Tooling Up for MicroElectroMechanical Systems (MEMS)

Advances in electrostatics, electrostatic-structural coupling, and reduced-order modeling

By Dale Ostergaard
Team Leader, Multiphysics Development Group, ANSYS, Inc.

With the rapid advancement and success of micromachining technologies for the fabrication of microsystem devices, there is an ever-increasing need for virtual prototyping tools. Complicating the issue is the need for sophisticated multiphysics analyses of the device components, as well as broad system-level simulations using electronic design automation (EDA) tools. Advancing multiphysics simulation tools for the design of micromachines as well as integrating their results into system-level EDA tools are therefore high priorities in computer-aided engineering (CAE).

The ANSYS software product line is well-suited for performing the myriad physics simulations microsystem components require. A previous article by Steve Groothuis highlighted the MEMS industry and presented applications of ANSYS usage in this market. Following that publication, ANSYS, Inc., announced partnerships with EDA tool providers Tanner EDA and MEMSCAP SA to help create a total MEMS simulation environment.

Electrostatic actuation resides at the core of many electromechanical microsystem components. At the micrometer level, electrostatic forces are significant and can be used to actuate components in comb drive resonators, micromirrors, actuators, or gyroscopes. Pressure sensors also work through the detection of capacitance changes induced by electrostatic forces, which are generated when fluid pressure changes cause membrane deflections. The simulation of electrostatic actuation involves static computations of electrostatic fields to obtain capacitance and force, as well as coupled electrostatic-structural simulations to arrive at a self-consistent solution. Ultimately, the dynamic performance of the component is required in order to perform system-level simulation. Therefore, accurate and fast algorithms are required to perform transient dynamic solutions of coupled electromechanical components.

ANSYS 5.6 represents a milestone in the development of simulation tools that accurately characterize and simulate static and dynamic performance of electrostatically actuated MicroElectroMechanical System (MEMS) devices. This article will review these enhancements and the benefits they offer the MEMS designer.

Electrostatic Field Simulation

Historically, there has been considerable controversy over whether boundary elements or finite elements are best-suited to solve unbounded electrostatic field problems. Accurate characterizations of both the near-field and far-field are required. Near-field accuracy is needed to properly compute electrostatic forces on structural members. Both near- and far-field characterizations are required for accurate capacitance computations. In addition, multiple dielectric materials modeling is required to simulate oxide layers, substrates, and other dielectrics. Symmetry also plays an important role in cutting down on required modeling geometry. Both periodic and aperiodic boundary condition capabilities are required as well. Through Release 5.5, ANSYS offered two- and three-dimensional second-order finite elements to model the near-field (PLANE121, SOLID122, SOLID123) as well as infinite

finite elements to model the far-field (INFIN110, INFIN111).

Adaptive P-elements

ANSYS 5.6 introduces two new elements to the electrostatics family: a brick/wedge element, SOLID128, and a tetrahedral element, SOLID127. These elements use an adaptive p-element formulation to converge the field solution to a user-prescribed criterion by automatically increasing the polynomial order of the element. The user may prescribe global convergence criteria, such as energy (which is proportional to capacitance), as well as the total electrostatic force over a particular component. In addition, the user may prescribe local convergence criteria at nodes for voltage, electric field, or electric flux density. A new dual-reconstruction algorithm has been employed to accurately assess local solution error and is valid for dissimilar material boundaries. The p-elements are compatible with node coupling and constraint equations for use in periodic and aperiodic boundary conditions.

The adaptive p-element formulation answers concerns over solution accuracy and mesh refinement by automatically compensating for mesh deficiency. In addition, the convergence criteria can be tailored to the user's needs. By setting the convergence criteria for energy, the user typically can retrieve accurate capacitance with only one or two adaptive loops. By setting the convergence criteria for force, the user can retrieve accurate forces on a body required for coupled electrostatic-structural simulations. These electrostatic simulations typically require more adaptive loops than those based on energy convergence.

