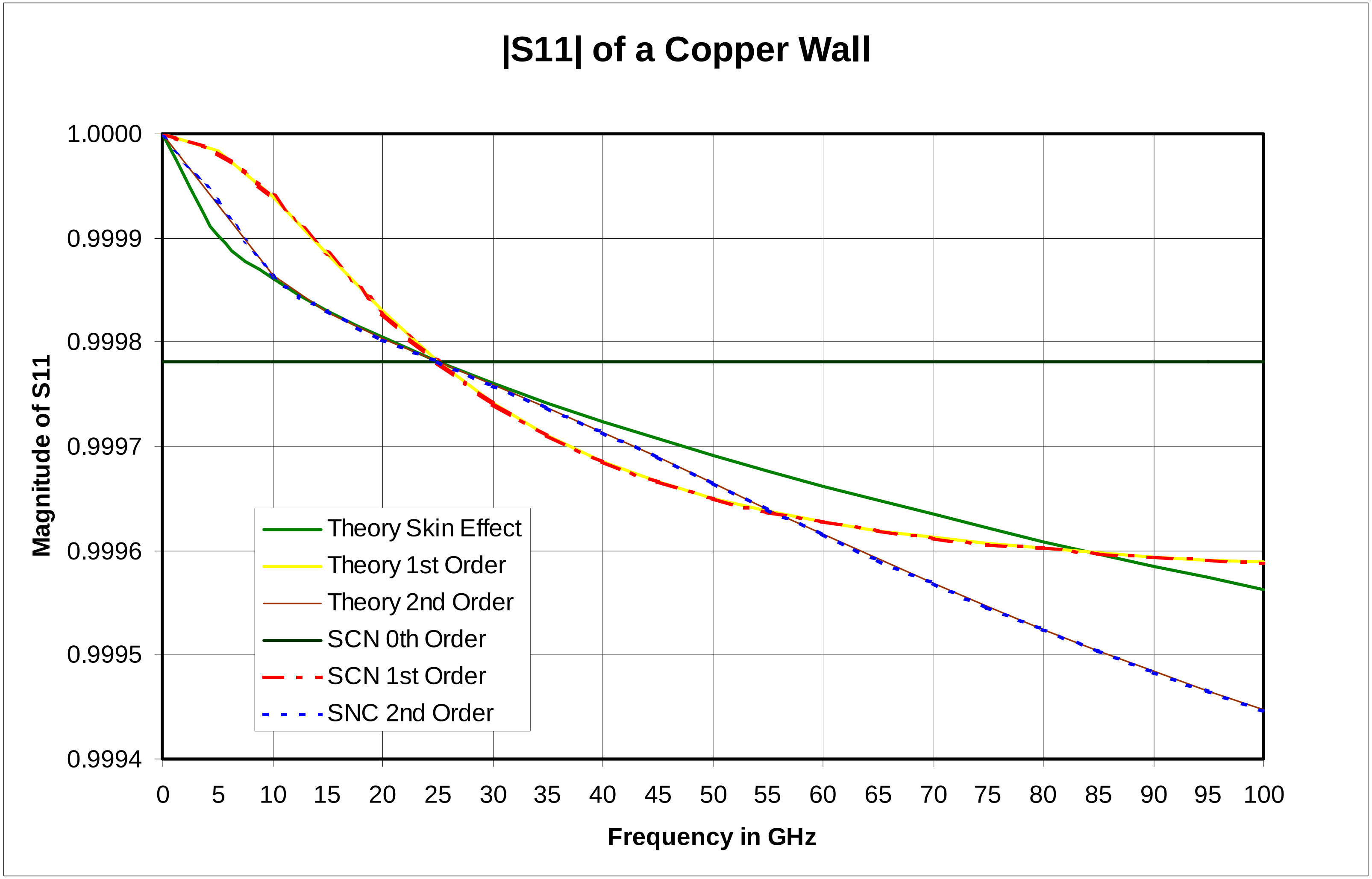
1st- order model of lossy boundary:
The surface impedance is approximated by one $R_1$ and $L_1$ in parallel.



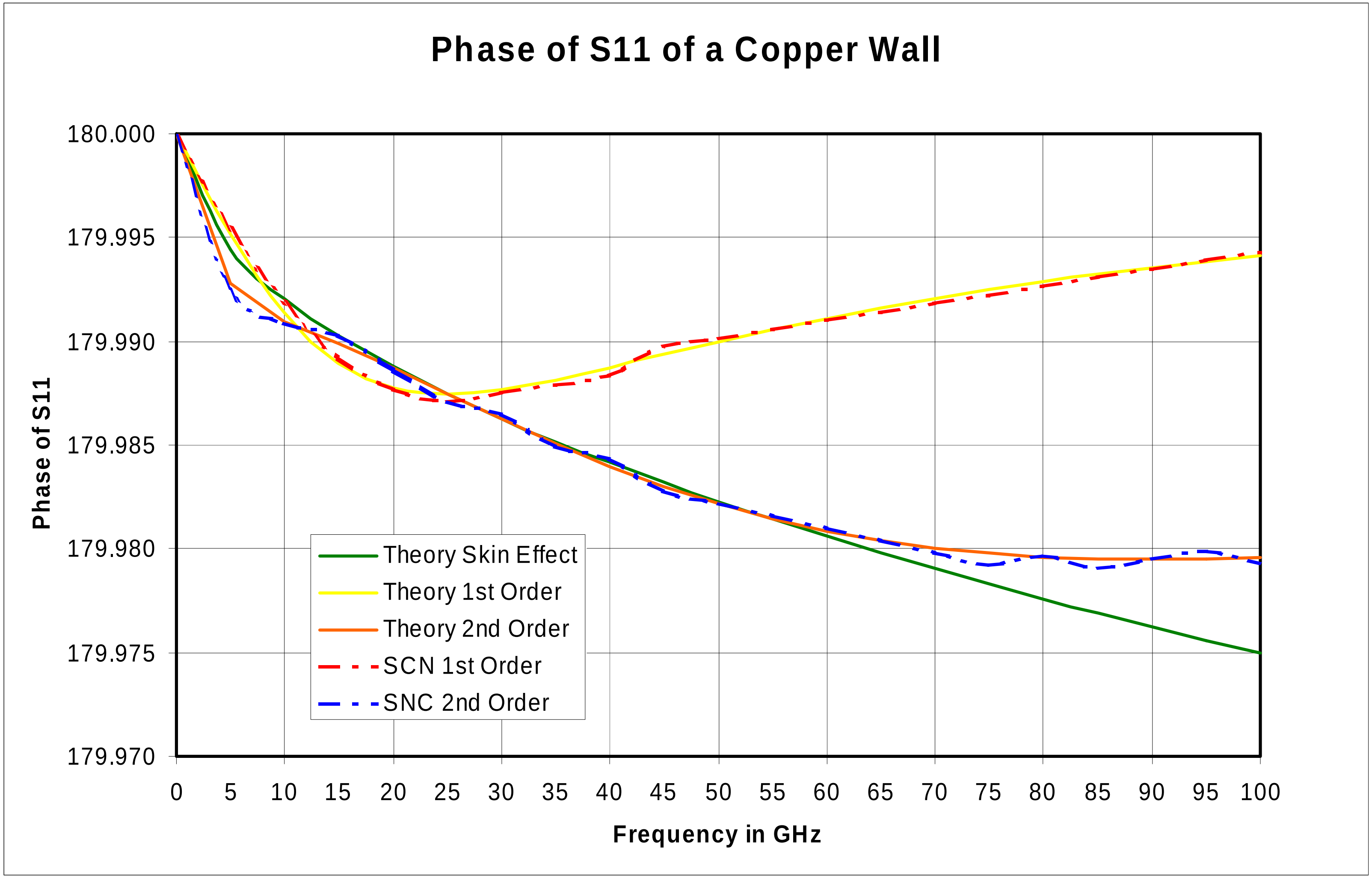
2nd- order model of lossy boundary:
The surface impedance is approximated by $R_1$ and $L_1$ in series with $R_2$ and $L_2$.

MEFiSTo integrates the differential equation of this eq. circuit (recursive convolution) (see Newsletter vol.2 no. 2 on website)

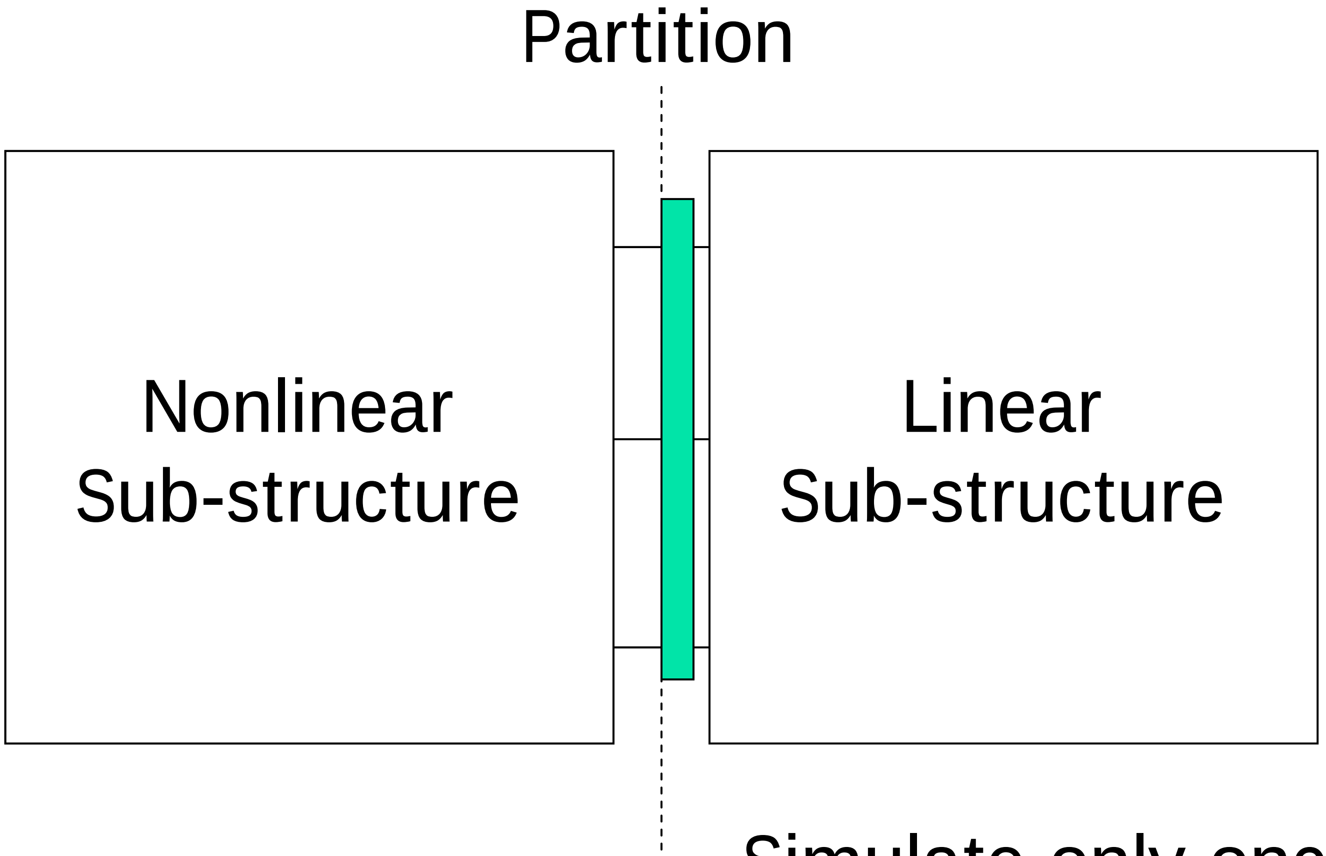
# Dispersive Surface Impedance



# Dispersive Surface Impedance



# Application of Diakoptics



Partition

Nonlinear
Sub-structure

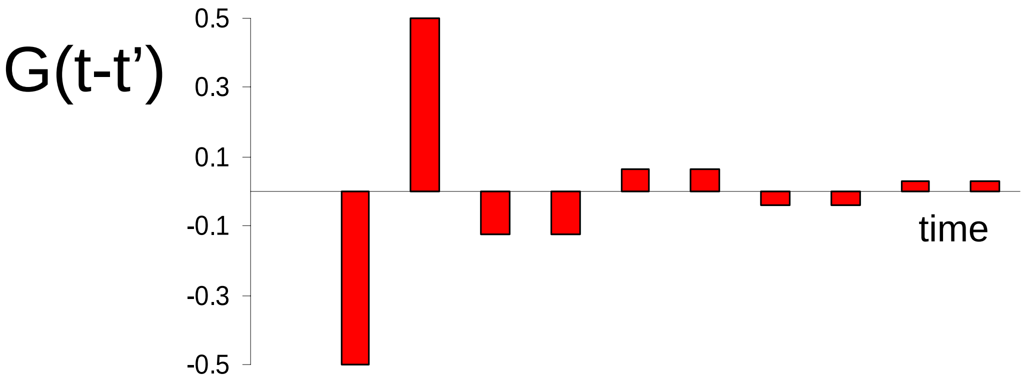
Linear
Sub-structure

Simulate only once,
Store impulse response

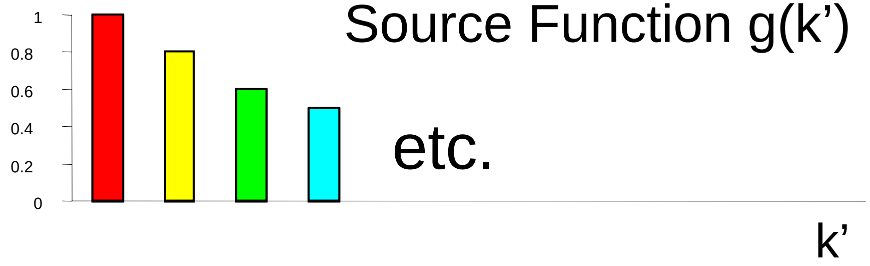
Reconnect by convolution

# A TLM Impulse Response

Impulse Excitation



Impulse Response (Green’s Function)



