COMSOL CONFERENCE ROTTERDAM2013

Virtual Prototyping of a Microwave Fin Line Power Spatial Combiner Amplifier

Alberto Leggieri, Franco Di Paolo, Davide Passi

Department of Electronic Engineering
University of Roma "Tor Vergata"
00133 Roma Italy, email alberto.leggieri@uniroma2.it

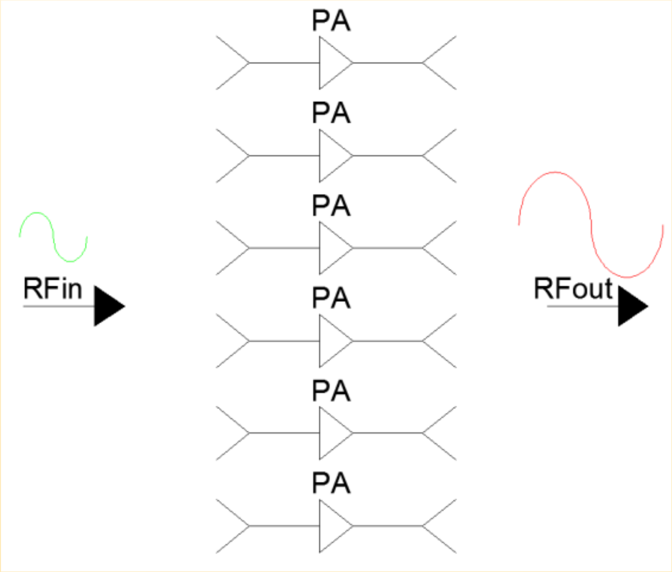
UNIVERSITA' DEGLI STUDI DI ROMA TorVergata

COMSOL

Outline

- Spatial Power Combiners (SPC)
- Motivations
- Fin Taper SPC Features
- Use of COMSOL Multiphysics
- Simulation Results
- Conclusions

Spatial Power Combiners (SPC)



Energy in low loss EM modes and combined in Dielectric → **low combining losses**

Many amplifiers → **high available power outputs**

Much available space → **heat sinking facilitations and device compactness**

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 TL SPC architecture, the energy is distributed in low loss electromagnetic modes and combined in the TL dielectric, using a single stage of power combining; this reduces the ohmic combining losses and makes it quite independent to the number of devices [1]

Motivations

Multiphysics Simulation Ambient

Working conditions:

Thermal Expansion

↑↓

Mechanical Stresses

Electromagnetic behavior

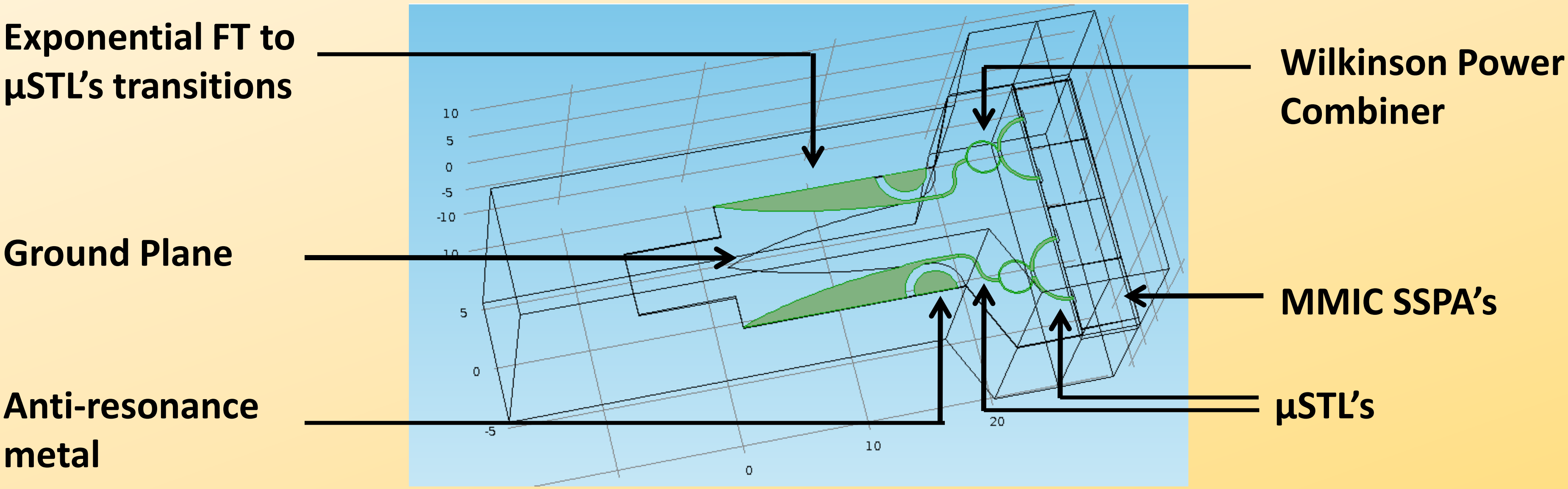
High Power = Many amplifiers in a SPC = Heat = Thermal expansion = Stress and displacement

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.

This virtual prototyping consists in a Multiphysics simulation which shows the electromagnetic operation of the proposed SPC under thermal stress working conditions.

Fin Taper SPC Features

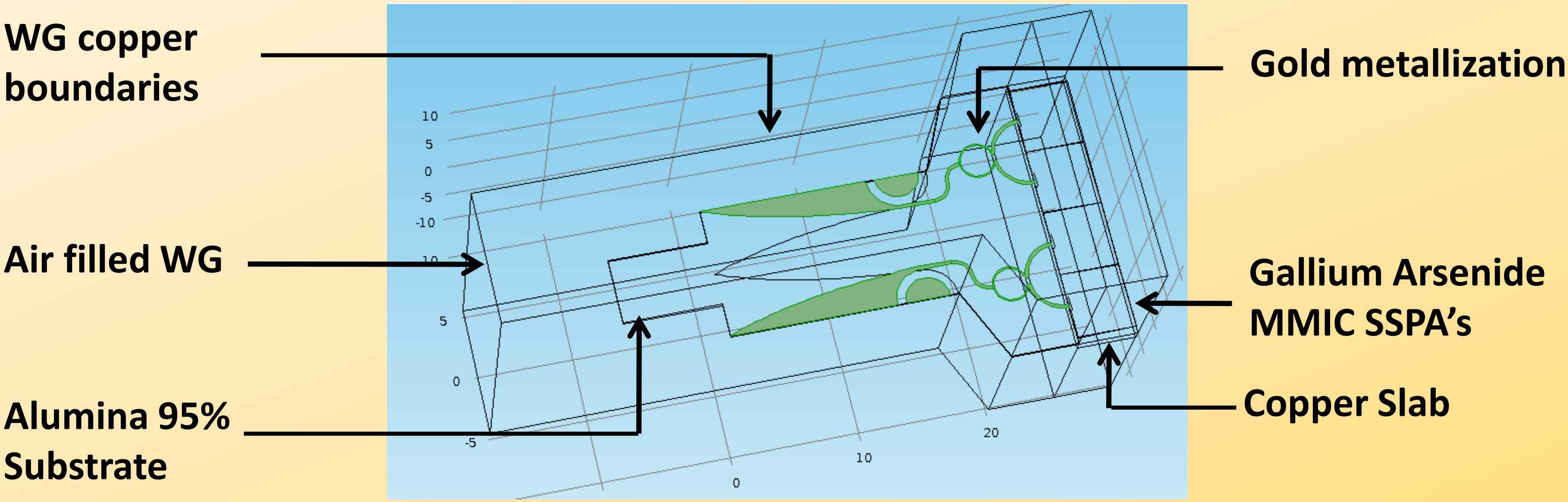
Fin Taper SPC Structure



In the proposed device, 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 Taper (FT) to µSTL's transitions placed inside an air filled rectangular Waveguide (WG), in order to be spatially combined by exciting the rectangular WG fundamental mode. 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].

