Simulations of Selected Devices and their Couplings using Existing Tools

Wim Schoenmaker, Peter Meuris
Rick Janssen, Wil Schilders
Monica Selva Soto, Caren Tischendorf

MAGWEL, NXP, Köln University

February 4, 2009
Simulations of Selected Devices and their Couplings using Existing Tools

Wim Schoenmaker, Peter Meuris
(MAGWEL)

Rick Janssen, Wil Schilders (NXP)

Monica Selva Soto, Caren Tischendorf
(University of Cologne)

February 4, 2009

Contents

1 Introduction 4

2 Review of the electromagnetic modeling 5
  2.1 Use of potentials .......................... 5
  2.2 S-parameters, Y-parameters, Z-parameters .... 7

3 A simple conductive rod 8
  3.1 Why is it interesting? ...................... 8
  3.2 Simulation set up ........................... 9
  3.3 Results .................................... 9
  3.4 What did we learn? .......................... 12

4 Strip line above a conductive plate 12
  4.1 Why is it interesting? ...................... 12
  4.2 Simulation set up ........................... 14
  4.3 Results .................................... 15
  4.4 Effects of finite strip thickness .............. 17
    4.4.1 Finite $t_M$ results .................... 18
  4.5 Running the adapter ........................ 19
  4.6 Simulations with Opera - VectorFields ....... 20
  4.7 What did we learn? .......................... 32
1 Introduction

This document describes the work performed in WP3 Task 3.1. In order to support compact model building, we need ab-initio field solving to verify / falsify approximations that are made, while building the compact models. In the frequency domain, a compact modeling procedure was developed in the projects CODESTAR and CHAMELEON-RF, such a procedure was not yet successfully implemented in the time domain. The main stumbling block is the so called Courant limit\(^1\) which requires unrealistically small time steps for progressing in time.

In this work package, the transient regime is addressed by implicit methods. This means that a time step is integrated using both the end-point as well as the begin point of the time interval. Our method is inspired by the transient simulation technique which is used in Technology Computer Aided Design (TCAD). In particular, there a backward finite difference is combined with the trapezoidal rule, leading to a rather enhanced time-step size.

However, the construction of the transient solver does not start from scratch but is build on available software tools that are available from MAGWEL and the University of Köln and NXP. Starting from these resources, it is possible to 'short-circuit' huge amounts of development effort (reuse) that otherwise would be spent on activities not directly related to the required work in this task but are indispensable for computing industrial applications. The following set of library components that are available are:

- Structure loading and simulation set-up
- Mesh generation
- Linear system solving
- Field viewing
- Mesh analysis
- Time-step integration

Of course, in order not to turn the work into a futile exercise it is required to evaluate the status of these tools with respect to what is available. In particular, the purpose of task 3.1 is three-fold: The following questions need to be addressed.

- Are the software tools sufficiently mature and competitive to justify its use in the present work?
- Are there critical issues that need to be resolved before the software can be successfully applied to transient simulations?
- Is the present available software capable of dealing with EM coupling in a sufficiently satisfactory way? (The frequency domain is under consideration, because the transient regime is the goal of the work)

---

\(^1\)condition for convergence while solving certain PDEs numerically, arises when using explicit time-stepping methods.
• Are there specific alerts detected that need to be taken into account while performing the development work?

Above questions can be viewed as part of a SWOT (strengths, weaknesses, opportunities, and threats) analysis. The last issue deals with identifying warnings in the development of the new tools by taking into account experiences collected in earlier software development cycles.

In order to give answer to above questions, we follow a very practical approach: by computing a collection of structures taken from various resources using different simulation tools, we obtain a good insight into the positioning of the source codes that are available in the project in comparison to tools from other providers. Moreover, we learn about the strengths and weaknesses of the tools and gain understanding on which aspects need to be improved irrespective of the task related to the building of the transient solver. The cases are selected on the following criteria: (1) availability, (2) "intriguing" (the benchmark challenges insight) and (3) compact model availability. Using these criteria we have selected the following set as benchmarks:

• Simple conductive rod
• Two parallel transmission lines
• Strip line above a conductive plate
• Coax configuration
• Inductor with grounded guard ring
• Inductor with narrow winding above a patterned semiconductor layer

It is important to realize that some of these benchmarks have been acquired under NDA. As a consequence we cannot release the data that deals with the processing of the structure. However, it suffices to say that the structures are loaded with equal setting of the technology files into the different simulators.

The simulations are done in the frequency regime.

2 Review of the electromagnetic modeling

2.1 Use of potentials

Our starting point will be the equations of Maxwell:

\[
\begin{align*}
\text{Gauss’ law:} & \quad \nabla \cdot \mathbf{D} = \rho \quad (1) \\
\text{Absence of magnetic monopoles:} & \quad \nabla \cdot \mathbf{B} = 0 \quad (2) \\
\text{Maxwell-Faraday:} & \quad \nabla \times \mathbf{E} = -\frac{\partial \mathbf{B}}{\partial t} \quad (3) \\
\text{Maxwell-Ampère:} & \quad \nabla \times \mathbf{H} = \mathbf{J} + \frac{\partial \mathbf{D}}{\partial t} \quad (4)
\end{align*}
\]
where \( D, E, B, H, J \) en \( \rho \) are the electric induction, the electric field, magnetic induction and magnetic field, current density and charge density.

The following constitutive laws are used:

\[
B = \mu H, \quad D = \varepsilon E
\]  

The charge density \( \rho \) and current density \( J = 0 \) in insulating materials and the charge density \( \rho \) consists of a fixed background charge. In conductive domains we rely on the current continuity:

\[
\nabla \cdot J + \frac{\partial \rho}{\partial t} = 0
\]

If these conductive domains are metallic, then we apply Ohm’s law for the connection between electric field intensity and current density. It should be noted that this is not the most general expression and for Hall devices a magnetic field dependence must also be included. However, here we limit ourselves to situations where the magnetic fields are sufficiently weak in order to ignore Hall currents:

\[
J = \sigma E
\]

It should also be noted that the charge density and current densities are determined by the physical character of the materials under study. For example leakage currents can flow in insulating layers and the current-field relation is highly non-linear since tunneling mechanisms play an important role. Semiconductors also have more general current-field relations as given above and these will be discussed later.

