Instituto Tecnológico
y de Estudios Superiores de Occidente

Reconocimiento de validez oficial de estudios de nivel superior según acuerdo secretarial
15018, publicado en el Diario Oficial de la Federación del 29 de noviembre de 1976.

Departamento de Electrónica, Sistemas e Informática
Maestría en Diseño Electrónico

Matlab Driver for Design Parameterization and Optimization of Microstrip Circuits in PowerSI

TRABAJO RECEPCIONAL que para obtener el GRADO de
MAESTRA EN DISEÑO ELECTRÓNICO

Presenta: MARISOL CABRERA GÓMEZ

Director: JOSÉ ERNESTO RAYAS SÁNCHEZ

Tlaquepaque, Jalisco. Septiembre de 2019.

TÍTULO: **Matlab Driver for Design Parameterization and Optimization of Microstrip Circuits in PowerSI**

AUTORA: Marisol Cabrera Gómez

Ingeniero en Comunicaciones y Electrónica (Universidad de Guadalajara, México)

DIRECTOR DEL TRABAJO: José Ernesto Rayas Sánchez

Departamento de Electrónica, Sistemas e Informática, ITESO

Ingeniero en Electrónica (ITESO, México)

Maestro en Sistemas Electrónicos (ITESM Campus Monterrey, México)

Doctor en Ingeniería Eléctrica (Universidad McMaster, Canadá)

Senior, IEEE

NÚMERO DE PÁGINAS: vii, 30

Dedication

To my husband Daniel, for his unconditional support during my studies, who always encouraged me to reach my goals.

To my children Carmina and Logan, who have been my motor and my best motivation.

To Dr. José Ernesto Rayas Sánchez, for his guidance in the development of this work.

To Intel, for its support in pursuing of my Master's degree.

Summary

The present study case focusses in the generation of a Matlab driver for parameterized full-wave electromagnetic (EM) simulations performed in Cadence® Sigrity™ PowerSI (PowerSI), illustrating its usage to analyze several high-frequency circuits, such as microstrip filters and antennas, as well as for design optimization of a spiral inductor. The proposed methodology drives Power SI in batch mode, generates a base file with .SPD extension that can be edited in text format through Matlab commands, sets the simulation parameters and runs the simulation without opening the Power SI graphical user interface (GUI), and finally reads the simulation results for plotting or further processing with Matlab. The application of the proposed Matlab driver for PowerSI is illustrated by simulating several microstrip circuits. Furthermore, the proposed driver enables the possibility of sweeping the dimensions and other physical characteristics of the simulated circuits. Additionally, this study case describes the implementation of an optimization technique that enhances fine-tuning of the geometry for a particular spiral inductor structure simulated in a 3D EM-FEM solver module inside Power SI, namely, Cadence® Sigrity™ PowerSI 3D-EM® (PowerSI 3D-EM). Then this works compares the 3D simulation results from our simulator with the results reported in previous work in Sonnet, a 2.5D tool that uses the MoM. Finally, the propose study case presents an optimization case employing the Broyden-based input space mapping algorithm, better known as aggressive space mapping (ASM), to obtain the geometrical dimensions of the spiral inductor structure that satisfies some given design specifications.

Contents

Introduction

Full-wave and quasi-static electromagnetic (EM) simulation tools are frequently used for a wide variety of electromagnetic interference (EMI) and electromagnetic compatibility (EMC) problems. The most fundamental numerical methods of these EM simulation tools were developed in the 60s: finite elements method (FEM) [Silvester-69], method of moments (MoM) [Harring-67], and finite integration technique (FIT) [Microwaves101-15], among others. These methods allow simulations at a physical level, which provides great precision for circuits that work at frequencies up to GHz, or in more general terms, those circuits whose physical size is comparable with the wavelength of the maximum simulated frequency, which are called "electrically large" circuits. One of the latest full-wave EM tools introduced to the market is Cadence® Sigrity™ PowerSI® [Cadence-18a].

Chapter I presents a brief introduction to the full-wave EM simulator PowerSI, and the generation of a Matlab driver that manipulates the simulation set-up and enables the generation of microstrip structures in TCL format using a text editor.

Chapter II analyzes some high-frequency circuits, including microstrip filters and antennas, in order to test the Matlab driver. It specifies how the coordinates for each circuit are calculated and how to enable the parameterization option.

Chapter III focuses on the implementation of the aggressive space mapping (ASM) optimization technique for a spiral inductor optimal design, which includes the use of a coarse model, implemented in APLAC, versus a fine model, implemented in Power SI exploiting the proposed driver.

Finally, some general conclusions are presented, followed by an appendix with the most fundamental code for ASM optimization of the spiral inductor.

1.   PowerSI Full-Wave Simulator

The Cadence® Sigrity™ PowerSI® [Cadence-18a] simulator provides full-wave EM analysis of integrated circuit (IC) packages and printed circuit boards (PCBs). It is considered as a frequency-domain power- and signal-integrity tool. It is able to accurately simulate physical phenomena typically encountered in high-speed interconnects and power delivery networks, such as simultaneous switching noise (SSN), signal coupling, decoupling capacitor implementations, and catch design regions that are under or over target voltage levels. PowerSI is especially suitable for system level EM simulations, exhibiting fast computational speed with unique patented methodologies. PowerSI supports S, Z, and Y parameters model extraction and facilitates AC analysis to assess voltage distribution across ground planes. Additionally, Cadence® Sigrity™ PowerSI 3D-EM® (PowerSI 3D-EM) [Cadence-18b] is a 3D FEM full-wave solver inside PowerSI, this optional module uses the same GUI than PowerSI and provides accurate EM solutions for complicated three dimensional structures, the cloud-ready tool make use of the adaptive finite element mesh (FEM) refinement technology that provides faster simulation speed than other popular 3D tools.

The proposed Matlab driver for PowerSI and PowerSI 3D-EM is intended to provide parameterization capabilities to the user and avoids using the GUI by simulating the circuits in batch mode. This chapter is based on [Cabrera-Gómez-19].

1.1.   PowerSI Simulation Set-up

It is possible to assign the general options for the simulator in text format, including the file manager, the layout manager, and the simulation manager, among others. With a utility script in Matlab is generated an initial file with .SPD extension that can be opened with Power SI and loads the general simulation setup in text format, as illustrated in Fig. 1-1. This is comparable to filling out the PowerSI options tool in the software's GUI, as shown in Fig. 1-2.

```
Title - PowerSI file for version 2000.18
* PowerSI 17.1.5.08031.101946   005

* Please do NOT edit special void criteria manually.
.DoglegHoleThreshold = 0.001500
.ThermalHoleThreshold = 0.001500
.SmallHoleThreshold = 0.001500
.ViaHoleThreshold = 0.001500
.AutoDetectAntipad = Yes
.DropShapesOfUnselectedNets

* Transient description line
.Transient FinalTime = 0.216817ns Timesteps = 100 Viewstep = 1
+  DC = Yes Window = Yes PlaneSkinEffect = Yes TransmissionLineMetalLoss = No NonTotalReflectionBoundary = Yes
+  DielectricLossDispersion = No InterPlaneCoupling = No  ViaCapacitanceActivated = Yes  CoplanarLineModeling = No ShapeGapModeling = yes
+  AddFloatingPlaneIfNeeded = No AutoRLGCEnabled = No PassivityCheck = No Fmax_TransmissionLine = 5.00GHz  TransmissionLineMode = No DT = 2.16817e-12

* External Circuit Solver description lines

* Mesh description line
.Mesh Pkg = Package1 Mesh_X = 200 Mesh_Y = 200
```