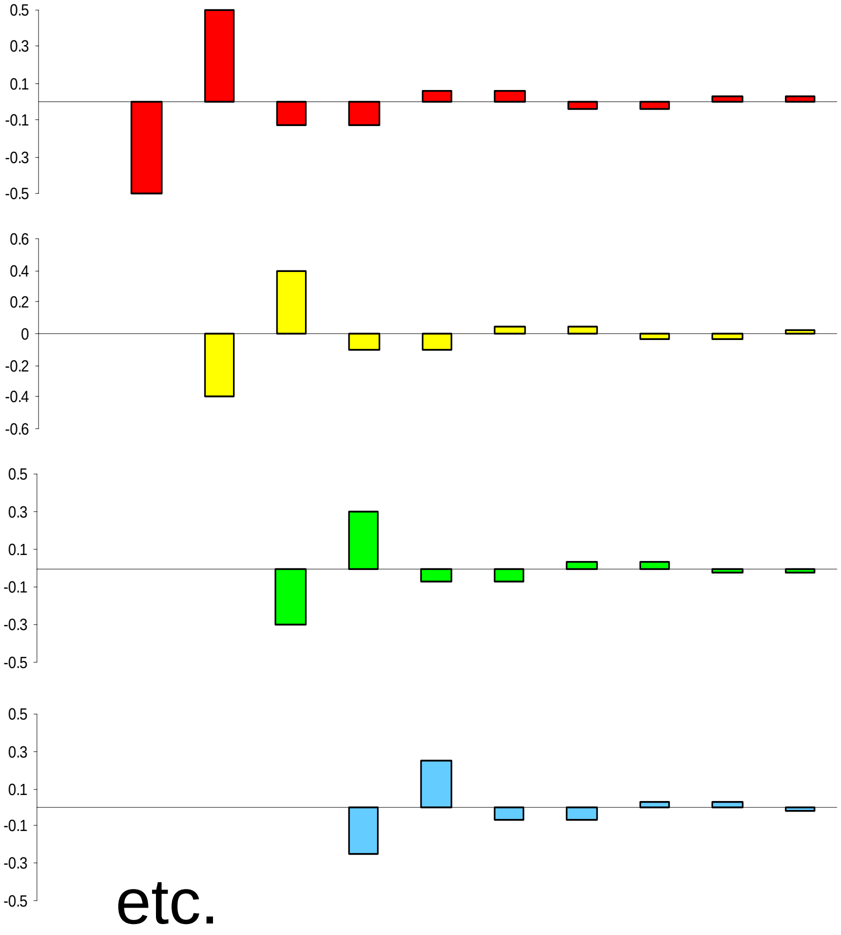
Source Function g(k')

etc.

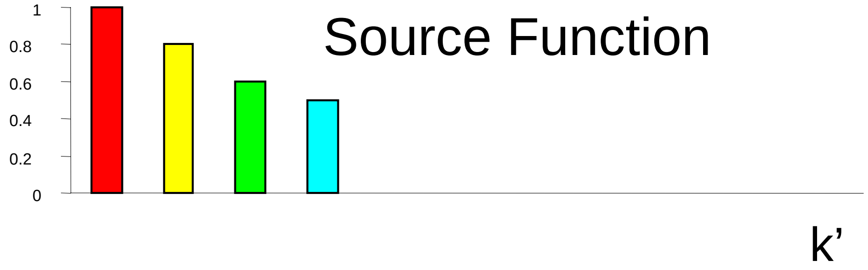
$$\int_0^t G(t,t')\,g(t')\,dt' = f(t)$$

$$t = k\Delta t \;;\quad t' = k'\Delta t$$

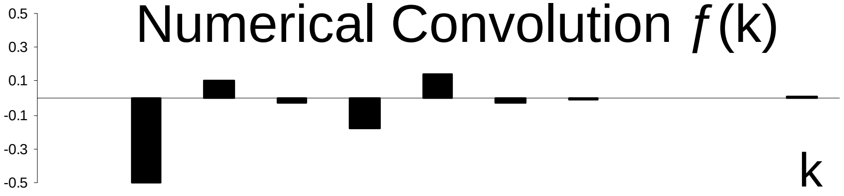
$$\sum_{k'=0}^{k} G(k,k')\,g(k') = f(\mathrm{k})$$



etc.

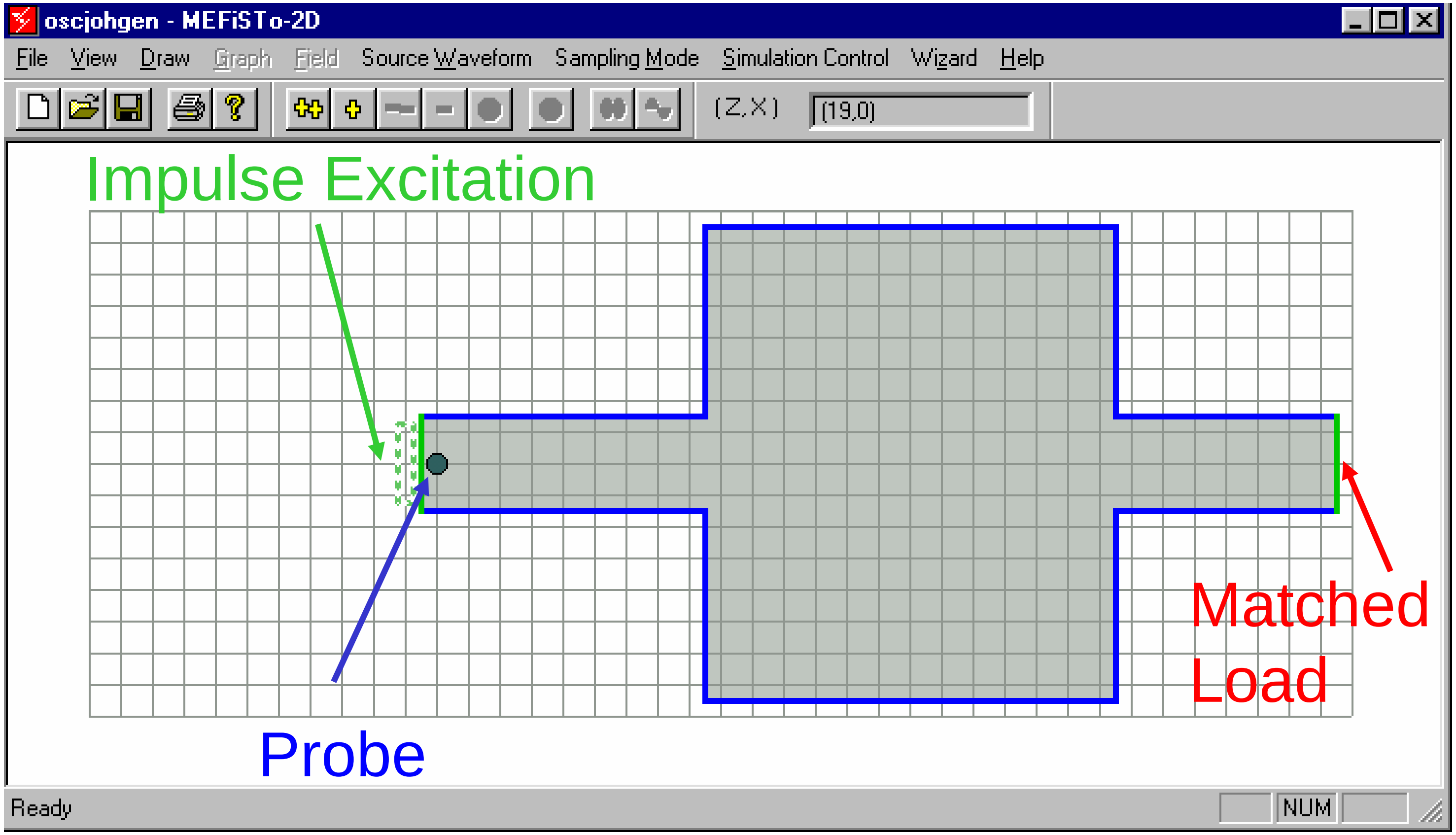


Source Function



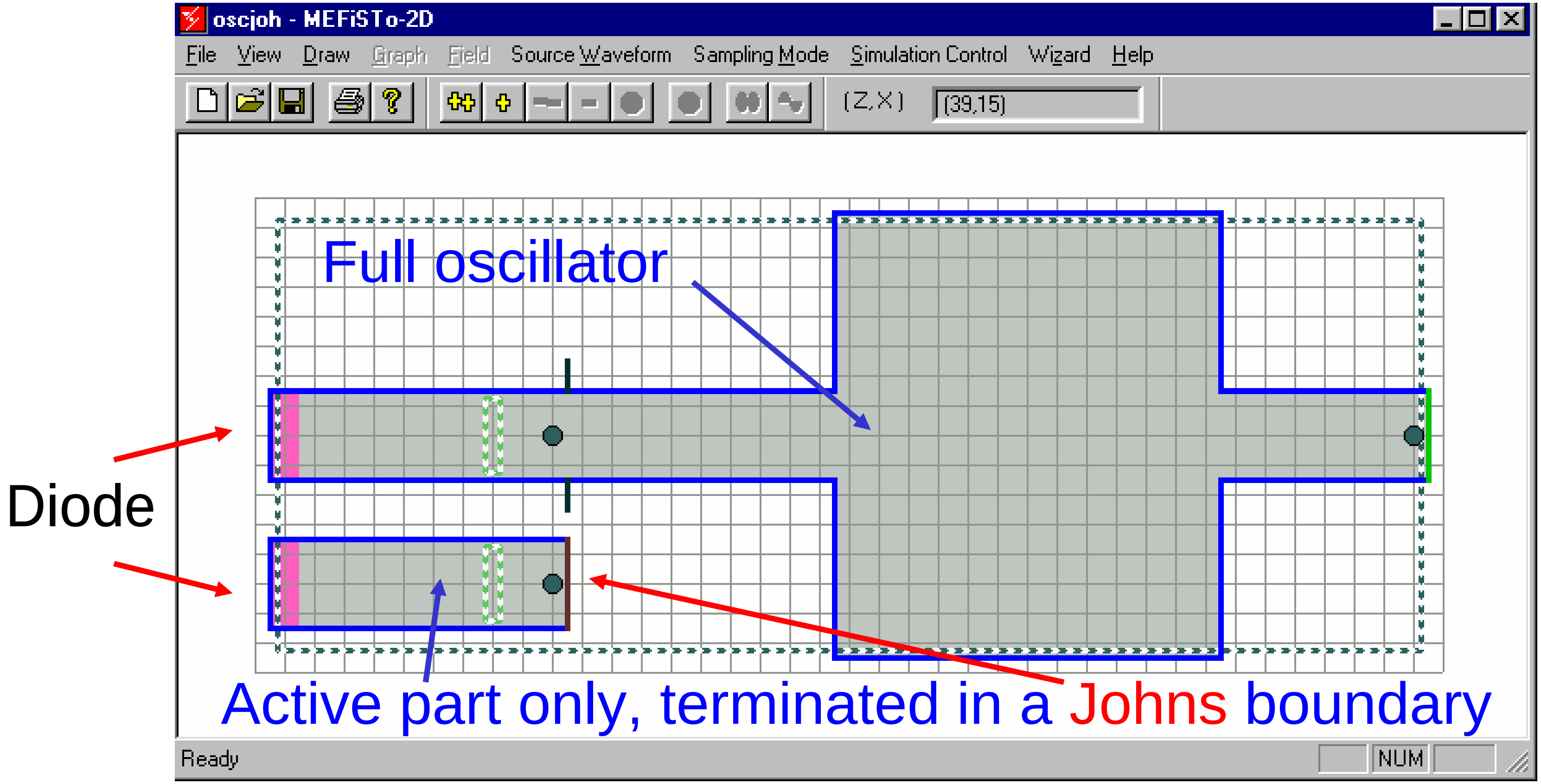
Numerical Convolution $f$(k)