We introduce the scalar potential \( V \) en the magnetic vector potential \( A \) that satisfy

\[
B = \nabla \times A \quad \text{and} \quad E = -\nabla V - \frac{\partial A}{\partial t}
\]

then the Maxwell equations become in these variables:

- **for insulators:**

\[
-\nabla \cdot \left[ \varepsilon \left( \nabla V + \frac{\partial A}{\partial t} \right) \right] = 0 \quad (10)
\]

\[
\nabla \times \frac{1}{\mu} (\nabla \times A) = -\varepsilon \frac{\partial}{\partial t} \left( \nabla V + \frac{\partial A}{\partial t} \right) \quad (11)
\]

- **for conductors:**

\[
-\nabla \cdot \sigma \left( \nabla V + \frac{\partial A}{\partial t} \right) = \frac{\partial}{\partial t} \left( \nabla \cdot \varepsilon \left( \nabla V + \frac{\partial A}{\partial t} \right) \right) \quad (12)
\]

\[
\nabla \times \frac{1}{\mu} (\nabla \times A) = -\sigma \left( \nabla V + \frac{\partial A}{\partial t} \right) - \varepsilon \frac{\partial}{\partial t} \left( \nabla V + \frac{\partial A}{\partial t} \right) \quad (13)
\]
In order to determine the S-matrix, a rather straightforward procedure is followed. For that purpose a collection of ports is needed and each port consists of two contacts. A contact is defined as a collection of nodes that are electrically identified. A rather evident appearance of a contact is a surface segment on the boundary of the simulation domain. A slightly less trivial contact consists of two or more of these surfaces on the boundary of the simulation domain. The nodes that are found on these surfaces are all at equal potential. Therefore, although there may be many nodes assigned to a single contact, all these nodes together generate only one potential variable to the system of unknowns. Of course, when evaluating the current entering or leaving the contact, each node in the contact contributes to the total contact current. Assigning prescribed values for the contact potential can be seen as applying Dirichlet’s boundary conditions to these contacts. This is a familiar technique in technology CAD. Outside the contact regions, Neumann boundary conditions are applied. Unfortunately, since we are now dealing with the full system of Maxwell equations, providing boundary conditions for the scalar potential will not suffice. We also need to provide boundary conditions for the vector potential. Last but not least, since the set of variables $V$ and $\mathbf{A}$ are not independent, setting a boundary condition for one variable has an impact on the others. Moreover, the choice of the gauge condition also participates in the appearance of the variables and their relations. A convenient set of boundary conditions is given by the following set of rules:

- **contact surface**: $V = V|_c$. To each contact area is assigned a prescribed potential value.
- **outside the contact area on the simulation domain**: $\mathbf{D}_n = 0$. There is no electric field component in the direction perpendicular to the surface of the simulation domain.
- **For the complete surface of the simulation domain**, we set $\mathbf{B}_n = 0$. There is no magnetic induction perpendicular to the surface of the simulation domain.

We must next translate these boundary conditions to restrictions on $\mathbf{A}$. Let us start with the last one. Since there is no normal $\mathbf{B}$, we may assume that the vector potential has only a normal component on the surface of the simulation domain. That means that the links at the surface of the simulation domain do not generate a degree of freedom. It should be noted that more general options exist. Nevertheless, above set of boundary conditions provides the minimal extension of the TCAD boundary conditions if vector potentials are present.

In order to evaluate the scattering matrix, say of an N-port system, we iterate over all ports and put a voltage difference over one port and put an impedance load over all other ports. Thus the potential variables of the contacts belonging to all but one port, become degrees of freedom that need to be evaluated. The following variables are required to understand the scattering matrices, where $Z_0$ is a real impedance that is
The simple conductive rod is a circular bar with constant conductance. In the static regime, a magnetic field is created, that behaves as \(1/r\) outside the rod. As a consequence, the total magnetic energy diverges, i.e.

\[
E_{\text{magn}} = \int dr \mathbf{B} \cdot \mathbf{H} \propto \int_0^\infty dr r^2 (1/r)^2
\]
Therefore, we would like to learn if the simulation tool can deal with such configurations. Furthermore, the MAGWEL solver has a unique discretization algorithm based on a two-fold application of Stokes’ law in order to deal with the term $\nabla (1/\mu \times \nabla \times \mathbf{A})$. Recently this term was programmed also for unstructured grids. This benchmark will test if these implementations can accurately produce analytic known results.

3.2 Simulation set up

The simulation set up is done with the following parameters :

- Wire cylindrical
- Radius 2 micron
- Length 100 micron
- Sigma=$10^8$ S/m
- $\mu_0 = 4\pi \times 10^{-7}$ H/m

An illustration of the wire is shown in Fig. 2.

3.3 Results

Check if the B field matches the analytic expectation. Outside the wire we have :

$$B(r) = \frac{4\pi \times 10^{-7} \times I}{2\pi r}$$ (18)
We put a voltage of $V=10$ volt over the wire. This gives the following values for the current and resistance.

<table>
<thead>
<tr>
<th>parameter</th>
<th>value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Analytic current</td>
<td>$1.257861 \times 10^2$ A</td>
</tr>
<tr>
<td>Analytic resistance</td>
<td>$7.95 \times 10^{-2}$ Ohm</td>
</tr>
</tbody>
</table>

The meshing is done using a two-dimensional Delaunay scheme with 3D extrusion. The mesh in shown in Fig. 3 and a zoom-in of the mesh is shown in Fig. 4.

The result of the simulation is:

<table>
<thead>
<tr>
<th>parameter</th>
<th>value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Analytic current</td>
<td>$1.23323376 \times 10^2$ A</td>
</tr>
<tr>
<td>Analytic resistance</td>
<td>$8.10876 \times 10^{-2}$ Ohm</td>
</tr>
</tbody>
</table>

We use Plotmtv to generate B-field magnitudes along a line through the center of the wire. The result is shown in Fig. 5. The MGW results (red) are the outcome of a full numerical simulation on this mesh. Then there are two analytic curves. The green curve (analytic num 1) refers to an analytic computation using the current as computed numerically on this mesh, whereas the blue curves also refers to the current computed analytically.

The $1/r$ drop is very well reproduced. This is only possible if the curl-curl operator is correctly implemented (ignoring finite-size effects), no loss of B field propagation of the inverse operator is observed. The numerical B field (red) is in agreement with the
Figure 4: Zoom-in of the mesh in the MAGWEL editor

Figure 5: View of the magnetic field strength
analytic current (blue). The numerical current value induces an analytic B field (green) in agreement with the numerical B field (red).

We expect that at 1 GHz the conclusion is still more or less valid, the skin depth = 1.59 micron < R= 2 micron.

The next plot is a scan over frequencies from 1KHz to 1GHz. It is very boring up to 100 MHz and then there is a collapse.

3.4 What did we learn?

From this benchmark we learned that the numerical treatment of the curl-curl operator is leading to correct analytic results in the static regime. No specific techniques for correcting or calibrating the outcome is needed.

