Virtual Prototyping of a Microwave Fin Line Power Spatial Combiner Amplifier

A. Leggieri*[1], F. Di Paolo[1], and D. Passi[1]
[1]Department of Electronic Engineering, University of Rome "Tor Vergata", Italy
*A. Leggieri: Department of Electronic Engineering, University of Rome "Tor Vergata",
Via del Politecnico, 1, 00133, Rome, Italy, alberto.leggieri@uniroma2.it, leggieri@ing.uniroma2.it

Abstract: This paper describes the Virtual Prototyping based on a COMSOL Multiphysics simulation for a novel X-Band Fin Taper (FT) Spatial Power Combiner (SPC) Amplifier.

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 TEM mode, in order to be amplified by a Solid State Power Amplifiers.

The power dissipation of the MMIC amplifiers produces a considerable temperature increase, stresses and strains with consequent displacement of the structures, which alter the desired behavior of the device. These multiple effects have been investigated at the same time.

The model is organized by using Thermal Stress (HT), Moving Mesh (MM) and Electromagnetic Waves (EMW) COMSOL modules.

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

Keywords: Finline structures, Rectangular Waveguide, Spatial Power Combiner, Heat Transfer in Solids, Structural Mechanics.

1. Introduction

SPATIAL POWER COMBINING is a suitable approach to design High Power Amplifiers in the High frequency range. In comparing to the binary combining, this solution offers several advantages as high device compactness, low combining losses, higher available power outputs and heat sinking facilitations [1].

In the described device, the power provided by four Monolithic Microwave Integrated Circuit (MMIC) Solid State Power Amplifiers (SSPA's) is carried by microstrip transmission lines (µSTL's) to two Wilkinson Power Combiners. After this first binary combining stage, the outgoing power is sent to two opportune Fin Tapers placed inside an air filled rectangular Waveguide (WG), in order to be spatially combined by exciting the WG fundamental mode. Several circuital and technological solutions have been adopted. In order to reduce the combining loss and size, exponential FT to µSTL's transitions has been considered [2], using antipodal configuration. In order to improve the operative band, a parasitic void has been implemented by inserting an anti-resonance metal in the antipodal transition profile [3].

Both µSTL's and FL hot and ground conductors are made of gold and printed on an Alumina Al2O3 95% substrate. The SSPA's are made of Gallium Arsenide and, in order to ensure the right heat sinking, are placed on a copper slab. WG is made of copper. The combining structure is printed on a substrate card, and several of these cards can be placed inside a single WG. This principle ensures low combing losses, easy integration, heat sinking, space saving and great output powers. The proposed SPC structure is reported in Figure 1.

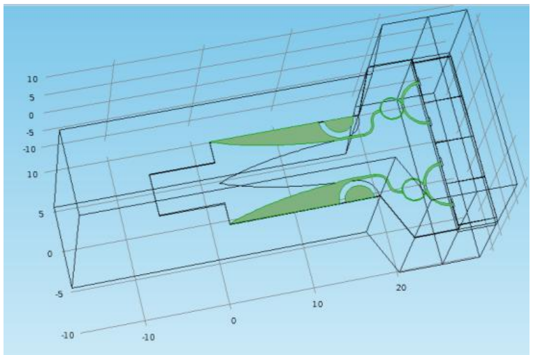

Figure 1. The proposed SPC

2. Use of COMSOL Multiphysics

The power dissipation of the MMIC SSPA's produces a considerable temperature increase and induces a thermal expansion of both the PA's and the connected structure. If the SSPA's temperature exceeds the maximum allowed value (specified by the SSPA's vendor), an amplification failure or device damage may occur.

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 Electromagnetic (EM) behavior of the structure.

In order to perform a Radiofrequency (RF) characterization of this SPC, a five-port network Scattering (S) parameters representation is needed. Therefore a frequency domain (FD) analysis is mandatory and it besides allows to estimate the power density distribution of the lowest mode in the transversal cross section of the WG and FT in the steady state condition, including the μSTL ports.

