Finite Element Based Electrostatic-Structural Coupled Analysis with Automated Mesh Morphing

V. I. Zhulin[1], S.J. Owen[2], D.F. Ostergaard[1]
[1]ANSYS, Inc., 275 Technology Drive, Southpointe, Canonsburg PA. 15317
[2]Sandia National Labs, Parallel Computing Sciences, Albuquerque, NM 87185-0847

ABSTRACT

A co-simulation tool based on finite element principles has been developed to solve coupled electrostatic-structural problems. An automated mesh morphing algorithm has been employed to update the field mesh after structural deformation. The co-simulation tool has been successfully applied to the hysteric behavior of a MEMS switch.

1 INTRODUCTION

In a coupled electrostatic-structural problem, electric forces act on mechanical elements resulting in structural deformations, which in turn, change the electric field and forces. Traditionally, the finite elements (FE) technique is used to solve the structural problem and boundary elements (BE) for the electric domain [1]. BE is preferred because the FE solution requires meshing the electrostatic domain. However, FE simulation of the electrostatic domain has definite advantages over BE when the domain is large, complicated, and contains dielectrics. This is the case for many practical micro electromechanical structures where computer memory requirements and solver speed make BE impractical. To exploit the advantages of FE simulation a mesh morphing technique and its control algorithm, commonly known as a co-simulation tool, has been developed to automate the updating of the electrostatic mesh. The morphing algorithm has been implemented in the ANSYS/Multiphysics program for solving 2D and 3D electrostatic-structural simulations. The paper will focus on the mesh morphing algorithm which now enables the use of finite element methods in the electrostatic domain.

2 CO-SIMULATION TOOL

The co-simulation tool solves the electrostatic field domain and the structural domain separately. A simple relaxation algorithm [1] is used to ensure a self-consistent electromechanical analysis. Finite element methods are used for both the electrostatic and structural domains. Coupling is achieved by forces computed using the Maxwell Stress Tensor technique. Forces are passed from the electrostatic mesh to the structural nodes. The algorithm requires node-to-node compatibility between the electrostatic mesh and the structural mesh. The electrostatic domain can consist of second order h-elements or p-adaptive elements. The structural domain may consist of first or second order elements. Monitoring the change in stored energy in the electrostatic domain as well as the change in the maximum structural displacement regulates convergence. All linear and non-linear structural finite element methods available in the ANSYS program may be employed in a co-simulation problem.

3 MESH MORPHING ALGORITHM

After each iteration of the co-simulation tool, it is necessary to adjust the electrostatic field finite element mesh. This was accomplished by developing a *mesh morphing* tool that automatically updates the electrostatic mesh based on the current structural displacements. This is done by either (1) keeping the original element connectivity and adjusting the node locations within the electrostatic domain, or (2) forming a completely new mesh within the electrostatic region while maintaining the boundary between structural and electrostatic domains.

In the first approach, a combination of smoothing techniques are used to adjust node locations. Laplacian smoothing [2] is a technique commonly used for improving finite element meshes by iteratively adjusting node locations to the centroid of their surrounding nodes. The surrounding nodes are typically defined by those connected by a finite element edge. For this application, the Laplacian smoothing proved to be not as effective, tending to bunch elements around the structural deformations. A modified area/volume based smoothing technique was employed which adjusts node locations based on the centroid of the adjacent elements. For example, For node P at location $\mathbf{P_x}$, its smoothed location is defined as:

$$\mathbf{P_x} = \frac{\sum_{i=1}^{n} w_i \mathbf{C}_i}{\sum_{i=1}^{n} w_i} \qquad (1)$$

where $\mathbf{C}_i$ is the centroid of element i adjacent P, and w_i is the area or volume of element i. The total number of elements adjacent to P is given by n. For nodes on the boundary, the above definition may be used, but a subsequent projection to the associated curve or surface is necessary. After first applying the structural displacements to the nodes on the boundary of the electrostatic domain, two or three iterations of equation (1) can be applied to all nodes of the electrostatic mesh within proximity of the deformation. Although in most cases equation (1) will improve the surrounding elements, there is no guarantee that the resulting elements will be optimal or, on occasions, even better than a previous iteration. To ensure that elements are improved, equation (1) is only applied if the minimum quality, Q_{min}, of any element adjacent P, improves. Q_i can be any valid shape measure for finite elements, such as Jacobian ratio. For efficiency, the radius ratio, for tetrahedral elements, given by 3r/R, where r is the inscribed sphere radius and R is the circumscribed sphere radius, is used. Reference [3] presents a closed form expression of the metric in terms of the tetrahedron's node locations. Other distortion metrics [4,5] may be employed based on the shape of the elements or application for which the morphing technique will be applied.

