

Electro-Mechanical Analysis of MEMS Devices with CoventorWare and Abaqus

Summary

The computational analysis of MEMS (Micro Electro Mechanical systems) devices poses distinctive challenges, requiring software that provides flexible modeling tools, enables the coupling of multiple physical phenomena, and considers the integration of the devices into their macro-scale surroundings. To meet these requirements, Abaqus partners with developers of commercially available MEMS software by providing the necessary finite element analysis capabilities to these packages.

Coventor is a market leader in MEMS design and analysis software. CoventorWare is an integrated suite that employs Abaqus for structural, thermal, and piezoelectric analyses, as well as the structural component of coupled electromechanical and package analyses. In this technology brief two example electro-mechanical analyses of MEMS devices with CoventorWare and Abaqus are presented.

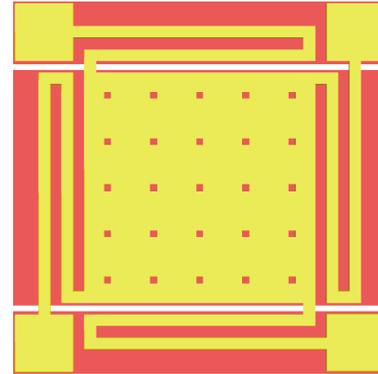
Background

MEMS are miniaturized devices that consist of both mechanical and electronic parts. One or more optical, mechanical, electronic, thermal, or magnetic functions can be incorporated on a single chip. These systems can sense, control, and actuate on a micro scale and can function individually or be arrayed on a macro scale.

Integrated circuit process sequences are used to fabricate the electronics, and the micromechanical components are fabricated using compatible "micromachining" processes that selectively etch away parts of the silicon wafer or add new structural layers to form the mechanical and electro-mechanical devices.

The multiphysics nature of MEMS simulation makes the design process very challenging and has led to the development of design tools that specifically address the needs of this community.

CoventorWare provides an electrostatic analysis capability using the boundary element method and a mechanical analysis capability using the finite element method. Dassault Systèmes SIMULIA Corp. partners with Coventor and provides the finite element component of CoventorWare, enabling its structural, thermal, and piezoelectric analysis capabilities. In addition, CoventorWare includes an efficient, coupled electro-mechanical analysis capability in which the finite element and boundary element solutions are linked in an iterative fashion. CoventorWare is



Key Abaqus Features and Benefits

- Ability to perform structural and modal analysis of micro-scale devices.
- Geometrically nonlinear analysis capability, including contact.
- Incorporation of thin film residual stresses via initial stress capability.

an integrated MEMS analysis package; while CoventorWare can run in conjunction with an existing Abaqus installation, a separate Abaqus installation is not required.

In this technology brief the electromechanical performance of two MEMS devices is analyzed with CoventorWare. The first device is a suspended plate variable capacitor (Figure 1), which is used in accelerometers, gyroscopes, and tunable capacitors for radio frequency (RF) applications.

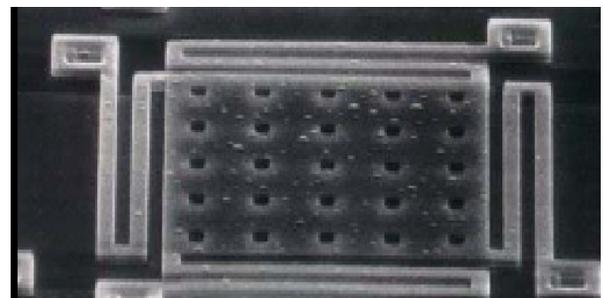


Figure 1: Variable capacitor (courtesy of QinetiQ)

The second device is an electrostatic switch (Figure 2), which is used in micro-relays for wired communication systems and passive RF switches for wireless communications.

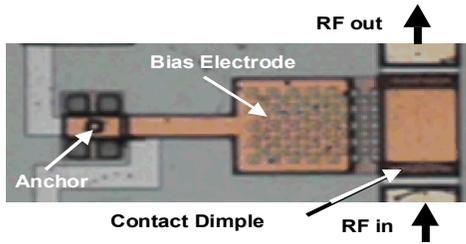


Figure 2: RF switch (courtesy of HRL)

Analysis Approach

CoventorWare includes a unique, MEMS-specific analysis flow to address system and device design.