Fig. 1-1   General simulation setup in text format generated in Matlab utility script.
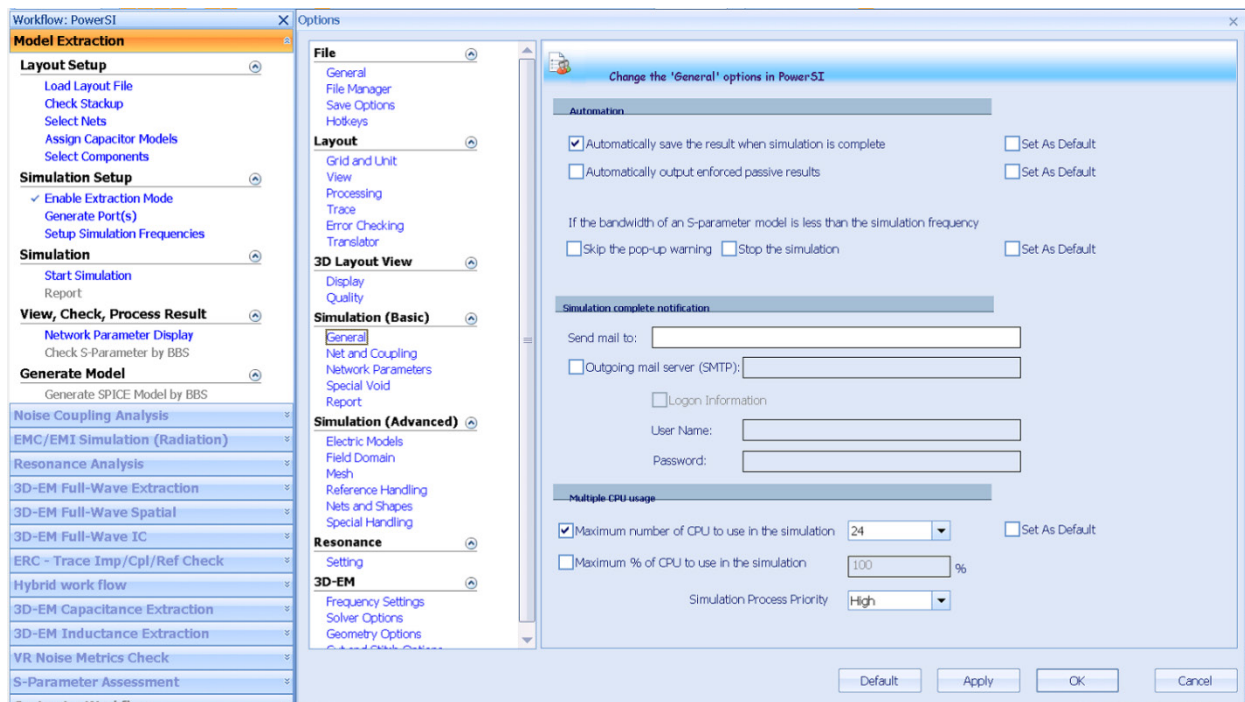

Fig. 1-2   General simulation setup using the PowerSI GUI.

The initial simulation set-up includes an adaptive meshing to ensure solution converge and accuracy. Its internal adaptive algorithm refines the mesh based on solutions from previous mesh and only those elements with high solution errors are redefined [Sigrity-11]. For all the microstrip structures analyzed in this report, the target delta S is 0.01, the adaptive refinement percentage is 20%, and the max number of adaptive mesh iterations is 20. Additionally, the finite element formulation in Power SI 3D-EM allows to choosing between zero-order elements, where E-field

is constant along edges and linear inside elements, or first-order elements, where E-field is linear along edges and quadratic inside elements. It was selected zero-order elements for all the simulations in this report.

1.2. Generating a Microstrip Structure

For the generation of a microstrip structure, the simulation software requires the location for each corner of the microstrip traces. Using a Matlab utility script, the X and Y locations for all the vertices of the circuits are automatically generated. It is important to mention that the coordinates must be selected in the correct order and the nodes must be adjacent to each other.

Unlike the traditional Signal Integrity (SI) tools that treat power and ground planes as ideal with fixed voltages, the PowerSI simulator fully considers all signal and plane effects simultaneously, which requires the construction of a ground plane as a reference for the microstrip circuit; the ground plane size is calculated based on the physical dimensions of the circuit.

1.3. Simulation Boundary Box

The analysis of planar circuits can be performed in a completely enclosed environment: a metallic shielding with a rectangular box. The size for the boundary box should be calculated based on the dimension of the microstrip structures to be simulated; it is recommended to be about twice the X and Y dimensions and about four times the height of the dielectric substrate. These recommendations are usually adequate to avoid EM interaction between the microstrip structure and the enclosing box, as well as to avoid box resonances; however, they can be adjusted for each particular application according to the microstrip circuit geometry and the simulated frequency band.

1.4. Parameterization

Once the initial file with .SPD extension is created, a Matlab utility script modifies the file and includes the dimensions of the structure as a set of variables, which allows the user to perform

parameterized models. This initial file modified by Matlab also includes the material properties, such as the dielectric constant, conductivity, etc., which can also be parameterized.

1.5.   Power SI in Batch Mode

In order to execute Power SI in batch mode [Sigrity-11], it is needed to open Cadence's Sigrity Suite Manager Window. After that, it is possible to execute the program from Matlab with simple commands, as demonstrated in Fig. 1-3. The user can select between PowerSI and PowerSI 3D-EM solver by only modifying the PSIexe variable in Matlab.

```
PSIexe = 'C:\Cadence\Sigrity2017\tools\bin\PowerSI.exe';
PSIProjectFileName = [out_Filename '.spd'];
system([PSIexe ' /b ' PSIProjectFileName]);
```

Fig. 1-3   Matlab commands to enable PowerSI in batch mode.

2. Driver Application Cases

In this Chapter, the full-wave EM simulation of several high-frequency planar structures is realized in PowerSI by exploiting the proposed Matlab driver. First, a microstrip patch antenna is simulated, explaining in detail the required Matlab files. Secondly, a couple of basic microstrip filters are simulated using the Matlab driver for PowerSI. The material in this chapter is based on [Cabrera-Gómez-19].

2.1. Edge-Fed Patch Antenna

Consider the edge-fed patch antenna illustrated in Fig. 2-1 [Grayaa-05]. It uses a relative dielectric constant $\varepsilon_r = 2.2$, a substrate height $H = 0.8$ mm, and a negligible loss tangent.
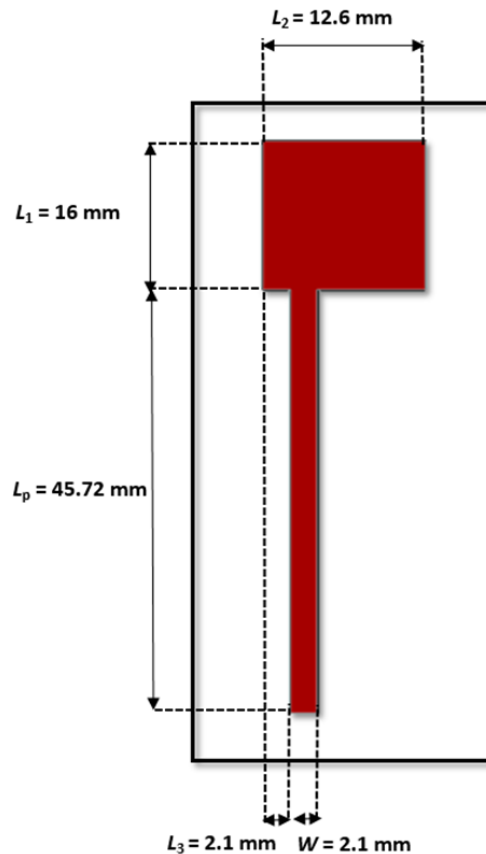

Fig. 2-1   Dimensions edge-patch antenna.