4 Strip line above a conductive plate

4.1 Why is it interesting?

When computing capacitances using the MAGWEL field solver there is a tendency for overestimation. This problem is not seen for capacitors which extend in two dimensions all the way over the simulation domain, i.e. for infinitely large parallel plates, but when finite segments are taken, then the measured capacitance is below the computed capacitance. Therefore, this benchmark is of interest to get a grip on this systematical computational error which is related to fringe capacitance. The benchmark is of interest for support and selection of adaptive meshing strategies.
The strip line theory is explained in full detail in an online book: http://www.ece.rutgers.edu/ orfanidi/ewa/ch10.pdf : pp. 401-403. This modeling is due to Hammerstad and Jensen [14, 12].

The model has the following ingredients:

- Effective relative permeability: $\varepsilon_{\text{eff}}$
- Characteristic impedance: $Z$
- Relations between $Z$, $L$ and $C$

The capacitance, $C$, per unit length is $C = \varepsilon \frac{w}{2}$, where $\varepsilon = \varepsilon_0 \ast \varepsilon_r$ and $\varepsilon_0 = 8.85418 \times 10^{-12}$ F/m.

A cross-sectional view of the strip line above a plate is shown in Fig. 7. The parameter $w$ is the width and $h$ is the height of the dielectric layer. Furthermore the parameters $\eta$ and $\eta_0$ are defined as:

$$\eta = \sqrt{\frac{\mu}{\varepsilon}}, \quad \eta_0 = \sqrt{\frac{\mu_0}{\varepsilon_0}},$$

where $\mu = \mu_0 = 4\pi \times 10^{-7}$ H/m.

The contacting is shown in Fig. 8.

The modeling is based on the geometrical parameter $u = w/h$ and the effective material parameter $\varepsilon_{\text{eff}}$:

$$\varepsilon_{\text{eff}} = \frac{\varepsilon_r + 1}{2} + \frac{\varepsilon_r - 1}{2} \times (1 + \frac{10}{u})^{-ab}$$

$$Z = \frac{\eta_0}{2\pi \sqrt{\varepsilon_{\text{eff}}}} \ln \left[ \frac{f(u)}{u} + \sqrt{1 + 4 \frac{1}{u^2}} \right] = \frac{Z_0(u)}{\sqrt{\varepsilon_{\text{eff}}}}$$

The variables $a$ and $b$ are given below:

$$a = 1 + \frac{1}{49} \ln \left[ \frac{u^4 + (\frac{u}{22})^2}{0.432} \right] + \frac{1}{18.7} \ln \left[ 1 + \left( \frac{u}{18.1} \right)^3 \right]$$

Figure 7: Layout of the parallel strip above a conductive plate
These expressions were programmed and the textbook examples are confirmed (Page 403 of the on-line book S.J. Orfanidis [22]) For example:

\[
b = 0.564 \left( \frac{\varepsilon_r - 0.9}{\varepsilon_r + 3} \right)^{0.053}
\]

(24)

The value of C is in F/m.

4.2 Simulation set up

In MAGWEL we take 1 micron length strip line

<table>
<thead>
<tr>
<th>variable</th>
<th>value</th>
</tr>
</thead>
<tbody>
<tr>
<td>h</td>
<td>1 micron</td>
</tr>
<tr>
<td>w</td>
<td>10 micron</td>
</tr>
<tr>
<td>\varepsilon_r</td>
<td>3.9</td>
</tr>
<tr>
<td>T_{metal}</td>
<td>0.5</td>
</tr>
<tr>
<td>T_{air}</td>
<td>1 micron</td>
</tr>
</tbody>
</table>
4.3 Results

The results of this set up are shown below:

<table>
<thead>
<tr>
<th>variable</th>
<th>value</th>
</tr>
</thead>
<tbody>
<tr>
<td>MAGWEL</td>
<td>3.951870e-16 F</td>
</tr>
<tr>
<td>Hammerstad and Jensen</td>
<td>3.958997e-016 F</td>
</tr>
<tr>
<td>Parallel plate model $C = \varepsilon A/h$</td>
<td>3.4531302e-16 F</td>
</tr>
</tbody>
</table>

Static capacitance results

Note that the result from the simple parallel-plate model has a substantial error. A few plots of the electric field are shown in Fig. 9 and Fig. 10.

At first sight, the result for the capacitance looks very acceptable. However, we have not accounted for the finite thickness of the strip (here we used 0.5 micron) nor did we take into account a substantial layer of air (1 micron). So far, the height of the strip is not accounted for by Hammerstad and Jensen. By taking very thin strips, we can deduce from the equations of Hammerstad and Jensen (HJ) that $\varepsilon_{eff}^{HJ} < \varepsilon_r$. When $w/h \rightarrow \infty$ then $\varepsilon_{eff}^{HJ} \rightarrow \varepsilon_r$. We have studied this limit and found serious deviations between the analytic and numerical results. The reason is that there also needs to be included a sufficient amount of air in the field solver set up. We re-computed the capacitance with $t_{metal} = 0.01$ micron, $t_{air} = 10$ micron. In Fig. 11, the spread out of the electric field is illustrated.
Figure 10: Layout of the parallel strip above a conductive plate

Figure 11: Separation in two domains of the electric field. red: $E > 0.01E_{max}$, blue: $E < 0.01E_{max}$
The mesh is shown in Fig. 12.
Also an impression of the mesh is given in Fig. 13.

4.4 Effects of finite strip thickness

A follow-up paper of Hammerstad and Jensen [12] deals width the impact of the finite
strip thickness. The modified theory is given below. The input parameters are:

- $w$: strip width,
- $h$: thickness of the dielectric layer,
- $t_M$: thickness of the metal strip.

The normalized thickness is $t = t_M/h$. Furthermore we define:

$$u_1 = u + \delta u_1$$
$$u_r = u + \delta u_r$$

in which:

$$\delta u_1 = \frac{t}{\pi \ln \left( 1 + \frac{4e^{(1)}_u}{t \cdot \coth^2 (\sqrt{6.517} u)} \right) }$$
$$\delta u_r = \frac{1}{2} \left( 1 + \frac{1}{\cosh \sqrt{\frac{\varepsilon}{u} - 1}} \right) \delta u_1$$
The parameter $Z_0(u, t, r)$ is defined as

$$Z_0(u, t, r) = \frac{Z_{01}(u_r)}{\epsilon_{eff}(u_r, r)}$$

(25)

where the function $Z_{01}$ was defined as (see eq.(22))

$$Z_{01}(u) = \frac{\eta_0}{2\pi} \ln \left[ \frac{f(u)}{u} + \sqrt{1 + \frac{4}{u^2}} \right]$$

(26)

A thickness-dependent permittivity is given by

$$\epsilon_{eff}^{t_M} = \epsilon_{eff}(u_r, \epsilon_r) \left( \frac{Z_{01}(u_1)}{Z_{01}(u_r)} \right)^2$$

(27)

The capacitance is obtained from

$$C = \frac{\sqrt{\mu_0 \epsilon_0} \times \epsilon_{eff}^{t_M}}{Z}$$

(28)

4.4.1 Finite $t_M$ results

Hammerstad and Jenssen, with a correction for the strip thickness, give for the choice of parameters: $w=10$, $\epsilon_r = 3.9$ and $h=1$, the following results:
We refined the capacitance computation. Indeed, the results that where obtained in our first experiment are accidentally in agreement. The HJ model is an underestimation because the thickness of the metal was not included and the MAGWEL simulation was an underestimation because the air layer was only 1 micron (instead of ‘infinity’).

