Transactions on the Built Environment vol 12, © 1995 WIT Press, www.witpress.com, ISSN 1743-3509

3D coupled electro-mechanical simulations in microstructures

T. Angulo, S. Marco, J. Samitier
EME Electronic Materials and Engineering, Departament de Físcia Aplicada i Electrònica, Universitat de Barcelona, Barcelona, Spain

Abstract

A finite element approach has been developed to solve the problem of structural displacements due to forces from electrostatic attraction. The method can be applied to typical MEMS structures and has been specially thought to determine the pull-in voltage for that subset of systems in which the structure can be taken into collapse. The FE package ANSYS and subroutines programmed in its own language have been the only tools used for this purpose. So, no interface between different program packages is required. Another trait is the three dimensional capability, which allows studies to be performed without geometrical restrictions on complex structures and includes boundary effects of fields. The different parts of the procedure and some alternatives are discussed, mainly the problem of remeshing the deformed structure. As an example, it has been used to determine the attraction cycle of a line-addressable torsional micromirror.

1 Introduction

Micro-electro-mechanical systems (MEMS) are often designed on scales at which electrostatic forces are capable of moving or deforming some parts of the systems. Electrostatic actuation is used advantageously in many of these devices but can also bring problems with it. In any case, the behaviour of structures under the effects of these forces has to be studied in order to be able to predict exact displacements or critical voltages.

When devices are too complex to use analytical formulae or some

Transactions on the Built Environment vol 12, © 1995 WIT Press, www.witpress.com, ISSN 1743-3509

approximations are not desirable, numerical simulations are the best alternative. A 3D coupled simulation is required. But finite element analysis of structures subject to electrostatic forces is difficult because the electrostatic force is non-uniform and changes as the structure deforms.

As currently there is no commercially available finite element analysis package that combines mechanical and electrostatic forces, several university systems have been developed (Schwarzenbach[1], Gilbert[2]). The different methods proposed to solve the problem involve more than one program package with the subsequently need of interfaces to transfer the computed data; others require the obtention of an expression for the forces in terms of the element displacements (Artz[3]). Our contribution to this subject is the development of a coupled solver for 3D or 2D electro-mechanics where only one package is required: the FEA program ANSYS.

2 General procedure

The design of microsystems fabricated with integrated circuit technology combined with micromachining needs a software tool with the following requirements: automatic generation of a 2D or a 3D geometrical model, as input for the simulation, an efficient implementation of the algorithm realizing the pertinent physical coupling effects, and postprocessing capabilities, which complete the microsystem simulation.

Both, electrostatic and structural analysis, are executed by ANSYS 5.0 and a group of subroutines has been programmed in order to build a self-consistent method. The programming language used is APDL (ANSYS Parametric Design Language).

The procedure developed consists of an iterative process. The two fields are coupled by applying the results for forces from an electrostatic analysis as loads to the mechanical analysis. The displacements obtained from the mechanical analysis are then transferred to a new electrostatic model (the deformed structure), resulting in increased electrostatic forces. This procedure is continued until the structure reaches equilibrium or collapse takes place. That is evaluated in the subroutine 'convergence'. The generation of the meshes used in the successive electrostatic analyses has required the development of several subroutines in APDL. They compute the mesh configuration using different algorithms. A flow diagram of the general method with one of these algorithms is shown in figure 2.

The electrostatic analysis is performed with element type SOLID122. It's a three-dimensional finite element with 20 nodes and quadrilateral shape. The mechanical analysis is performed with its equivalent structural element SOLID95. While it is often used a 2D analysis of a cross section of the

Transactions on the Built Environment vol 12, © 1995 WIT Press, www.witpress.com, ISSN 1743-3509

structure, and then assumed the behaviour along the normal axis, these elements permit true 3D simulation. It allows the analysis of that structures in which there are no symmetries to simplify the problem. To simulate a two-dimensional problem the 2D version would be easily available just changing the element types to PLANE82 for structural and PLANE121 for electrostatic analysis.

Contact elements (CONTAC49) are also used in order to prevent an structure from deflecting beyond a rigid surface. This element simulates surface-to-surface contact and its geometry is a pyramid with the base being a quadrilateral.