2. DRIVER APPLICATION CASES

Using the coordinates from the utility script for the edge-fed patch antenna and selecting the nodes order as showed in Fig. 2-2 , it is possible to obtain the X and Y coordinates of the microstrip structure vertices; the results are shown in Table I.
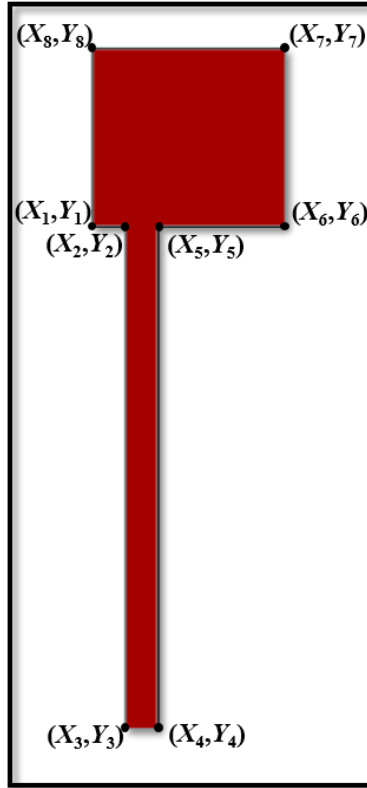

Fig. 2-2   Generic coordinates for vertices for the edge-patch antenna.

TABLE I
EDGE-FED PATCH COORDINATES CALCULATION
$W = 2.1$ mm, $L_1 = 16$ mm, $L_2 = 12.6$ mm, $L_3 = 2.1$ mm, $L_p = 45.72$ mm

X	Y	X value	Y value
X_1	Y_1	0	0
X_2	Y_2	L_3	0
X_3	Y_3	L_3	$-L_p$
X_4	Y_4	$L_3 + W$	$-L_p$
X_5	Y_5	$L_3 + W$	0
X_6	Y_6	L_2	0
X_7	Y_7	L_2	L_1
X_8	Y_8	0	L_1

The initial file base.spd is generated in Matlab and it contains the general simulation setup in text format, however, it does not contain any characteristic of the microstrip circuit, as shown in Fig. 2-3 . Subsequently, the Fun_build_SPD.m utility script uses the base file to generate a new file, for this example EdFed_patch.spd, where all dimensions are added as well as the characteristics for the edge fed rectangular patch antenna, as illustrated in Fig. 2-4 .

For this particular case, the stack-up information assumes copper for metal layer (1⁄2 oz = 0.7 mils = 17.78 um). Plane02 layer is used for the microstrip circuit, Medium01 for dielectric and Plane01 for ground reference.

```
* Shape description lines

* Layer description lines

* Node and Trace description lines

* Via description lines
```

Fig. 2-3   Portion of initial SPD file in text format, without including the microstrip characteristics.

```
* Shape description lines
.Shape Shape001
Box01::VSS+ -12.600000mm -107.440000mm 37.800000mm 185.160000mm
.EndShape

.Shape Shape002
Polygon01::VCC+ 0.000000mm 0.000000mm 2.100000mm 0.000000mm 2.100000mm -45.720000mm 4.200000mm -45.720000mm
+ 4.200000mm 0.000000mm 12.600000mm 0.000000mm 12.600000mm 16.000000mm 0.000000mm 16.000000mm
.EndShape

* Layer description lines
Plane02 Thickness = 17.780000u Conductivity = 58000000.000000 Shape = Shape002 Color = 0xA00000 Permittivity = 1
Medium01 Thickness = 800.000000u Conductivity = 0 Permittivity = 2.200000 LossTangent = 0.000000e+000
Plane01 Thickness = 17.780000u Conductivity = 58000000.000000 Shape = Shape001 Color = 0xA00000 Permittivity = 1

* Node and Trace description lines
Node05::VCC X = 3.310000mm Y = -45.620000mm Layer = Plane02
Node06::VSS X = 3.310000mm Y = -45.620000mm Layer = Plane01

* Via description lines
```

Fig. 2-4   Portion of SPD file in text format modified in Matlab, including the characteristics for the edge-fed patch antenna.

2. DRIVER APPLICATION CASES

In Fig. 2-5 it is shown the magnitude of the reflection coefficient S_{11} obtained directly from PowerSI and manipulated and plotted in Matlab, for this edge fed antenna. There is some small discrepancy at the resonant frequency with respect that one reported in [Grayaa-05], from which the antenna dimensions were obtained. Nevertheless, this is expected since there are differences in the numerical calculation methods and the meshing resolutions used in [Grayaa-05] and those used in our Power SI simulation.
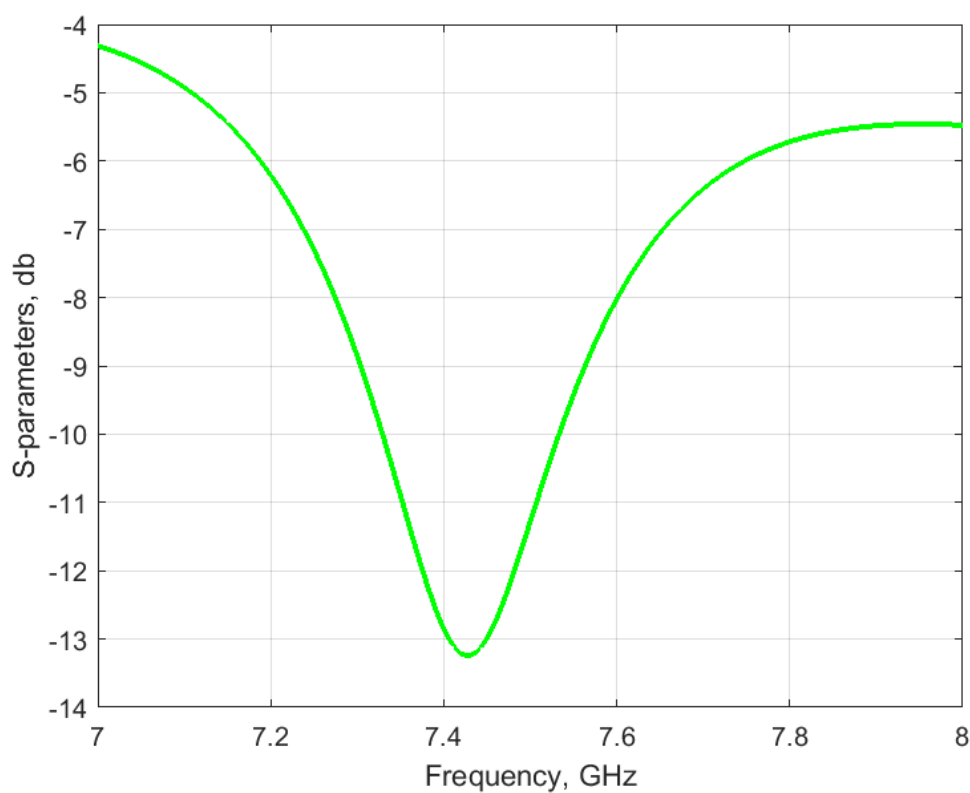

Fig. 2-5   Plot of the magnitude of the reflection coefficient S_{11} for the Edge-fed patch antenna generated with the Matlab driver.