A system-level parametric design study is first completed. Analytical MEMS device models are created using parametric model libraries. These libraries provide mathematical functions that define the device geometry and material properties necessary to model the global behavior of the MEMS components.

The device behavior is evaluated simultaneously with that of the electronic control circuitry, the environment, and other surrounding systems.

After the preliminary system-level design is complete, specific device design issues are addressed with detailed multiphysics analyses performed with the finite element/boundary element solvers.

CoventorWare, with its unified user interface, allows the creation of the complex, layered geometries specific to MEMS, automatic generation of the desired meshes, and execution of the required multiphysics analysis. The present study focuses on preliminary device designs that have already been established with parametric behavior models.

Variable Plate Capacitor Analysis and Results

The device consists of a capacitor plate suspended over another capacitor plate that is fixed to a substrate. Four folded tether arms maintain the position of the first plate (Figure 1). When a DC bias voltage is applied across the plates, the suspended plate is drawn toward the fixed plate. The tether arms, anchored with bond pads, provide the necessary restoring forces that return the suspended plate to its original position when the voltage is removed. The tether arms also provide the upper plate with the necessary electrical connections for maintaining a bias voltage. An air gap of 1 micron separates the capacitor plates.

The suspended plate is a three-layer stack with a layer of low stress silicon nitride sandwiched between metal layers. The stack has a thickness of 2.4 microns and in-plane dimensions of 100 x 100 microns. Etch holes, 3 x 3 microns in dimension and on a 20-micron pitch, facilitate the process of removing the sacrificial layers that are deposited during the manufacturing process. In the present analysis it is assumed that the etching process has been completed and that all surplus material has been removed, thus forming the air gap between the capacitor plates.

A modal analysis of the suspended capacitor plate is performed. The mode of interest is the out-of-plane “bounce” mode. Depending on the application, this mode will decide the optimal operating frequency for the device. For example, if the device is used as an accelerometer, the operating frequency should be well within the out-of-plane mode to prevent the capacitor plates from contacting each other.

The electro-mechanical response of the device is then examined. When an excessive bias voltage is applied across the capacitor plates, the suspended plate is susceptible to being pulled into contact with the fixed plate. This occurrence will adversely affect the functionality of the device. Therefore, determining the “pull-in” voltage is essential in deciding the operating voltage range of the capacitor. Finally, the simulation results are compared to experimental data obtained from devices fabricated at QinetiQ.

A two-dimensional layout of the device is shown in Figure 3, and the three-dimensional model is shown in Figure 4. The two-dimensional layout, the three-dimensional model, and the mesh are created using CoventorWare. The specifications of the manufacturing process are used to generate the three-dimensional model from the two-dimensional layout. Specifically, information on the materials in use and the sequence of manufacturing operations allow the dimensions of the device to be determined completely. The model shown in Figure 4 is based on the INTEGRAM metal-nitride sacrificial surface micromachining process developed by QinetiQ. Models with and without etch holes are created and meshed.

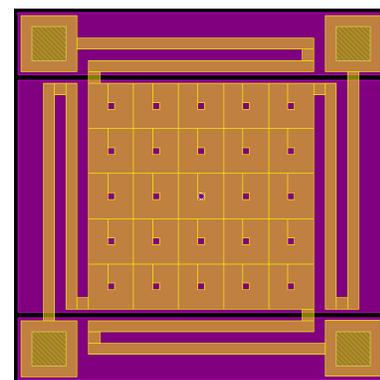


Figure 3: Two-dimensional layout of the variable capacitor

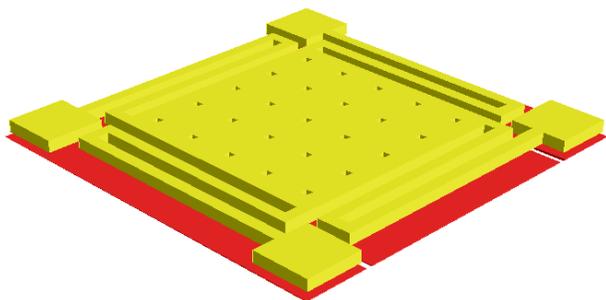


Figure 4: Variable capacitor model with etch holes

Determining the out-of-plane vibration mode

The modal analysis is performed by Abaqus within CoventorWare. It is found that the mesh without etch holes could be used for the modal analysis by changing the effective material density to account for the mass of the material occupying the etch holes. The results show good agreement (<5 percent difference) in frequencies with simulations that included etch holes while significantly saving analysis time. The first frequency of the nominal design (that which does not consider the effects that process variations can have on device layer thickness, air gap, etc.) is predicted to be 80 kHz and is significantly lower than the higher modes. The predicted mode shape and displacement magnitudes are shown in Figure 5. Damping effects have not been considered in this case study.