A Thermal Stress (TS) stationary analysis allows to determine the temperature and the deformation when the heat generated by the MMIC's power dissipation has been diffused on all the reachable SPC components and the system has become thermally stable, since the external temperature is by then steady over all the SPC outer boundaries which are exposed to the external environment.

A Finite Element Method (FEM) based Multiphysics simulation using COMSOL can couple TS and EM analysis by Moving Mesh (MM) dedicated interface and storing temperature information. In order to decrease computational time and resources maintaining accuracy, the device model is organized by using several strategies allowed by COMSOL. The architecture of the model is based on TS, MM and Electromagnetic Waves (EMW) COMSOL modules.

In order to ensure a great model reliability, all the materials are temperature dependent, except for the alumina, since the vendor provides only a single number per parameter valid in the range from 20°C to 300°C, this range is comprised in the temperature range of simulation, which is from 20°C to 140°, and has been evaluated previously with a single heat simulation.

The mesh is composed in different settings: Air domain is set with minimum element size of 0.6 mm. In order to perform an accurate computation of the port field, the rectangular port boundary is set with a minimum size of 0.1 mm, which is $3.3{\cdot}10^{-2}\ \lambda$, where λ is the wavelength at 10 GHz. The substrate domain is set to 0.6 mm and the metallization boundaries to 0.01mm which is the minimum edge that compose the Wilkinson circles. Then the MMIC domain is set to 0.1 mm, the height of the device, and the holder to 06 mm as the WG.

The lumped port boundaries are set at 1μm which is 1/250 of the substrate height minimum edge of the SPC which is 10^{-4} m long and is $3{\cdot}10^{-5}\ \lambda$. By using the default normal settings of COMSOL mesh interface for other mesh features, an accurate discretization has been reached with non excessive computational cost.

2.1 Thermal Stress

An innovation in the TS and MM modeling with COMSOL is about the WG walls representation and its behavior regard the thermal expansion. This consists in using opportune condition described by a combination of Heat Transfer in Fluids, Highly conductive Layer and structural Fixed Constraints in the TS interface.

WG walls are intended as in a stationary temperature regime, cooled by the external environment, so they are non deformable by thermal stress and consists in virtual fixed constraints, since the WG domain is described only to transfer heat from the internal solid to the external WG walls. By using this condition, walls meshing are avoided.

The only necessary fixed constraints remains the contact boundaries between the substrate and the WG external walls, in order to appreciate the all the compressive forces induced by the thermal expansion, although the WG is modeled as an air domain with no any thermal linear elastic material feature. In the MM module, the domain free mesh of WG results such as a fixed constraint of the walls. This strategy significantly allows to relieve computation complexity, resulting in a reliable modeling, that symmetry conditions are not indispensable.

The TS module is employed to describe the Thermal-Structural formulation of the problem by using the following TS module features [4]:

- Thermal Linear Elastic, default: The combining structure and the SSPA's are modeled in this feature in order to compute all the thermal-stress dynamics over its constituent materials. The solid model is intended as isotropic and the structural transient behavior as quasi-static.

- Heat Sources: Each MMIC SSPA represents a constant volume heat source. The heat power density is calculated from the SSPA's Power Added Efficiency (PAE) at its maximum power output, resulting in $Q=8.7[GW/m^3]$ for a dissipated power of $P_d=20[W]$.

- Highly conductive Layer: The metallization which constitutes the µSTL and the FT's is modeled as a highly heat conductive layer, in order to avoid the meshing of the thin printed conductors volumes, saving computational cost. The specified thickness is 17 µm for both the hot and the ground conductors.

- Heat transfer in Fluids: The air inside the WG is used to model only the heat transfer from the combining structure to the external environment. The formulation refers to the Gas-Liquid fluid type. WG is modeled as an air domain.

- Fixed constraints: The contact between the substrate and the WG external walls has been intended to be fixed, in order to appreciate the all the compressive forces induced by the thermal expansion, although the WG is modeled as an air domain with no any thermal linear elastic material feature. By imposing this condition, the simulation can show the substrate deformation when it is placed inside a cooled WG but heated by the amplifier power dissipation.