2.2. Microstrip Filters

2.2.1   Microstrip Low-Pass Filter

Consider now the low-pass microstrip filter depicted in Fig. 2-6 [Sheen-90], [D'Inzeo-79], whose simulation is done in the frequency range 2-12 GHz. We use the following dimensions: W_1 = 2.413 mm, W_2 = 2.54 mm, L_1 = 5.65 mm, L_2 = 12.257 mm, L_p = 10 mm. It uses a relative dielectric
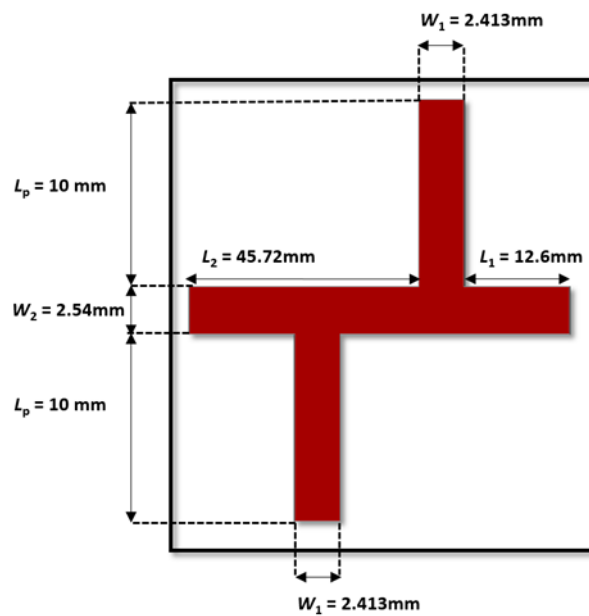

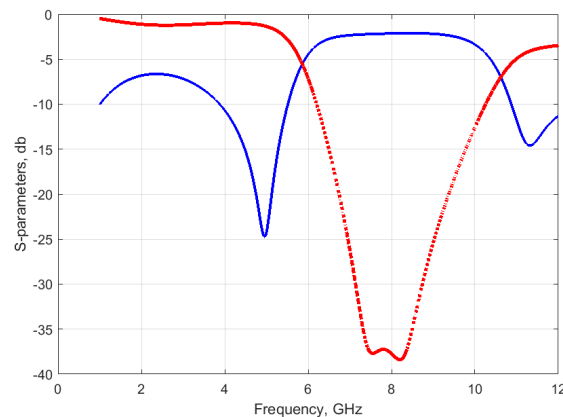Fig. 2-6   Dimensions for the microstrip low pass filter.

Fig. 2-7   Plot of scattering parameters $|S_{11}|$ (continued blue trace) and $|S_{21}|$ (dotted red trace) generated from Matlab driver for the microstrip low pass filter.

2. DRIVER APPLICATION CASES

TABLE II
LOW PASS FILTER COORDINATES CALCULATION
$W_1 = 2.413$ mm, $W_2 = 2.54$ mm, $L_1 = 5.65$ mm, $L_2 = 12.257$ mm, $L_p = 10$ mm

X	Y	X value	Y value
X_1	Y_1	0	0
X_2	Y_2	L_2	0
X_3	Y_3	L_2	0
X_4	Y_4	$L_2 + W_1$	L_p
X_5	Y_5	$L_2 + W_1$	0
X_6	Y_6	$L_2 + W_1 + L_1$	0
X_7	Y_7	$L_2 + W_1 + L_1$	$-W_2$
X_8	Y_8	$W_1 + L_1$	$-W_2$
X_9	Y_9	$W_1 + L_1$	$-W_2 - L_p$
X_{10}	Y_{10}	L_1	$-W_2 - L_p$
X_{11}	Y_{11}	L_1	$-W_2$
X_{12}	Y_{12}	0	$-W_2$

constant $\varepsilon_r = 2.2$, a substrate height $H = 794$ mm, and a negligible loss tangent. Table II shows the formulas used by our coordinate's calculator utility script, and Fig. 2-7 shows the magnitude of the scattering parameters S_{11} (blue) and S_{21} (red) obtained from our Matlab driver.

2.2.2 Microstrip Band-Pass Filter

Consider a microstrip bandpass filter depicted in Fig. 2-8. This filter has been reported in [Hennings-06], [Koziel-06], and [Gutiérrez-Ayala-10] with some small variations. Its simulation is done for frequencies between 4.5-5.5 GHz. The dielectric substrate is made of Alumina with a relative dielectric constant $\varepsilon_r = 9$, a substrate height $H = 0.66$ mm, and a negligible loss tangent. All microstrip lines used in the filter have the same width $W = 0.7$ mm. The feeding microstrip lines have a length $L_p = 10W$. Main resonators have lengths $L_1 = 6.6096$ mm, $L_2 = 4.6042$ mm, $L_3 = 6.2821$ mm, and $L_4 = 4.8314$ mm, with a separating gap $S_g = 0.1271$.

12

In contrast with the previous two examples, this band-pass filter consists of two separated polygons. This means that it is necessary to generate coordinates for both polygons: 8 pairs of coordinates and values for each polygon. We assume X_9 as the initial coordinate for the second polygon and it is equivalent to $X_9 = L_1 + W + L_2 + S_g$. Table III includes the calculation formulas for both polygons and Fig. 2-9 shows the magnitude of the scattering parameters S_{11} (blue) and S_{12} (red) obtained from our Matlab driver.
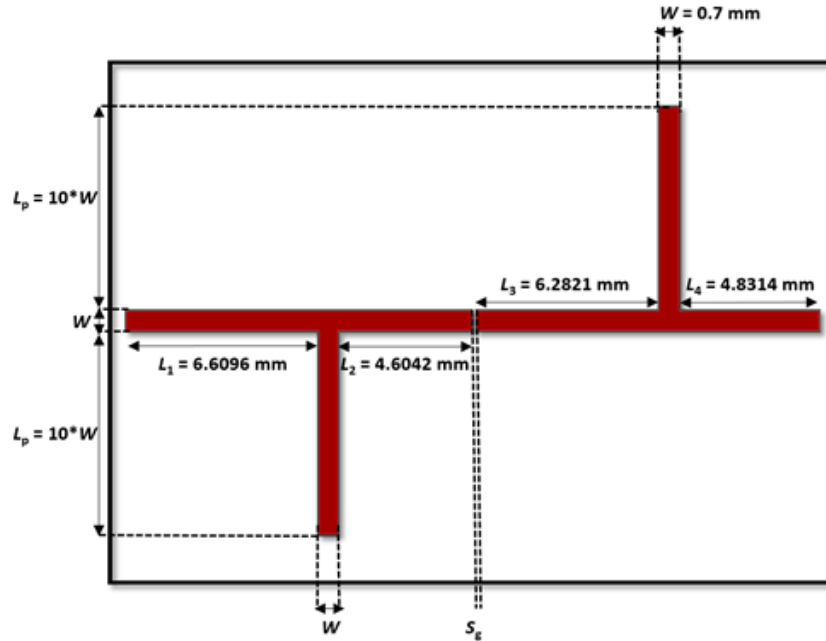

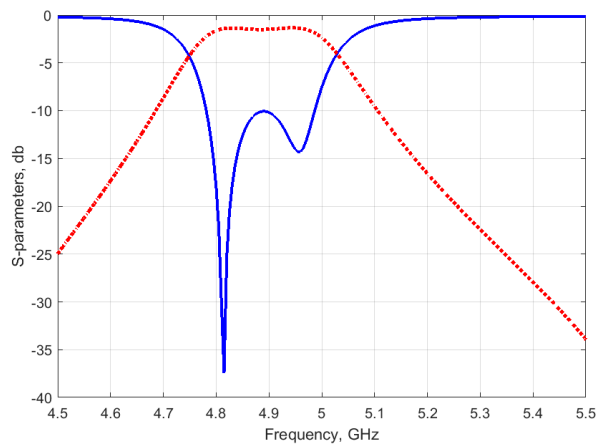Fig. 2-8   Dimensions for the microstrip low pass filter.