With the above constraint placed on the smoothing algorithm, the overall quality of the elements is guaranteed not to degenerate from one smoothing iteration to the next. It does not, however, guarantee that it will improve. For cases where structural displacements result in regions of very poor element quality, an optimization technique is employed. A steepest descent, iterative approach is used to move P in a gradient direction to increase Q_{min} as much as possible. Details of combined Laplacian/Optimization based techniques can be found in [4,6]. Since optimization-based smoothing is more time consuming than equation (1), it is only invoked where element quality, Q_{min}, is sufficiently small.

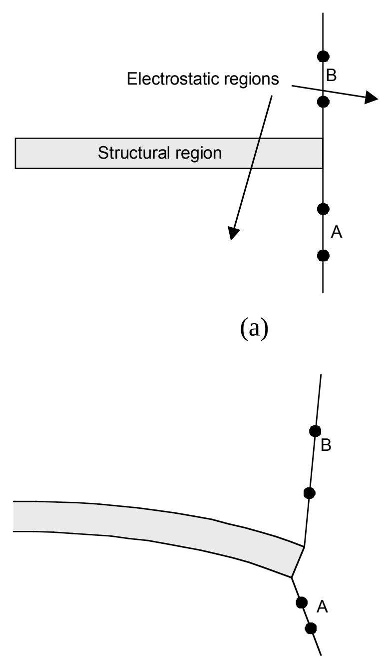
Depending on the magnitude of the structural displacements, it can also be beneficial to incrementally apply the structural displacements to the electrostatic nodes on the boundary. With larger displacements, the electrostatic elements can become inverted, resulting in failure of the smoothing algorithm. To accommodate this, the displacement factor, δ is computed as $e_{min}/2d_{max}$ where e_{min} is the smallest element edge length in the region of displacement and d_{max} is the magnitude of the largest displacement. Before each set of smoothing iterations, the location of the structural nodes, $\mathbf{S}_j$, at increment j, can be given as

$$\mathbf{S}_j = \mathbf{S}_i + \delta\left(\mathbf{S}_f - \mathbf{S}_i\right) \quad (2)$$

where $\mathbf{S}_i$ is the structural node location before applying displacements, and $\mathbf{S}_f$ is the final displacement location. The number of increments, N, is then given as $d_{max}\delta$. With the location of the structural nodes set, the smoothing can then be applied, generally with a higher success rate than had the final displacement, $\mathbf{S}_f$, been used directly.

The second approach used in the mesh morphing tool is to completely remesh the electrostatic region. This alternative is used when the computed number of increments, N, becomes very large. In these cases, the structural displacements are great enough that the smoothing technique becomes ineffective or very inefficient. Remeshing is also employed if during the incremental smoothing operation, the distortion metric, Q, of the elements fall below a threshold value. To effect the remeshing operation, the internal elements are first deleted leaving only the boundary. In two-dimensions this is a polygonal boundary defined by the outermost finite element edges. In three-dimensions, a set of surface facets remains. The boundary is then sent to an appropriate mesh generator [7,8,9] for new elements to be created.

For cases where the structural displacements require that the boundary node locations be modified, additional methods are employed to update these before generating the internal elements. For example, in Figure 1(a), the initial non-displaced structural region is touching lines A and B of the electrostatic region. When the structure is displaced, as in Figure1(b), the node locations on lines A and B must also be updated. This can be done through additional smoothing or by explicitly setting the new node locations based on parametric interpolation.



(b)

Figure 1: Structural displacements that effect non-structural geometric features
(a) original configuration (b) deformed configuration

The mesh morphing tool has been implemented for both 2D and 3D domains for triangle, quadrangle, tetrahedron, hexahedron and prism shaped elements. It has proven effective on a wide range of geometry and computed displacements within a commercial software environment.

4 CO-SIMULATION EXAMPLE

An electro-mechanical hysteric response of a MEMS switch is used to illustrate the co-simulation tool with mesh morphing. Figure 2 illustrates a _ symmetry FE model of a beam above a ground plane (bottom of model). The air region is meshed using triangular elements and the beam using quad elements. The largest downward deflection occurs at the symmetry plane.