# Response of Passive Circuit



The impulse response of the passive part of an oscillator is computed, picked up by the probe and stored

# Connection to the Active Diode



Full oscillator

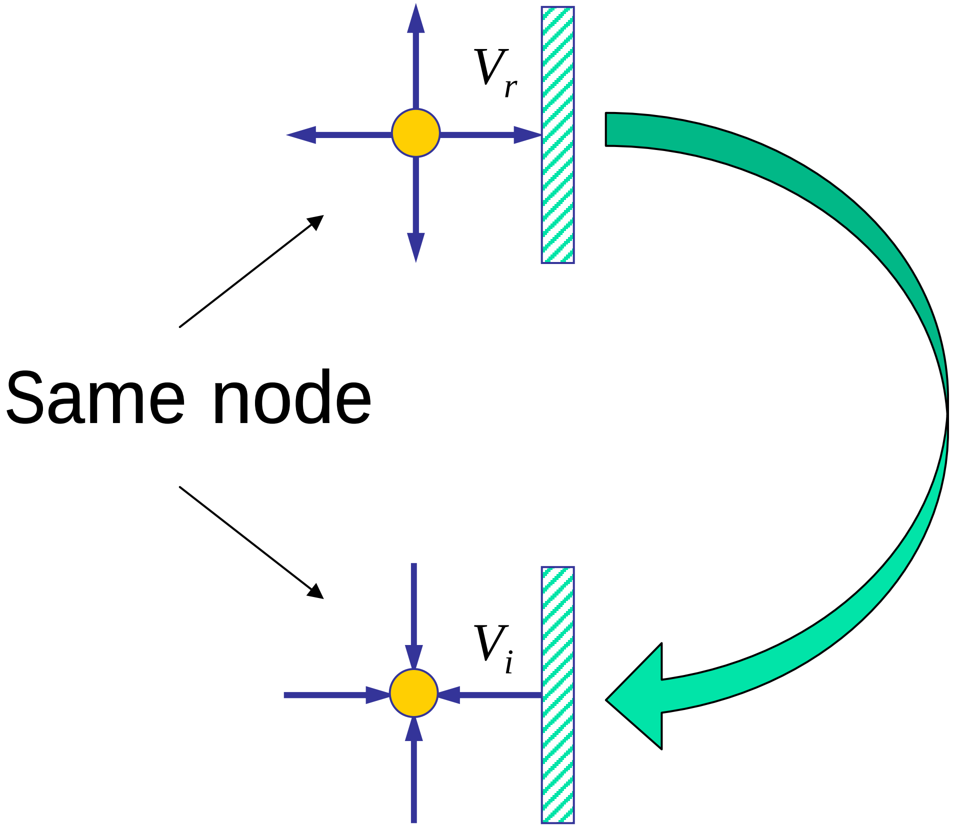
Diode

Active part only, terminated in a Johns boundary

The active part of the oscillator is terminated by convolution in a Johns boundary characterized by the impulse response of the passive part. It behaves exactly as the full oscillator

# TLM Modeling Environment

## TLM boundary operation framework



Same node

$$V^i = \Gamma_{impulse} \times V^r$$

$$V^i = h(k) * V^r(k)$$

$$\vdots$$

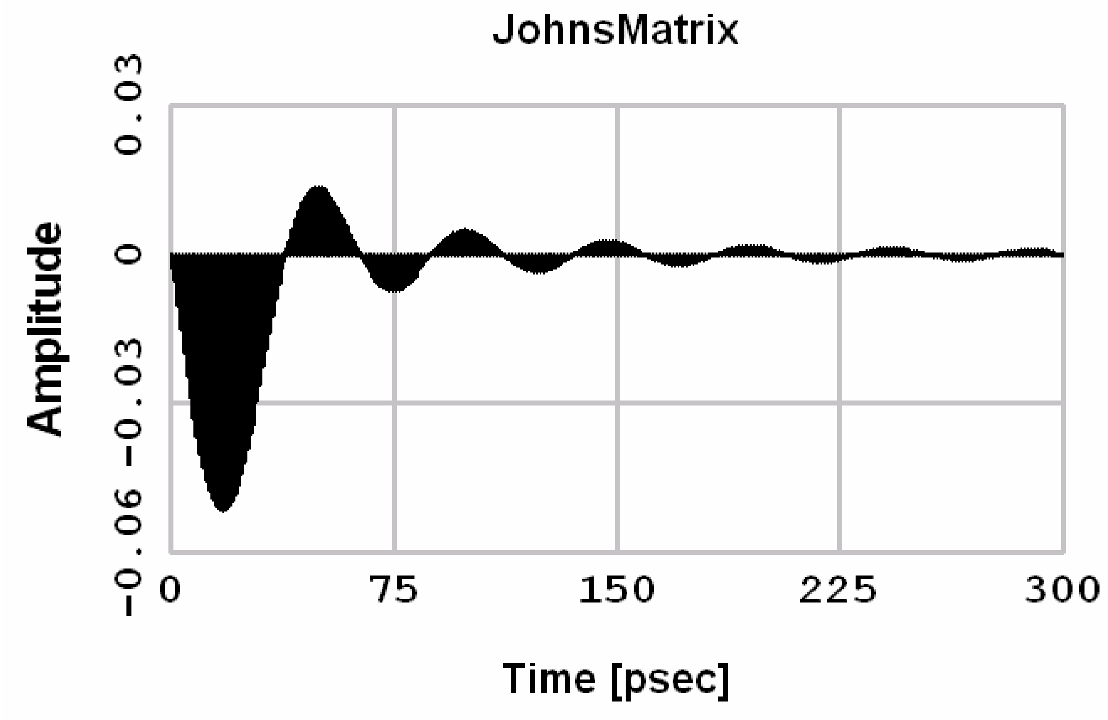
$$V^i = SPICE\left(V^r(k)\right)$$

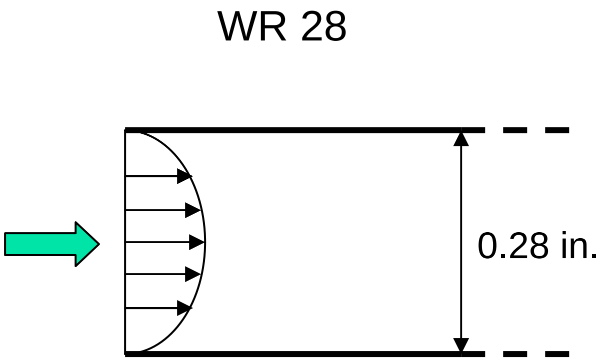
$$V^i = \mathrm{Thermal}\left(V^r(k)\right)$$

# Wideband Waveguide Load

$$Z(f) = \frac{Z_0}{\sqrt{1-\left(\frac{f_c}{f}\right)^2}}$$

Waveguide Characteristic Impedance



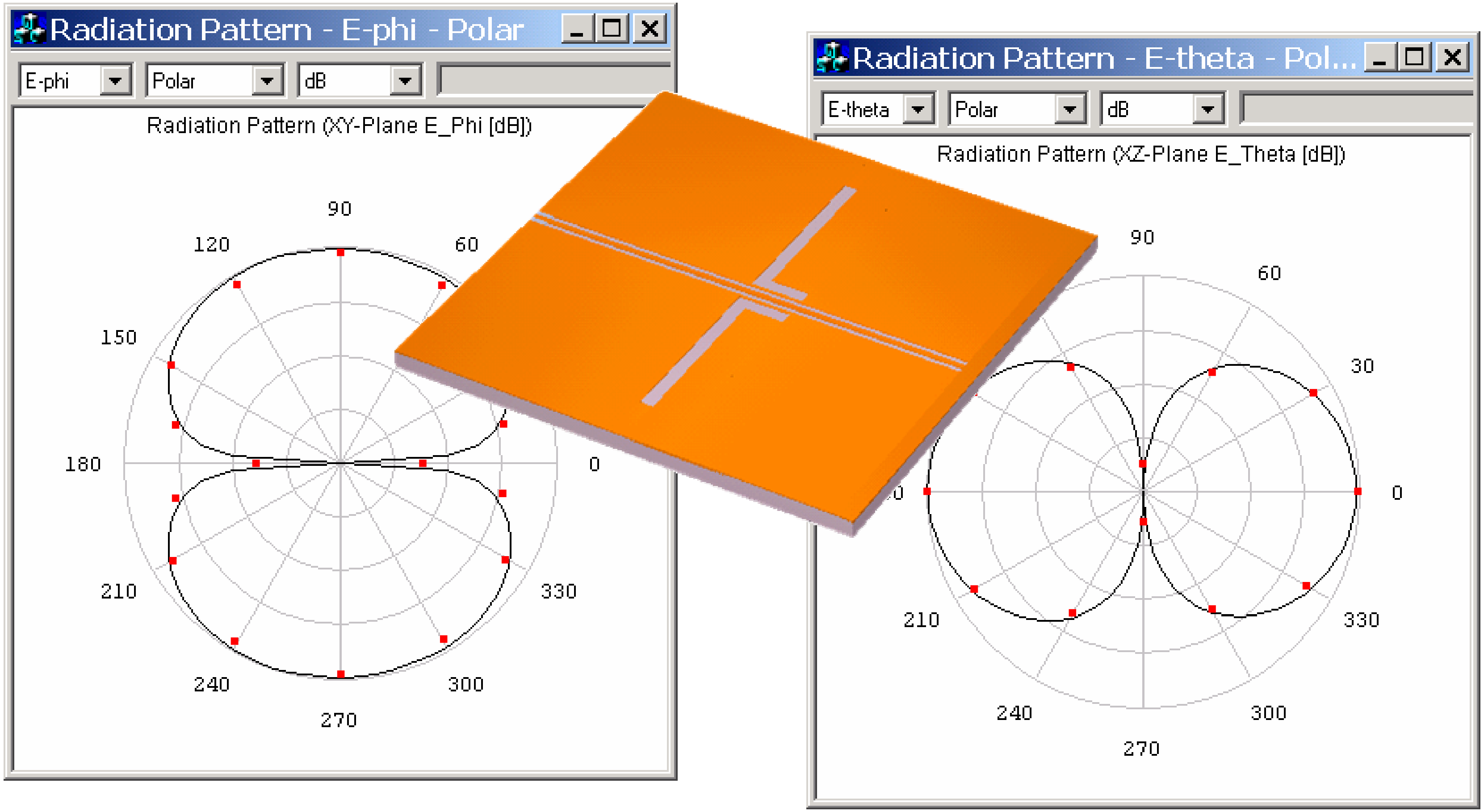


Modal Impulse Response of a WR28 Waveguide (Modal Green’s Function)

# Radiation Pattern

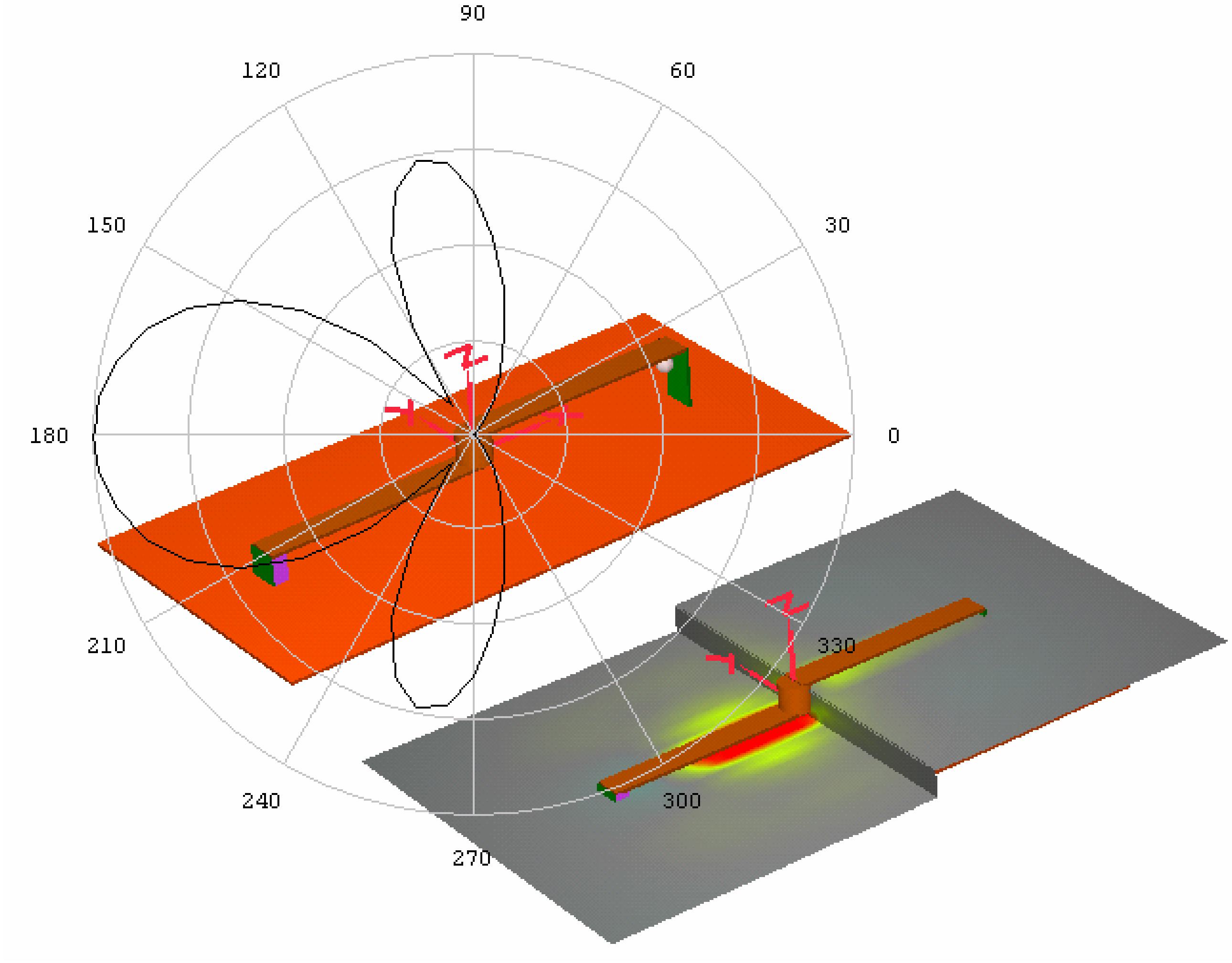
- The far field of a radiating structure is computed using the Helmholtz integrals,
- The source terms in the Helmholtz integrals are the tangential fields on an absorbing boundary,
- The tangential fields are obtained in the time domain using TLM, and then Fourier transformed,
- The far field radiation patterns can be obtained over a finite time interval and updated during the simulation, showing their evolution in time.