By taking a layer of air of 10 micron we obtain the following numerical results:

<table>
<thead>
<tr>
<th>$t_M$</th>
<th>$C_{GW}$ in $\text{F/m}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.01</td>
<td>3.964446958408e-16 $\text{F/m}$</td>
</tr>
<tr>
<td>0.5</td>
<td>4.015557092879e-16 $\text{F/m}$</td>
</tr>
</tbody>
</table>

This last number is in close agreement with the corrected HJ result:

$$t_M = 0.5, \quad C_{\text{HJ corrected}} = 4.02602544041718e \times 10^{-16} \text{ F/m}$$

4.5 Running the adapter

Start from the most-coarse mesh

```
mesh
  <uniformMesh>
    <nX>1</nX>
    <nY>1</nY>
    <nZ>1</nZ>
  </uniformMesh>
  <minXdiff>0.001</minXdiff>
  <minYdiff>0.001</minYdiff>
  <minZdiff>0.001</minZdiff>
  <maxNrMeshRefinements>4</maxNrMeshRefinements>
  <relativeNrNewNodesLimit>5</relativeNrNewNodesLimit>
  <minNrNewNodes>100</minNrNewNodes>
</mesh>
```

<table>
<thead>
<tr>
<th>Capacitance</th>
<th>refinement level</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.038975073324e-15</td>
<td>no refinement</td>
</tr>
<tr>
<td>6.966594046402e-16</td>
<td>1 sweep</td>
</tr>
<tr>
<td>5.296263512904e-16</td>
<td>2 sweeps</td>
</tr>
<tr>
<td>4.525636287160e-16</td>
<td>3 sweeps</td>
</tr>
<tr>
<td>4.211632499841e-16</td>
<td>4 sweeps</td>
</tr>
</tbody>
</table>
Result moves towards analytic result when the mesh refines.

We now know where we want our refinement to occur. The choice of metric does not yet generate the refinement at this desired location. 

Notes: How does the aspect ratio tag interfere with the refinement request? Adaptation was based on HF testing. The C plots were for static capacitances, but they are very similar to 1 GHz plots for V and E.

4.6 Simulations with Opera - VectorFields

Strip line simulations with Opera-2D

The commercial software of Vector Fields [2] provides design solutions for a wide variety of electromagnetic applications. Opera is a state-of-the-art software package for the modeling of static and time varying electromagnetic fields, using 2D and 3D finite elements. For benchmarking purposes for the strip line model, the 2D electrostatic adaptive solver was used, where capacitance calculations can be performed in 2 ways:

- Energy calculation: \( C_1 = 2E_{electric}/V^2 \)
- Gauss' law: \( C_2 = \oint D \cdot dA \)

For the adaptive meshing stopping criterion, we use the Internal Error \(< 0.5\%\) criterion, which means that the relative difference between the electric field solution in each mesh element adjacent to a node and the averaged field in a node is smaller than 0.5%. The meshing is based on a triangular Delaunay algorithm. Furthermore, we put the electrostatic potential \( V=0 \) on the bottom of the substrate (ground plane) and \( V=1 \) on the strip line, while the rest of the boundary has a Neumann condition.

Fig. (14) shows the model (using different scales for the X- and Y-axis).

The next figures, Fig. (15,16), where X- and Y-axis now have the same scale, shows the potential distribution and electric fields (arrows).

In the table we plot the different capacitance values \( C_1 \) and \( C_2 \) where we vary the Error and air height. The Hammerstad and Jensen value is 4.0260e-016 F/m for the strip line case with thickness \( T=0.5 \mu m \). On average a number of 15 iterations in the mesh adaptation process were needed to get an internal accuracy of less than 0.5%, with a final average (of \( C_1 \) and \( C_2 \)) difference of 0.2 % compared to the Hammerstad & Jensen value, which of course may also contain some error itself.

<table>
<thead>
<tr>
<th>( T = 0.5 \mu m ), air height</th>
<th>Internal Error</th>
<th>( C_1 ) (in e-16 F/m)</th>
<th>( C_2 ) (in e-16 F/m)</th>
<th># elem.</th>
<th># nodes</th>
</tr>
</thead>
<tbody>
<tr>
<td>2.5 ( \mu m )</td>
<td>2 %</td>
<td>3.9089</td>
<td>3.9152</td>
<td>11151</td>
<td>5726</td>
</tr>
<tr>
<td>2.5 ( \mu m )</td>
<td>1 %</td>
<td>3.9093</td>
<td>3.9121</td>
<td>28745</td>
<td>14558</td>
</tr>
<tr>
<td>2.5 ( \mu m )</td>
<td>0.5 %</td>
<td>3.9096</td>
<td>3.9108</td>
<td>96719</td>
<td>48611</td>
</tr>
<tr>
<td>2.5 ( \mu m )</td>
<td>0.2 %</td>
<td>3.9106</td>
<td>3.9111</td>
<td>285335</td>
<td>143043</td>
</tr>
<tr>
<td>6.5 ( \mu m )</td>
<td>0.5 %</td>
<td>3.9821</td>
<td>3.9831</td>
<td>116517</td>
<td>58468</td>
</tr>
<tr>
<td>11.5 ( \mu m )</td>
<td>0.5 %</td>
<td>4.0054</td>
<td>4.0061</td>
<td>189073</td>
<td>94736</td>
</tr>
<tr>
<td>16.5 ( \mu m )</td>
<td>0.5 %</td>
<td>4.0121</td>
<td>4.0127</td>
<td>149111</td>
<td>74716</td>
</tr>
<tr>
<td>50 ( \mu m )</td>
<td>0.5 %</td>
<td>4.0181</td>
<td>4.0189</td>
<td>165182</td>
<td>82744</td>
</tr>
</tbody>
</table>

Numerical results of the OPERA simulations
In the next table we plot $C_1$ and $C_2$ and also the Hammerstad & Jensen values for various values of the strip line thickness, with average (of $C_1$ and $C_2$) differences of 0.02–0.2 % with the H-J values.