- Temperature: The temperature of the SPC outer boundaries is constant and equals the external temperature, in order to model the thermal steady state.

- Heat flux: All external boundaries are used to compute the inward heat flux, with a heat transfer coefficient to the external environment of 5 Wm-2K-1. This feature allows to save more computational cost.

2.2 Moving Mesh

The MM module is employed to give the rules on how to move the mesh in function of the displacement computed by the TS analysis, by employing the Arbitrary Lagrangian Euleran formulation (ALE). In order to perform the RF analysis on the deformed geometry, the MM module uses the following features [5]:

- Prescribed deformation: The combining structure and the SSPA's represent the volumes subjected to deformation. The displacement vectors (u, v, w) computed by the TS module are employed to specify this volumetric deformation. Its prescribed Mesh displacement is set to dx = u, dy = v, dz = w.

- Free deformation: The air volume (which is not subjected to any structural elastic formulation by the TS analysis) is free to move. The initial deformation is set to $dx_0 = 0$, $dy_0 = 0$ and $dz_0 = 0$.

- Prescribed Mesh Displacement: This condition specifies that the boundary is to be deformed by the thermal stress computation, though is attached to the free deformation air boundary. The combining structure and the SSPA's boundaries are subjected to deformation. This superficial displacement has been specified by setting the prescribed mesh displacement to d_x = u, d_y = v, d_z = w.

2.3 Electromagnetic Waves

The EMW module is employed to describe the EM modeling of the SPC combining space and is applied only to the combining structure: SSPA's with copper slab support and terminal WG section are excluded from the EMW model.

Microstrip propagating modes are non-TEM very complex to calculate, which can difficultly computed by using a numeric port, generating very high non linear problem and absorbing high computational resources.

A significant innovation in RF modeling with COMSOL, produced in this study, is the µSTL ports representation: Fringe effects are computable, by introducing a Perfect Magnetic Conductor (PMC) boundary condition close to the lumped ports at the interface with the WG back boundary which is assigned to a Scattering Boundary condition (SBC). By using this strategy, fringe electric field can exist out of the lumped port, subtracting power to the field inside the port boundary, so that it decrease of the fringe field amplitude and the in-port electric field amplitude, probed by the simulator, can have the right value, as the port was numeric.

This results in a simple computation of the electric field so that all the structure can be simulated without symmetry representation, increasing output precision. However by setting the port parameter sweep, for scattering parameter computation, only to the first three ports a great speed increase is allowed since the second and the fifth ports are located symmetrically respect the WG symmetry axis, parallel to the long side WG planes; the same applies to the third and fourth ports.

In order to describe the EM behavior of the PSC, the following RF module features are been employed [6]:

- Rectangular Port: The rectangular WG port is represented as a rectangular WG TE10 mode port.

- Lumped Element: The resistors of the Wilkinson power divider are represented as user defined lumped elements.

- Lumped port: the μSTL ports are represented as lumped ports. As the wavelength is much smaller than the port width, a local quasi-static approximation can be done and a lumped port can be used. These ports are defined vertically in order to model the interface between the SPC and the SSPA's, a particular Perfect Magnetic Conductor condition is imposed near the Lumped ports to allow the fringe effect. The ports are set as voltage driven showing 50Ω characteristic impedance. By using this strategy, an enough accurate computation of the non-TEM port mode is ensured and much computational cost is saved.

- Perfect Magnetic Conductor: Microstrip lines carries a non-TEM EM mode, which is typically analyzed with the quasi-TEM representation as a superposition of TE and TEM modes. The non transverse component of the fields flows out from the substrate dielectric and fringing in the air. In order to consider the fringe components of the electric field close to the lumped port boundaries, Perfect Magnetic Conductor (PMC) boundary conditions has been imposed on the peripheral boundaries between the microstrip ports and the waveguide back open boundary, in which scattering boundary condition is placed.

  The typical estimated fringe fields of a μSTL, with width w and height h, extends to 3/2w out of the hot conductor metallization and to 2h out of the dielectric height. The peripheral boundaries, assigned to PMC, have been set to have a height of 3h and a width of 4w.