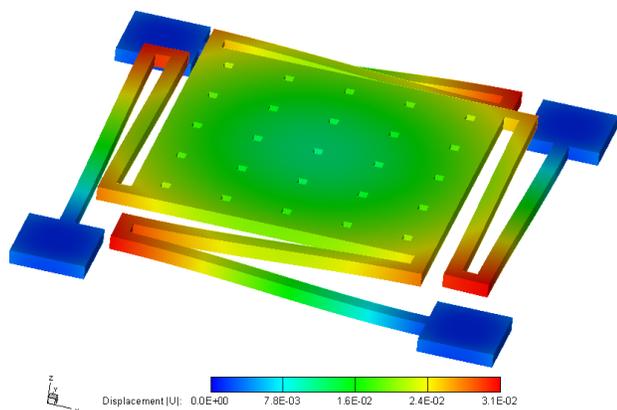


Figure 5: Displacement magnitude in the out-of-plane "bounce" mode for the variable capacitor

Coupled electrostatic-structural analysis

The electromechanical response is evaluated in a coupled manner that combines the electrostatic and mechanical solutions of the device. The solver achieves its accuracy by iterating on a sequence of electrostatic and mechanical solutions. Each new solution is compared against the previous iteration until convergence is reached.

A linear voltage ramp trajectory is applied to the suspended capacitor plate, and the electromechanical response is calculated. At "pull-in" the electrostatic forces overcome the mechanical restoring forces and the plate snaps, or pulls in, to the ground plate. The structural response of the suspended plate becomes highly nonlinear once contact is established. In this situation numerous solver iterations may be required to achieve convergence. Because the objective of the analysis is to determine when pull-in occurs, the response with contact is not considered. Figure 6 shows a contour plot of displacement magnitude on the suspended plate just before pull-in occurs. The predicted pull-in voltage for the nominal design is between 7.25 and 7.5 volts. A range of 0.25 volts is specified because the last stable point of the solution occurs at 7.25 volts and a resolution of 0.25 volts is used in the solution process. A further refinement of this range can be achieved by using a smaller voltage step.

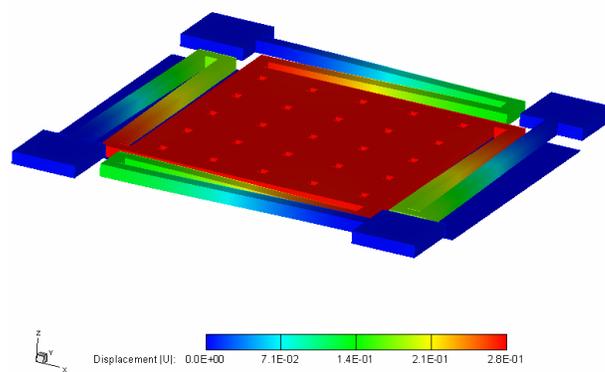


Figure 6: Contour plot of displacement magnitude when pull-in occurs

Figure 7 compares the results of the current simulation with experimental data obtained from QinetiQ. Twenty variable capacitors are tested. The measured pull-in voltages on these devices range from 6.8 to 7.8 volts, which is in good agreement with the predicted results.

