

HOT WIRE

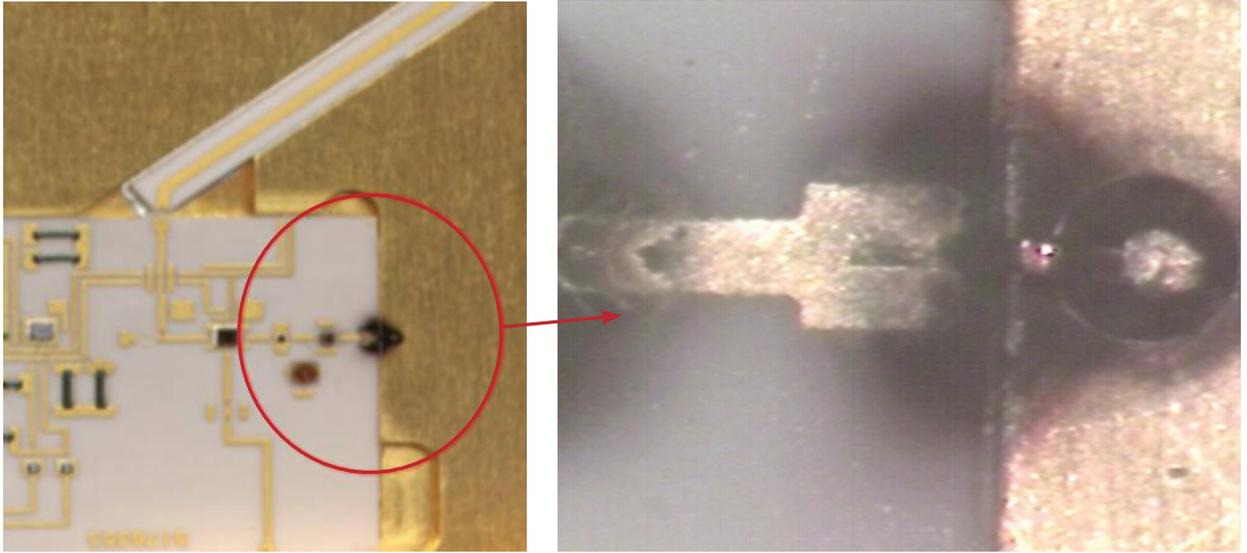
Multiphysics simulation helps to achieve robust electronics design for high-power antennas and microwave components.

By Amedeo Larussi

Senior Principal Electrical Engineer
Raytheon Corporation, Goleta, U.S.A.

The aerospace and defense industry is charged with delivering advanced electronics systems faster and at a lower cost than ever before. Antenna and microwave design engineers must balance competing requirements for reduced size, high power delivery, rock-bottom cost and excellent reliability. The result is that antennas and microwave components operate at higher power levels and higher frequencies while being contained in smaller form factors. The inevitable outcome is increased risk that temperatures will greatly impact product performance. Traditionally, electrical design and thermal design are the responsibility of two different groups, each operating

Antennas and microwave components operate at higher power levels and higher frequencies while being contained in smaller form factors.



▲ Photos (normal left, magnified right) show damage to the microwave junction.

with their own separate requirements and analysis tools – and with only limited cross-group interaction. This common failure to more fully account for design dependencies has, in some documented cases, resulted in serious product malfunctions.

For example, the heat generated by microwave components can increase the dielectric loss tangent of some materials; the consequence is more heat production and the potential for a runaway reaction. In extreme cases, product failure could prevent a mission from being accomplished or even cause loss of life. Combining electrical, thermal and structural simulations often provides unprecedented insight toward preventing failures and improving product performance. Raytheon Corporation – a technology and innovation leader specializing in defense, security and civil markets throughout the world – uses comprehensive robust electronic design solutions to improve the reliability of its products, reduce time to market, and control engineering and manufacturing costs.