<table>
<thead>
<tr>
<th>air height = 50 $\mu m$</th>
<th>H&amp;J</th>
<th>$C_1$</th>
<th>$C_2$</th>
<th>Aver. Diff. %</th>
</tr>
</thead>
<tbody>
<tr>
<td>T= 0.4 $\mu m$</td>
<td>4.01623</td>
<td>4.01016</td>
<td>4.0115</td>
<td>0.13</td>
</tr>
<tr>
<td>T= 0.3 $\mu m$</td>
<td>4.00544</td>
<td>4.0010</td>
<td>4.0024</td>
<td>0.09</td>
</tr>
<tr>
<td>T= 0.2 $\mu m$</td>
<td>3.99330</td>
<td>3.99046</td>
<td>3.9916</td>
<td>0.06</td>
</tr>
<tr>
<td>T= 0.1 $\mu m$</td>
<td>3.97033</td>
<td>3.97802</td>
<td>3.9787</td>
<td>0.20</td>
</tr>
<tr>
<td>T= 0.01 $\mu m$</td>
<td>-</td>
<td>3.96148</td>
<td>3.9638</td>
<td>-</td>
</tr>
<tr>
<td>T= 0 $\mu m$</td>
<td>3.95900</td>
<td>3.95926</td>
<td>3.9600</td>
<td>0.02</td>
</tr>
</tbody>
</table>

Numerical results of the OPERA simulations for various strip line thicknesses

To get an idea of the adaptation process and the meshes, see the following figures:

Mesh4: 881 elements, evaluated error 10.40742 %
Figure 14: Illustration of the model with X and Y at different scales.

Figure 15: Illustration of the potential with X and Y at the same scale.

Figure 16: Illustration of the electric field with X and Y at the same scale.
Figure 17: Mesh1: 436 elements, evaluated error 18.21371 %.

Figure 18: Mesh2: 532 elements, evaluated error 15.04066 %.

Figure 19: Mesh3: 671 elements, evaluated error 17.95522 %.
Figure 20: Mesh4: 881 elements, evaluated error 10.40742 %.

Figure 21: Mesh5: 1079 elements, evaluated error 8.484277 %.
Figure 22: Mesh6: 1411 elements, evaluated error 7.563303 % and with zoom-in.
Figure 23: Mesh7: 1955 elements, evaluated error 6.604616 % and with zoom-in.
Figure 24: Mesh8: 2706 elements, evaluated error 4.726004 % and with zoom-in.
Figure 25: Mesh9: 3647 elements, evaluated error 3.773939 %.

Figure 26: Mesh10: 4967 elements, evaluated error 3.268908 %.
Figure 27: Mesh11: 7100 elements, evaluated error 2.078323 % and with zoom-in.
Figure 28: Mesh12: 9894 elements, evaluated error 2.127898 %.

Figure 29: Mesh13: 13752 elements, evaluated error 1.517278 %.
Figure 30: Mesh14: 18945 elements, evaluated error 1.209618 %.

Figure 31: Mesh15: 24736 elements, evaluated error 0.908549 %.
4.7 What did we learn?

Many effects influence the final result of the fringe capacitance, including air layer, mesh size, strip thickness, domain size, etc. Refined analytic and refined numerical results converge towards each other.

Difference: $(C_{MGW} - C_{HJ}) / C_{HJ} = 0.011 / 4.02 = 0.003 = 0.3\%$

This is very important for deciding on stopping criteria for adaptive meshing methods. Furthermore, we have demonstrated that the MAGWEL solver can compete with OPERA (and therefore with any other finite-element based field solver) as far as accuracy vs. node consumption is concerned.

5 Coax configuration

5.1 Why is it interesting?

The simple coax example is used for testing a circular geometry. In the MAGWEL solver these circles are converted to polygons, and either a staircase Manhattan approximation is used or a general unstructured Delaunay mesh. This example enables us to compare with analytical expressions.

5.2 Simulation set up

The simulation set up is done with the following parameters:

- Cylindrical coax
- Inner radius 1 micron ($R_0$)
- Inner radius of outer conductor 5 micron ($R_1$)
- Outer radius of outer conductor 5.5 micron ($R_2$)
- Length 10 micron
- $\mu_0 = 4\pi \times 10^{-7}$ H/m
- $\mu_r = 1$

An illustration of the wire is shown in Fig. 32
5.3 Results

We compare the static simulation results with the analytical magnetic field (Fig. 33).

\[ \begin{align*}
0 \leq r \leq R_0 : \quad B(r) &= \frac{B_{\text{max}} r}{R_0} \\
R_0 \leq r \leq R_1 : \quad B(r) &= \frac{B_{\text{max}} R_0}{r} \\
R_1 \leq r \leq R_2 : \quad B(r) &= \frac{B_{\text{max}} R_0}{r} \left( \frac{R_2^2 - r^2}{R_2^2 - R_1^2} \right)
\end{align*} \]  

(29) 

(30) 

(31)

Also the resulting inductance per unit length is analytically known as:

\[ L = \frac{\mu_0}{2\pi} \ln \left( \frac{R_1}{R_0} \right) + \frac{\mu_0}{2\pi} \left( \frac{R_2^2}{R_2^2 - R_1^2} \right)^2 \ln \left( \frac{R_2}{R_1} \right) - \frac{\mu_0}{4\pi} \left( \frac{R_2^2}{R_2^2 - R_1^2} \right) \]  

(32)

and for the given configuration this results in a value of \( L_{\text{analytical}} = 3.7837 \times 10^{-7} \) H/m. This result is reproduced by MAGWEL simulations using different meshes (Manhattan and Delaunay), to give the following results:

<table>
<thead>
<tr>
<th>nodes</th>
<th>( L_{\text{sim}} )</th>
<th>error</th>
<th>mesh type</th>
</tr>
</thead>
<tbody>
<tr>
<td>8448</td>
<td>3.7709 ( 10^{-7} ) H/m</td>
<td>0.38%</td>
<td>Delaunay</td>
</tr>
<tr>
<td>31000</td>
<td>3.7768 ( 10^{-7} ) H/m</td>
<td>0.18%</td>
<td>Delaunay</td>
</tr>
<tr>
<td>13920</td>
<td>3.7372 ( 10^{-7} ) H/m</td>
<td>1.40%</td>
<td>Manhattan</td>
</tr>
<tr>
<td>209760</td>
<td>3.7950 ( 10^{-7} ) H/m</td>
<td>0.15%</td>
<td>Manhattan</td>
</tr>
</tbody>
</table>

The RF simulations for the same structure get exactly the same resulting inductance, when we stay below a frequency of 1 GHz. For higher frequencies, extra phenomena play a role, like skin effect, current re-distributions, etc.
5.4 What did we learn?

From this benchmark we learned that the numerical treatment of the magnetic operator is leading to correct analytic results in the static regime. Typical circular geometries are adequately simulated with the MGW software.

6 Inductor with grounded guard ring

6.1 Why is it interesting?

This design considers an inductor that is shielded by a closed-loop grounded guard ring. The layout is shown in Fig. 35. When an alternating current is injected into the inductor, an induced current in opposite phase is induced in the closed loop. Although the loop is grounded at both ends, the vector potential is still present and therefore the induced current exists despite the fact that the loop is grounded. This configuration is also instructive to obtain an in-depth understanding of the measurement set-up for S-parameters.