  By imposing this condition, the electric field near the microstrip ports is allowed to have non transverse component, respecting the fringe effect of the microstrip line.

- Impedance boundary condition: The external WG boundaries are modeled in order to consider the losses due to the partial penetration of the electric field in the lossy material which constitutes the WG walls. This condition allows to not include an external domain for modeling the volume penetrated by the electric field, avoiding the meshing and saving computational cost. The specified thickness of the WG boundaries is 1.5 mm.

- Transition boundary condition: The thin gold layers of the μSTL and the FT metallization are modeled in order to allow for a discontinuity in the fields across the interface so that both losses and a phase shift across the interface can be considered. By using these features the meshing of the thin printed conductor layers is not required, and the layers are modeled as boundaries. The thickness is specified in the Transition boundary condition as 17 μm, and the losses are taken from the material conductivity.

- Scattering boundary condition: The back WG boundary will be connected to the same but mirrored structure or can be left open. The plane wave which crosses this boundary must be free to feed forward. This condition makes a boundary transparent for a scattered wave and potential resonances are avoided.

2.4 Analysis

In order to consider the computed temperature resulted from the TS analysis, the information has been inserted as the default temperature in all the modules in the RF module settings, where required. By the MM module the meshes of the model has been moved in function of the displacement computed by the TS analysis, in order to couple the TS an RF simulation by performing the RF analysis on the deformed geometry. The RF FD analysis has been performed between 7 and 13 GHz. The solver is organized in performing two steps: First, a stationary analysis to compute the thermal TS and MM in fully coupled mode, then a FD step to perform a stationary analysis which calculates the electric fields and the scattering parameters of the SPC.

3. Results

3.1 Temperature and Heat Flux

By imposing a power dissipation of 20 W, the TS stationary analysis has shown a maximum temperature over the SSPA's of 141°C, perfectly respecting the maximum temperature allowed and the maximum power output of the chosen SSPA's. The temperature on the combining and amplifying structures is shown in Figure 2, and isothermal contours in Figure 3. The Power dissipations is ensured by proper copper carriers. The Heat is directed towards the external WG walls as shown in Figure 4.

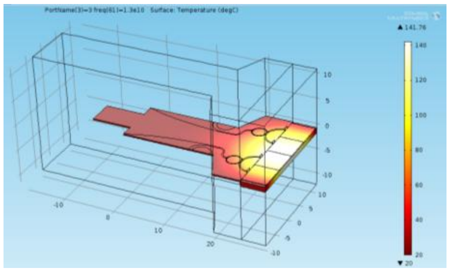

Figure 2. Temperature.

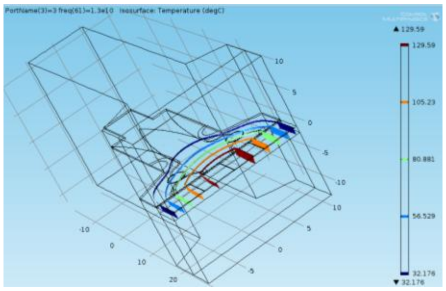

Figure 3. Temperature - Isothermal Contours.

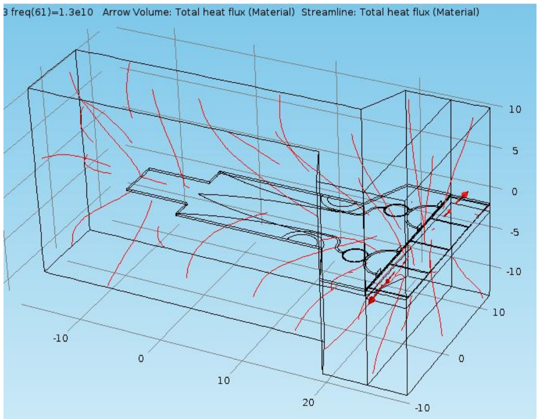

Figure 4. Total Heat Flux

3.2 Stress and Displacement

By receiving the stored Temperature the fully coupled stationary analysis, has been shown the following results. The deformation scale has been increased in order to better show the displacements. In the following figures, black outlines represent the original conformation, and the stained volume represents the deformed structure.