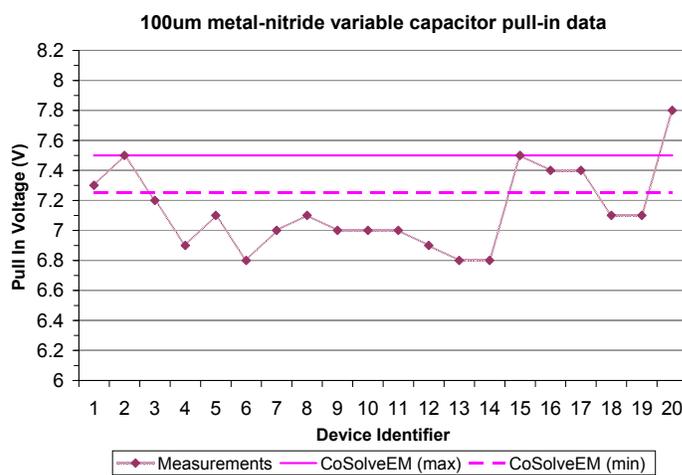


Figure 7: Measured pull-in voltage and simulation results

RF Switch Analysis and Results

The operating principle of a MEMS switch is shown in Figure 8. In the “OFF” state there is no electrical connection between the RF lines. In the “ON” state the switch is in electrical contact with the transmission lines via metallic dimples located near the free end of the switch.

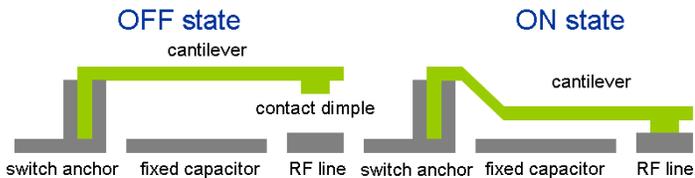


Figure 8: Schematic diagram of an RF switch

There are several ways to actuate the switch and achieve the ON state, including electrostatic, electromagnetic, and electrothermal mechanisms. The present analysis demonstrates a switch that has been actuated by electrostatic forces. The switch consists of capacitor plates separated by an air gap. One of the plates is fixed to the substrate, while the other is suspended above by a mechanical lever arm. The suspended plate is a multilayer structure. It consists of a structural nitride (silicon nitride) beam that is anchored at one end in cantilever fashion. The nitride layer is sandwiched between two metal layers. The bulk of the lower metal layer serves as the capacitor plate. The contact dimples are located in a small isolated section of the lower metal layer close to the free edge of the lever arm. The top metal layer serves primarily to balance the residual manufacturing stresses in the lower metal layer while also providing electrical connections to the upper electrode. The RF signal lines are a part of the switch substrate; the RF performance will not be considered in this study.

The operating principle is similar to that of the suspended capacitor plate. When an electric potential is applied across the capacitor plates, the switch deflects down in response to the electrostatic forces. At small applied potentials these forces are balanced by the restoring mechanical forces in the cantilever, and the switch remains in the OFF state. The electrostatic force varies inversely with the air gap separation. With increasing potential, the electrostatic forces increase due to the decreasing air gap between the capacitor plates, eventually causing the cantilever to establish contact at the “snap through” voltage.

Characterizing the electromechanical performance of the RF switch is a critical design task. The device parameters usually explored to meet customer specifications are the switch geometry, the multilayer stack thickness, the air gap thickness, and the actuation area. In addition, since these devices are very sensitive to thickness variations and the effect of residual thin film stresses, it is important to include inherent process variations in the device design. CoventorWare includes several convenient tools for studying these variations.

In the present case the performance of a nominal switch design is evaluated. The in-plane dimension of the switch is 100 x 175 microns. The multilayer stack thickness is 3 microns, and the air-gap separation is 2 microns. The contact dimples are 0.5 microns thick and 1.5 microns from the contact pads on the substrate. Figure 9 shows the three-dimensional model of the device created in CoventorWare using appropriate mask layouts, fabrication process information, and material properties.

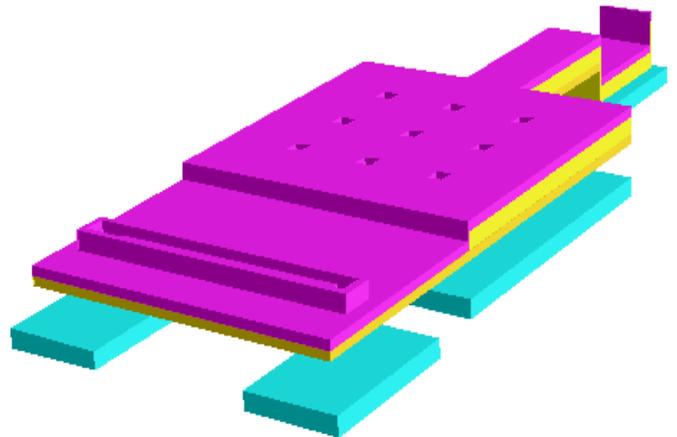


Figure 9: RF switch model
(Z-direction exaggerated for clarity)

Coupled electrostatic-structural analysis

Since the coupled electrostatic-mechanical analysis can be computationally expensive, the model is simplified. Much of the switch topology is removed, and the anchor region is replaced with appropriate boundary conditions. The applied load is a voltage cycle, ramping linearly to 80 volts and returning linearly to 0 volts.