Use of COMSOL Multiphysics

Fin Taper SPC 3D Model - Materials



Both µSTL’s and FL hot and ground conductors are made of gold and printed on an Alumina Al_2O_3 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.

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.

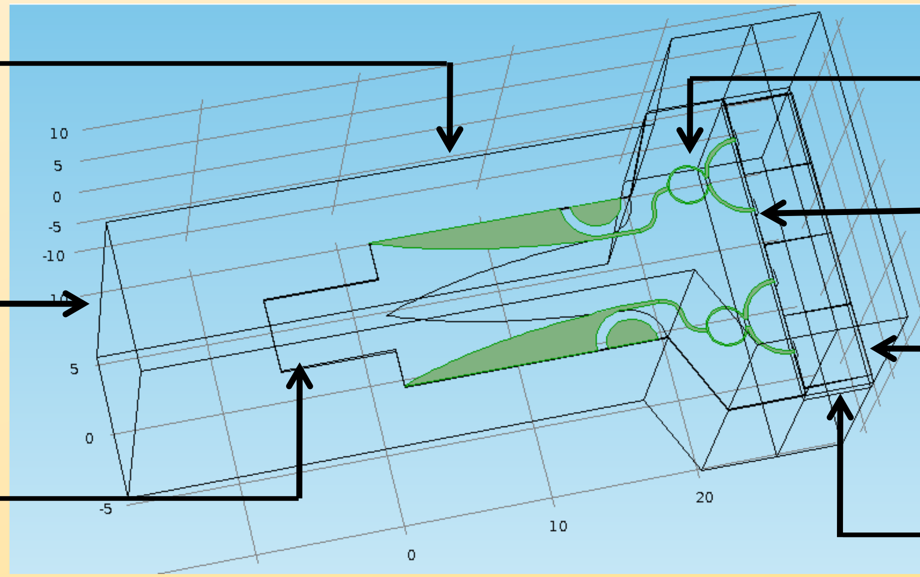
Use of COMSOL Multiphysics

Fin Taper SPC 3D Model - Mesh

Air filled WG
mES= 0.6 mm

Input Port
mES= 0.1 mm

Alumina 95%
Substrate
mES= 0.6 mm



Gold metallization
mES= 0.01mm

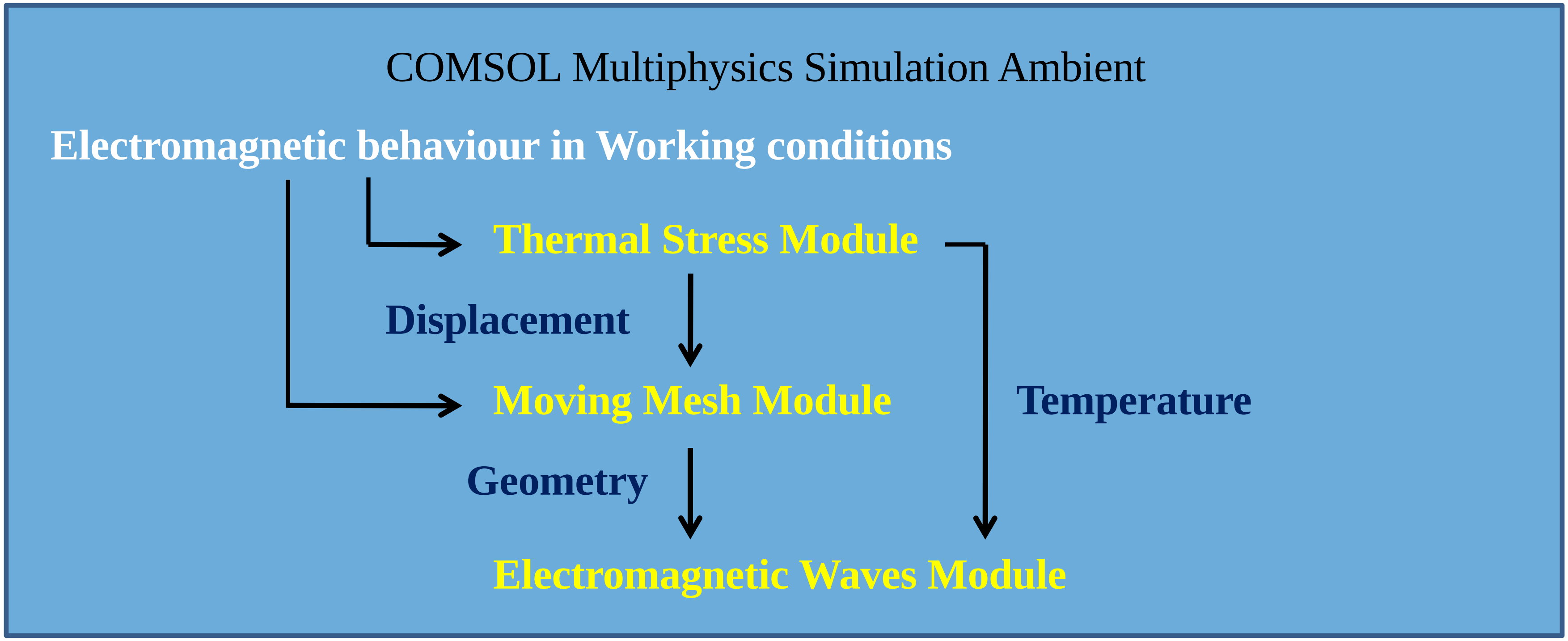
Lumped port
mES=1µm

MMIC SSPA's
mES= 0.1 mm

Copper Slab
mES= 0.6 mm

In order to obtain an accurate simulation and saving computational cost, the mesh is composed in different settings: Air domain is set with minimum element size (mES) of 0.6 mm and the rectangular port boundary with a minimum size of 0.1 mm which is $3.3 \cdot 10^{-2}$ λ, where λ is the wavelength at 10 GHz. The substrate domain is set to 0.6 mm and the metallization boundaries to 0.01mm. MMIC domain is set to 0.1 mm, the height of the device, and the holder to 0.6 mm. 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 $3 \cdot 10^{-5}$ λ.