A contact element placed at the beam mid-span is used to maintain a minimum gap. Figure 3 illustrates a close-up view of the mesh at the beam mid-span before excitation. Figure 4 shows the morphed mesh at the pull-in voltage. Figure 5 shows similar results but using the re-mesh algorithm. The displacement vs. voltage of the beam tip for the hysteresis simulation is shown in Figure 6. The results compare favorably to those performed in a previous study [10].

The co-simulations have been applied to 2-D and 3-D models producing similar results. Table 1 summarizes the CPU time for one co-simulation loop for the different models. The number of loops to converge (defined by a 1% variation in displacement between loops) at any given voltage level varies between 2 and 40 (pull-in). When run as a series of incremental voltages to map out the hysteric behavior, the number of loops at each voltage level decreases. Solution times are for an HP Kayak XW PC with a 300 MHz. processor.

Dimension	**Shape**	**Number of Elements**	**CPU/Loop**
2D	Quad	321	4.6 sec.
2D	Tri	5085	33 sec.
3D	Hex	7760	275 sec.

Table 1: Solution time per co-simulation loop for switch model

REFERENCES

[1] Aluru, N.R., White, J., "Direct Newton Finite-Element/Boundary-Element Technique For Micr-Electro-Mechanical-Analysis", Solid-State Sensor and Actuator Workshop, Hilton Head, SC, June 2-6, 1996, pp. 54-57.

[2] Field, D. A., "Laplacian smoothing and Delaunay triangulations", Commuications in Applied Numerical Methods., 4, 709-712. (1988)

[3] Joe, B., "Construction of Three-Dimensional Improved-Quality Triangulations Using Local Transformations", Siam Journal of Scientific Computing, 16, 1292-1307 (1995).

[4] Canann, S. A., Tristano, J.R., Staten, M.L., "An Approach to Combined Laplacian and Optimization-Based Smoothing for Triangular, Quadrilateral, and Quad-Dominant Meshes", Proceedings, 7th International Meshing Roundtable, 1998, pp. 479-494,.

[5] Knupp, Patrick M., "Matrix Norms and the Condition Number: A General Framework to Improve Mesh Quality via Node Movement", Proceedings, 8th International Meshing Roundtable, 1999, pp.13-22

[6] Freitag, Lori A., "On Combining Laplacian and Optimization-Based Mesh Smoothing Techniques", Trends in Unstructured Mesh Generation, ASME, AMD- 220, 37-43 (1997)

[7] Tristano, Joseph R., Steven J. Owen and Scott A. Canann, "Advancing Front Surface Mesh Generation in Parametric Space Using a Riemannian Surface Definition", Proceedings, 7th International Meshing Roundtable, 1998, pp.429-445.

[8] Owen, Steven J, M.L. Staten, S.A. Canann and S. Saigal, "Q-Morph: An Indirect Approach to Advancing Front Quad Meshing", International Journal for Numerical Methods in Engineering, 44, 1317(1999).

[9] George, P.L., F. Hecht and E. Saltel, "Automatic Mesh Generator with Specified Boundary", Computer Methods in Applied Mechanics and Engineering, 92, 269-288 (1991).

[10] Gilbert, J.R., Ananthasuresh, G.K., Senturia, S.D., "3D Modeling of Contact Problems and Hysteresis in Coupled Electro Mechanics", MEMS 1996, pp. 127-132

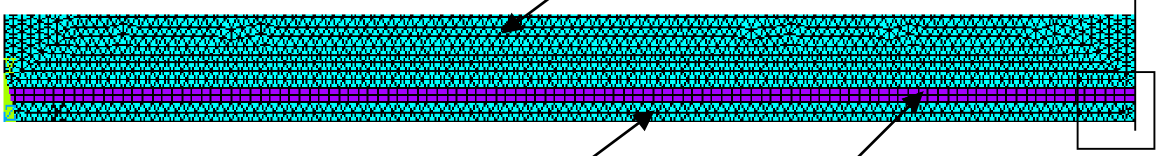


Figure 2: Half -symmetry model.