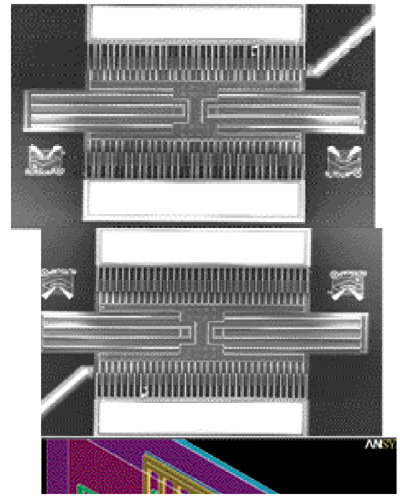
Far-Field Treatment

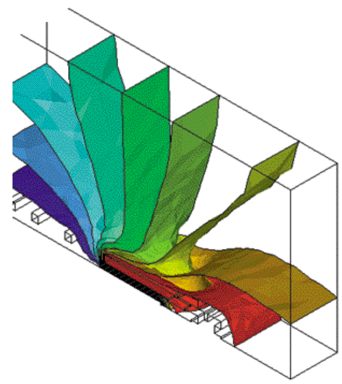
Users have found that modeling the open boundary of an electrostatic problem with infinite finite elements is computationally burdensome in three-dimensional space. In addition, the accuracy of the infinite finite elements is dependent on the shape and location of the elements as well as the limitations of the shape functions modeling the far-field decay. Boundary elements typically handle open boundaries with far greater accuracy then infinite elements. However, they are hampered by the resultant full-system matrix, which comes with substantial memory and processing requirements. Even advanced accelerator schemes for solving boundary integral equations are hampered by system memory requirements and CPU needs.

Release 5.6 debuts a new hybrid finite element/Trefftz formulation (hereafter referred to as Trefftz).

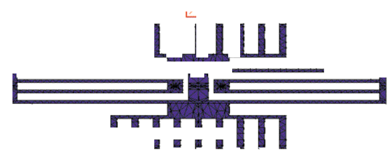
Analysis of a MEMS Filter

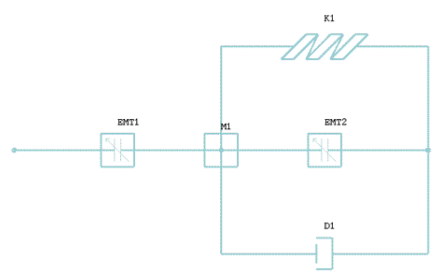
A MicroElectroMechanical System (MEMS) filter resonator is modeled to compute frequency response characteristics. A SAT model of the device is retrieved from the MEMSCAP SA MEMS Pro product. The comb drive is analyzed to compute the capacitance as a function of the stroke using a one-tooth symmetry model.



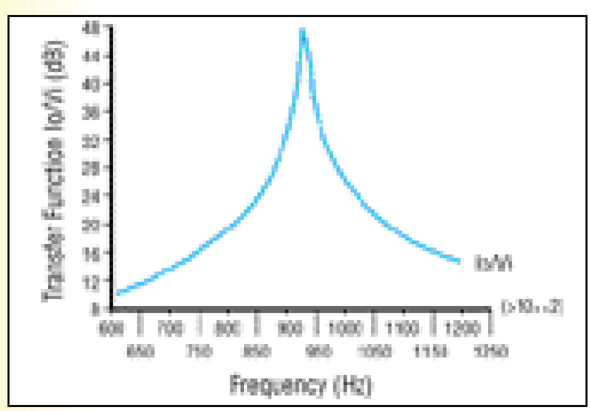


The capacitance-stroke relationship is used as input to the new TRANS126 transducer element. For coupled electromechanical simulation, the drive and pickup combs are replaced by transducer elements. The remaining mechanical structure can be reduced to a substructure to gain efficiency in solution. Alternatively, since the response is primarily governed by a single mode, a complete reduced-order model can be constructed using the new electromechanical circuit builder.