- The maximum stress is near the oblique sectors and is 1GNm^{-2}, as shown in Figure 5 and 6.

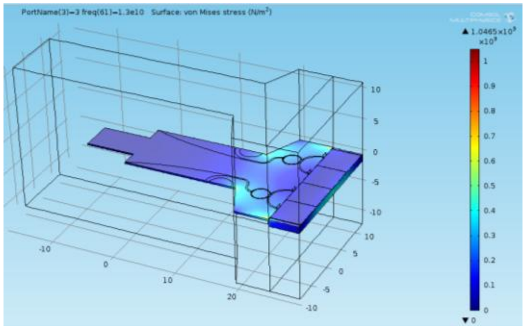

Figure 5. Stress

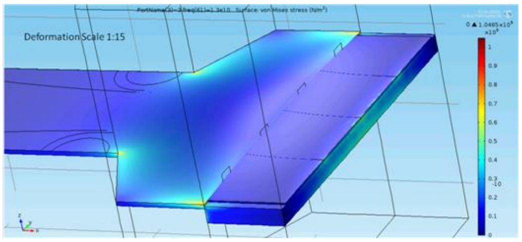

Figure 6. Stress - Magnified scale

- The maximum total displacement is located near the interface between the SSPA's and the SPC and the terminal boundary of the substrate in the TE10 direction of propagation, which is 33.3 µm.

  The maximum displacement of the metallization along the direction of the electric filed vector (y) is 2.8 µm, and 3.5 µm on the wave propagation direction.

  Displacements are described in Figure 7 and 8.

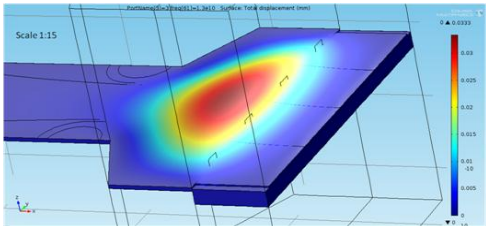

Figure 7. Displacement - Magnified scale

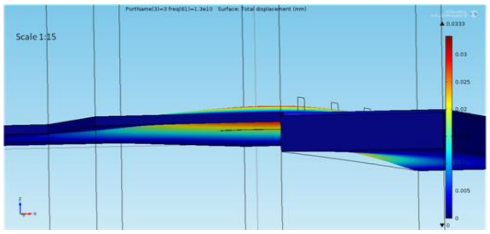

Figure 8. Displacement - Profile - Magnified scale

These values are very small respect the wavelength and the microstrip dimensions, so will result negligible from the guiding properties of the structure, as shown by the following scattering parameters analysis.

On the other hand, this displacement value is completely incompatible with the GaAs survivability to displacement: for such reason, an interface layer is needed between the back of the GaAs MMIC and the copper carrier: used materials are CuW or CuMo.

3.3 Electric field plot

The simulation output shows the field power density distribution of the lowest mode in the transversal cross section of the WG and the field on the FT in the steady state condition, including the µSTL ports. Figure 9 shows this plot referring to the frequency of 10 GHz.

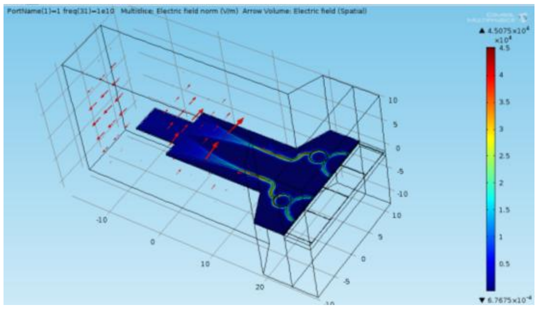

Figure 9. Electric field plot.

3.4 Scattering parameters

S-parameters versus frequency matrix without hating and in working conditions are plotted respectively in Figures 10 and 12. The transmission S-parameters are also plotted in figures 11 and 13 for a better view. Both deformations and temperature increase cause a negligible decrease of the RF efficiency in the operative band of the SPC, as shown below. This result confirms the well design of the SSPA's copper support slab and the right choice of Alumina as substrate, instead of Duroid.