Use of COMSOL Multiphysics

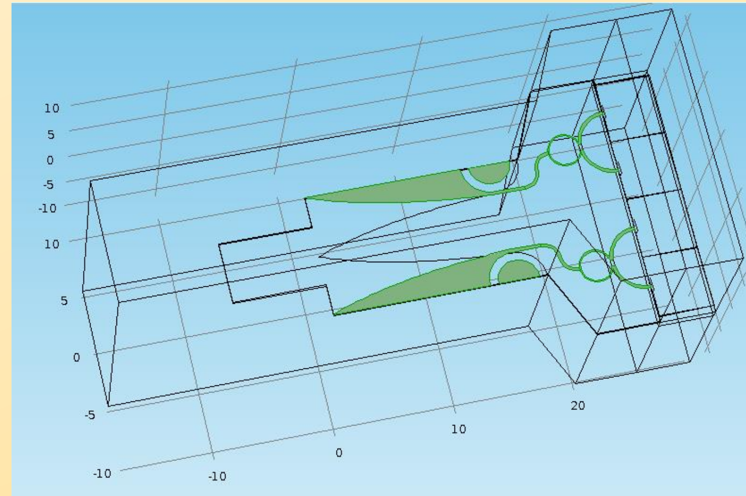


COMSOL can couple TS and EM analysis by Moving Mesh (MM) dedicated interface and storing temperature information.
The TS module calculates temperatures and displacements [4].
The MM module moves the mesh in function of the displacement computed by the TS analysis [5]. TS and MM are employed in a single stationary analysis.
The EMW calculates the Electric field and Scattering Parameters of the SPC [6], by performing a Frequency domain stationary analysis on the new mesh and temperature.

Use of COMSOL Multiphysics

Thermal Stress Module

Temperature and displacement computation applied to the entire structure



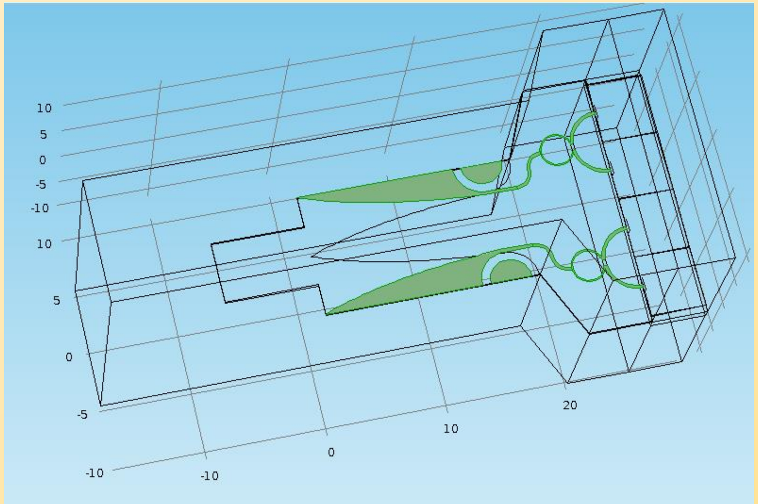
- Thermal Stress (*ts*)
  - Thermal Linear Elastic Material 1
  - Free 1
  - Thermal Insulation 1
  - Initial Values 1
  - Heat Transfer in Fluids 1 → **Air which fills the WG internal volume**
  - Temperature 1 → **WG external walls cooled by the external environment**
  - Highly Conductive Layer 1 → **Printed Circuit Metallization (17µm)**
  - Fixed Constraint 1 → **Contact between substrate and WG external walls**
  - Heat Source 1 → **MMIC SSPA $Q=8.7 GW/m^3$ since $P_d=20$ W**

WG walls are in a stationary temperature regime, cooled by the external environment (25°C), are not deformable by thermal stress, since the WG domain is described only to transfer heat from the internal solid to the external WG walls. The only necessary fixed constraints remains the contact boundaries between the substrate and the WG external walls.

Use of COMSOL Multiphysics

Thermal Stress Module

Temperature and displacement computation applied to the entire structure



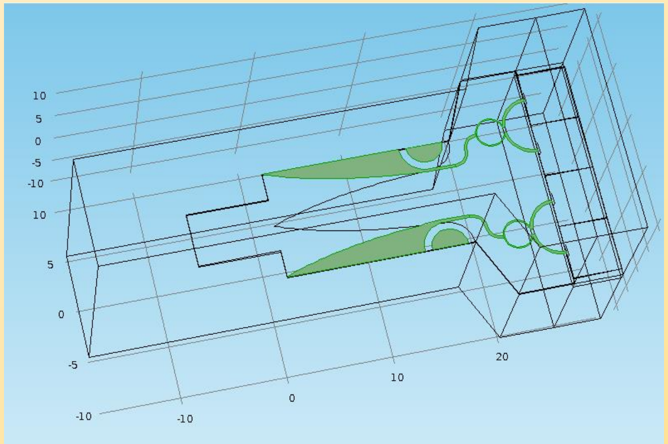
- Thermal Stress (ts)
  - Thermal Linear Elastic Material 1
  - Free 1
  - Thermal Insulation 1
  - Initial Values 1
  - Heat Transfer in Fluids 1 → **Air which fills the WG internal volume**
  - Temperature 1 → **WG external walls cooled by the external environment**
  - Highly Conductive Layer 1 → **Printed Circuit Metallization (17µm)**
  - Fixed Constraint 1 → **Contact between substrate and WG external walls**
  - Heat Source 1 → **MMIC SSPA $Q=8.7[GW/m^3]$ since $P_d=20[W]$**

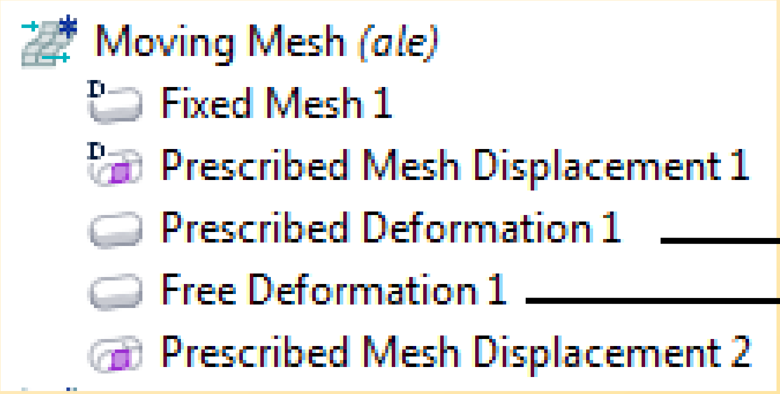
Each MMIC SSPA represents a constant volume heat source. The heat power density is calculated from the SSPA's Power Added Efficiency (PAE) at its maximum power output, resulting in $Q=8.7[GW/m^3]$ for a dissipated power of $P_d=20[W]$.

Use of COMSOL Multiphysics

Moving Mesh Module

Mesh displacement computation applied to the entire structure





- Prescribed Deformation 1 → **Combining structure and SSPA's volumes**
- Free Deformation 1 → **Air which fills the WG internal volume**
- Prescribed Mesh Displacement 2 → **Combining structure and SSPA's surfaces adjacent to the air volume**

The combining structure and the SSPA's represent the volumes subjected to structural formulation by TS analysis.