The predicted switch ON voltage is 25 volts. Figure 10 shows displacement magnitude results at this voltage for the simplified model.

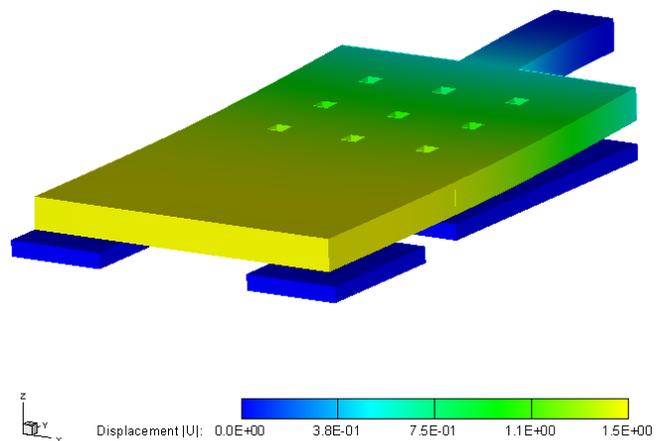


Figure 10: Contour plot of displacement magnitude at 25 volts

Figure 11 shows a plot of the maximum Z-displacement in the model with applied voltage during the cycle. When the ON state is established, the displacement is approximately -1.6 microns. The switch stiffens significantly upon initial contact, resulting in very little displacement thereafter. A complete pull-in is observed at 60 volts. In this state contact has been established by the capacitor plates as well as the contact dimples. The voltage must be reduced below the ON value to alleviate this condition.

From Figure 12 the contact forces predicted at each contact dimple at the beginning of the ON state are about $48 \mu\text{N}$ and increase with operating voltage. Contact resistance is minimized with increased contact force; the information presented in Figure 11 and Figure 12 helps to determine the operating voltage range of the switch.

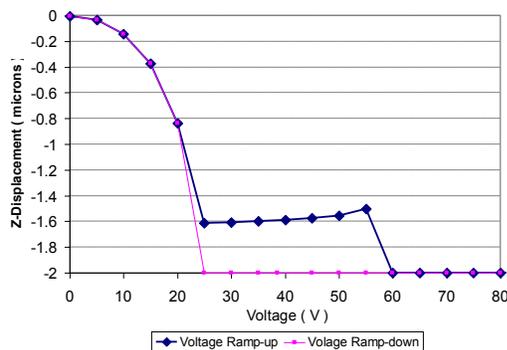


Figure 11: Maximum Z-displacement of upper electrode with applied voltage

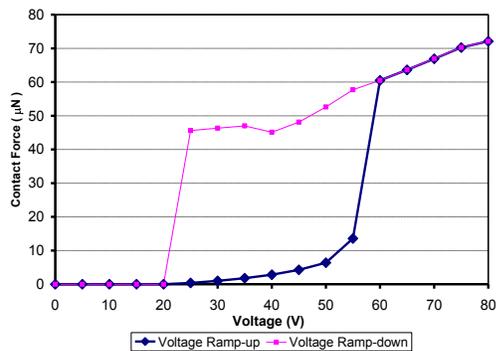


Figure 12: Contact force on each contact dimple with applied voltage

Coupled electrostatic-structural analysis: hot switching

In addition to evaluating the actuation voltage of the switch, it is also important to determine the highest RF line voltage that can be applied before hot switching occurs. Hot switching will occur when the RF line voltage is large enough to generate an electrostatic force sufficient to actuate the switch. This force is not generated between the capacitor plates; rather, it is generated between the RF line and the contact bar that supports the contact dimples.