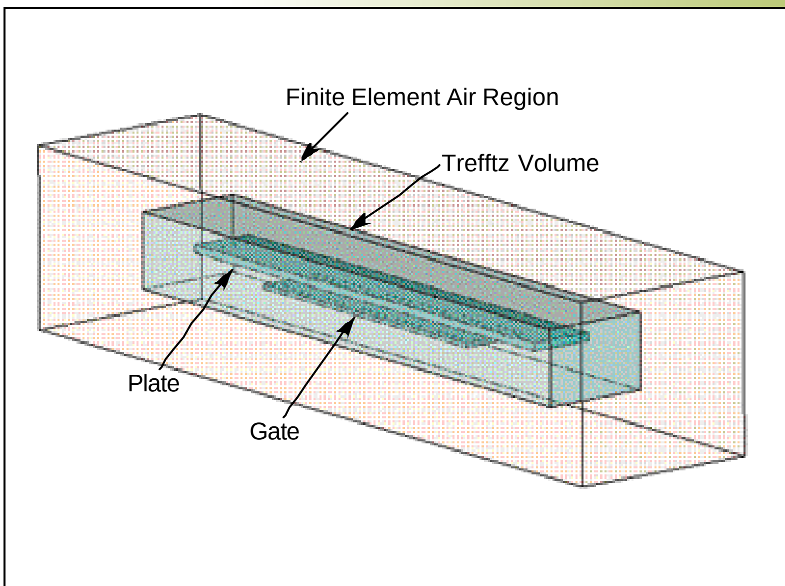




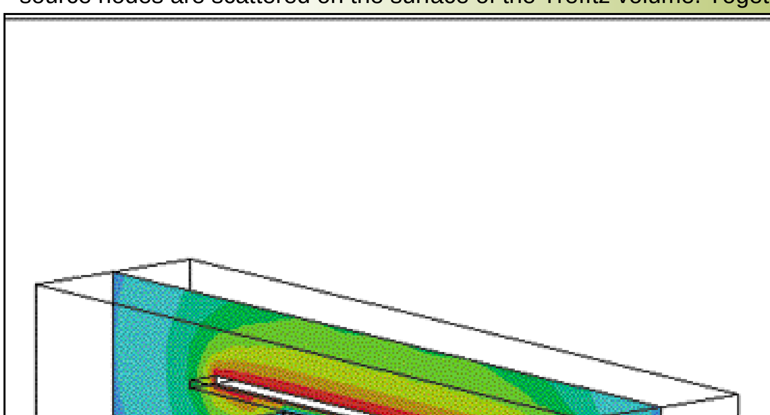
The transfer function relating the input voltage to the output current is required for the electronic system design. This task is accomplished by executing a coupled electromechanical harmonic sweep over the desired frequency range and extracting voltage and current from the TRANS126 elements.



Coupled Simulation

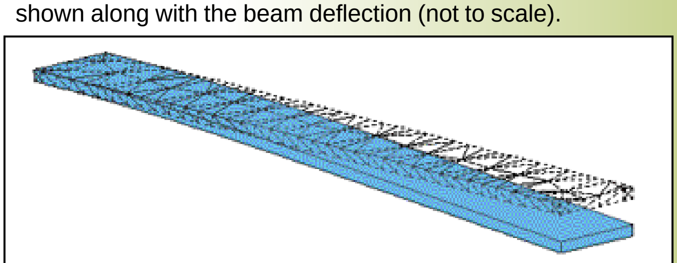


A cantilever beam is positioned above a gate. Applying a voltage to the beam will deflect the beam towards the grounded gate. The finite element domain is truncated a short distance from the modeled region. Trefftz source nodes are scattered on the surface of the Trefftz volume. Together



with the surface of the exterior air elements, a Trefftz substructure is automatically created to accurately model the open boundary.

The new ESSOLV command is used to solve the coupled electrostatic-structural problem to obtain the beam deflection for the applied voltage. A display of the voltage distribution through the midspan of the beam is shown along with the beam deflection (not to scale).



This technique combines the efficiency and sparse nature of finite elements for modeling the near-field domain with the Trefftz boundary integral formulation for modeling the open domain. The Trefftz functions are evaluated by fictitious sources (degrees of freedom) distributed between the conductors and the exterior of the finite element domain. These source nodes are combined with the surface facets of the exterior finite element model to create a Trefftz substructure and a set of constraint equations.

The whole process of creating Trefftz source nodes has been vastly simplified; substructure and constraint equation generation is entirely automated. Typically, only a few Trefftz sources are required (20-100), which produce only a very small, dense matrix in the total system matrix. Thus, solver efficiency is preserved, and solution accuracy is greatly enhanced. In particular, the open domain energy is accurately computed for capacitance computations. In addition, multiple finite element regions can be modeled independently and "tied" together with a single Trefftz domain. Hence the interaction of multiple conductor regions separated by large distances can be accurately simulated without requiring the user to mesh the large air region between conductors. Trefftz capability is available for both the traditional h-based electrostatic elements and the new adaptive p-elements.

Capacitance

The computation and extraction of capacitance for multiconductor systems is essential for characterizing electrostatic field interaction. Typically, a designer will want to compute lumped capacitance values for use in a circuit-based simulator. Both self-capacitance and mutual capacitance between conductors are required to construct a lumped-equivalent representation of the system. ANSYS 5.6 offers a new solver command macro (CMATRIX), which can automatically solve a series of simulations and extract lumped capacitance values. The user only is required to identify the conductors by grouping the surface nodes of the conductors into node components. The command macro will then use the components to systematically compute a series of simulations and extract capacitance.