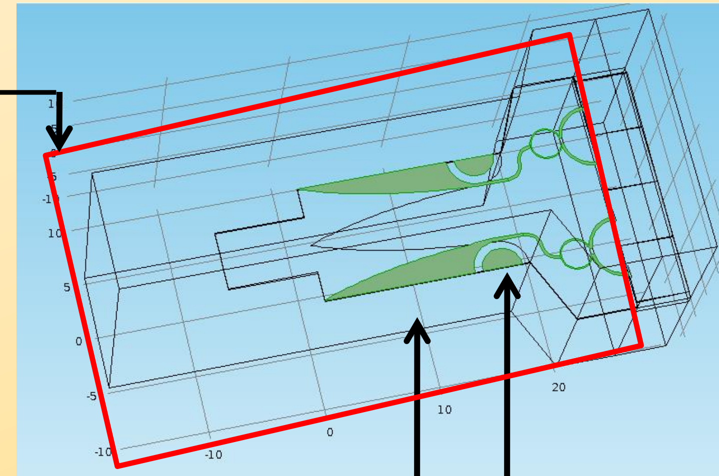
The air volume is free to move, since is subjected only to heat transfer in fluid formulation by the TS analysis.

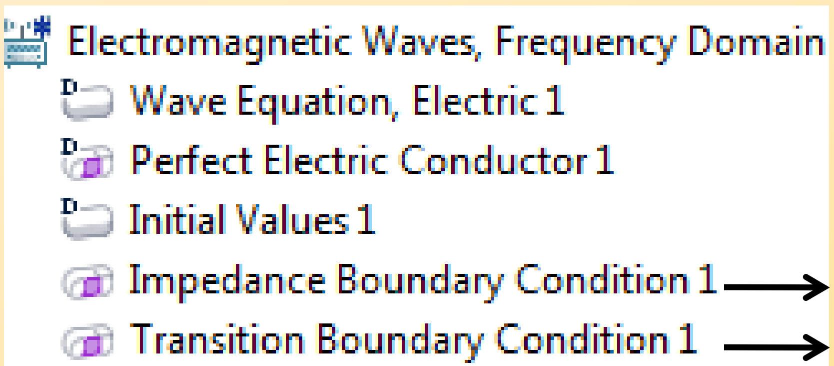
Combining structure and SSPA's surfaces adjacent to the air volume are deformed by the thermal stress computation, though are attached to the free deformation air boundary.

Use of COMSOL Multiphysics

Electromagnetic Waves Module

Electric field calculation in the combining space
Applied only to the combining structure:
SSPA's with copper slab support and terminal WG section are excluded.





Impedance Boundary Condition 1 → **External WG boundaries (1mm)**

Transition Boundary Condition 1 → **Printed Circuit Metallization (17μm)**

The external WG boundaries are modeled in order to consider the losses due to the partial penetration of the electric field in the lossy walls material.

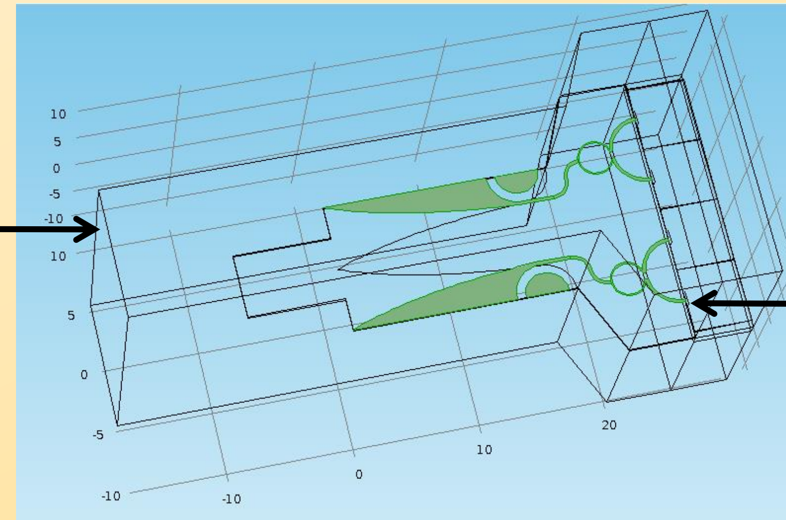
The thin gold layers of the μSTL and the FT metallization are modeled in order to allow for a discontinuity in the fields across the interface.

By employing these conditions, walls and printed circuit domains can be not included to the model.

Use of COMSOL Multiphysics

Electromagnetic Waves Module

Electric field calculation in the combining space



Port 1 **Rectangular WG TE_{10} mode Input port.** →

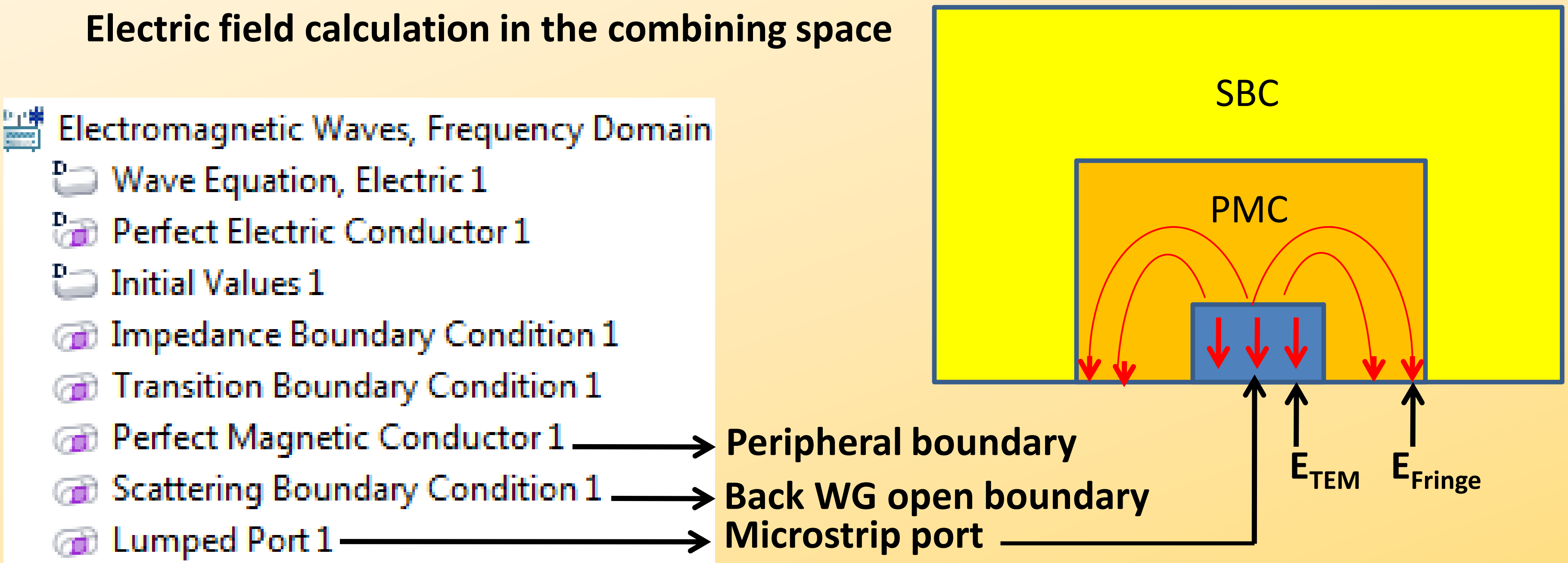
Electromagnetic Waves, Frequency Domain
- Wave Equation, Electric 1
- Perfect Electric Conductor 1
- Initial Values 1
- Impedance Boundary Condition 1
- Transition Boundary Condition 1
- Perfect Magnetic Conductor 1 → **Peripheral boundary for Fringe effect consideration**
- Scattering Boundary Condition 1 → **Back WG open boundary**
- Lumped Port 1 → **Microstrip port: Interface between SSPA and µSTL**