6.2 Simulation set up

The structure is simulated using the Y-parameter extraction method. This design has a high-resistive substrate. Therefore, the decay of the field strength is rather slow and
Figure 34: View of the magnetic field strength
a full stack of 625 micron substrate is included in the simulation. The experimental data of the substrate is: $\sigma = 0.1 \text{ S/m}$. The computed curves are obtained by using 1) the MAGWEL solver with calibrated substrate resistance, 2) the MAGWEL solver using an integrating factor method.\(^2\) Finally, the results obtained using Agilent Momentum [4] are shown. The curves are identified with the labels EXP, OPTIM, INTFAC, MOM respectively.

6.3 Results

In Figs. 36, 37 the Y-parameters are shown. The curves of MAGWEL and the measurements agree for Re(Y11) and Momentum disagrees. The curves for Im(Y11) and Re(Y12) from MAGWEL and Momentum agree but both deviate from the measured data. Finally for Im(Y12) all curves agree. The 'noisy' behavior of the MAGWEL results for Re(Y12) is the result of setting up the simulation such that the computation was truncated too early because of too relaxed stopping conditions. This is the subject of further analysis.

In Figs. 38, 39, 40 the S-parameters are shown. The Y-parameter information can be transformed into compact-model parameter information. For details, we refer to the next section. Here we plot the 2-port resistance, the 2-port inductance and the Q-factor. The results are shown in Fig. 41.

---

\(^2\)The integrating factor method is a newly developed theory for discretizing the Maxwell equations. It takes the singular perturbation character of the differential operator for high frequencies into account.
6.4 What did we learn?

From this simulation work we learned that the implementation of the Hodge operators is very important for getting results that are within 1% accuracy for the inductance. A naive implementation gives errors of 10%. These errors can be counteracted by using an enhancement factor for the permeability $\mu_r$. This factor is tuned once at zero frequency. Furthermore, in order to get results that are very accurate for inductors with wide windings, the integrating factor method improves the results at higher frequency. This is in agreement with the expectations since the integrating factor becomes important for high frequency and large mesh elements.
Figure 36: Re (Y11) and Im (Y11)
Figure 37: Re $(Y_{12})$ and Im $(Y_{12})$
Figure 38: Re (S11) and Im (S11)
Figure 39: Re (S12) and Im (S12)
Figure 40: Re (22) and Im (S22)
Figure 41: The inductance, resistance and Q factor of the inductor with grounded closed-guard ring.
7 Inductor with narrow winding above a patterned semiconductor layer

7.1 Why is it interesting?

This inductor is interesting because the narrow windings require a rather advanced meshing strategy. There are numerous calibration issues involved for the simulation of these type of inductors. The layout of the inductor is shown in Fig. 42.

However, the details of this design are found in the Nwell pattern that is supposed to limit the eddy currents in the substrate. The underlying idea is that conductive patterns will induce an equipotential in the top layer and thereby induce a collapse of the eddy currents.

7.2 Simulation set up

This simulation work spun out of research from the projects CODESTAR and CHAMELEON-RF. There are a number of issues that were not fully appreciated while performing the measurements. For instance, detailed knowledge on the guarding and contact engineering is missing. Therefore we have experimented with numerous layout variations. Figures (44) and (45) show two variations.

7.3 Results

The following figures show how good we can reach the experimental data. Furthermore, a comparison is given using HFSS results [3]. Figures (46), (47), (48), (49), (50) and (51) show the S-parameters. We have several techniques at out disposal for obtaining these parameters. A direct method exploits the termination of all but one port and extract the S-parameters. This simulation method mimics the experimental situation. Another technique that is more robust for simulation, grounds all but one port and extracts the Y-parameters. The latter method is applied here. Both methods give equal
Figure 43: 2D View of the Nwell pattern

Figure 44: 2D View of the Nwell pattern
results.

Of more interest are the compact-model parameters that can be extracted from these S-parameters. For that purpose we convert the S-matrix to the Y- and Z-matrix.

Depending on the way of using the inductor in a circuit, there are two definitions of the inductance. In the CODESTAR project the inductance was defined using a 1-port approach:

\[
R_{\text{CODESTAR}} = \Re \left( \frac{1}{Y_{11}} \right) \quad (33)
\]

\[
L_{\text{CODESTAR}} = \frac{1}{\omega} \Im \left( \frac{1}{Y_{11}} \right) \quad (34)
\]

For the 2-port approach we use the following definitions as was also done in CHAMELEON-RF

\[
R_{\text{CHAMELEON}} = -\Re \left( \frac{1}{Y_{12}} \right) \quad (35)
\]

\[
L_{\text{CHAMELEON}} = -\frac{1}{\omega} \Im \left( \frac{1}{Y_{12}} \right) \quad (36)
\]

These formulas can be easily applied using python scripts.

```python
# this script uses Y-parameters and converts to S- and Z-parameters
```
Figure 46: Re($S_{11}$)

Figure 47: Im($S_{11}$)
Figure 48: Re(S12)

Figure 49: Im(S12)
Figure 50: Re(S22)

Figure 51: Im(S22)
#!/usr/bin/env python2.4
import os
import math
import scipy
from scipy.linalg import inv, det, eig
import math

Yparam = scipy.zeros((2,2), complex)
Zparam = scipy.zeros((2,2), complex)
Sparam = scipy.zeros((2,2), complex)

# help matrices
M1 = scipy.zeros((2,2), complex)
M2 = scipy.zeros((2,2), complex)

refImpedance =50;
F = scipy.zeros((2,2), complex)
F[0,0]=refImpedance
F[1,1]=refImpedance
ONES = scipy.zeros((2,2), complex)
ONES[0,0]=1.0
ONES[1,1]=1.0
times=50

Y0 = scipy.zeros( (2,2), complex)
Z0 = scipy.zeros( (2,2), complex)
Y0ref = 1/refImpedance

Zparam = scipy.linalg.inv(Yparam)
M1 = -Z0+Zparam
M2 = scipy.linalg.inv(Z0+Zparam)
Sparam = scipy.dot(M1, M2)

# CHAMELEON-RF definition of R and L using Y12 <-> Yparam[0,1]
R = -(1.0/Yparam[0,1]).real
L = -(1.0/Yparam[0,1]).imag/(2.0*3.1415*freq)
# CODESTAR definition of L using Y12 <-> Yparam[0,1]
R1 = (1.0/Yparam[0,0]).real
L1 = (1.0/Yparam[0,0]).imag/(2.0*3.1415*freq)
# Quality factor (same for CODESTAR and CHAMELEON-RF)
Q= (1.0/Yparam[0,0]).imag /(1.0/Yparam[0,0]).real