Fig. 2-9   Plot of scattering parameters $|S_{11}|$ (continued blue trace) and $|S_{21}|$ (dotted red trace) generated from Matlab driver for the microstrip low pass filter.

2. DRIVER APPLICATION CASES

TABLE III
BAND PASS FILTER COORDINATES CALCULATION
$W = 0.7$ mm, $L_p = 10W$, $L_1 = 6.6096$ mm, $L_2 = 4.6042$ mm, $L_3 = 6.2821$ mm, $L_4 = 4.8314$ mm,
$S_g = 0.1271$ mm

X	Y	X value	Y value
X_1	Y_1	0	0
X_2	Y_2	L_1	0
X_3	Y_3	L_1	$-L_p$
X_4	Y_4	$L_1 + W$	$-L_p$
X_5	Y_5	$L_1 + W$	0
X_6	Y_6	$L_1 + W + L_2$	0
X_7	Y_7	$L_1 + W + L_2$	W
X_8	Y_8	0	W
X_9	Y_9	$L_1 + W + L_2 + S_g$	0
X_{10}	Y_{10}	$X_9 + L_3 + W + L_4$	0
X_{11}	Y_{11}	$X_9 + L_3 + W + L_4$	W
X_{12}	Y_{12}	$X_9 + L_3 + W$	W
X_{13}	Y_{13}	$X_9 + L_3 + W$	$W + L_p$
X_{14}	Y_{14}	$X_9 + L_3$	$W + L_p$
X_{15}	Y_{15}	$X_9 + L_3$	W
X_{16}	Y_{16}	X_9	W

3. Design Optimization Case

This section focuses on the exploitation of the proposed Matlab driver to implement an optimization technique for designing microstrip structures simulated in the 3D full-wave electromagnetic field solver PowerSI. The optimization method implemented is a space mapping (SM) method, more specifically the aggressive space mapping (ASM) approach to design. A probe of concept is done by simulating a spiral inductor for its EM optimization. The material in this chapter is based on [Leal-Romo-17a] and [Leal-Romo-17b].

3.1. Spiral Inductor Design

In this section we discuss the design of a 3D spiral inductor structure for high-power applications in PowerSI 3D-EM, which will be used as the fine model for the ASM optimization to adjust its geometrical dimensions to meet the power design specification. To perform such task, the spiral inductor is based on inductor 5 from [Froglu-11], which is illustrated in Fig. 3-1. The parameters and material properties of the inductor structure are described in Table IV.

The inductance value of interest is measured at 13.5 MHz, which is a common frequency for high power Industrial, Scientific and Medical (ISM) applications. However the resonance frequency point is expected to happen at around 90 MHz, the table V summarizes the parameters obtained from our PowerSI 3D-EM simulation against the simulated parameters reported in
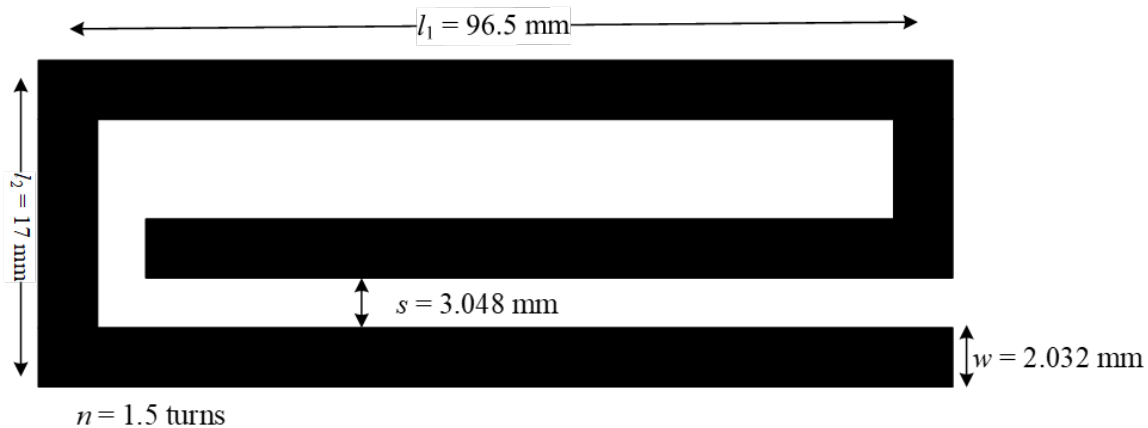

Fig. 3-1   Spiral inductor geometry and dimensions for high-power applications [Froglu-11].

3. DESIGN OPTIMIZATION CASE

TABLE IV
SPIRAL INDUCTOR PHYSICAL PARAMETERS AND MATERIAL PROPERTIES

Variable Name and Units	Initial Value
L_1 (mm)	96.5
L_2 (mm)	17
S (mm)	3.048
W (mm)	2.032
H (mm)	2.54
t (mm)	0.03048
n (turns)	1.5
ε_r (FR4)	9.8
ρ (CuAu)	0.7066
σ (S/m)	5.8×10^7
$\tan \delta$	0

TABLE V
SONNET *VS* PSI-3D RESUTLS

Simulator	L (nH) @ 13.5MHz	F_r (MHz)
Sonnet [5]	132.78	92.5
PSI-3D	130.5	89.12

[Froglu-11]. As it is observed in Table V, both numerical results are very similar. In our case the inductance measured at 13.5 MHz is off by 2 nH and there is a small difference of almost 3 MHz in the resonance frequency, with respect to Sonnet's simulation results. This slight discrepancy in the results can be attributed to the nature of the simulators employed. In our case, we used a zero order model from a 3D FEM solver, while [Froglu-11] employs Sonnet, a 2.5D tool that uses the MoM.

3.2. Optimizing Spiral Inductor using Aggressive Space Mapping

The Matlab driver is implemented for the optimization of the spiral inductor by executing

the Aggressive Space Mapping (ASM) algorithm [Bandler-95], [Rayas-Sanchez-16] as depicted in Fig. 3-2 . This algorithm starts by optimizing a coarse model of the original structure, denoted as $R_c(x_c)$, to optimally satisfy some given specifications. The coarse model response is in R_c, while the coarse model design parameters are in x_c. Then, the optimal design of the coarse model, x_c^*, is introduced as new geometrical parameters to PSI-3D, to run a simulation of the fine model $R_f(x_f)$. With this information, the algorithm reads the S-parameters from the fine model contained in vector R_f using our driver and converts them into Z-parameters. Next, coarse model design parameters are extracted and converted into Z-parameters to match the current fine model response. If the extracted parameters are sufficiently close to the optimal coarse model design, the algorithm ends, otherwise, it continues by updating the fine model design parameters using Broyden's formula [Broyden-65], simulating again the PSI-3D structure with the new geometrical parameters.
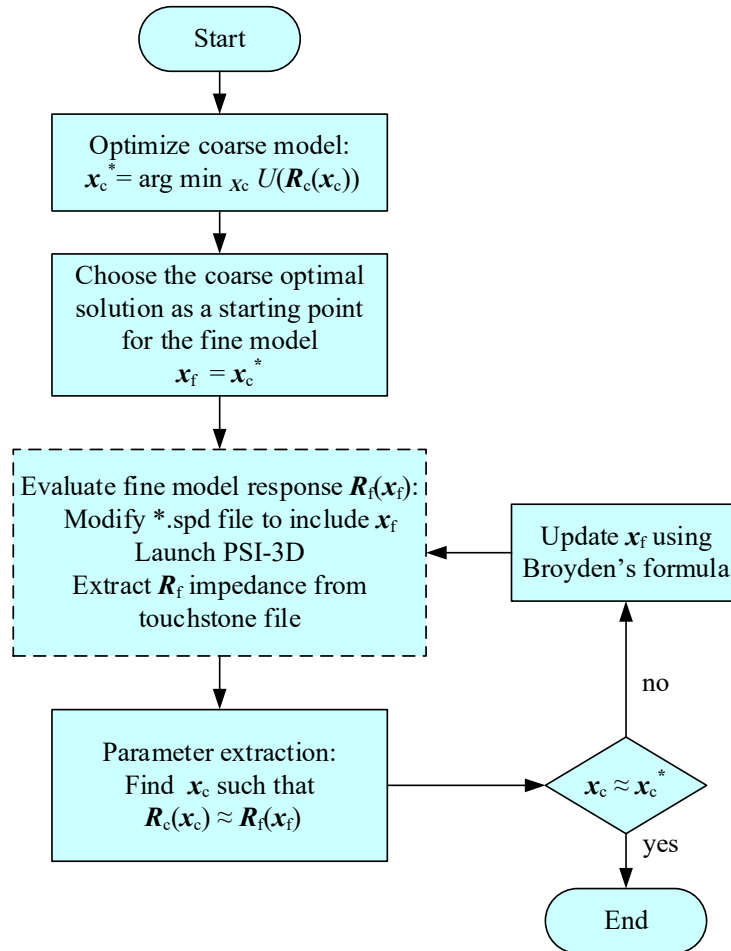
Fig. 3-2   Aggressive space mapping (ASM) algorithm as used with our Matlab-PSI-3D driver.

