Virtual Prototyping of a Microwave Fin Line Power Spatial Combiner Amplifier

A. Leggieri[1], F. Di Paolo[1], D. Passi[1]

[1]University of Rome "Tor Vergata" - Department of Electronic Engineering, Rome, Italy

Abstract

Introduction
This paper describes the Virtual Prototyping (VP) based on a COMSOL Multiphysics® (MP) simulation of a novel Microwave (MW) Fin Taper (FT) Spatial Power Combiner (SPC) Amplifier. Virtual prototype is obtained by an accurate modeling and simulation of the analyzed structure, making respect future construction features and ensuring the maximum computational ligh.

The analyzed system is waveguide WG based, and uses FT Probes to convert the energy of a rectangular WG EM fundamental mode to a Microstrip Transmission Line (µSTL) TEM mode, in order to be amplified by a Solid State Power Amplifier (SSPA) [1].

The power dissipation of the MMIC SSPA's produces a considerable temperature increase and induces a thermal expansion of both the PA's and of the connected structure. The thermal expansion of the materials may also induce significant stresses and strains with consequent displacement of the guiding and combining structures, which alter the desired (EM) behavior of the device.

Use of COMSOL Multiphysics®
Since the SPC efficiency depends critically on the FT profile and the SSPA's operating temperature, a Multiphysics (MP) approach is necessary, in order to study the effect of these multiple influencing factors.

In order to decrease computational time and resources while maintaining accuracy, the device model is organized by using several strategies allowed by COMSOL Multiphysics®. The architecture of the model is based on Thermal Stress (TS), Moving Mesh (MM) and Electromagnetic Waves (EMW) interfaces.

A significant innovation in RF modeling with COMSOL Multiphysics®, produced in this study, is the µSTL ports representation: Fringe effects are computable with lumped ports, by introducing opportunely distributed Perfect Magnetic Conductor (PMC) boundary condition close to the lumped ports with a Scattering boundary condition on the EM wave outgoing WG boundary [2].

Another innovation is in the TS and MM modeling about the WG walls representation and its

behavior with regard to thermal expansion. This consists of using opportune condition described by a combination of Heat Transfer in Fluids, Highly conductive Layer and structural Fixed Constraints in the TS interface [3].

WG walls, in a stationary temperature regime, can consist in fixed constraints, although the WG is modeled as an air domain with no any elastic material feature. The WG domain free mesh of the MM interface results such as a virtual fixed constraint on the walls [4].

This strategy significantly allows to relieve computation complexity, avoiding wall meshing and resulting in a reliable modeling.

Results
Both deformations and temperature increase cause a decrease of the RF efficiency in the operative band of the SPC. The small decrease of performance confirms the well design of the SSPA's copper support slab and the right choice of Alumina as substrate, instead of Duroid.

Conclusions
The FL SPC technology has been studied using FEM Multiphysics simulation implemented on COMSOL Multiphysics®, and many aspects has been investigated at the same time such as mechanical stress and thermal expansion together with the EM behavior of SPC.

The in-frequency behavior of the electric field and S-parameters has been computed in thermal stress operative conditions.

Reference

[1] Ha Trong Than, George W. Sun, Geovanni S. Cuellar, Jiyang Zeng, Nate T. Schultz, Michael E. Moya, Younkyu Chung, Blythe C.Deckman, Michael P. DeLisio "Design and Performance of a 600-W –Band Amplifier Using Spatially Combined GaAs FETs for Satellite Communications", IEEE JOURNAL OF SOLID-STATE CIRCUITS, VOL. 47, NO. 10, OCTOBER 2012, 0018-9200/$31.00 © 2012 IEEE.
[2] COMSOL RF Module User's Guide Version : May 2012 COMSOL 4.3.
[3] COMSOL Structural Mechanics Module User's Guide Version : May 2012 COMSOL 4.3.
[4] COMSOL Multiphysics User's Guide; Version: 2010 COMSOL 4.0

Figures used in the abstract

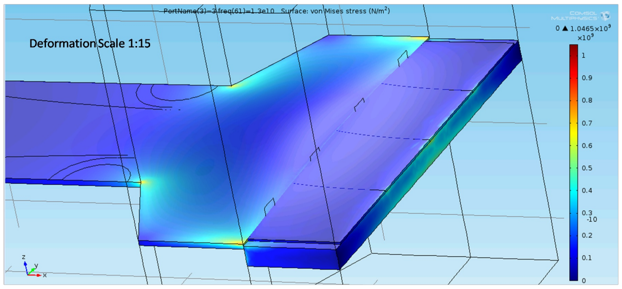

Figure 1: Stress

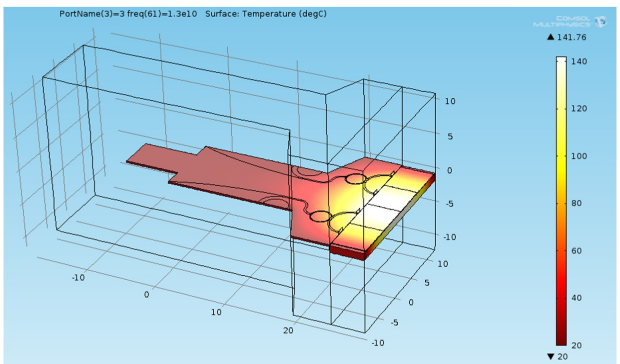

Figure 2: Temperature

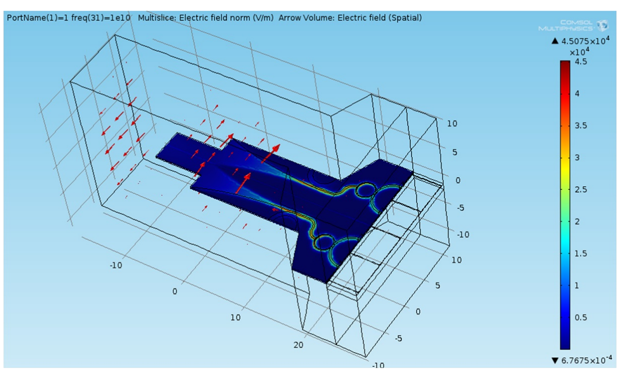

Figure 3: Electric Field

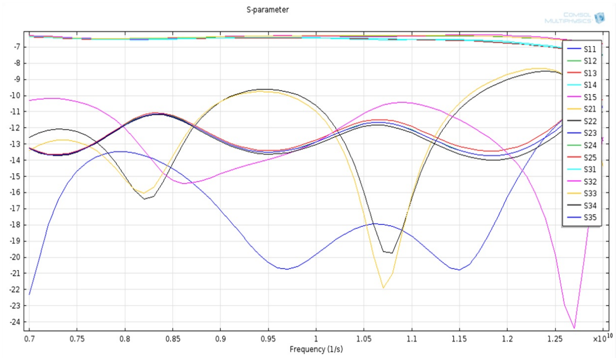

Figure 4: Scattering Parameters