3 Remeshing the deformed structure

The most complicated step in all this procedure is to remesh the model once it has been deformed by the electrostatic forces. It can't be done in a standard way by ANSYS because of some peculiarities of our problem which will be explained later. As a result, another four meshing algorithms have been created, one with air regions included in structural analyses and three without it. All have been programmed in APDL for their easy integration in all the loop process, eliminating the need of any other CAD package. Figure 1 shows a brief description of the different alternatives.

For best understanding of the discussion about these different methods we will refer to the example of a line-addressable torsional micromirror. It's an aluminized single crystal silicon mirror suspended over an air gap by two thin beams that permit a mirror rotation of $\pm 7.6°$. Below the mirror are an address electrode and a landing electrode.

A. Structural Analysis including Air

A first approach to the behaviour of the device can be obtained by using the full model as input for both analyses, electrostatic and mechanical. In the structural analysis the air is modeled as an elastic solid. The Young's modulus of the air must be set then to a value small enough to make the solid-modelling of the air behaving like a fluid as best as possible, but also large enough to avoid numerical problems. So, a compromise must be reached. The modelling of the air as a real fluid is not possible because the only element available in ANSYS 5.0 simulates a 'contained' fluid, which is not our case. Besides, this element does not fit in the model because it is a 8-node element.

That way, the structural analysis provides not only the displaced mirror, but also the deformation of the air in the gap.

As an advantage it's the simplest way to generate the mesh of the new model for the next iteration. As we obtain the new positions for all the nodes,

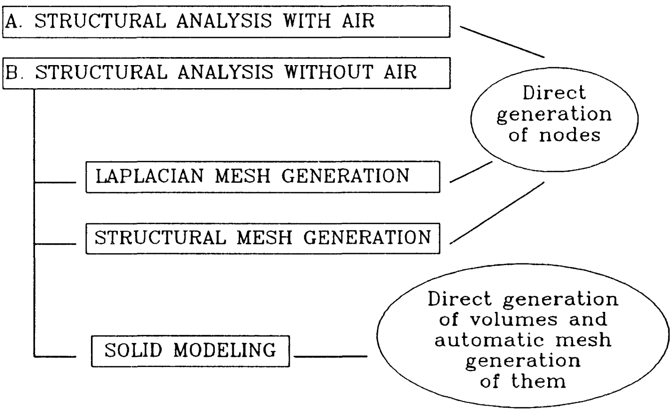

Figure 1: User-programmed subroutines to perform mesh generation of the deformed structure.

the direct mesh generation is easy.

On the other hand, this method assumes that the air is offering resistance to the movement of the micromirror, which is not true in quasi-static conditions since it's not confined. Some error might be present at the results for that reason.

But a part from that errors, there is another disadvantage, specially important when the ANSYS version used has a limited wavefront: a very accurate mesh is not allowed because the full mesh is analyzed each time (in the structural analysis and in the electrostatic analysis), wasting CPU time and memory or disk space.

B. Structural Analysis without Air

To avoid the problems exposed above, structural analyses must be performed excluding the air gap. So, only one part of the mesh is analyzed each time (the mirror plate in the structural analysis and the air gap in the electrostatic analysis). As a result only the displacements for the nodes of the mirror are obtained. Next step is now to mesh the new geometry of the air gap in order to perform the following electrostatic analysis. The main objective is to have a mesh keeping the continuity of the nodes on the interface between the mirror and the air gap.

Three algorithms to get the new positions for the air nodes have been developed:

Solid Modelling ANSYS has available a technique for meshing based on solid modelling: geometry is built and automatic mesh generation can be performed.

It works well when the model can be freely meshed, i.e., if the positions for the nodes are not relevant. But in our case one part of the model has its nodes defined from the mechanical analysis (the solid structure), while another part is still unmeshed (the air). This mixture makes the procedure be complicated. Automatic mesh generation fails because the resultant global mesh does not have continuity in the nodes which are common to both parts of the model. So a subroutine has been developed. Its objective is to be able of generating a mesh with prescribed nodes using the capabilities of the ANSYS automatic mesh generation, but controlling as much parameters as needed to keep continuity.