3. DESIGN OPTIMIZATION CASE

3.2.1   Coarse and Fine Models

As we mentioned in section 3.2 the ASM optimization method requires the use of two models, for this application the coarse model is created in APLAC, a high-frequency circuit simulator, and the fine model in PowerSI 3D-EM. The proposed methodology efficiently drives the coarse model (APLAC) directly from Matlab, as well as the fine model (PowerSI 3D-EM), without utilizing the software graphical user interface (GUI).

The coarse model implemented in APLAC is shown in Fig. 3-3. One simulation of this inductor in APLAC takes less than a second using a Xeon® computer server.

A 3D view of the spiral inductor structure as implemented in PowerSI 3D-EM is depicted on Fig. 3-4. It is the same spiral inductor built in APLAC with all geometrical dimensions parameterized, such that they can be altered by the optimization algorithm implemented in Matlab. The inductor is contained inside a boundary box at least four times taller than the total height of the substrate plus the inductor and the reference plane and two times wider and larger than the spiral inductor structure, such that the electromagnetic fields in the structure do not interact with the enclosing box. Two 50-Ω lumped ports perpendicular to the input-output of the inductor are
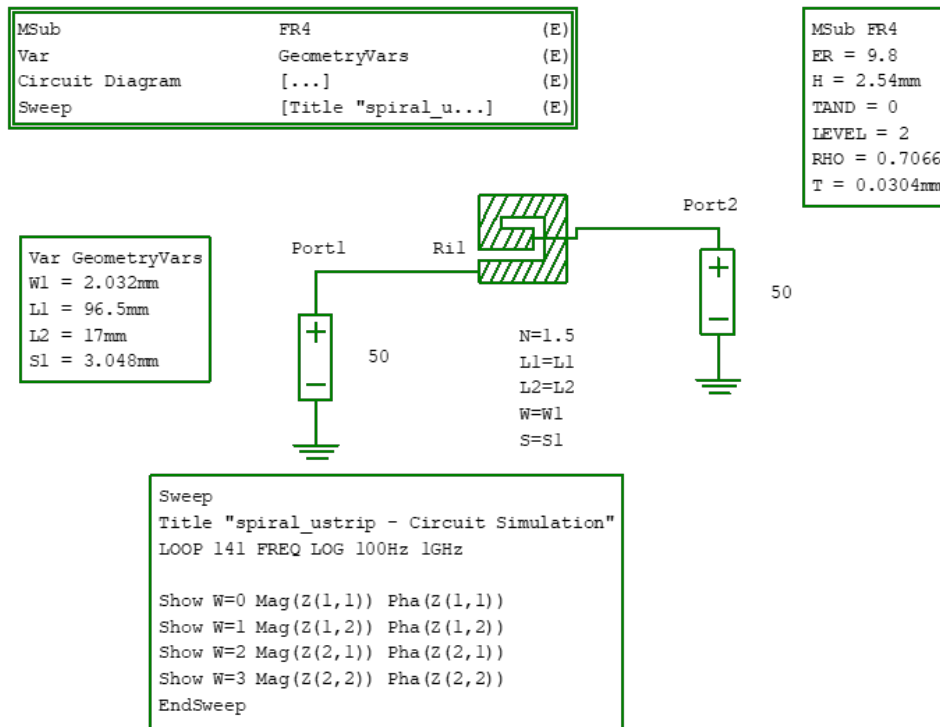

Fig. 3-3   Spiral inductor circuit as implemented in APLAC, taken here as the coarse model.
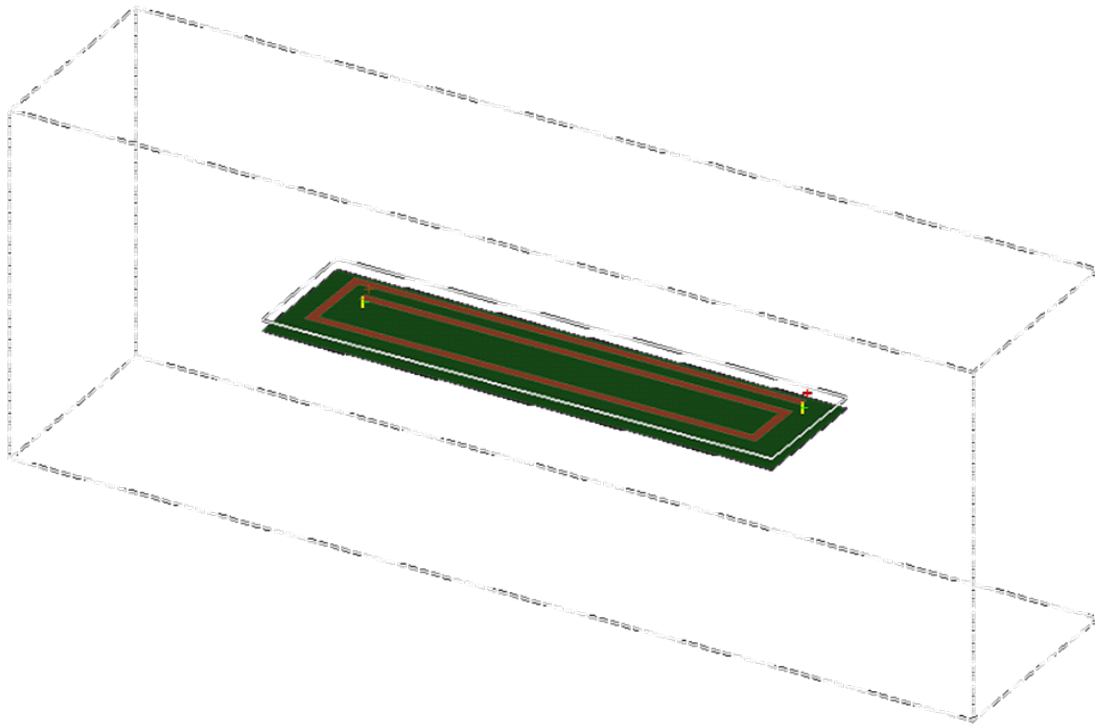
18

Fig. 3-4   Implementation of the 1.5 turns spiral inductor (fine model) in PSI-3D.

employed. One simulation (one frequency sweep from 100 Hz to 1 GHz with 141 frequency points) of this inductor in PowerSI 3D-EM consumes 5:47 minutes with the same server computer used for the coarse model, including the construction of the initial mesh, one adaptive iteration, with two minimum number of converged iterations per simulation.