50
The Q-factor is shown in Fig. (56).
Finally, we also plot the $Y$-parameters.
This simulation work has learned us numerous facts. First of all, in order to position the simulation tools with respect to each other it is important to eliminate all disputable uncertainties. In this example, there were two important aspects. The first deals with guard ring design and the second one deals with the contact design. Both uncertainties led to some guess work and in particular, this guess work manifests itself in the definition of the ports. As a consequence, both the MAGWEL solver as well as HFSS produce outcomes that do not match the measurements over the full frequency range. Nevertheless, by pursuing this simulation case, we have obtained a much better understanding of which aspects play an important role in obtaining successful Q-factor results. Moreover, we were able to define a calibration strategy for good inductor simulation. The strategy is based on first getting the static inductance and resistance in place. Next the determination of the Q-factor is set by an accurate modeling of the substrate. In practice it means that the substrate resistance and permittivity must be accurately determined. Finally, the high-frequency behavior must be settled by obtaining a good value for the inter-winding capacitive coupling. Based on these observations we conclude that accurate simulation at high frequency requires adaptive meshing. Without adaptive meshing the capacitive couplings are over-estimated. We suspect that this is the reason why the 2-port inductance of Fig. (57) shows a resonance at a too low resonance frequency. Since HFSS is equipped with adaptive measurement facilities, this resonance is shifted to a higher frequency since the over-estimation of the inter-winding capacitance is avoided. In order to verify/falsify this interpretation of the results we repeated the simulation and replaced the permittivity of the oxide with 1.0, This leads to a smaller inter-winding capacitance and that should give a higher self-resonance frequency. In Figs. (61), (62) and (63), the compact model parameters are shown. These plots demonstrate that the above explanation is correct. Of course, other deviations are seen because the oxide also is important for the coupling to the substrate.
It should be noted however, that it remains a challenge to get all the curves right over the full frequency range. Besides, the specific knowledge that was generated for simulating inductors above substrates, a few more interesting facts were gathered:

- At low frequencies, the computation of inductance requires that Hodge operator effects are taken into account.
- At high frequencies a modified discretization is needed.
- S-parameter, Y-parameter based simulations provide equivalent compact model results.
Figure 55: R using Y11

Figure 56: Q factor
Figure 57: Re(Y11)

Figure 58: Im(Y11)
Figure 59: Re(Y12)

Figure 60: Im(Y12)
Figure 61: L using Y12 and L using Y11
Figure 62: R using Y12 and R using Y11
Figure 63: Q-factor
8 Simulation of single and coupled oscillators: phase macro-model approach

8.1 Why is it interesting

Design of integrated RF circuits requires detailed insight in the behavior of the used components. Unintended coupling and perturbation effects need to be accounted for before production, but full time-domain simulation of these effects is expensive or infeasible. Oscillators are one of the main building blocks which are running at high frequencies (in GHz range), hence full time-domain simulations require a very small time step and are time consuming. Here we present a method to build non-linear phase macromodels for voltage controlled oscillators. These models can be used to accurately predict the behavior of individual and mutually coupled oscillators under perturbation, at much lower costs than full circuit simulations. In this section we consider LC oscillators, see Section 8.2.1, where the L and C parameters of a given oscillator are computed with the MAGWEL software. The approach is illustrated by numerical experiments with realistic designs.

8.2 Mathematical model

A general free-running oscillator can be expressed as an autonomous system of differential equations:

\[ \frac{dq(x)}{dt} + j(x) = 0, \]
\[ x(0) = x(T), \]

where \( x(t) \in \mathbb{R}^n \) are the state variables, \( T \) is the period of the free running oscillator, which is in general unknown, and \( q, j : \mathbb{R}^n \rightarrow \mathbb{R}^n \) are (non-linear) functions describing the oscillator’s behavior. The solution of (37) is called periodic steady state (PSS) and is denoted by \( x_{pss} \). A general oscillator under perturbation can be expressed as a system of differential equations

\[ \frac{dq(x)}{dt} + j(x) = b(t), \]

where \( b(t) \) are perturbations to the free running oscillator. For small perturbations \( b(t) \) it can be shown [10] that the solution of (38) can be computed as

\[ x_p(t) = x_{pss}(t + \alpha(t)), \]

where \( \alpha(t) \) is called the phase shift. The phase shift \( \alpha(t) \) satisfies the following scalar non-linear differential equation:

\[ \dot{\alpha}(t) = V^T(t + \alpha(t)) \cdot b(t), \]
\[ \alpha(0) = 0, \]

with \( V(t) \in \mathbb{R}^n \) being the perturbation projection vector (PPV) [10] of (38) and \( n \) is the system size. Using this simple and numerically cheap method one can do many kinds of analysis for oscillators, e.g., injection locking, pulling, a priori estimate of the locking range [10, 19].
8.2.1 LC oscillator

For many applications oscillators can be modeled as an LC tank with a non-linear resistor as shown in Fig. 64. This circuit is governed by the following differential equations:

\[
C \frac{dv(t)}{dt} + \frac{v(t)}{R} + i(t) + S \tanh\left(\frac{G_n}{S} v(t)\right) = b(t),
\]

(41a)

\[
L \frac{di(t)}{dt} - v(t) = 0,
\]

(41b)

where \(C\), \(L\) and \(R\) are the capacitance, inductance and resistance, respectively, which are calculated by using the MAGWEL software. The nodal voltage is denoted by \(v\) and the branch current of the inductor is denoted by \(i\). The voltage controlled non-linear resistor is defined by \(S\) and \(G_n\) parameters, where \(S\) determines the oscillation amplitude and \(G_n\) is the gain.

8.2.2 Mutual inductive coupling

Let us consider the two mutually coupled LC oscillators shown in Fig. 65. The inductive coupling between these two oscillators can be modeled as

\[
L_1 \frac{di_1(t)}{dt} + M \frac{di_2(t)}{dt} = v_1(t),
\]

(42a)

\[
L_2 \frac{di_2(t)}{dt} + M \frac{di_1(t)}{dt} = v_2(t),
\]

(42b)

where \(M = k \sqrt{L_1 L_2}\) is the mutual inductance and \(|k| < 1\) is the coupling factor. In this section all the parameters with a subindex refer to the parameters of the oscillator with the same subindex. If we combine the mathematical model (41) of each oscillator with (42), then the two inductively coupled oscillators can be described by the following
Figure 65: Two inductively coupled LC oscillators.

differential equations

\[
\begin{align*}
C_1 \frac{dv_1(t)}{dt} + v_1(t)/R + i_1(t) + S \tanh\left( \frac{G_n}{S} v_1(t) \right) &= 0, \\
L_1 \frac{di_1(t)}{dt} - v_1(t) &= -M \frac{di_2(t)}{dt}, \\
C_2 \frac{dv_2(t)}{dt} + v_2(t)/R + i_2(t) + S \tanh\left( \frac{G_n}{S} v_2(t) \right) &= 0, \\
L_2 \frac{di_2(t)}{dt} - v_2(t) &= -M \frac{di_1(t)}{dt}.
\end{align*}
\]