This new capacitance tool is essential for characterizing electrostatic actuation. The spatial derivative of capacitance along the stroke (or motion direction) of an electrostatic actuator is directly proportional to the force acting on the actuator. By running a series of electrostatic simulations for different strokes, or deflections, the user can obtain a complete characterization of the force on the actuator drive. These data can be used as input for a simplified "actuator" element to run static, harmonic, and transient dynamic simulations.

Coupled Electrostatic-Structural Simulation

For MEMS devices such as comb drives, RF switches, pressure sensors, microfluidic sensors, etc., equilibrium between electrostatic forces and the companion structural reaction forces must be resolved to obtain a consistent solution. Until now, utilization of finite element methods to accomplish this simulation has been hampered by the requirement to move or adjust the field (electrostatic) mesh to conform to structural displacements. Prior to ANSYS 5.6, users worked around this issue by defining the electrostatic region with structural properties to allow a consistent structural solution with deformation of both the electrostatic mesh and the structural mesh.

Mesh-Morphing

ANSYS 5.6 debuts a new mesh-morphing capability that will automatically adjust the electrostatic field mesh surrounding a mechanical structure in accordance with the structural displacements. Morphing is activated by selecting solid model areas or volumes that will morph or by directly selecting elements to undergo morphing. The algorithm will attempt to move the existing field mesh to align with the structural deformation. If the deformation is too great, the solid mesher will automatically remesh the region to provide a consistent field mesh aligned with the deformed structure. Thus the morphing tool will handle both small and large structural movement and is suitable for a wide range of coupled electrostatic-structural simulations. The morphing tool can be applied to other coupled simulation solutions, such as fluid-structural and magnetic-structural.

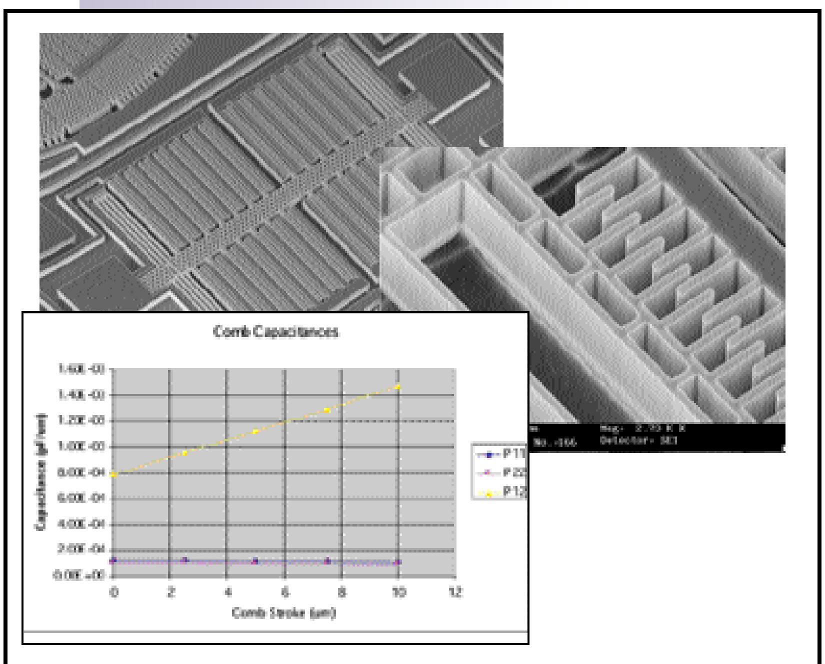
Coupled-Solver Tool

To automate coupled electrostatic-structural solutions, a new solver command macro, ESSOLV, has been developed. The tool utilizes the physics file capability in ANSYS. Users can set up a physics environment for each of the electrostatic and structural domains, and write these environments to physics files (PHYSICS command). The macro retrieves these physics files and performs a coupled-field solution. Both structural and electrostatic convergence criteria can be defined for the coupled-field solution. The solver tool works with all mechanical and electrostatic field elements as well as with a Trefftz domain. Typical applications include mapping of capacitance vs. displacement, pull-in simulation (determining the voltage level at which the structural device closes), and hysteresis effects.

