Integrating Circuit Simulation with EIT FEM Models

Alistair Boyle[1] and Andy Adler[2]

[1]School of Electrical Engineering and Computer Science, University of Ottawa, Ottawa, Canada aboyle2@uottawa.ca
[2]Systems and Computer Engineering, Carleton University, Ottawa, Canada

Abstract: This work presents two methods of co-simulating the FEM-based EIT model and SPICE-based circuit models in impedance imaging.

1 Introduction

Integrating circuit simulation into the Finite Element Method (FEM) models used for Electrical Impedance Tomography (EIT) simulations may improve analysis of the complete electrical system. Combining these two models is possible because both linear circuits (at a fixed frequency) and EIT forward simulations use the same underlying numeric tools, the Cholesky decomposition (for symmetric matrices) and LU decomposition (for unsymmetric matrices), which are both incorporated into the left divide operation in Matlab. The combined matrix is block diagonal with one sparse block for the FEM model and one sparse block for the circuit model. Off-diagonal entries are used to connect the two models at the "wires" which are the individual nodes that would normally be driven by the input vector or have difference measurements calculated.

2 Modified Nodal Analysis in EIDORS

SPICE-based circuit simulation tools such as `ngspice` use Modified Nodal Analysis (MNA) to solve linear circuits [1]. Non-linear circuits require additional steps: a DC solution to determine the operating point, then a linearization of non-linear models at the operating point, and insertion of the linearized model into the matrix. Non-linear elements such as transistors and diodes have a variety of models associated with them (BSIM3, BSIM4, SIMSOI, PSP, HICUM, MEXTRAM). Linear elements are often enough to model complex circuit behaviour, for example op-amp frequency response.

As an initial implementation, only the linear circuit elements (resistors `R`, inductors `L`, capacitors `C`, ideal current `I` and voltage `V` sources, current `H F` and voltage `E G` controlled sources) have been integrated into EIDORS. A "standard"[1] SPICE netlist reader takes a SPICE netlist (Listing 1) and transforms it into a matrix using "stamps" which are similar in construction to FEM elements; a standard N-terminal element is mapped to global nodes. The nodes of the stamp are variables that hold the nodal voltages and branch currents of each circuit element. Each type of element has a different "stamp." For linear elements, the Laplace representation gives a direct complex valued solution representing phase delay and voltage/current gain.

Listing 1: SPICE model of an ideal voltage controlled current source (`ec`) driving an electrode wire (RLC) into electrode #7 (`e7`)

```
Gs 6 0 ec 0 1.0
L4 5 6 4m
R1 e7 5 10
CL e7 0 250p
```

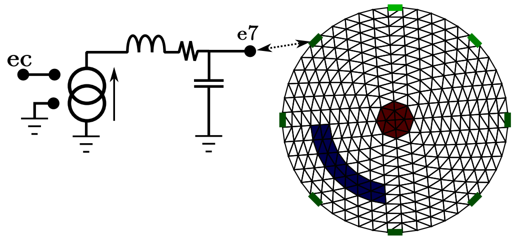

Figure 1: SPICE circuit model of wiring connected to the EIT forward model

3 Forward Modelling with SPICE

For a more complete forward model and stronger co-simulation capabilities, direct integration with SPICE would be preferable rather than re-implementing the feature-rich capabilities of a standard SPICE package. This integration is made challenging by Matlab's `.mex` file protections which isolate sub-processes and are tied to non-system compilers. Using a model reduction of the FEM for a system matrix

$$\begin{bmatrix} \mathbf{V}_A \\ \mathbf{V}_D \end{bmatrix} = \begin{bmatrix} \mathbf{A} & \mathbf{B} \\ \mathbf{B}^\intercal & \mathbf{D} \end{bmatrix}^{-1} \begin{bmatrix} 0 \\ \mathbf{I}_D \end{bmatrix} \quad (1)$$

$$\mathbf{D}' = \mathbf{D} - \mathbf{B}^\intercal \mathbf{A}^{-1} \mathbf{B} \quad (2)$$

$$\mathbf{V}_D = \mathbf{D}'^{-1} \mathbf{I}_D \quad (3)$$

the reduced matrix $\mathbf{D}'$ can be converted to a mesh network of $n(n-1)/2$ resistors (for n electrodes). This mesh is loaded into a SPICE simulation as a subcircuit which enables the full facilities of a non-linear SPICE simulation when evaluating EIT hardware. More complete simulations are possible without making a quasi-static assumption [2], but this Partial Element Equivalent Circuit approach has not been implemented here.

4 Conclusions

Inverse problems could benefit from modelling the nonlinear behaviour of the circuits used to transmit and acquire signals in the system. This work presents two methods of co-simulating the FEM-based EIT model and SPICE-based circuit models in impedance imaging.

Integrated circuit/FEM modelling may reduce calibration effort and enable more graceful aging of equipment over its service life. SPICE does not have any specialized capabilities for inverse modelling except through manual controls to characterize circuits. A method that unites the Jacobian for the FEM model with an accurate model of the circuit Jacobian requires equations for each (non-linear) circuit stamp.

EIT-into-SPICE `eit_spice` and SPICE-into-EIT `spice_eit` implementations are available in the EIDORS repository (`solvers/forward/`).

References

[1] T Quarles *PhD thesis*, UC Berkley, 1989
[2] A Ruehli *IEEE Trans. Microw. Theory Techn.* 22(3):216–221, 1974

[1]There are many SPICE dialects, but the linear components generally use common prefixes amongst the many SPICE implementations.