(43a) \hspace{1cm} (43b) \hspace{1cm} (43c) \hspace{1cm} (43d)

For small values of the coupling factor $k$, the right hand side of (43b) and (43d) can be considered as a small perturbation to the corresponding oscillator and we can apply the phase shift macromodel. Then we obtain the following simple non-linear equations for the phase shift of each oscillator:

\[
\begin{align*}
\dot{\alpha}_1(t) &= V_1^T(t + \alpha_1(t)) \cdot \begin{pmatrix}
0 \\
-M \frac{di_2(t)}{dt}
\end{pmatrix}, \\
\dot{\alpha}_2(t) &= V_2^T(t + \alpha_2(t)) \cdot \begin{pmatrix}
0 \\
-M \frac{di_1(t)}{dt}
\end{pmatrix},
\end{align*}
\]

(44a) \hspace{1cm} (44b)

where the currents and voltages are evaluated by using (39):

\[
\begin{align*}
[v_1(t), i_1(t)]^T &= x_{pss}^1(t + \alpha_1(t)), \\
[v_2(t), i_2(t)]^T &= x_{pss}^2(t + \alpha_2(t)).
\end{align*}
\]

(44c) \hspace{1cm} (44d)

8.3 Results

8.3.1 LC oscillator

Consider an oscillator with the following parameters $L = 0.93$ nH, $C = 1.145$ pF and $R = 1000$ $\Omega$. It has a free running frequency $f_0 = 4.8773 * 10^9$ and the amplitude of the
Consider the injected signal to be sinusoidal of the form
\[ b(t) = A_{\text{inj}} \sin(\omega_1 t), \] (45)
where \( A_{\text{inj}} \) is the amplitude of the injected signal and \( \omega = 2\pi f \) is the frequency in radians.

For the injection amplitude \(0.05 \cdot A_0\) and the injection frequency \(0.97 \cdot f_0\), then the oscillator under this perturbation will not lock, see Fig.66.

A slight change in the injection frequency will lock the oscillator. If the injection frequency is 2\% less than the oscillator’s frequency, i.e. \( f_1 = 0.98 \cdot f_0 \) then the oscillator will lock to the injected signal, see Fig. 67.

\subsection*{8.3.2 Mutually coupled oscillators}

Let us consider two oscillators with inductance and resistance \( L_1 = L_2 = 0.64 \text{nH}\) and \( R_1 = R_2 = 50 \Omega\), respectively. The first oscillator is designed to have a free running
frequency \( f_1 = 4.8 \) GHz with capacitance \( C_1 = 1/(4L_1\pi^2 f_1^2) \). Then the inductor current in the first oscillator is \( A_1 = 0.0303 \) A and capacitor voltage is \( V_1 = 0.5844 \) V. In a similar way the second oscillator is designed to have a free running frequency \( f_2 = 4.6 \) GHz with the inductor current \( A_2 = 0.0316 \) A and capacitor voltage \( V_2 = 0.5844 \) V.

For both oscillators we choose \( S_i = 1/R_i, G_n = -1.1/R_i \) with \( i = 1, 2 \). Simulation results with the phase shift macromodel are compared with Pstar (in-house NXP circuit simulator) simulations of the full circuit, hereafter called full simulation.

In all the numerical experiments the simulations are run until \( T_{\text{final}} = 6 \times 10^{-7} \) s with the time step \( \tau = 5 \times 10^{-13} \). Numerical simulation results of two inductively coupled oscillators for different coupling factors \( k \) are shown in Fig. 68. For small values of the coupling factor we observe a very good approximation with the full simulation results. As the coupling factor grows, small deviations in the frequency occur, see Fig. 68(d). Because of the mutual pulling effects between the two oscillators, a double sided spectrum is formed around each oscillator carrier frequency. The additional sidebands are equally spaced by the frequency difference of the two oscillators.

The phase shift \( \alpha_1(t) \) of the first oscillator for a certain time interval is given in Fig. 69. We note that it has a sinusoidal behavior. Recall that for a single oscillator under perturbation a completely different behavior is observed: in locked condition the phase shift changes linearly, whereas in the unlocked case the phase shift has a non-linear behavior different from a sinusoidal, see for example [20].
8.3.3 Inductively coupled oscillators under injection

As a final example let us consider two inductively coupled oscillators where in one of the oscillators an injected current is applied. Let us consider a case when a sinusoidal current of the form

$$I(t) = A \sin(2\pi(f_1 - f_{off})t)$$  \hspace{1cm} (46)

is injected in the first oscillator. Then (44) is modified to

$$\alpha_1(t) = V_T^T(t + \alpha_1(t)) \cdot \begin{pmatrix} -I(t) \\ -M \frac{di_2(t)}{dt} \end{pmatrix}$$,  \hspace{1cm} (47)

For a small current injection with $A = 10 \mu$A and an offset frequency $f_{off} = 20 \text{ MHz}$, the spectrum of both oscillators with the coupling factor $k = 0.001$ is given in Fig.70. We observe that the phase macromodel is a good approximation of the full simulation results.
Figure 70: Inductive coupling with injection and $k = 0.001$. Top: phase shift. Bottom: comparison of the output spectrum obtained by the phase macromodel and by the full simulation with a small current injection.
9 Conclusions

The goal of this document is to decide if the MAGWEL solver can serve as an appropriate base for constructing a transient solver in the ICESTARS project. We have shown that the test cases designed for testing elementary qualities give excellent results. However, for industrial examples, the situation is more involved. We have compared results from MAGWEL with results from Momentum and HFSS. The details of the results differ, but the overall conclusion is that the MAGWEL solver is able to generate outcomes of comparable quality as Momentum and HFSS. Whereas the MAGWEL solver is not as complete in terms of ease-of-use, adaptive frequency stepping and other conveniences that are collected over many years, the tool is indeed a good starting point for further development. On the other hand, the meshing facilities of the MAGWEL solver are very unique and useful for dealing with combined small and large scales into a single simulation set up. In particular, the simulation of high-resistive substrate inductors has been a serious challenge for many tools. The Q-factor computed by MAGWEL are in general somewhat too small at the peak value, giving a 'pessimistic' result, whereas other field solvers generate a too large (optimistic) result. Nevertheless, an important area of attention is definitely adaptive meshing which is essential for obtaining accurate results in the RF regime.
References


[33] K. Yee. Numerical solution of initial boundary value problems involving maxwell’s
equation in isotropic media. *IEEE Trans. on Antennas and Propagation*, AP-