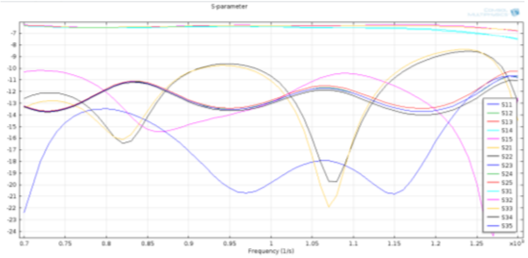

Figure 10. S-parameters - without heating.

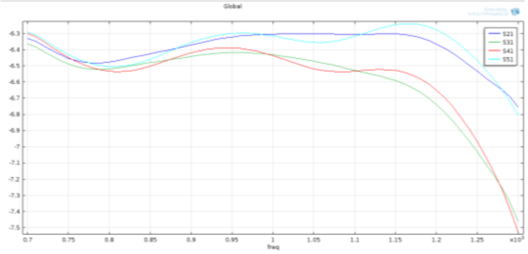

Figure 11. S-parameters - transmission - without heating.

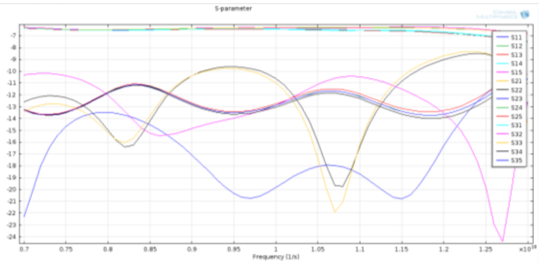

Figure 12. S-parameters - working conditions.

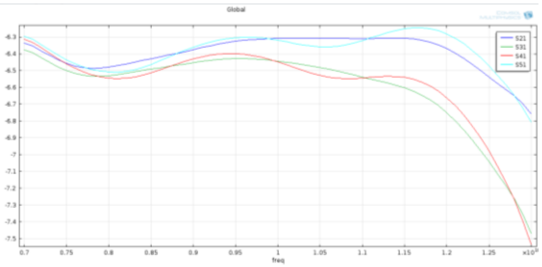

Figure 13. S-parameters - transmission - working conditions.

7. Conclusions

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

In order to decrease computational time and resources maintaining accuracy, the device model has been organized by using several strategies allowed by COMSOL.

A significant innovation in RF modeling with COMSOL, 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.

Another innovation is in the TS and MM modeling about the WG walls representation and its behavior regard the thermal expansion. This consists in using opportune condition described by a combination of Heat Transfer in Fluids, Highly conductive Layer and structural Fixed Constraints in the TS interface.

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 module results such as a virtual fixed constraint on the walls.

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

Temperature, stress and displacement have been computed in operative conditions.

The electric field power density has been calculated and plotted in cold and thermal-stress operative conditions.

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

Expected results are been obtained and, according to this simulation, the appropriate materials have been chosen in order to ensure the correct operation of the device in thermal stress affected working conditions.

8. References

1. Ha Trong Than et al. “Design and Performance of a 600-W C–Band Amplifier Using Spatially Combined GaAs FETs for Satellite Communications”, *IEEE Journal Of Solid-State Circuits*, VOL. 47, NO. 10, OCTOBER 2012.

2. B.Bhat, S.K.Koul: "Analysis, Design and Optimization of Fin Lines", Artech House, 1987.

3. George E. Ponchak, Alan N. Downey: “A New Model for Broadband Waveguide to Microstrip Transition Design”, *NASA Technical Memorandum 88905*, Lewis Research Center Cleveland, Ohio, December 1986.

4. *COMSOL Structural Mechanics Module User’s Guide* Version : May 2012 COMSOL 4.3.

5. *COMSOL Multiphysics User’s Guide,* Version: 2010 COMSOL 4.3

6. *COMSOL RF Module User’s Guide* Version: May 2012 COMSOL 4.3.