In a coupled hot switching analysis, an electric potential is applied between the “RF in” and the “RF out” lines. In the absence of a bias voltage between the capacitor plates, the predicted RF line voltage is 40 volts.

Hot switching is undesirable since it tends to wear out the contact and reduces the electrical life of the switch. Typically, a high “hot switching” voltage is desirable. However, this involves a trade-off with the actuation voltage and the contact resistance and requires an optimization study within the design parameter space.

Effect of residual stresses

MEMS devices are typically layered structures. In the fabrication process individual layers are often subjected to elevated temperatures. A mismatch of the thermal expansion coefficients of the materials used for each layer can result in significant residual stresses in the final structure.

In the present nominal switch design it is assumed that the residual stresses in the switch metal layers are identical. Since the metal layers have the same thickness and material properties, the switch initial configuration is flat. However, actual switches will exhibit some curvature in the initial configuration due to stress gradients and process variations.

Figure 13 shows the displacement magnitude results of a separate, structural-only analysis of the switch. In the analysis residual in-plane stresses are assumed to exist in each layer. The layer-wise distribution is such that there is a positive gradient through the thickness of the multilayer switch.

The stress gradient is introduced as an initial stress distribution. The stresses equilibrate during the analysis, causing the switch to deflect out-of-plane. Figure 13 shows that the free corners of the switch have the maximum displacement and curve away from the lower contact pads. Although a coupled electro-mechanical analysis is still required, it is clear that, as a result of the increased air gap between the capacitor plates, the switch ON voltage will increase. If the stress gradients result in excessive out-of-plane curvature, the switch may not actuate at all. Similarly, if the residual stresses induce curvature that effectively decreases the air gap, the operating voltage of the switch is lowered. In this case the switch contact resistance increases and the device may not meet the desired specifications.

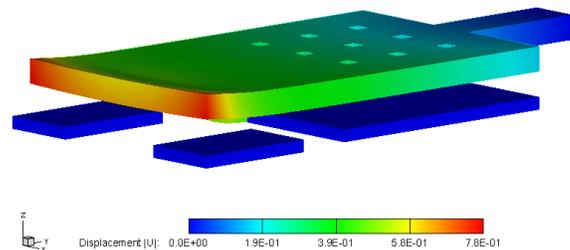


Figure 13: Effect of residual stress gradients on displacement

Acknowledgements

Dassault Systèmes SIMULIA Corp. would like to acknowledge Coventor, Inc., for providing access to CoventorWare and test case material. Additional acknowledgements are given to QinetiQ for providing experimental data.

References

1. Senturia, S., *Microsystems Design*, Kluwer Academic Publishers, Boston, 2001.
2. *INTEGRAM Design Handbook on Metal-nitride Surface Micromachining*, Version 1.1b, QinetiQ and Coventor, April 2004.

Abaqus References

For additional information on the Abaqus capabilities referred to in this brief, see the following Abaqus 6.11 documentation references:

- Analysis User's Manual
 - “Fully coupled thermal-stress analysis,” Section 6.5.4
 - “Natural frequency extraction,” Section 6.3.5
 - “Initial conditions,” Section 32.2
 - “Defining contact pairs in Abaqus/Standard,” Section 34.3.1

CoventorWare References

For additional information on the CoventorWare capabilities referred to in this brief, contact sales@coventor.com or visit <http://www.coventor.com>.

About SIMULIA

SIMULIA is the Dassault Systèmes brand that delivers a scalable portfolio of Realistic Simulation solutions including the Abaqus product suite for Unified Finite Element Analysis, multiphysics solutions for insight into challenging engineering problems, and lifecycle management solutions for managing simulation data, processes, and intellectual property. By building on established technology, respected quality, and superior customer service, SIMULIA makes realistic simulation an integral business practice that improves product performance, reduces physical prototypes, and drives innovation. Headquartered in Providence, RI, USA, with R&D centers in Providence and in Suresnes, France, SIMULIA provides sales, services, and support through a global network of over 30 regional offices and distributors. For more information, visit www.simulia.com

The 3DS logo, SIMULIA, Abaqus, and the Abaqus logo are trademarks or registered trademarks of Dassault Systèmes or its subsidiaries, which include ABAQUS, Inc. Other company, product, and service names may be trademarks or service marks of others.

Copyright © 2007 Dassault Systèmes