The back WG boundary remains open, then is transparent for a scattered wave and potential resonances are avoided; it is represented by a Scattering Boundary Condition (SBC). By employing the Lumped Element Boundary Condition, the resistors of the Wilkinson power divider are represented as user defined lumped elements.

Use of COMSOL Multiphysics

Electromagnetic Waves Module

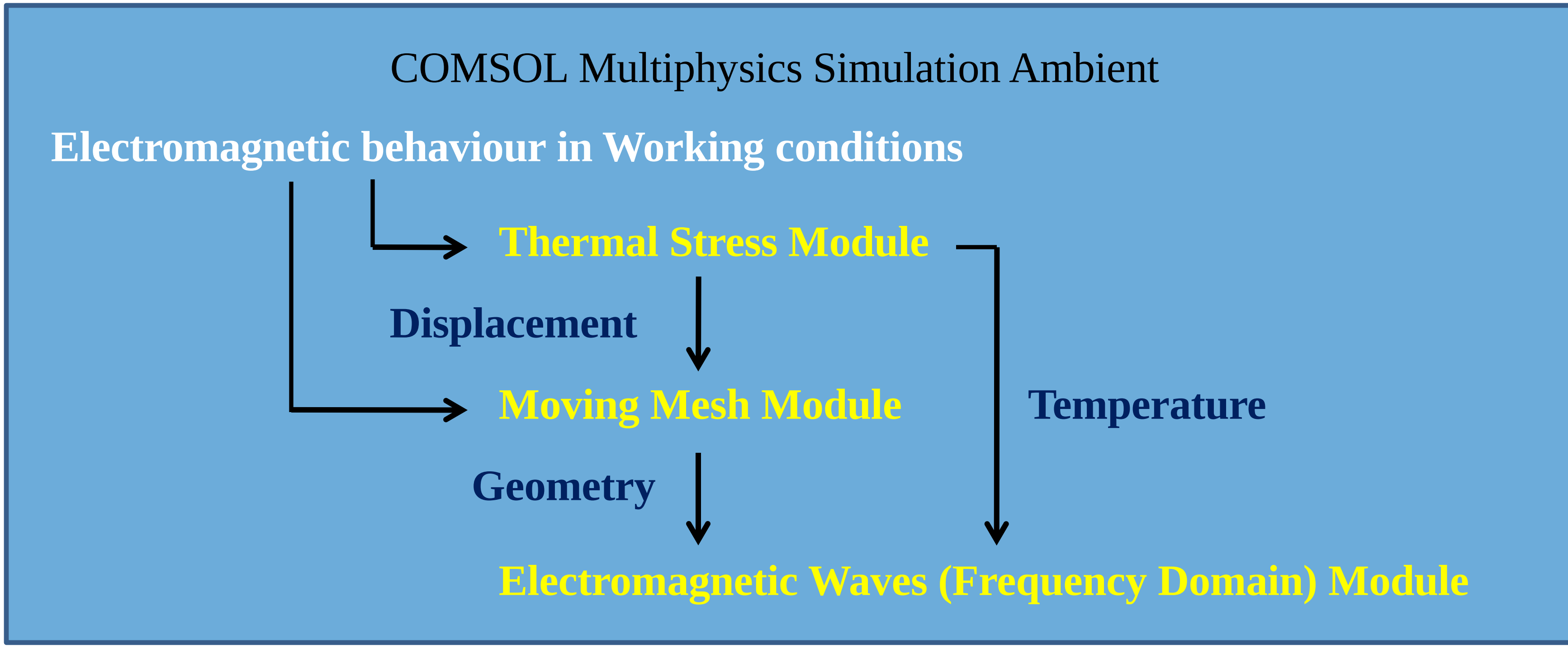
Electric field calculation in the combining space

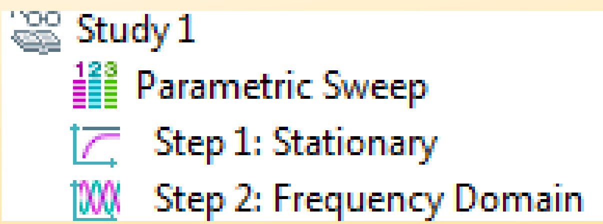


Fringe effects are considered by introducing a Perfect Magnetic Conductor (PMC) boundary condition on a peripheral boundary between the microstrip ports and the remaining waveguide back open boundary. Peripheral boundaries have a height of 3h and a width of 4w, where h and w are respectively the height and the width of the µSTL ports.

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 port electric field probed by the simulator, can have a very accurate value.