# Planar Slot Antenna

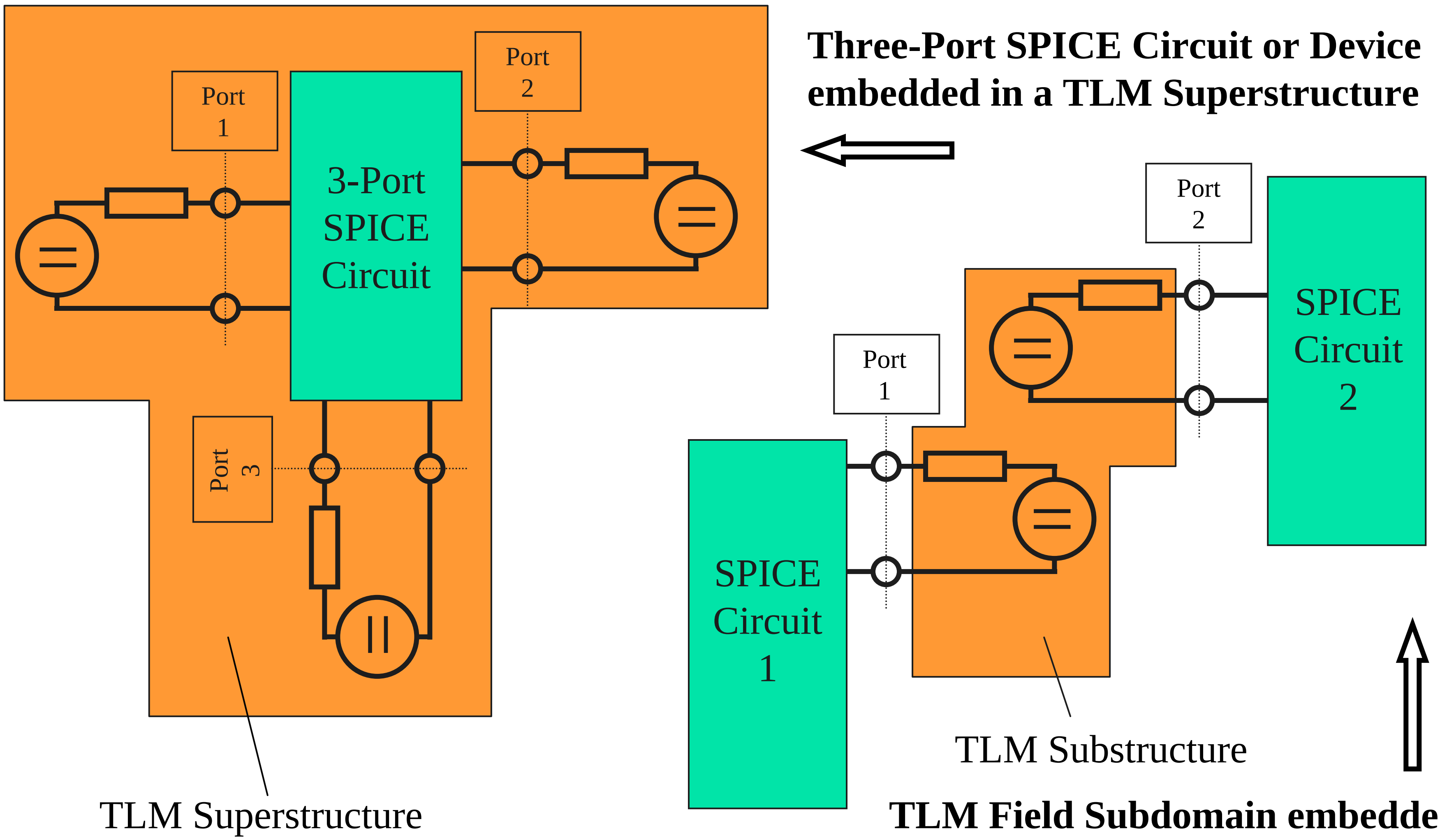


Measurements by Sierra-Garcia and J.J. Laurin, *Study of a CPW Inductively Coupled Slot Antenna*, IEEE Trans. Antennas and Propagation, vol. 47, pp. 58-64, Jan. 1999.

# Transient Radiation from Circuits



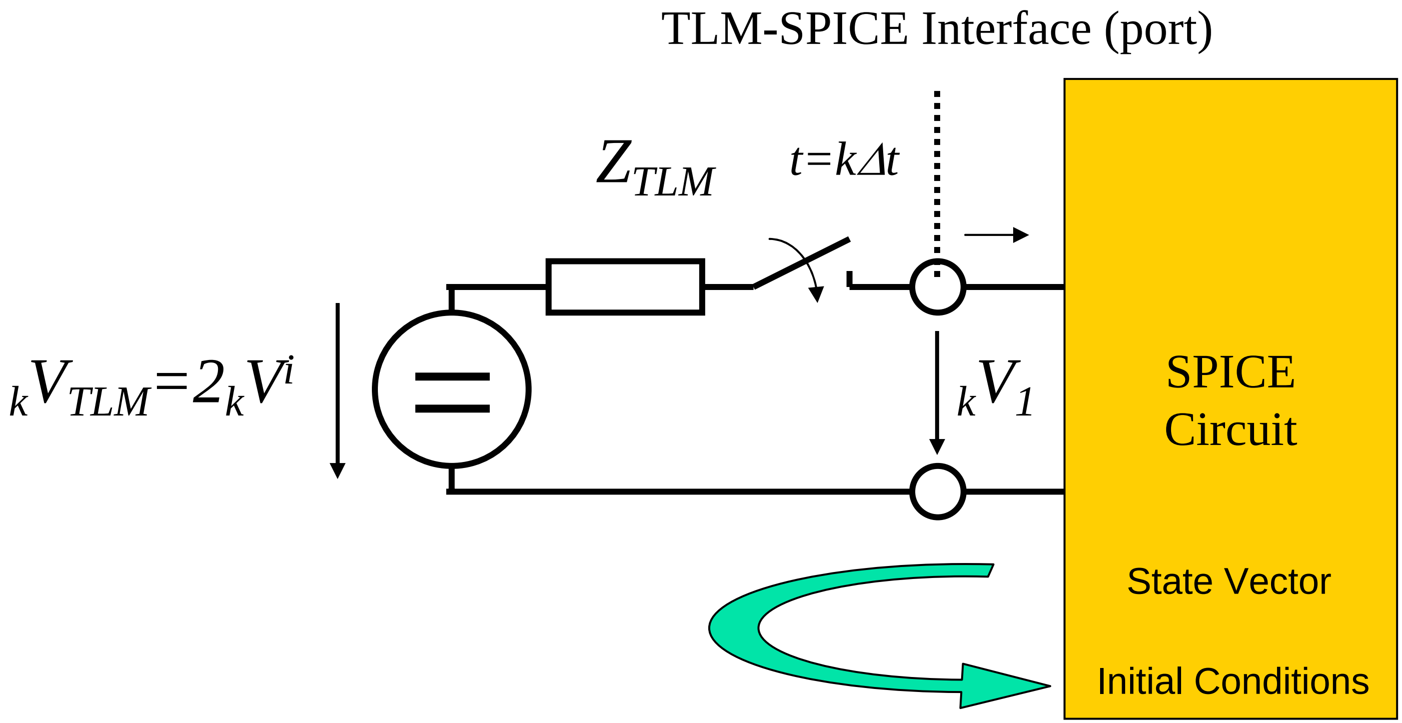
# General SPICE-TLM Interface



**Three-Port SPICE Circuit or Device embedded in a TLM Superstructure**

**TLM Field Subdomain embedded between two external SPICE Circuits.**

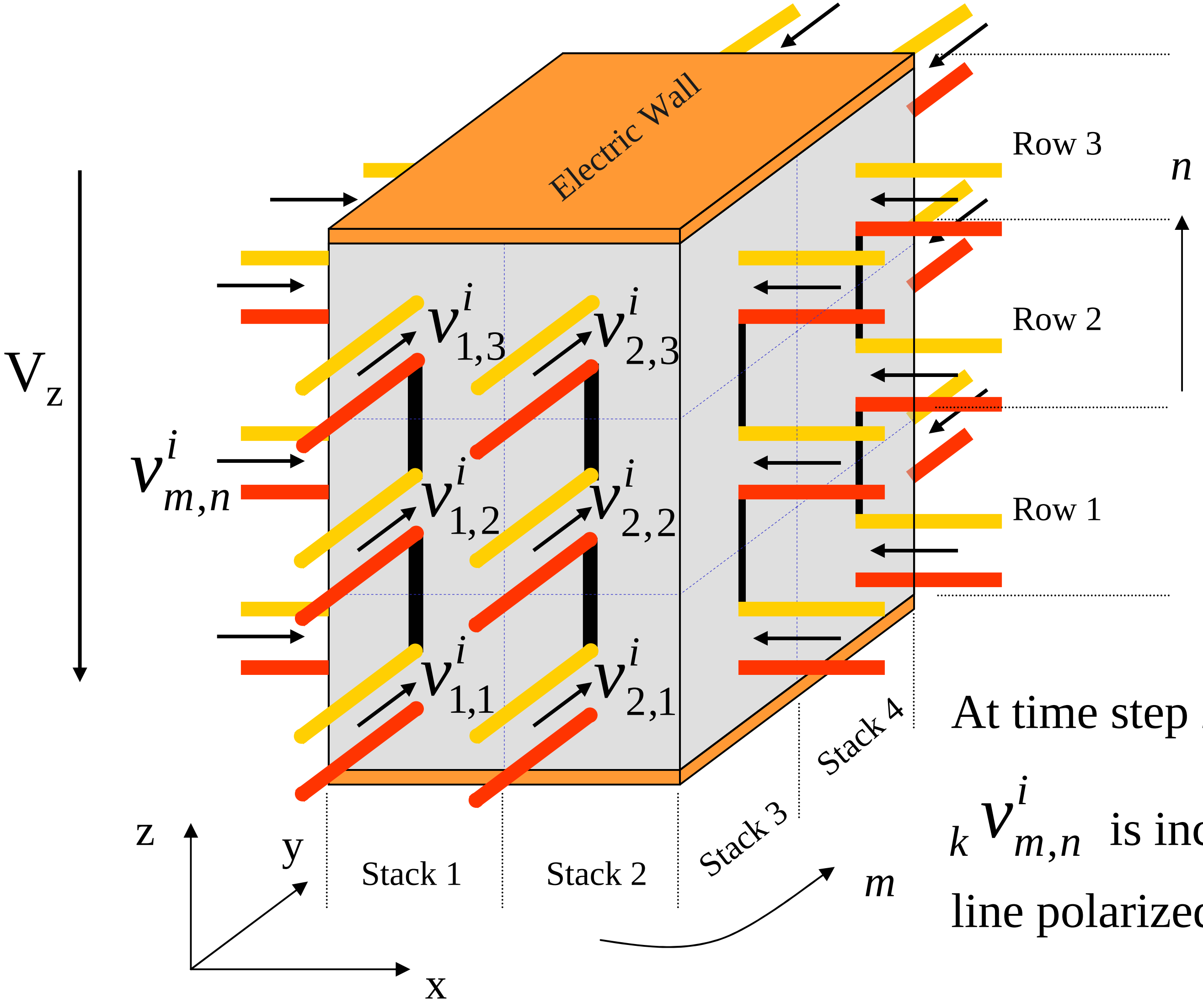
# TLM - SPICE Connection



The SPICE circuit may contain sources

The TLM impulses incident at the *kth* time step upon the SPICE port are represented by an equivalent voltage ${}_kV_{TLM}=2_kV^i$ connected to the SPICE circuit at time $k\Delta t$ through an equivalent real reference impedance $Z_{TLM}$ which depends on the port topology.