The procedure is based on the building of a number of volumes equal to the number of elements present on the mirror-air interface. Each volume is set to have just one element in the plane parallel to the interface, and any number of elements in the normal direction. Then, ANSYS automatically meshes each volume. The method is very tedious because the number of volumes is usually large and all the numbers and coordinates of the prescribed nodes must be achieved in a proper order to make connections of volumes. So, the procedure was only programmed to simulate the mirror with air located just below it, but not in a surrounding area.

This method works well but the laboriousness of the algorithm makes other procedures be preferable.

Structural Mesh Generation This algorithm is based on the same assumption that the first method explained: the air in the gap is modeled as an elastic solid. But now the mechanical analysis of the mirror has been yet performed independently from the air (the air has no influence in the structural analysis now), and this second structural analysis is just a trick to find the positions for the nodes in the air gap. The constraints are the displacements for the nodes that the mirror structure and the air have in common. This analysis can be performed either, starting from the original structure or starting from the previous mesh.

The mechanical properties assigned to the air play an important role in the shape of the resultant mesh. This shape depends on the difference between the Young's modulus defined in the volume just below the mirror and in the surrounding area respectively. It also changes if an isotropic or an orthotropic material is considered. When all the values for the Young's modulus are set equal and we are in a case of large deflections, the mesh we obtain is unusable for subsequent analyses. When some values are changed the resultant meshes are more regular.

However it presents some disadvantages. First, if the ANSYS version has a limited wavefront, the mesh can't be very accurate. The structural analysis of the air will limit the number of nodes because the air volume occupies the biggest part of the model, at least if a surrounding area is meshed to study border effects. It is due to the three degrees of freedom that there are in structural analyses.

In second place, building a good mesh is difficult with that method because the range of differences between mechanical properties can't be so wide as in the case of other properties (for example conductivities). Large differences in Young's modulus bring numerical problems. The best shapes got with this method are a little irregular. This irregularities are growing as the tilting angle increases, and it yields to bad results for torsion angles greater than the 70-80% of maximum.

Laplacian Mesh Generation The volume of air that we have to mesh has one surface (the interface) with nodes already defined. That means we have the x,y,z triplets for the nodes. Using this information we can perform three thermal analyses starting from the original non-deformed structure to obtain the positions for all the nodes within the volume. A flow diagram of the program with this algorithm implemented is shown in figure 2.

There is an identification of the values for coordinates with temperatures. We must assign successively the x coordinates, y coordinates and finally z coordinates as prescribed temperatures to the interface of our non-deformed model, and the x,y,z coordinates for the internal nodes will be computed, corresponding to the internal temperatures of each analysis. What we have just done is performed a numerical flux plot over the real geometry, with the flux lines defining elements and nodes. The resolution of that Laplacian equation gives as a result a gradient of temperatures where the values within the volume will be between the extreme values.

As a previous condition it is necessary that the structure is a relatively smooth surface.

In thermal analyses, the material properties are the conductivities. If the same value is assigned to all the model and in all directions (isotropic material) the resultant gradients lead to a bad mesh. But considering the air as an orthotropic material we are able to obtain better meshes according to the different sets of conductivity constants assigned. The best mesh, that is, a mesh as regular as possible, is got when besides being the air considered as an orthotropic material, different sets of constants are defined for the air located just below the plate and the surrounding air.

This method works well even for tilting angles near the maximum. It also has the advantage of allowing refined meshes in versions of ANSYS with limited wavefront because thermal analyses have only one DOF.

Transactions on the Built Environment vol 12, © 1995 WIT Press, www.witpress.com, ISSN 1743-3509