Use of COMSOL Multiphysics

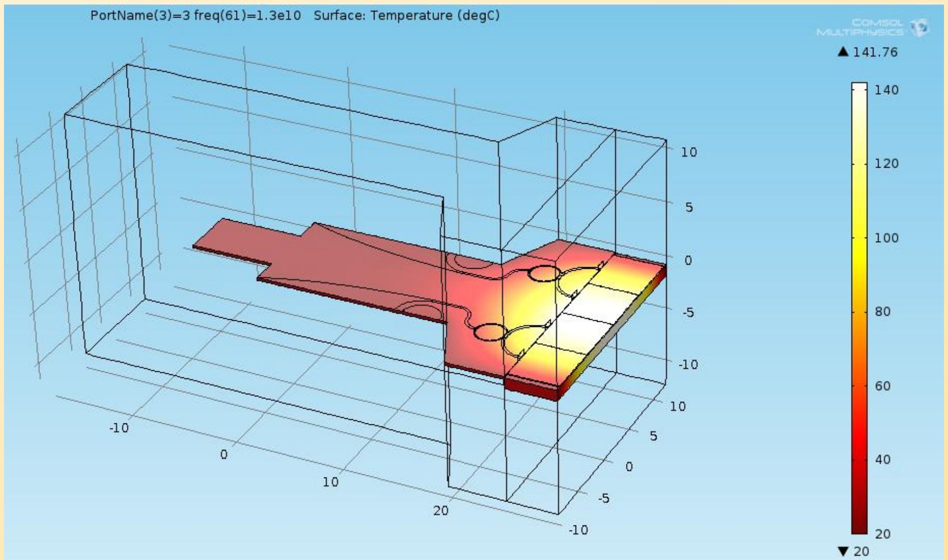


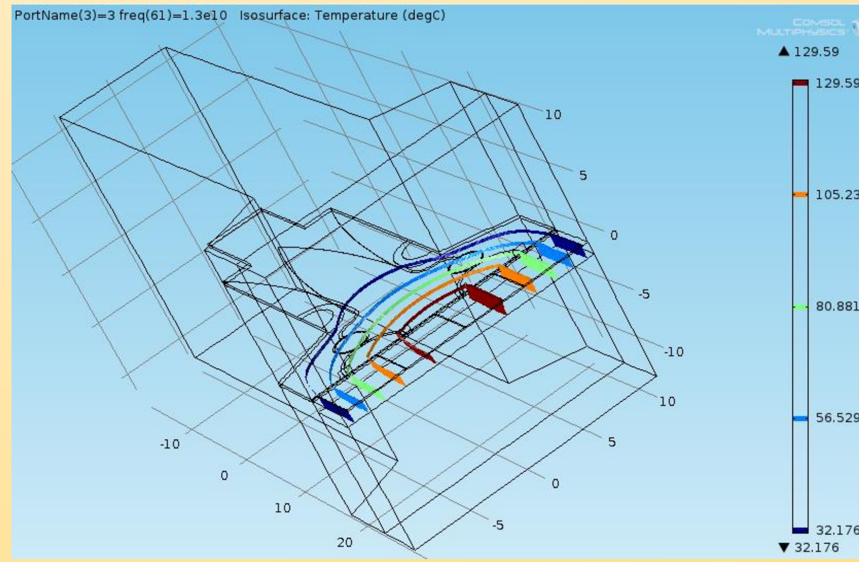


The solver is organized in performing two steps: First, a stationary analysis to compute the thermal TS and MM in fully coupled mode, then a Frequency Domain (FD) step.
The RF FD analysis has been performed between 7 and 13 GHz, by employing Parametric Sweep.

Simulation Results

Temperature

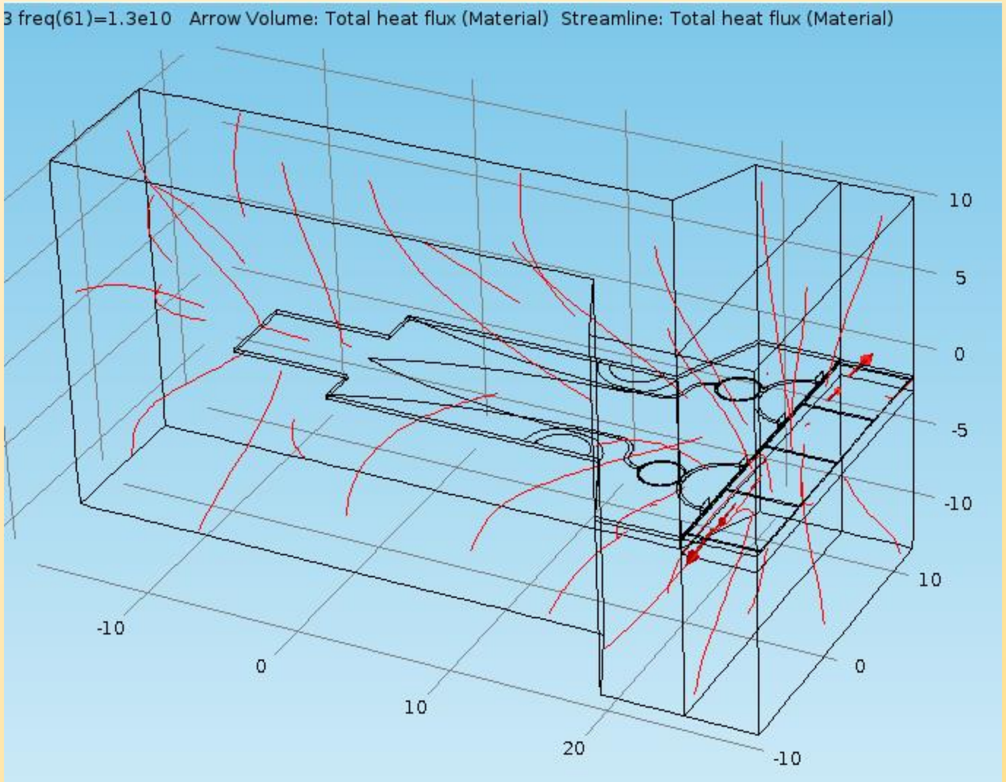




By imposing a power dissipation of 20 W, the TS stationary analysis has shown a maximum temperature over the SSPA's of 141°C, perfectly respecting the maximum temperature allowed and the maximum power output of the chosen SSPA's