# Voltage-Coupled SPICE-TLM

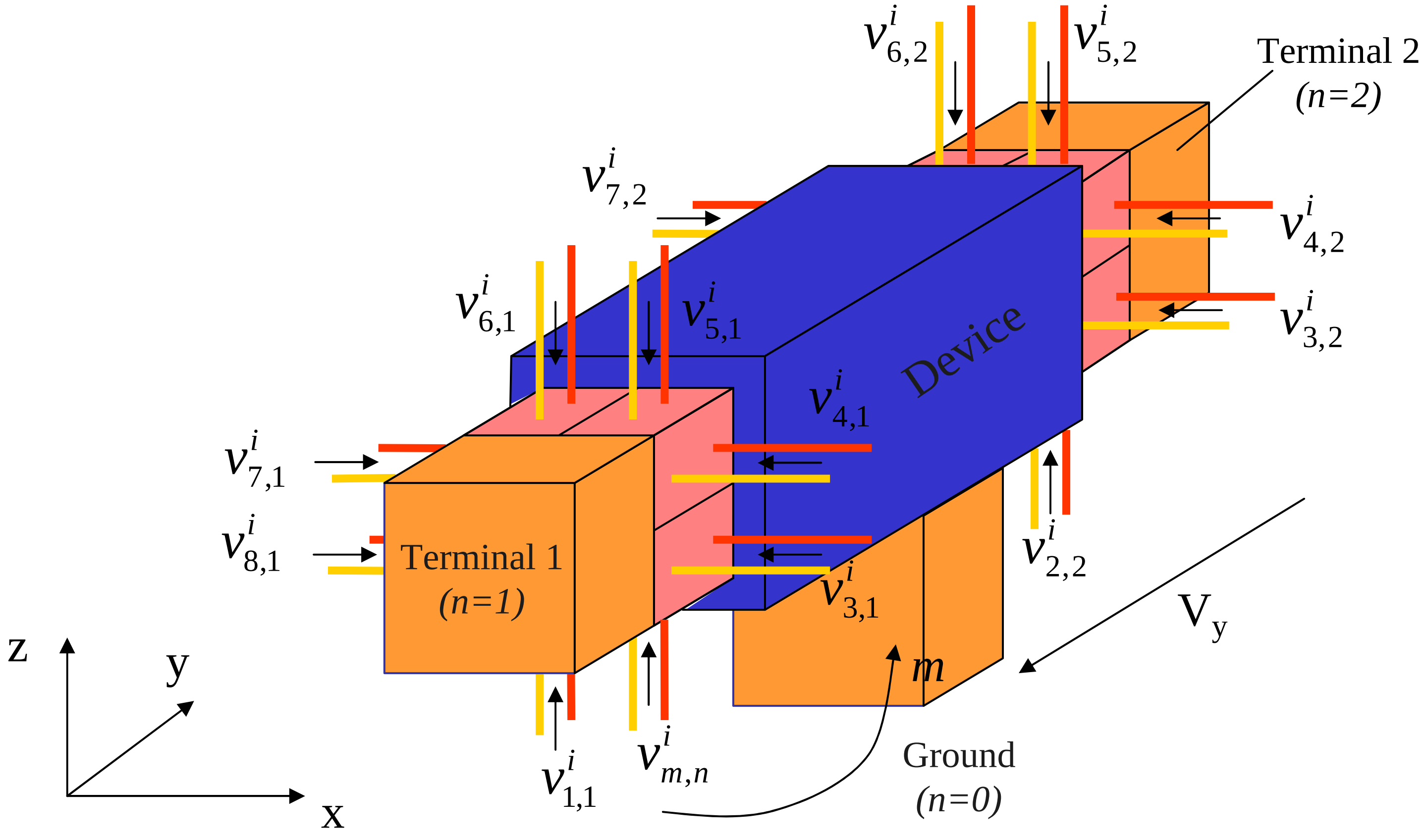


Dominant incident E-field component is parallel to the device voltage. (Example: Shunt diode in a coaxial or strip line)

At time step *k* a voltage impulse

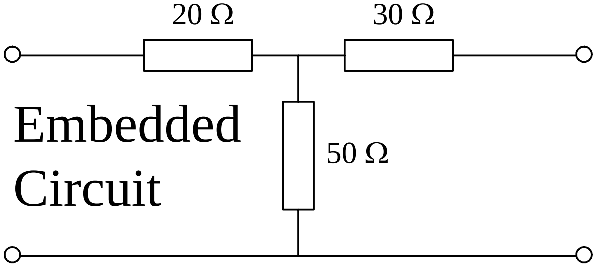
${}_k v^i_{m,n}$ is incident in each link line polarized in z-direction.

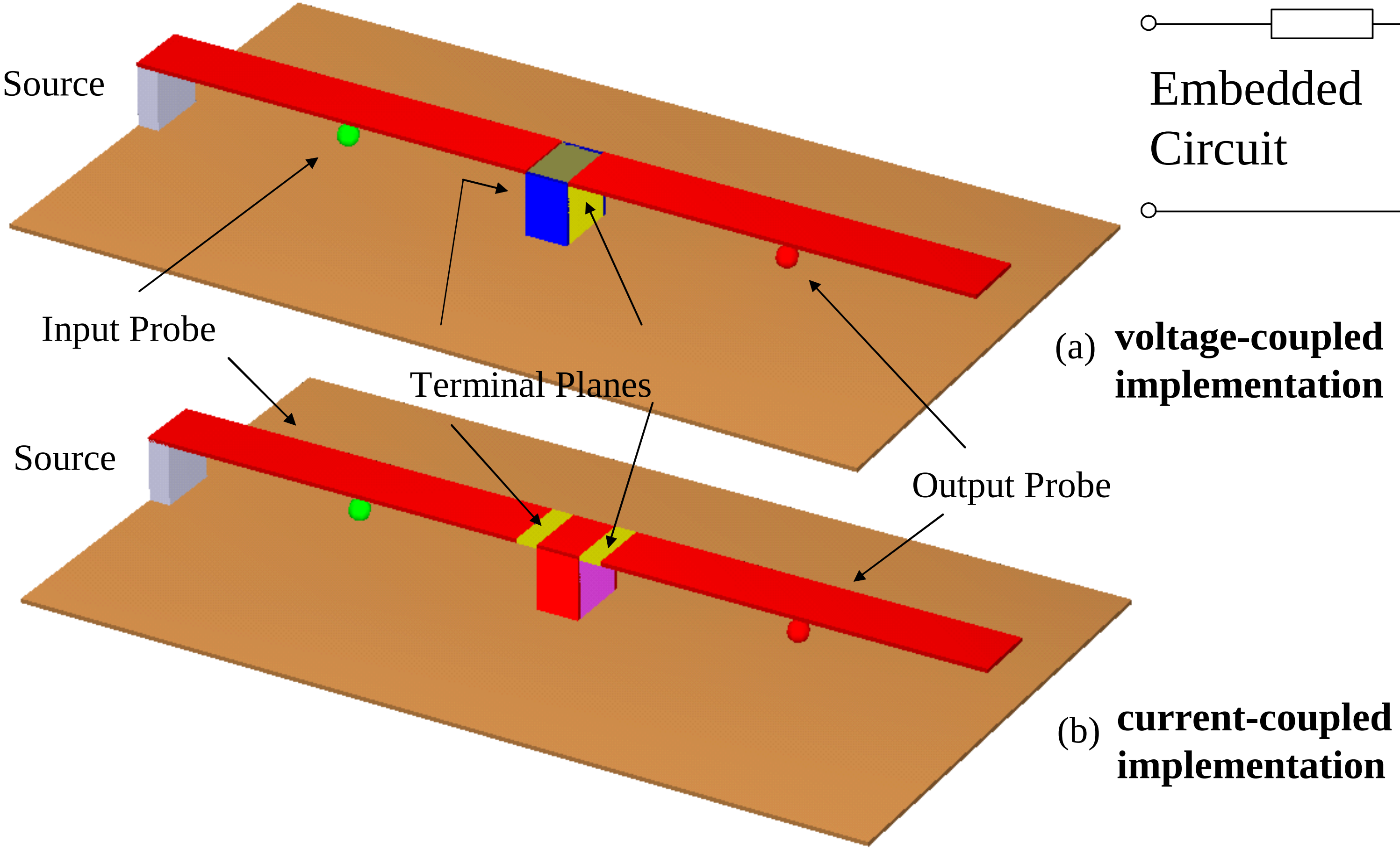
# Current-Coupled SPICE-TLM



The dominant incident electric field component is polarized perpendicular to the device voltage. (Example: transistor in a coaxial or strip transmission line)

# V- vs. I-Coupled SPICE-EM-bed



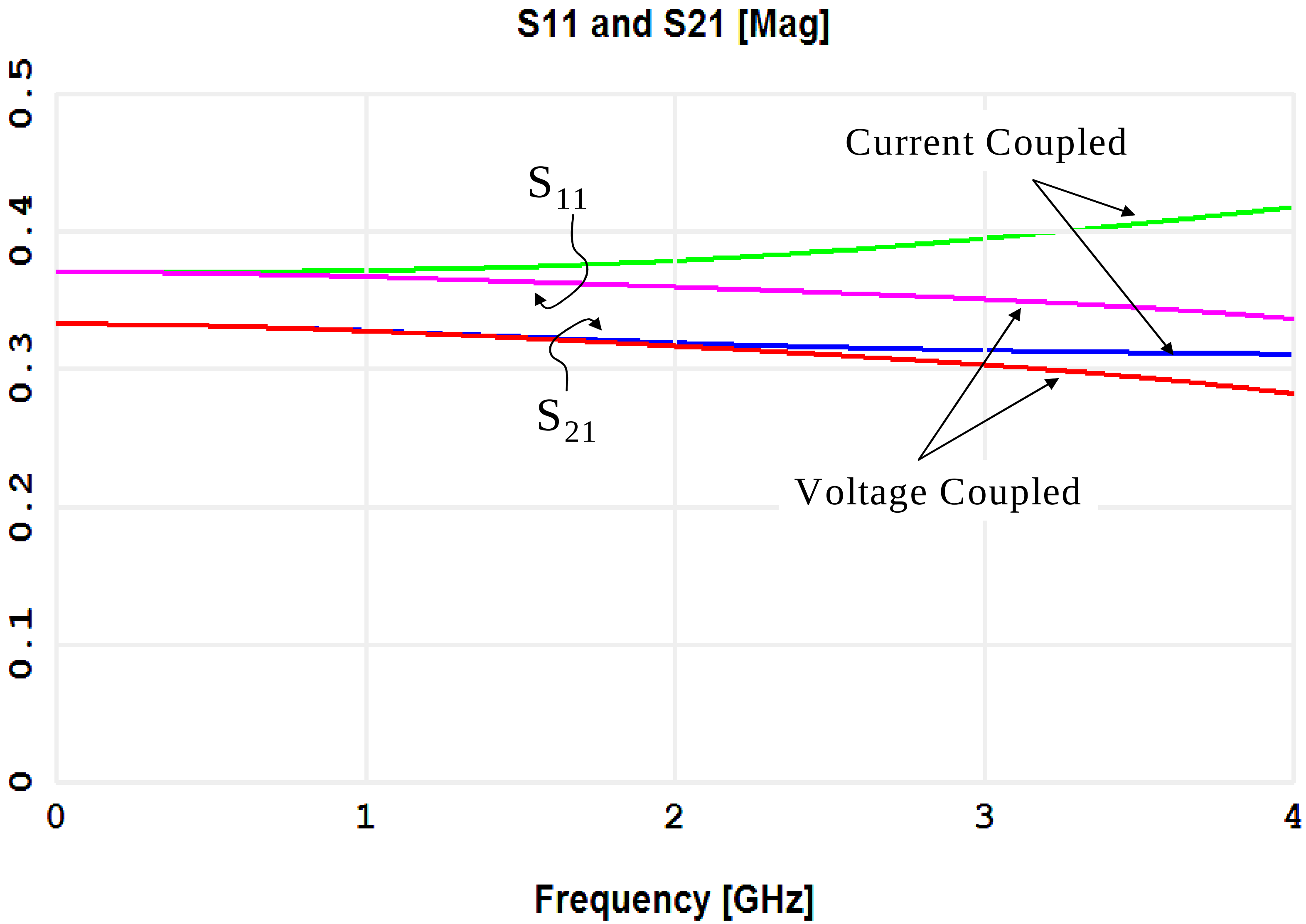


(a) **voltage-coupled implementation**

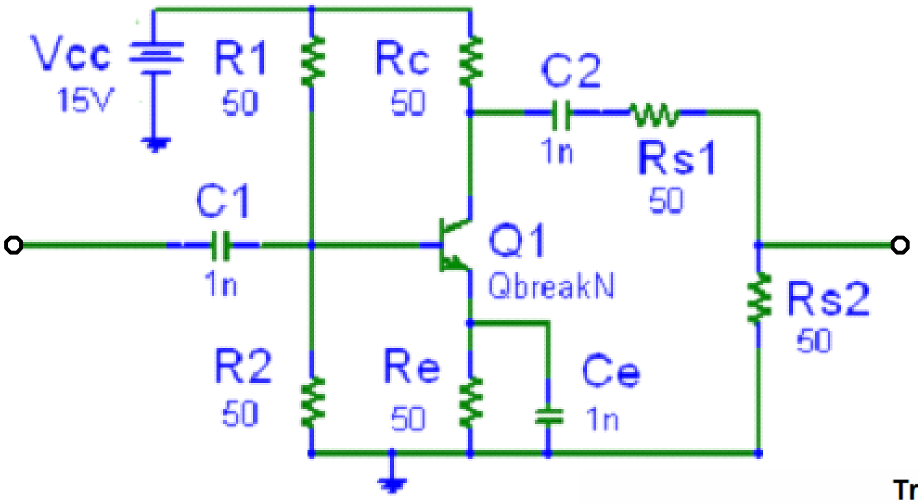
(b) **current-coupled implementation**

A two-port SPICE circuit is embedded in a microstrip.
The characteristic impedance of the microstrip is 126.12Ω.