MULTIPHYSICS SIMULATION – A BRIEF HISTORY

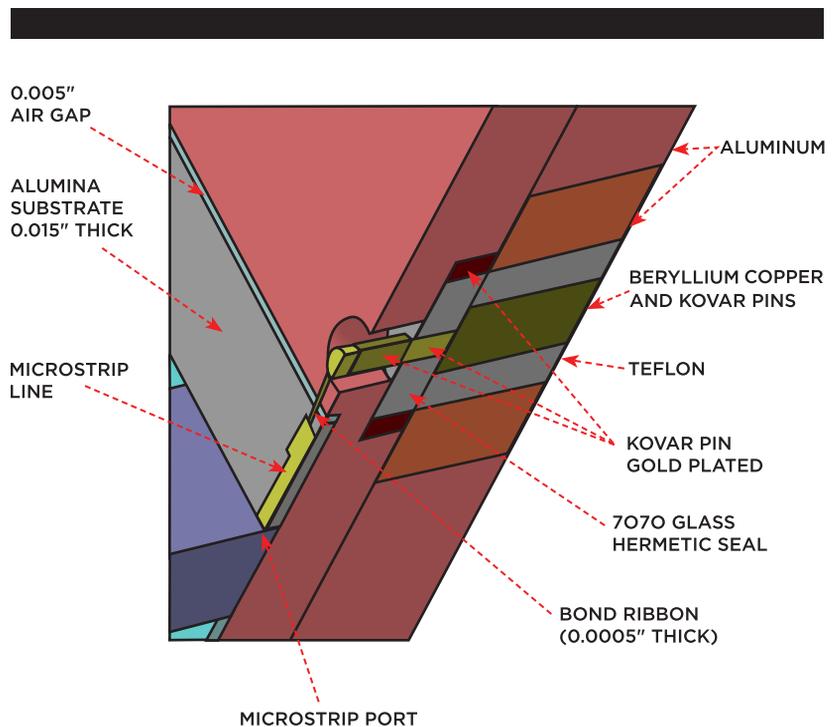
Engineers have long been interested in combining high-frequency electromagnetic simulation with thermal analysis, but before the turn of the millennium, there was no efficient way of doing so. About 2002, Raytheon management

encouraged investigation into the potential for coupled simulation capabilities. This led to the selection of ANSYS HFSS to enable coupling between electromagnetics and thermal analysis. Raytheon engineers began using the tools extensively to design microwave systems with excellent results. In 2007, the group needed to add vibration and fluid dynamics capabilities to the coupled analysis toolkit. With

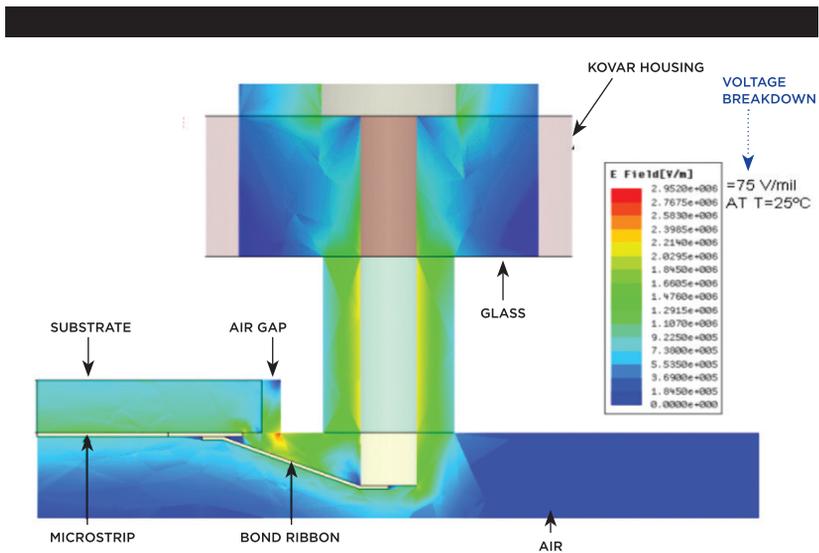


ROBUST ELECTRONIC SYSTEM DESIGN PRACTICES FOR AEROSPACE AND DEFENSE PRODUCTS
ansys.com/83wire

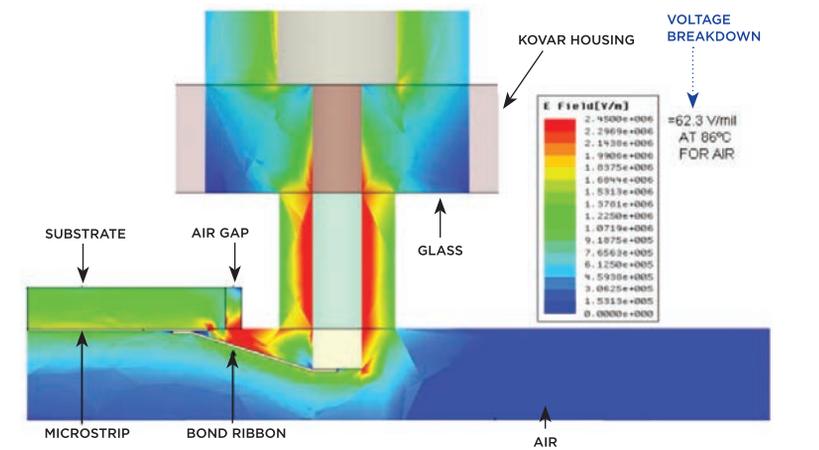
HFSS integrated into the broader ANSYS simulation portfolio (for example, ANSYS Mechanical and ANSYS Fluent), it was easy and intuitive to perform multiphysics simulations within ANSYS Workbench.