The performance and characterization of the dy-

Extracting Capacitance

A MEMS accelerometer consists of a series of capacitor plates (combs) attached to a rigid backbone structure and restoring springs. Stationary combs adjacent to the moving combs create a differential variable capacitor used to detect motion. An accurate electrostatic finite element simulation using the new CMATRIX command extracts self- and mutual-capacitance between the comb structures and ground for the anticipated stroke of the device. (Courtesy of Kionix Inc., Ithaca, NY.)



namic behavior of components for use in system simulation by EDA tools are critical to the MEMS industry. Full, dynamic, finite element solutions of coupled electrostatic-structural problems are prohibitively expensive and impractical. It is therefore necessary to simplify large, coupled-field problems and provide accurate harmonic and time-domain solutions in a fast and efficient manner. The simplification process is often referred to as reduced-order modeling (ROM). ANSYS has enjoyed the benefit of ROM for some time with the substructuring capability for linear systems. Substructuring allows large, linear, finite element models to be reduced to a subset of degrees of freedom by creating reduced substructure matrices. These matrices are introduced into an analysis as super-elements utilizing the MATRIX50 element.

Electrostatic-Structural Transducer

As mentioned earlier, the coupling between electrostatic fields and mechanical forces can be characterized by mapping the capacitance of the system as a function of the motion of the device. A simplification of the system coupling can be achieved by creating a "transducer" element, which captures the capacitance-stroke relationship. A new transducer finite element (TRANS126) has been developed that fully couples electrostatic and structural physics. It will convert electrostatic energy to mechanical energy and vice-versa, as well as store electrostatic energy. In this way, the fields are fully coupled. The transducer element takes on the form of a line element with voltage and structural degrees of freedom. Input for the element consists of a capacitance-stroke relationship that can be derived from electrostatic field solutions. If necessary, multiple transducer elements can be pieced together for three-dimensional characterization. Thus, the electrostatic field mesh is replaced by a set of transducer elements that act on the mechanical device. As a consequence of this tactic, a ROM of a coupled electrostatic-structural device simulation can be performed. Capabilities include static coupling; eigenvalue analysis of the system including "prestress" effects from DC bias voltages; small-signal AC harmonic analysis about a DC bias prestress voltage; and large-signal, nonlinear-transient analysis. Combined with substructuring, ANSYS offers a viable and accurate method for producing rapid ROM simulations.

Coupled Electromechanical Circuit Simulator

The lowest common denominator for ROM is complete characterization of devices in terms of lumped parameters. On the mechanical side, this would include spring, mass, and dampers; on the electrical side, it might include basic linear circuit elements such as resistors, capacitors, inductors, voltage sources, current sources, as well as more exotic elements. Coupling the two physics domains is accomplished by a coupled-physics transducer circuit element. In many cases, this type of simplification may be all that is necessary to characterize a system response; but in other, more complicated cases, it may require more rigorous characterization of the mechanical response using a finite element model attached to a circuit of lumped elements.

ANSYS 5.6 has bolstered the electrical circuit simulator introduced at Release 5.4 with mechanical elements and the transducer element to provide a coupled electromechanical circuit simulator. The circuit simulator provides a convenient method to create a lumped model of a coupled system. The simulator supports the elements COMB14, COMB39, MASS21, CIRCU124, and TRANS126. What is unique about the ANSYS circuit simulator is that it is finite element-based and uses the same physics degrees of freedom as ANSYS solid finite elements. This is in contrast to SPICE-type circuit simulators, which attempt to convert all physics into equivalent electrical characterization. In addition, the ANSYS circuit elements can be hooked directly into full two- or three-dimensional finite element models to capture more accurately the physics response of a system that cannot be easily reduced to lumped elements. The user is left with a wide range of options for ROM ranging from a pure lumped-element approach to combining the circuit elements with substructures, and finally coupling the circuit elements with full finite element models.

Further Reading

"Analyzing Microminiature Devices" by S. Groothuis, *ANSYS Solutions* magazine, Spring 1999.

"Hybrid Finite Element – Trefftz Method for Open-Boundary Analysis" by M. Gyimesi, I. Tsukerman, D. Lavers, T. Pawlak, D. Ostergaard, *IEEE Trans*. magazine, May 1996.

"Electro-Mechanical Transducer for MEMS Analysis in ANSYS" by M. Gyimesi, D. Ostergaard, Proceedings of the 1999 International Conference on Modeling and Simulation of Microsystems, April 19-21, 1999, San Juan, Puerto Rico.