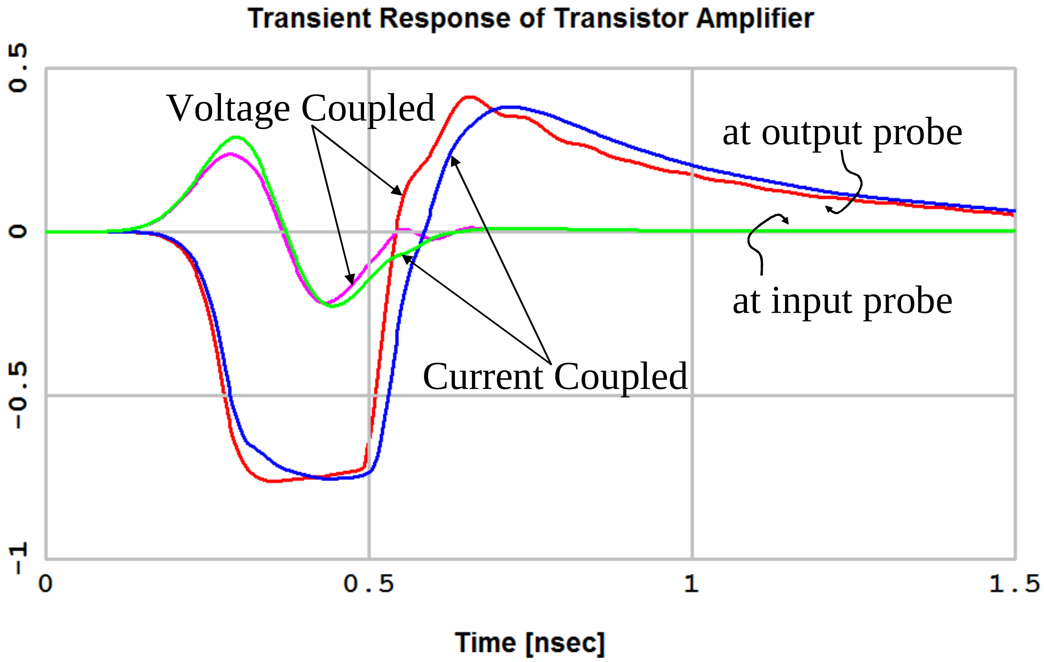
# S-Parameters of the Device



# Transient Amplifier Response

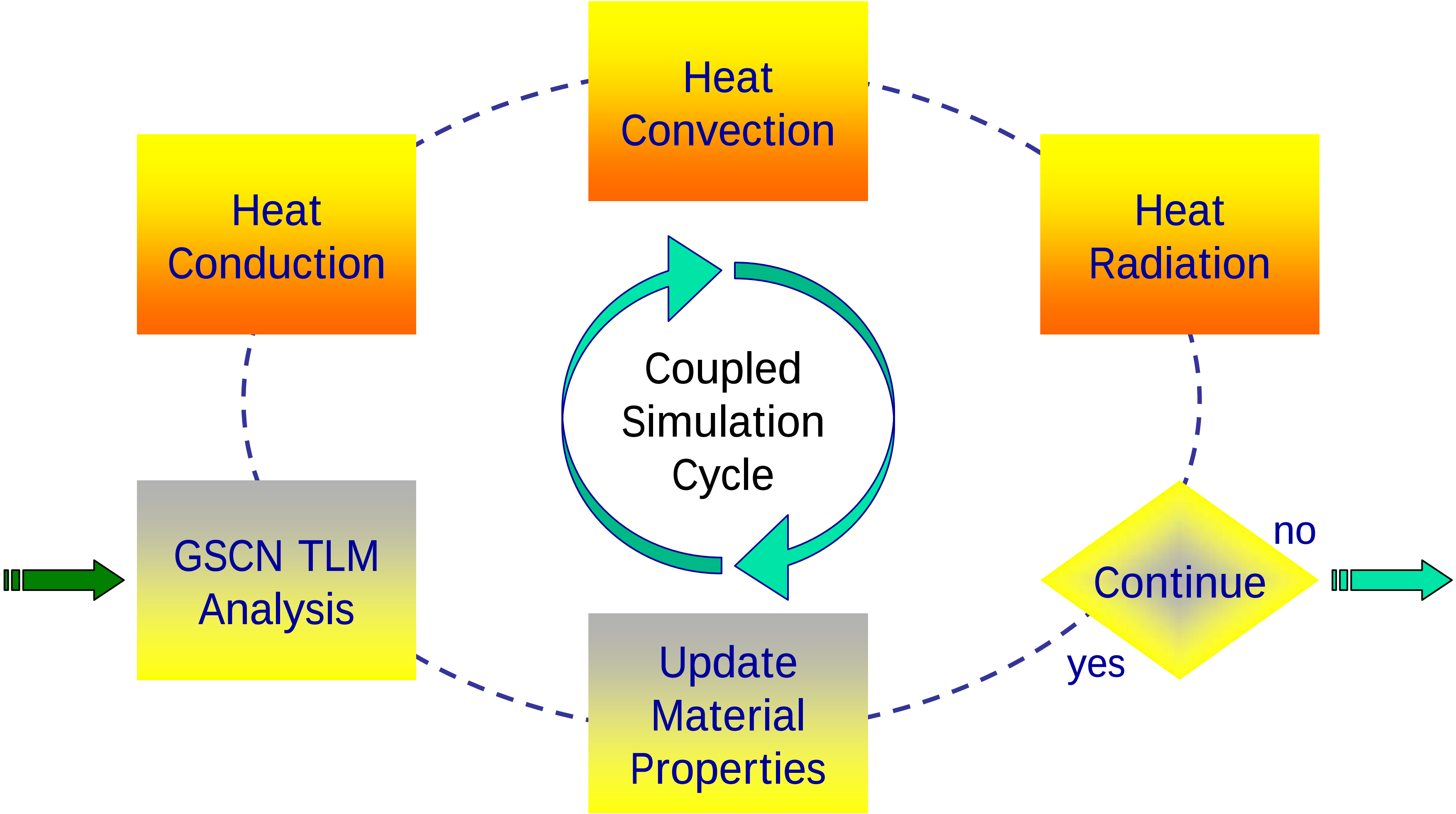


Embedded SPICE circuit



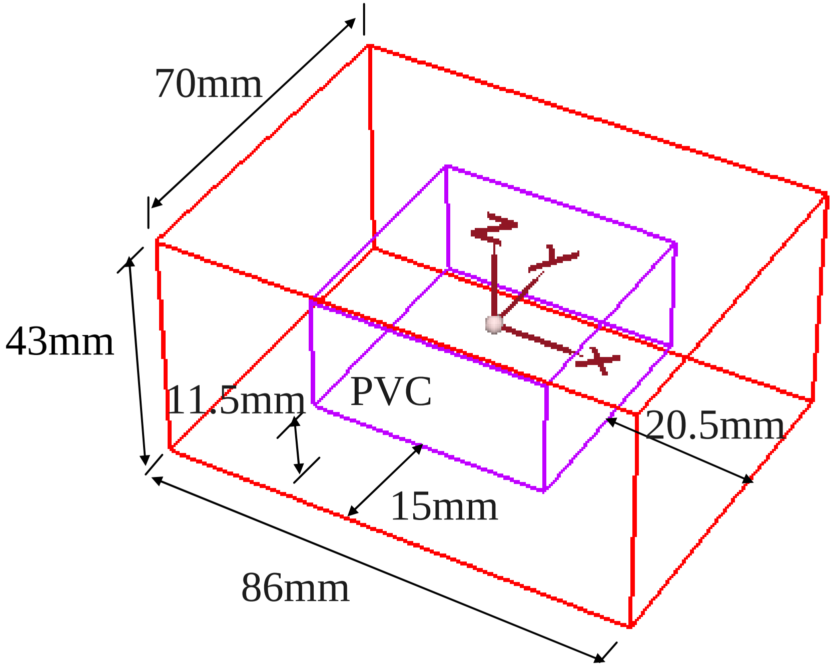
Transient response in same microstrip

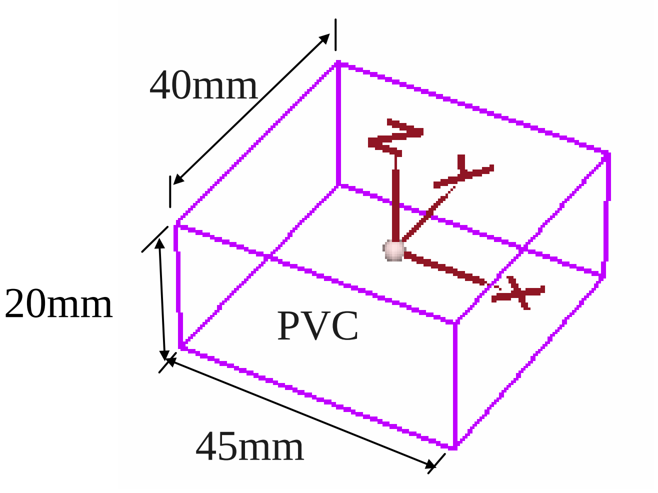
# Coupled EM-Thermal Engine



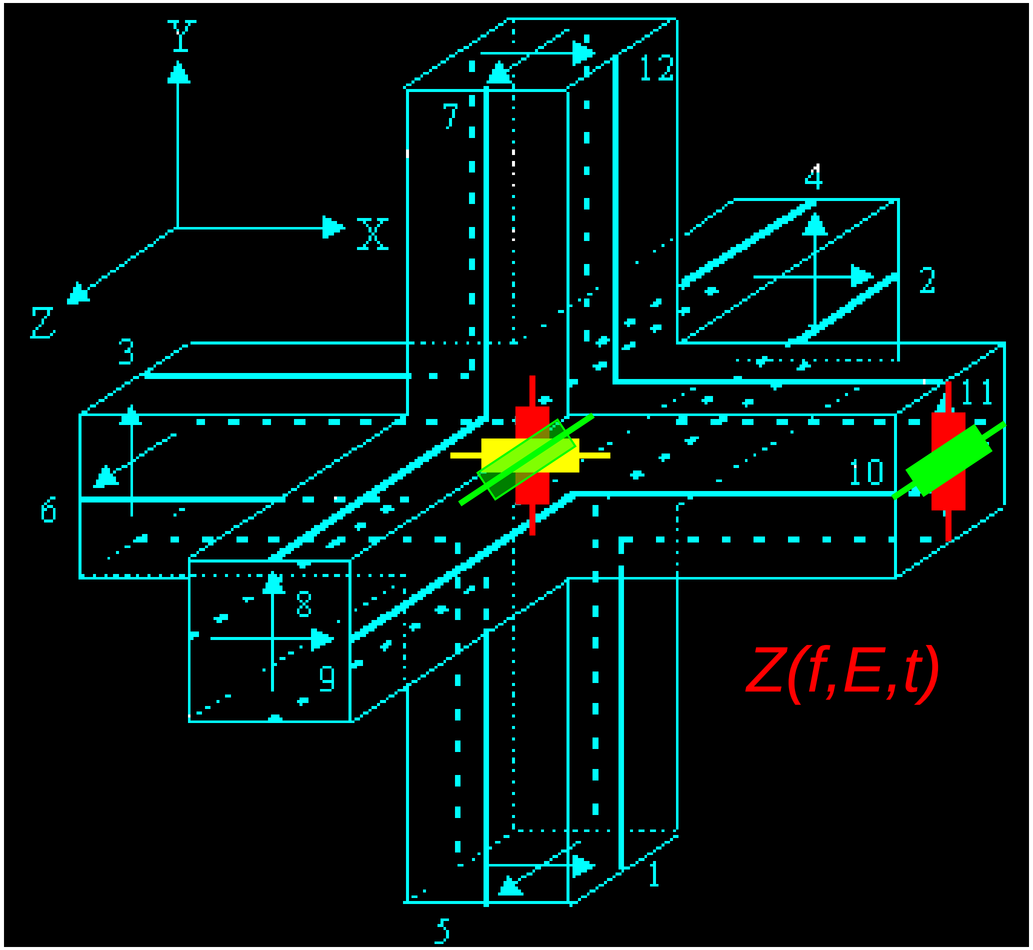
# Thermal Simulation Example

## PVC Block in a Rectangular Cavity





# Lumped Elements in 3D-TLM

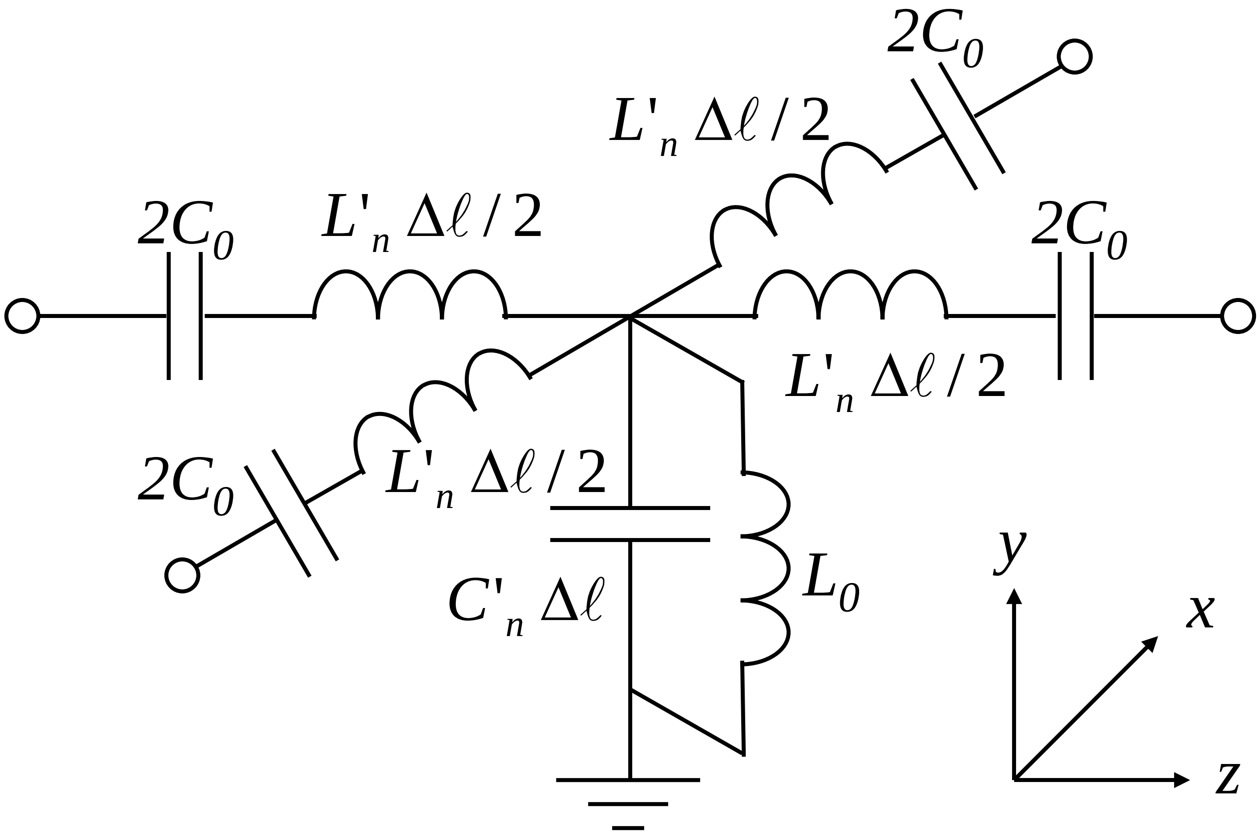