▲ ANSYS HFSS model of microwave junction



▲ Electrical field analysis at 25 C



▲ Electrical field analysis at 86 C

Raytheon engineers took advantage of integration capabilities in ANSYS Workbench to capture electromagnetic and thermal interdependencies.

VOLTAGE BREAKDOWN AT MICROWAVE JUNCTION

In an example from a recent project, a high-power signal is received by the antenna plane. The effective received radiation signal flows to a microwave feed circuit. Although both the electrical and thermal groups signed off on the design, a voltage breakdown occurred at a microwave junction, where a co-axial pin connects to a microstrip trace at the frequency of interest. Shortly after power was turned on, excessive heat destroyed

the connector. To address this, Raytheon engineers modeled the components in HFSS. This software accurately models microwave components, such as tuning screws and probes, to a fine level of detail. HFSS employs the finite element method, using small unstructured mesh elements when needed, along with large elements to reduce processing time without sacrificing accuracy. Adaptive meshing refines the mesh automatically in regions in which field accuracy needs to be improved.

Raytheon engineers imported initial design geometry from a computer-aided design (CAD) file. They defined the electrical properties of the materials, such as permittivity, dielectric loss tangent and bulk electrical conductivity for the Kovar® housing, alumina substrate, Teflon® insulator, and beryllium, copper and Kovar pins. Engineers then defined boundary conditions that specify field behavior on the surfaces of the solution domain and object interfaces. They defined ports at which energy enters and exits the model. HFSS computed the full electromagnetic field pattern inside the structure, calculating all modes and ports simultaneously for the 3-D field solution. (The dielectric properties of the materials are temperature dependent.) The HFSS electrical field analysis at 25 C showed that the electrical field in the area in which the failure occurred does not exceed 1.5×10^6 volts per meter (V/m), as compared to the 2.952×10^6 V/m value for voltage breakdown in air.

COUPLING ELECTRICAL AND THERMAL SIMULATION

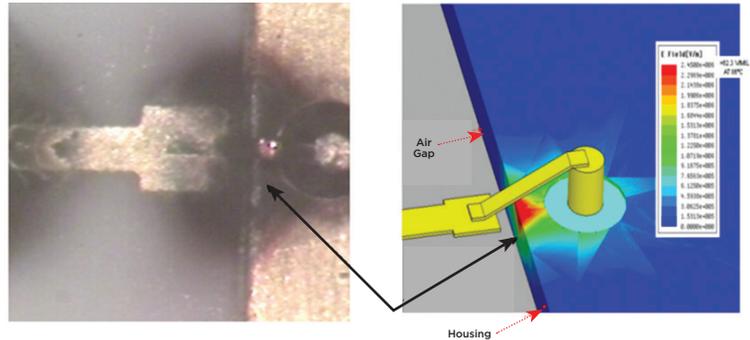
The real-life situation is more complex because ambient temperature affects the dielectric properties of the materials, and the dielectric properties of the materials affect the heat that is generated by microwave components. Raytheon engineers took advantage of the integration built into ANSYS Workbench between HFSS and ANSYS Mechanical to capture these interdependencies. The HFSS model was coupled to ANSYS Mechanical to perform a transient thermal simulation. Boundary conditions for natural convection cooling were added on the bottom face. The temperature distribution was used to perform a static structural analysis.

Engineers employed ANSYS Workbench coupling to apply temperature fields (determined by physical measurements) to ANSYS Mechanical to calculate the thermal stresses associated with these temperatures. The structural simulation showed high stresses and deformation up to $22 \mu\text{m}$ in the inner connector. Thermal analysis indicated that temperatures actually reached 86 C on the bond ribbon and the pin near the point where they connect, which translated into a lower breakdown voltage. Raytheon

engineers re-analyzed the components at 86 C using the dielectric properties at the higher temperature and discovered that the electrical fields in the area where the failure occurred exceeded the 2.45×10^6 value for voltage breakdown in air at this temperature.

The simulation results helped Raytheon engineers understand how the failure occurred, and they corrected the design to eliminate future failures. The team solved the electromagnetic model at the initial temperature, sent the electromagnetic loss to the thermal simulation to determine the impact of the losses on temperature, sent the temperatures back to the electromagnetic model to calculate losses on the new temperatures, and continued to iterate until steady-state temperature changes were reached. After a few more changes to the materials used in the product, the simulation showed that the design worked perfectly, and this was confirmed by physical testing. ▲

The simulation showed that the design worked perfectly; this was confirmed by physical testing.



▲ Multiphysics analysis accurately predicts voltage breakdown damage. Physical damage on left and simulation on right. Fields are plotted on the right.



- Consolidation using Remote Visualization
- Optimized HPC solutions for your Enterprise
- Gompute On Demand - The CAE Cloud solution
- Private, Public and Hybrid HPC Cloud computing

Discover Gompute- visit www.gompute.com