Simulation Results

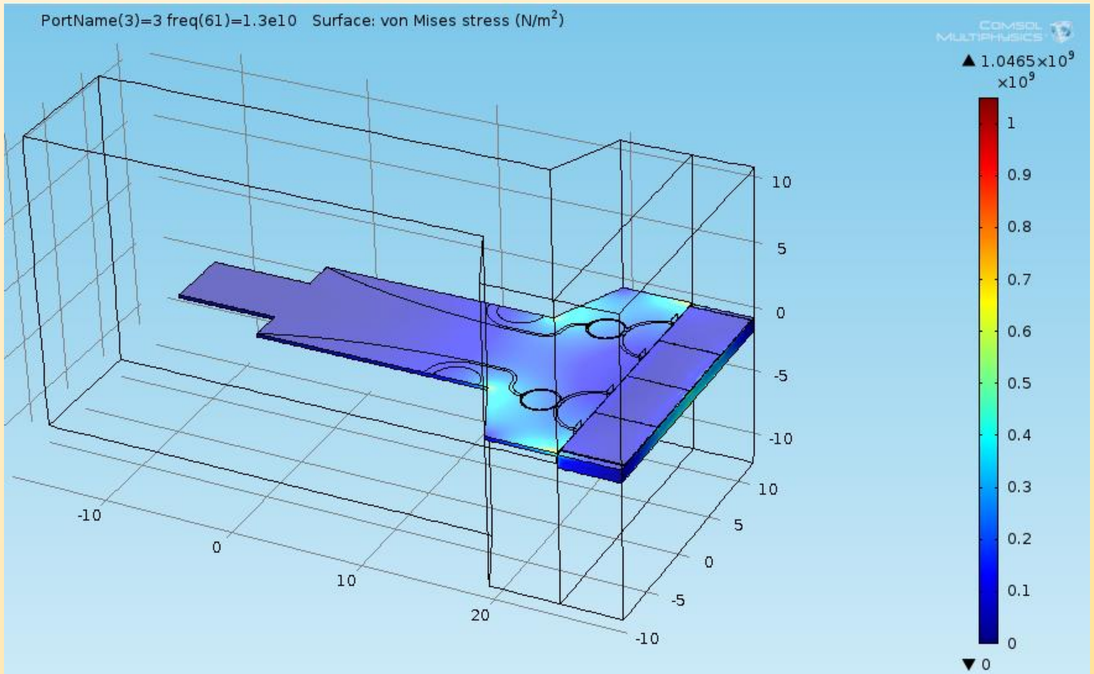
Heat Flux

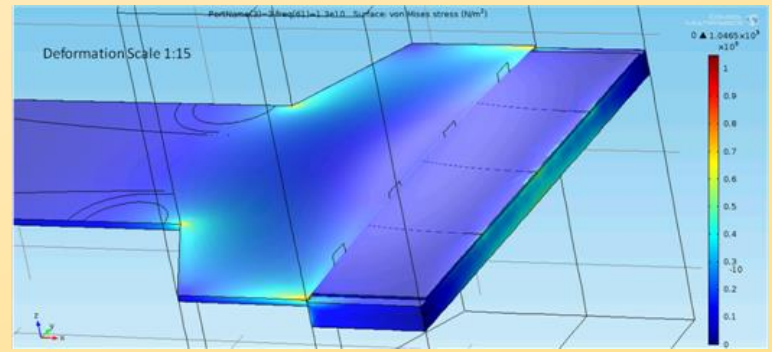


The Power dissipations is ensured by proper copper carriers. The Heat is directed towards the external WG walls

Simulation Results

Stress



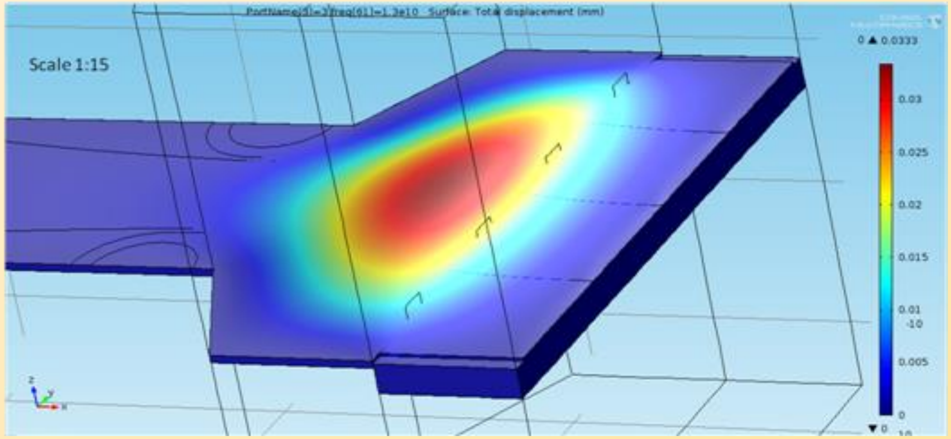


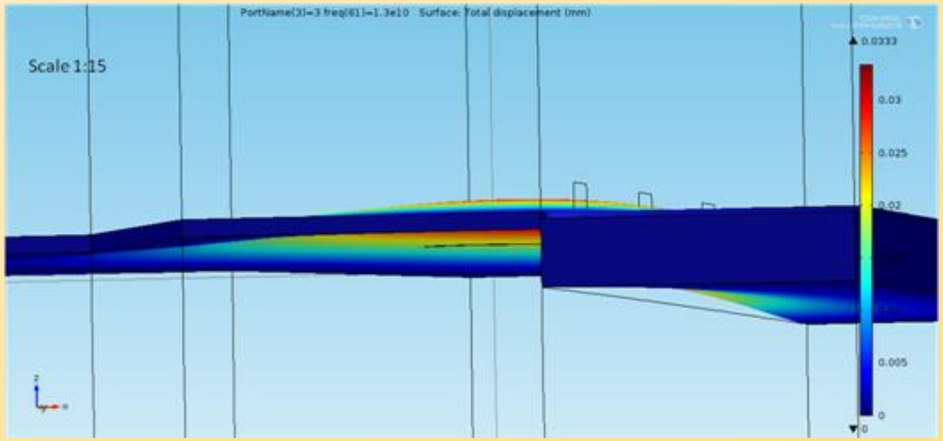
The deformation scale has been increased in order to better show the displacements.
In the figures, black outlines represent the original conformation, and the stained volume represents the deformed structure.

The maximum stress is near the oblique sectors and is 1GNm^{-2}

Simulation Results

Displacement





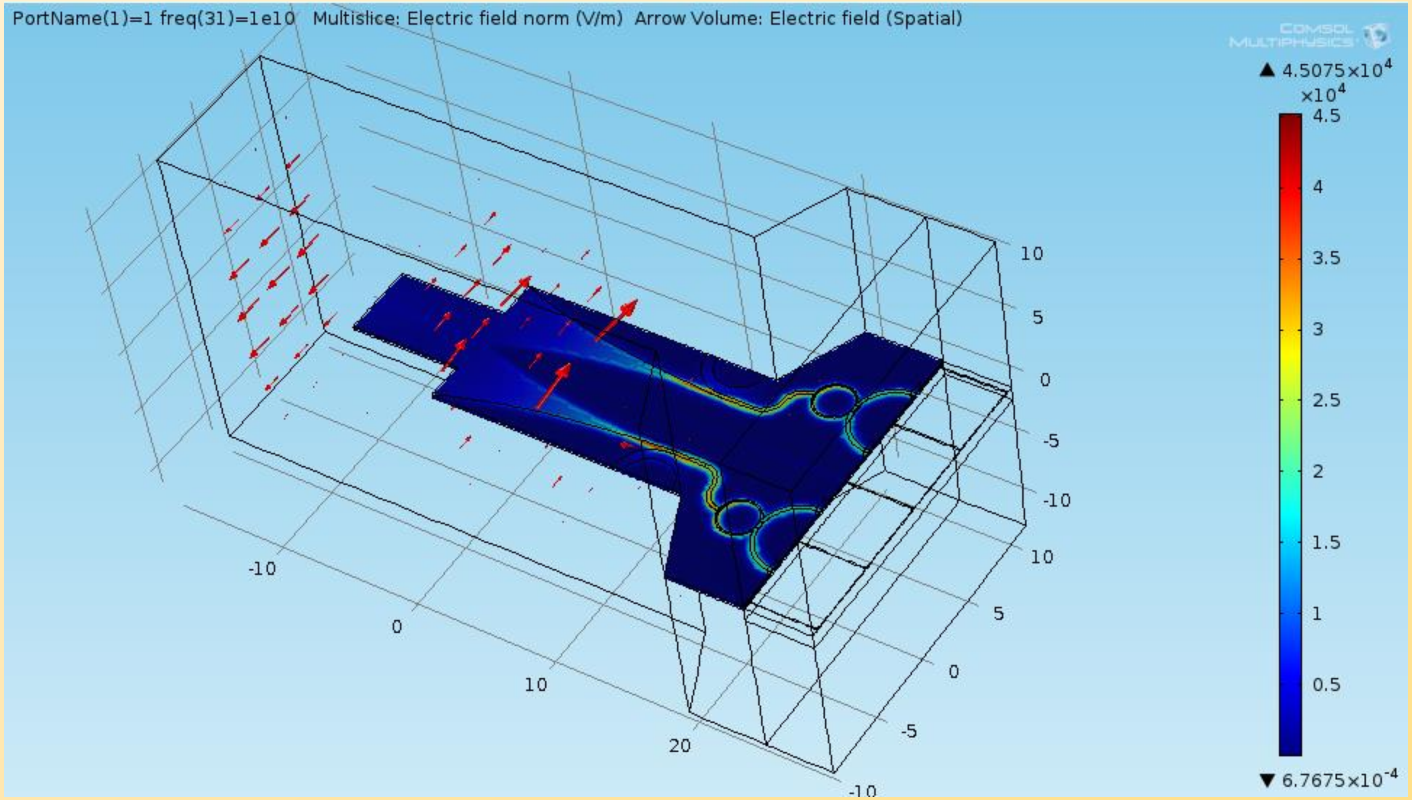
The maximum total displacement is located near the interface between the SSPA's and the SPC and the terminal boundary of the substrate in the TE10 direction of propagation, which is 33.3 µm.