One more benefit of non-GUI operation is the time saved by not having to load the project for graphical display, since, this operation can take long enough to impact the overall execution time.

3.2.2   Design Specifications

To comply with our own power delivery application, the optimization problem was formulated to meet the following specifications:

$$L_{AC} \leq 115 \text{ nH for } 10 \text{ MHz} \leq f \leq 16 \text{ MHz} \tag{3-1}$$

$$R_{AC} \geq 90 \text{ m}\Omega \text{ for } 100 \text{ Hz} \leq f \leq 1 \text{ kHz} \tag{3-2}$$

3. DESIGN OPTIMIZATION CASE

The subsequent parameters were used as optimization variables: $x = [w\ L_1\ L_2\ s]^T$, keeping fixed the following preassigned parameters: $y = [H\ t\ n\ \varepsilon_r\ \rho\ \sigma\ \tan\delta]^T$.

3.2.3 Space Mapping Techniques

As a result of applying ASM, the structure was optimized with 181 APLAC simulations using the fminsearch function in Matlab, and only 8 PowerSI 3D-EM simulations. Final dimensions of this structure are shown in Table VI, as well as the starting point for optimization (x_c^*). During the execution of this optimization method, no GUI interface was required.

3.2.4 Reading Results from PowerSI

Once PowerSI 3D-EM finished the simulation, it generates a Touchstone file containing a matrix of S-parameters, in this application case a ".s2p" file is generated. After that we read the S-parameters in Matlab and convert them into impedance parameters (Z-parameters).

Finally, the port at the output of the inductor is open-circuited to measure inductance and resistance parameters along the entire frequency range using the following equations:

$$\begin{bmatrix} V_1 \\ V_2 \end{bmatrix} = \begin{bmatrix} z_{11} & z_{12} \\ z_{21} & z_{22} \end{bmatrix} \begin{bmatrix} I_1 \\ I_2 \end{bmatrix} = [z] \begin{bmatrix} I_1 \\ I_2 \end{bmatrix} \tag{3-3}$$

$$z_{in} = z_{11} = \left. \frac{V_1}{I_1} \right|_{I_2=0} \tag{3-4}$$

where $[z]$ represents the complex impedance matrix of the form $R + jX$; where R is the real part or resistance and X is the imaginary part or inductor's impedance, and z_{11} is the input impedance when the output port is open.

The resulting L_{AC} and R_{AC} meet the desired behavior described by (3-1) and (3-2), as confirmed in Fig. 3-5 and Fig. 3-6 respectively.
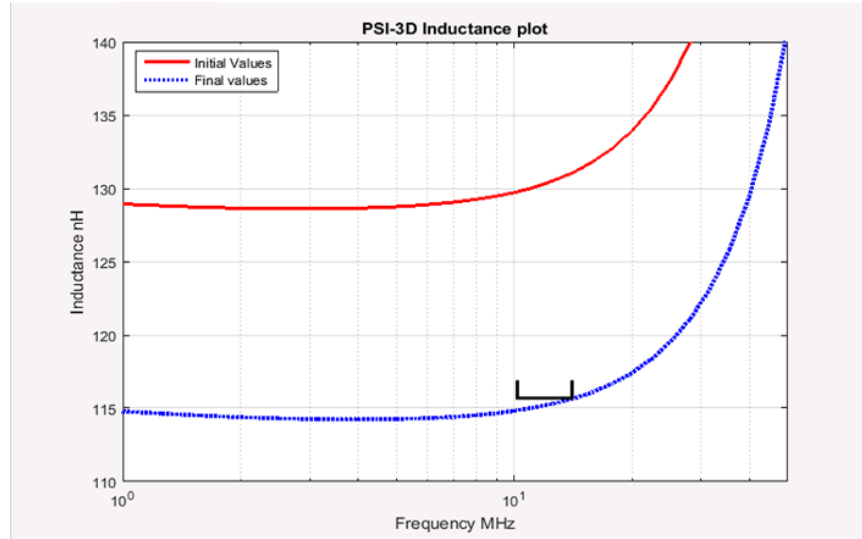
Fig. 3-5   Spiral inductor EM responses. L_{AC} before (red) and after (dotted blue) optimization.
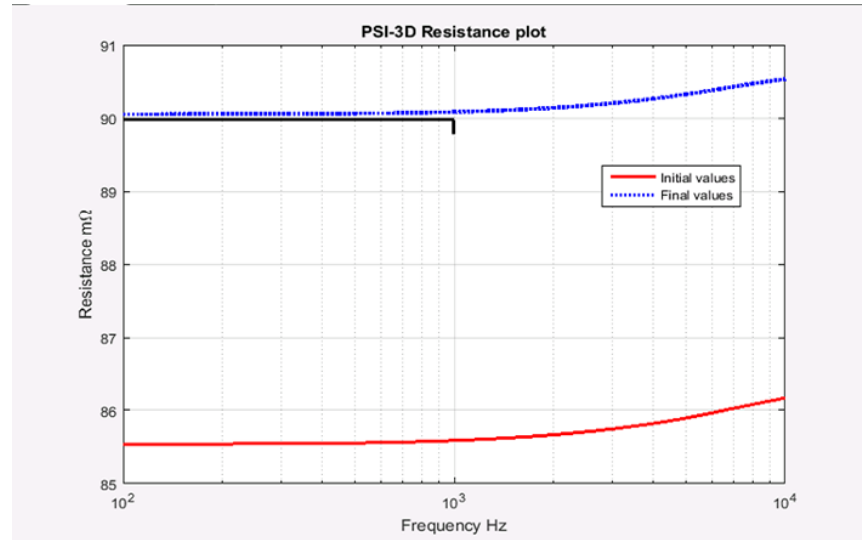

Fig. 3-6   Spiral inductor EM responses. R_{AC} before (red) and after (dotted blue) optimization.

TABLE VI
DESIGN PARAMETERS VALUES BEFORE AND AFETER OPTIMIZATION

Variable Name and Units	Initial Values	Final Values
L_1 (mm)	96.5	83.11
L_2 (mm)	17	16.22
S (mm)	3.048	4.48
W (mm)	2.032	1.659

Conclusions

This study case presented the generation of a driver for parametrized full-wave electromagnetic (EM) simulations in PowerSI driven from Matlab. This driver effectively creates text files with the dimensions and other characteristics of microstrip circuits and planar antennas. It also allows the parameterized simulation in PowerSI from Matlab of such structures. The proposed tool can be very useful to perform parametric studies and numerical optimization.

Additionally, the proposed Matlab driver for PowerSI was successfully tested by exploiting it in the implementation of the space mapping methodology for optimizing a microstrip spiral inductor structure in the 3D EM-FEM solver from PowerSI. This SM optimization methodology was tested with the successful implementation of the aggressive space mapping algorithm (or Broyden-based input space mapping) to fine tune the geometry of the spiral inductor structure using APLAC's equivalent circuit as our coarse model. The optimization achieved the desired results for the equivalent inductance and resistance in specific frequency ranges, with a few iterations of the fine model implemented in PowerSI 3D-EM, while the coarse model required more than a hundred of simulations to achieve the desired goal, consuming just a few minutes.