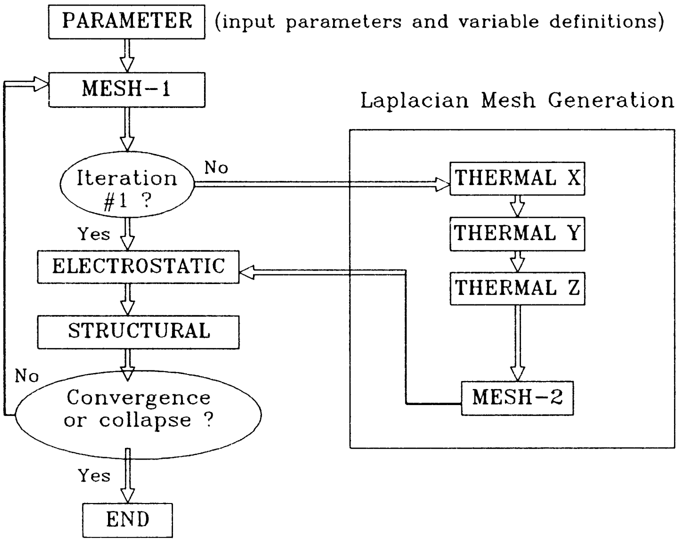

Figure 2: Flow diagram of the general procedure for coupled electro-mechanics simulation.

4 Application to line-addressable torsional micromirror

We have applied our method to analyze an aluminized single crystal silicon micromirror (Jaecklin[4]) which has a surface of 30x30 μm^2 and is suspended by 15 μm long and 0.4 μm wide beams. The thickness of silicon is 2 μm and the air gap is also 2 μm. We have chosen a 30 μm by 11 μm address electrode, and as a material an orthotropic silicon with the following constants:

Table 1.Material properties.

Elastic moduli (Pa)	$E_z = 1.302 \cdot 10^{11}$	$E_x = E_y = 1.689 \cdot 10^{11}$
Poisson's ratios	$\nu_{xy} = 0.064$	$\nu_{xz} = \nu_{yz} = 0.279$
Shear moduli (Pa)	$G_{xy} = 0.509 \cdot 10^{11}$	$G_{xz} = G_{yz} = 0.794 \cdot 10^{11}$

In order to get a first approach to the behaviour of the micromirror, we have meshed ignoring some parts of the device which are not expected to influence the results, as the aluminized mirror surface, the SiO insulator and the silicon substrate. So, we have simulated the silicon plate, the air gap and the address and ground electrodes.

Results

The behaviour of the micromirror has been simulated under the effects of 8 different voltages. In figure 3 the torsion angle is shown as a function of the applied voltage. It is obviously a non-linear problem. Abrupt tilting takes place when voltage is near the pull-in value. The graph is the typical one for a torsional micromirror in the attraction cycle.

Results of simulations also show how potential field, electric field and electrostatic forces depend on the tilting angle. For an applied voltage of 25 V the total torsion angle is about 0.1°. The maximum angle is 7.6°, so the mirror remains almost horizontal and a quasi-uniform electric field in the air gap is expected. On the contrary, for a voltage in the address electrode of 85 V the displacement is about 2.5° and field distributions are found to have a gradient that leads to a maximum value much greater due to the shorter distance. The electrostatic field intensity in that last case is shown in figure 4. The location of the electrodes is shown with arrows, being the address electrode between the two ground electrodes.

Figure 5 shows an exaggerated view of the micromirror tilt under a voltage of 45 V. Only half model has been meshed because of symmetry properties.

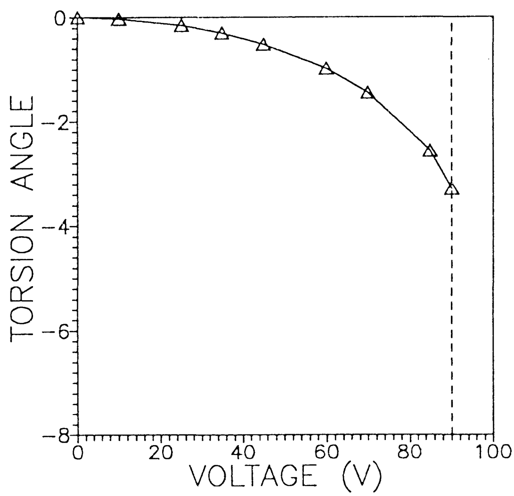

Figure 3: Angle (°) vs. voltage for the tilting mirror. The pull-in occurs beyond the dashed line.