These values are very small respect the wavelength and the microstrip dimensions, so will result negligible from the guiding properties of the structure.

On the other hand, is 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.

Simulation Results

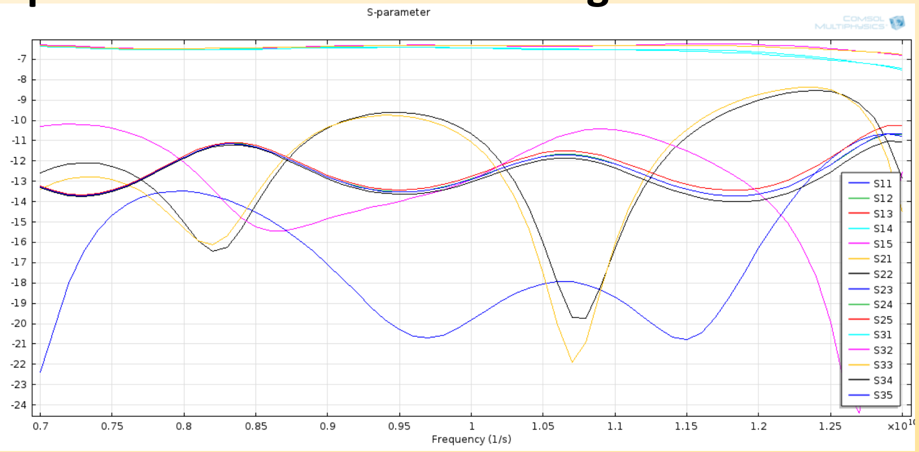
Electric Field

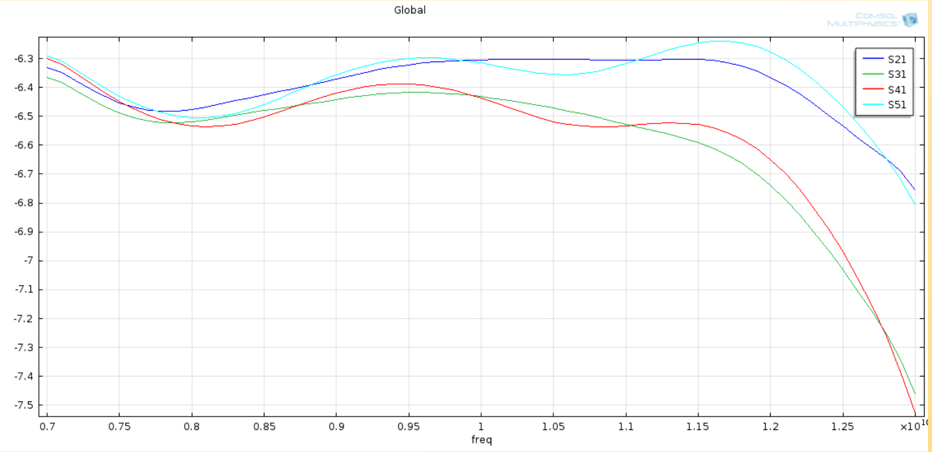


The simulation output shows the field power density distribution of the lowest mode in the transversal cross section of the WG and the field on the FT in the steady state condition, including the µSTL ports, referring to the frequency of 10 GHz.

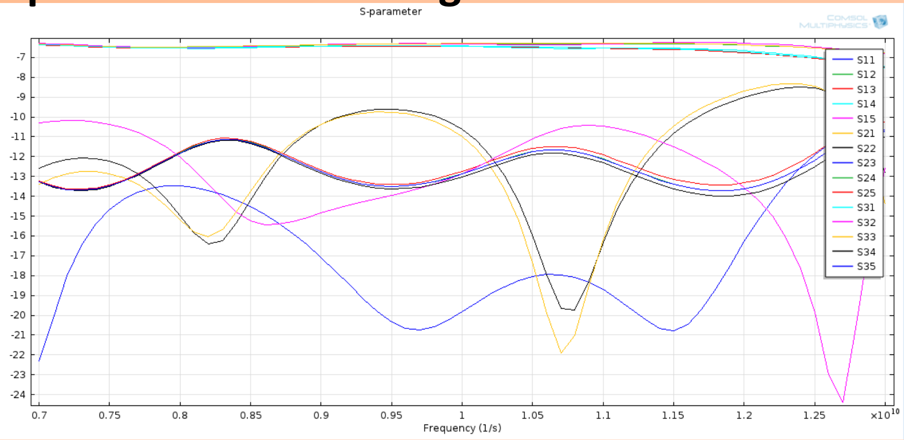
Simulation Results

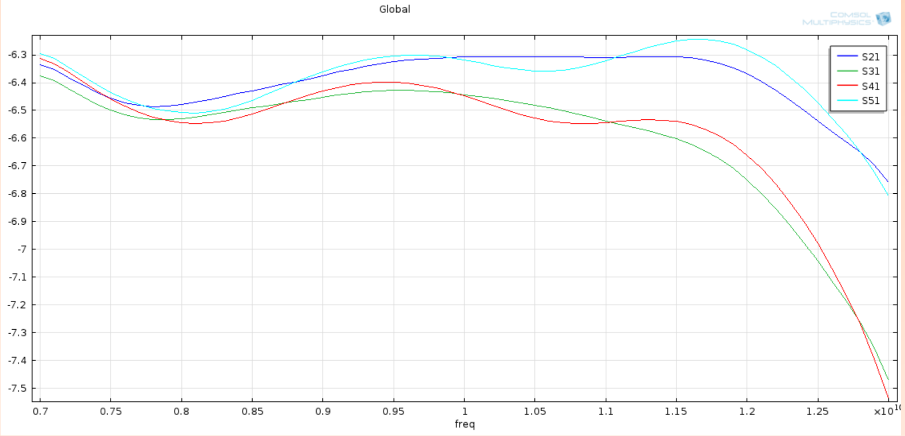
S-parameters - without heating





S-parameters – working conditions





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

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.

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

•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.

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.