Appendix

A. MATLAB CODE FOR AGGRESSIVE SPACE MAPPING OPTIMIZATION

```matlab
clear all;
clc;
echo on;

% x  = [W1,L1,L2,S1], design variables
x=[2.032 98.532 19.032 3.0480];
Lf=100;
x_tol = 1e-9;
h=[0; 0; 0; 0];
B=eye(max(size(x)));



% Circuit pre-assigned parameters
% z  = [H, Er, TAND, roCu, thck], pre-assigned parameters
z = [2.54 9.8 0.0 0.7066 30.48e-3];
% Simulation parameters
% Ps = [units IF FF FP], parameters of the simulator
Ps = [2 100 1 141];
% xf_4 = [2.225293328   93.34855682 18.39860019 3.008082083];

% Getting circuit response in Aplac
[fi,Ri,Li] = espiral_Aplac(x, z, Ps);

%-- Find optimal response using the coarse model
options = optimset('MaxFunEvals',1000,'MaxIter',1000,'TolX',1e-3,'TolFun',1e-
3);
[xc_ini,FunVal1,EF1,output1]=fminsearch('Obj_espiral_Aplac',x,options);
%
[xc_ini,FunVal1,EF1,output1]=fminimax('Obj_espiral_Aplac',x,[],[],[],[],lb,ub
,[],options);
[fc,Rc,Lc] = espiral_Aplac(xc_ini, z, Ps);

% -- Coarse model optimal solution is the starting point for fine model
xf=xc_ini;
count=0;

while (abs(Lf(103)-Li(103)))>x_tol

% -- Calculate fine model response

    In_Filename='spiral2';
    Out_Filename='spiral2_out';
    [ff,Rf,Lf] = Fun_build_ACI(In_Filename,Out_Filename,xf);
    count=count+1;

%-- Extract xc so that Lc ~= Lf using fminsearch.
```

```matlab
    Coarse_to_fine_OF = @(x) Coarse_to_fine_ERROR(Rf,Lf,x);
    options = optimset('MaxFunEvals',1000,'MaxIter',1000,'TolX',1e-
3,'TolFun',1e-3); %'TolFun',0.00001);
    xc = fminsearch(Coarse_to_fine_OF,xf,options);

%-- Broyden type variables update

    f = xc - xc_ini;
    B = B + (f'*h')/((h'*h)+1e-6);
    h= -B\f';
    xf = xf + h';

end

subplot(2,1,1);
plot(fi/1e9,Ri,'-r');
plot(f/1e9,R,'-r',f2/1e9,R2,'-b');
loglog(f,R,f2,R2);
semilogx(f,R,f2,R2);
grid on;

subplot(2,1,2);
plot(fi/1e9,Li,'-r');
plot(f/1e9,L,'-r',f2/1e9,L2,'-b');
loglog(f,L,f2,L2);
semilogx(f,L,f2,L2);
grid on;
```

Bibliography

[Bandler-95]   J. W. Bandler, R. M. Biernacki, S. H. Chen, R. H. Hemmers, and K. Madsen, "Electromagnetic optimization exploiting aggressive space mapping," *IEEE Trans. Microwave Theory Tech.*, vol. 43, pp. 2874-2882, Dec. 1995.

[Broyden-65]   C. G. Broyden, "A class of methods for solving non-linear simultaneous equations", *Math. Comp.*, vol. 19, pp. 577-593, 1965.

[Cabrera-Gómez-19]   M. Cabrera-Gómez and J. E. Rayas-Sánchez, "Matlab driver for parameterized full-wave EM simulations in PowerSI of microstrip circuits," Internal Report *CAECAS-19-06-R*, ITESO, Tlaquepaque, Mexico, Jun. 2019.

[Cadence-18a]   Cadence. (2018). *Cadence Sigrity PowerSI: Frequency-Domain Power- and Signal-Integrity Analysis* [Online]. Available: https://www.cadence.com/content/dam/cadence-www/global/en_US/documents/tools/pcb-design-analysis/sigrity-powersi-ds.pdf.

[Cadence-18b]   Cadence. (2018). *Cadence Sigrity PowerSI 3D EM Extraction Option* [Online]. Available: https://www.cadence.com/content/dam/cadence-www/global/en_US/documents/tools/ic-package-design-analysis/sigrity-powersi-3d-em-extraction-ds.pdf.

[D'Inzeo-79]   G. D'Inzeo, F. Giannini, and R. Sorrentino, "Novel microwave integrated low-pass filters," *Electron. Lett.*, vol. 15, no. 9, pp. 258-260, Apr. 26, 1979.

[Froglu-11]   A. Eroglu, "Planar inductor design for high power applications", *Progress in Electromagnetics Research B,* vol. 35, pp. 53-67, Oct. 2011.

[Grayaa-05]   K. Grayaa, T. Aguili, and A, Bouallegue, "Multiresolution source method for the analysis of planar microstrip structures," *Journal of Microwaves and Optoelectronics*, vol. 4, no. 2, pp. 111-122, Dec. 2005.

[Gutiérrez-Ayala-10]   V. Gutiérrez-Ayala and J. E. Rayas-Sánchez, "Neural input space mapping optimization based on nonlinear two-layer perceptrons with optimized nonlinearity," *Int. J. RF and Microwave CAE*, vol. 20, pp. 512-526, Sep. 2010.

[Harrington-67]   R.F. Harrington, "Matrix methods for fields problems", *Proc. IEEE*, vol. 55, no. 2, pp.136-149, Feb. 1967.

[Hennings-06]   A. Hennings, E. Semouchkina, A. Baker, and G. Semouchkin, "Design optimization and implementation of bandpass filters with normally fed microstrip resonators loaded by high-permittivity dielectric," *IEEE Trans. Microwave Theory Tech.*, vol. 54, no. 3, pp. 1253–1261, Mar. 2006.

[Leal-Romo-17a]   F. J. Leal-Romo, M. Cabrera-Gómez, D. M. García-Mora, and J. E. Rayas-Sánchez, "Design optimization of a 3D spiral inductor using space mapping," Internal Report *PhDEngScITESO-17-06-R (CAECAS-17-05-R)*, ITESO, Tlaquepaque, Mexico, May 2017.

[Leal-Romo-17b]   F. Leal-Romo, M. Cabrera-Gómez, J. E. Rayas-Sánchez, and D. M. García-Mora, "Design optimization of a planar spiral inductor using space mapping," in *Int. Conf. Electrical*

Performance of Electronic Packaging and Systems (EPEPS 2017), San Jose, CA, Oct. 2017, pp. 1-3.

[Microwaves101-15] Microwaves101. (2015). *Finite Integration Technique* [Online]. Available: https://www.microwaves101.com/encyclopedias/1163-finite-integration-technique.

[Rayas-Sanchez-16] J. E. Rayas-Sanchez, "Power in simplicity with ASM, tracing the aggressive space mapping algorithm over two decades of development and engineering applications," *IEEE Microwave Magazine*, vol. 17, pp. 64-76, Apr. 2016.

[Sigrity-11] *PowerSI User's Guide 11.1*, Sigrity Incorporated, CA, USA, 2011.

[Silvester-69] P. Silvester, "A general high-order finite-element waveguide analysis program", *IEEE Trans. Microwave Theory Tech.*, vol. 17, no. 4, pp. 204-210, Apr. 1969.

[Sheen-90] D. M. Sheen, S. M. Ali, M. D. Abouzahra and J. A. Kong, "Application of the three-dimensional finite-difference time-domain method to the analysis of planar microstrip circuits," *IEEE Trans. Microwave Theory Tech.*, vol. 38, pp. 849-857, July 1990.

[Koziel-06] S. Koziel, J. W. Bandler and K. Madsen, "A space-mapping framework for engineering optimization - theory and implementation," *IEEE Trans. Microwave Theory Tech.*, vol. 54, pp. 3721-3730, Oct. 2006.