Possibility 1:
Lumped Elements
connected to a Node
at the Center of the Cell
(3 polarization planes)

Possibility 2:
Lumped Elements
connected to Branches
at the Cell Border
(2 polarization planes)

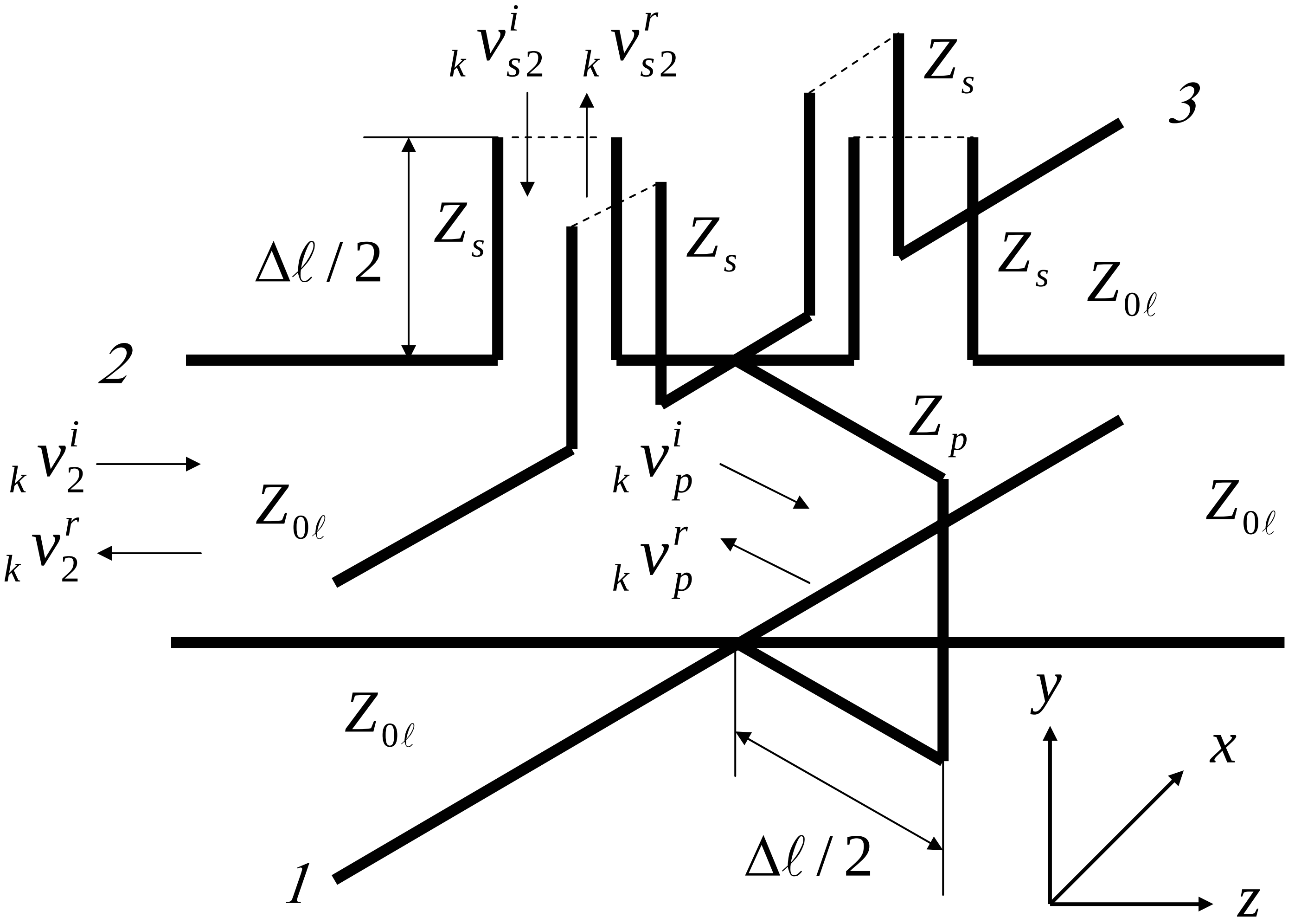
All impedances can be dispersive, nonlinear and time dependent

# Modeling of Metamaterials



Unit Cell of a 2D Metamaterial Model
(Eleftheriades et al.)