Transactions on the Built Environment vol 12, © 1995 WIT Press, www.witpress.com, ISSN 1743-3509

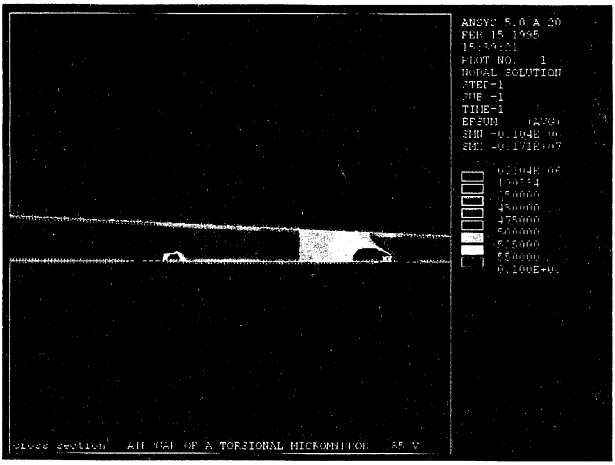

Figure 4: electrostatic field intensity in the air gap for V=85 V.

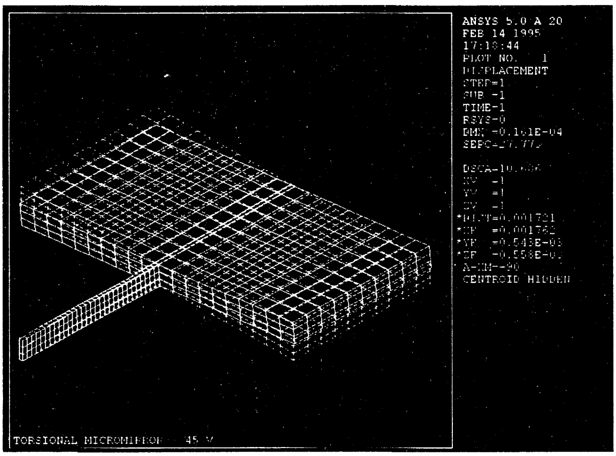

Figure 5: Exaggerated view of the deformed micromirror for V=45V.

5 Conclusions

With this method based on FE we are able to predict the behaviour of the micromirror, and it is general enough to be easily adapted to the simulation of other devices under the actuation of electrostatic forces. The procedure considers the fact that these are position-dependent forces.

Transactions on the Built Environment vol 12, © 1995 WIT Press, www.witpress.com, ISSN 1743-3509

Numerical simulations also consider many of these elements that are not included in analytic formulas, such as boundary effects of electrostatic forces. In our example they are not quite significative. That effects have been evaluated and represent just a 0.8% of the total force. But in some other cases as the design in surface micromachining they are too large to be neglected. The program is expected to provide reliable results in all that 3D cases in which analytical methods could fail because of the approximations.

A group of subroutines has been written to complete the procedure. These subroutines are necessary to generate the mesh for the electrostatic analyses in the successive iterations. Different techniques for meshing have been developed and after testing all the methods the "Laplacian Mesh Generation" has been chosen as the best one since it gives good results in all cases, while the others fail in specific limit situations as large tilting angles of the plate. But for analysis where this angle is still less than 70% of the maximum allowed all the methods are valid, though the "Structural Analysis including Air" gives just an approximated result.

References

1. Schwarzenbach, H.U., Korvink, J.G., Roos, M., Sartoris, G. & Anderheggen, E. A micro electro mechanical CAD extension for SESES, *Journal of Micromechanics and Microengineering*, 1993, **3**, 118-122.

2. Gilbert, J.R., Legtenberg, R. & Senturia, S.D. 3D Coupled Electro-mechanics for MEMS: Applications of CoSolve-EM, pp. 122 to 127, *Proceedings of IEEE on Micro Electro Mechanical Systems*, Amsterdam, the Netherlands, 1995, IEEE Press, 1995.

3. Artz, B.E. & Cathey, L.W. A finite element method for determining structural displacements resulting from electrostatic forces, pp. 190 to 193, *Proceedings of Technical Digest IEEE Solid State Sensors Workshop*, Hilton Head Island SC USA, 1992, IEEE Press, 1992.

4. Jaecklin, V.P., Linder, C., de Rooij, N.F., Moret, J.M. & Vuilleumier, R. Line-addressable torsional micromirrors for light modulator arrays, *Sensors and Actuators A*, 1994, **41-42**, 324-329.