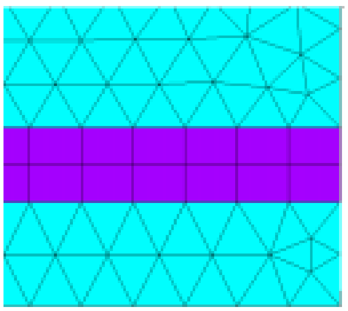


Figure 3: Close-up region

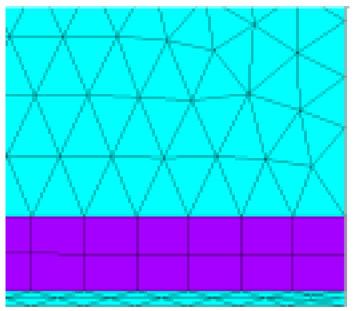


Figure 4: Mesh at pull-in
after morphing.

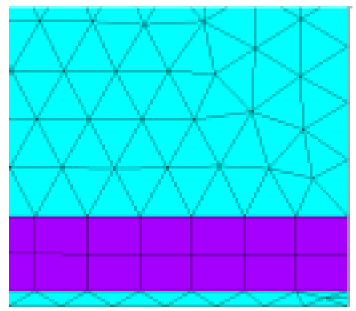


Figure 5: Mesh at pull-in
after remeshing.

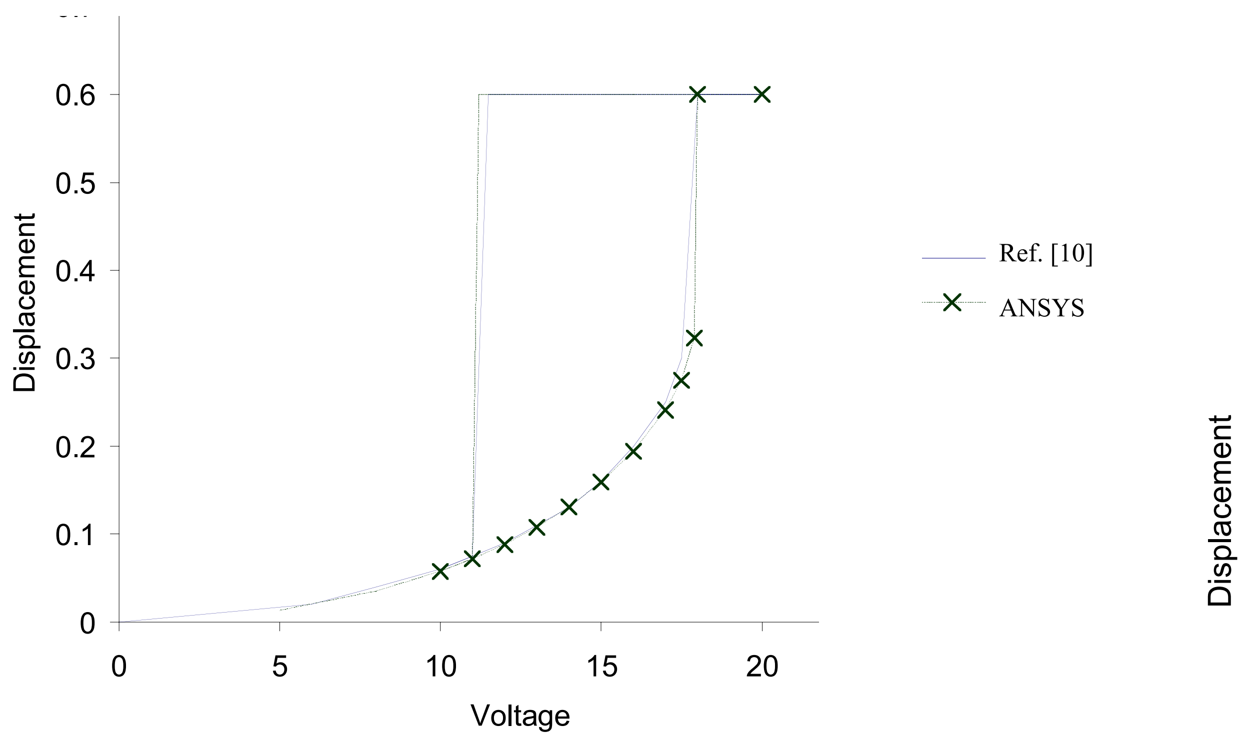


Figure 6: Hysteresis loop: displacement vs. voltage.