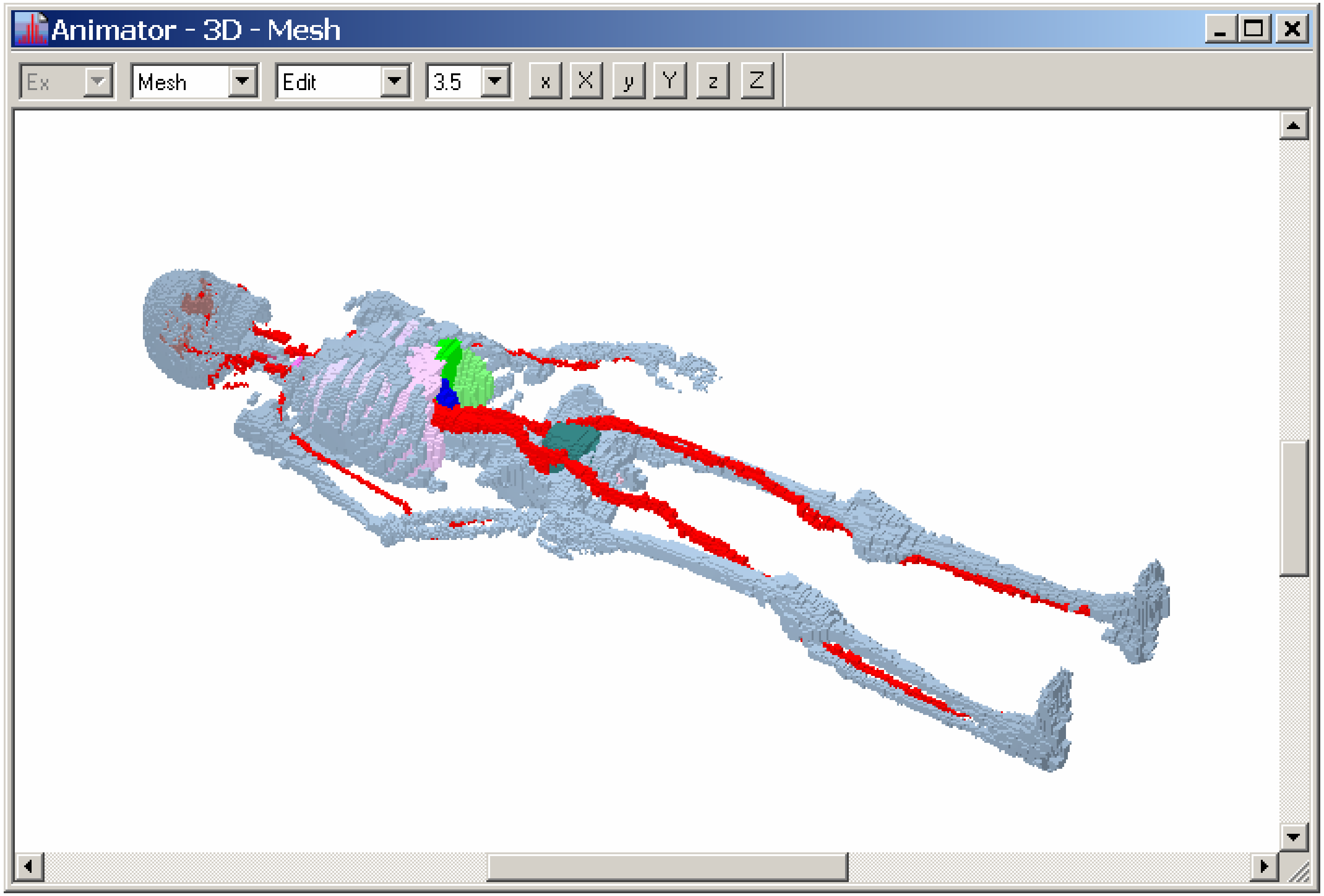
# Modeling of Metamaterials



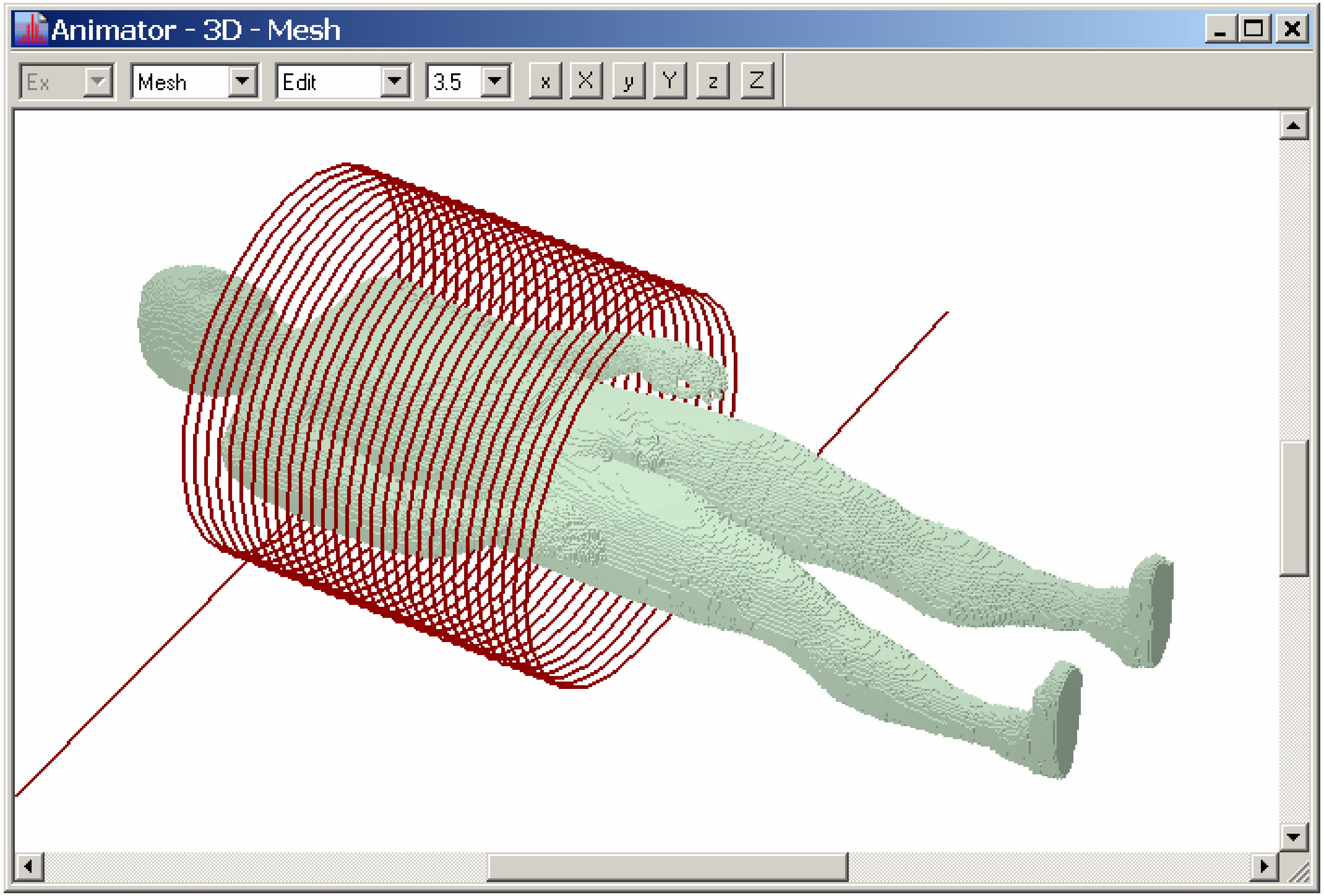
TLM Shunt Node Implementation of the Eleftheriades model

Right-Click on the yellow buttons and select “Open link in Browser”

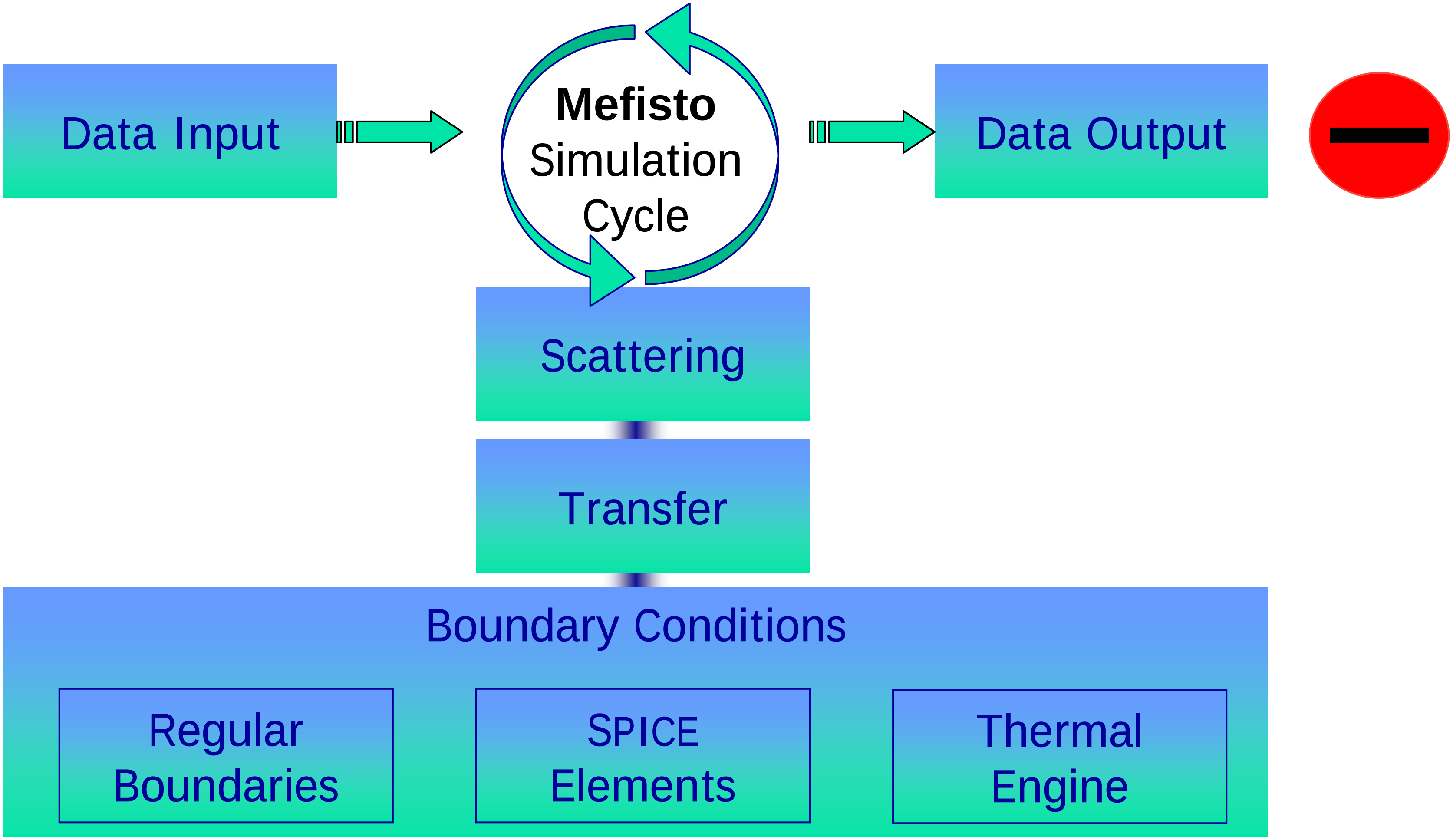
# Biomedical Application



# MRI Nerve Stimulation



# From Analysis to Synthesis?



Data Input

**Mefisto**
Simulation
Cycle

Data Output

Scattering

Transfer

Boundary Conditions

Regular
Boundaries

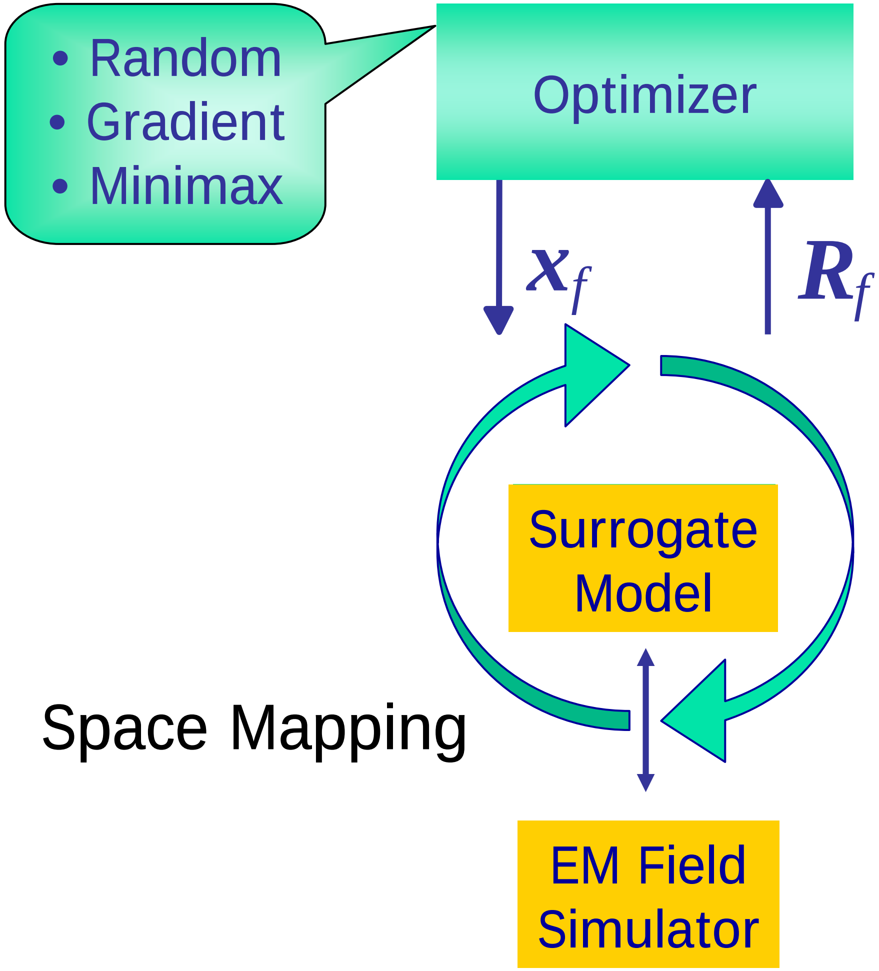
SPICE
Elements

Thermal
Engine

# Paradigm for Field Synthesis

- The classical approach to field-based optimization treats the electromagnetic simulator as a black box.
- The derivatives of the responses are obtained through finite differences, and parameters are optimized by minimizing the discrepancy between the analysis results and the desired specifications.
- MEFiSTo will employ a different approach to the design of microwave structures that employs direct time domain field synthesis.

# Optimization vs. Field Synthesis



- Random
- Gradient
- Minimax

Optimizer

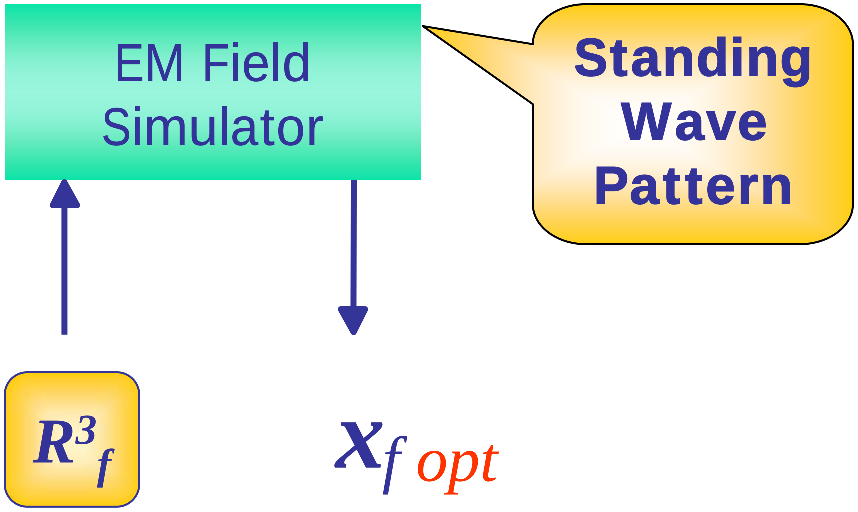
$x_f$

$R_f$

Surrogate Model

EM Field Simulator

Space Mapping



EM Field Simulator

**Standing Wave Pattern**

$R^3{}_f$

$x_{f\ opt}$

$x$ = *vector of designable parameters*

$R$ = *vector of responses*

# Summary and Conclusion

Essential Concepts for Modeling Advanced Mixed Signal Systems have been presented

- Technical Approach
- Integration of mixed signal problem into the circuit and field modeling domains
- Two-way flexible multi-level modeling environment
- Global Modeling / EM Interconnect Modeling /hybrid physics modeling/reduced-order model extraction/synthesis
- Several examples have been presented
- Technical collaboration is being developed