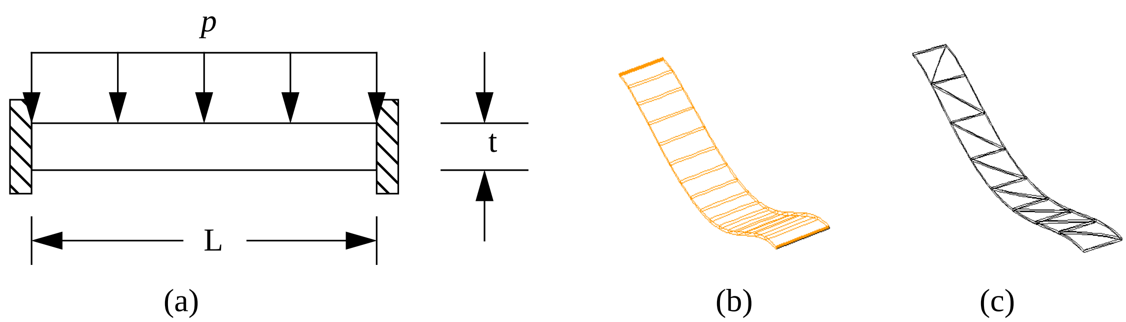


Fig. 3: The fixed-fixed plate test problem. The plate undergoes a maximum of 2 μm of deflection, which corresponds to a pull-in of a switch fabricated in the Poly1 layer in MUMPs. Geometric nonlinearity (large deflections and rotations) is accounted for in the simulations. (a) Length = 400 μm, width = 40 μm, thickness = 2 μm, pressure = $-5*10^{-9}$ N/μm^2, Young's Modulus = 0.136 GPa, Poisson's ratio = 0.23. (b) Deformed hexahedral mesh. (c) Deformed tetrahedral mesh. [scale factor of 20 on the displacements]

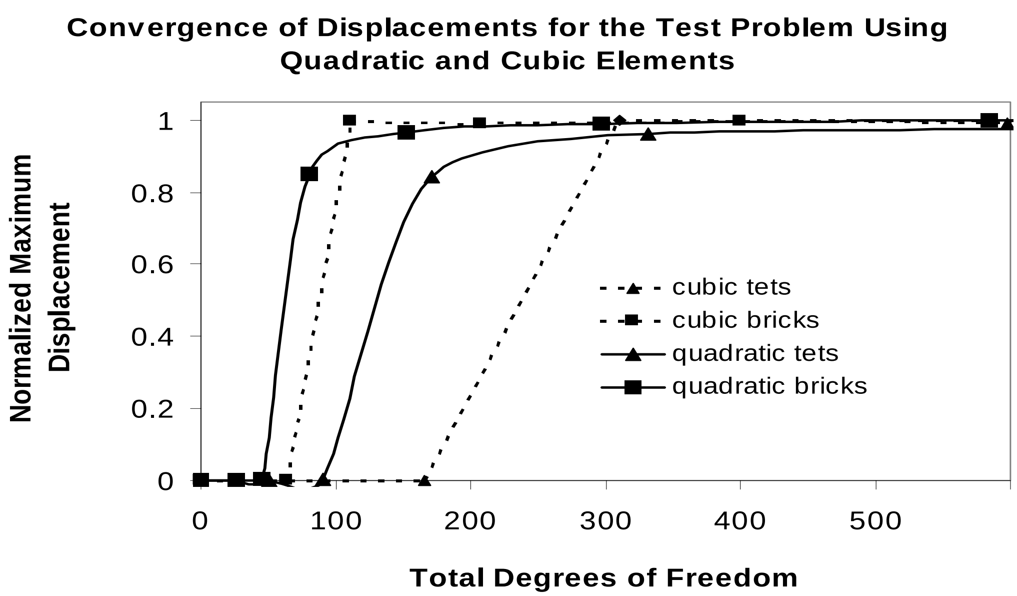
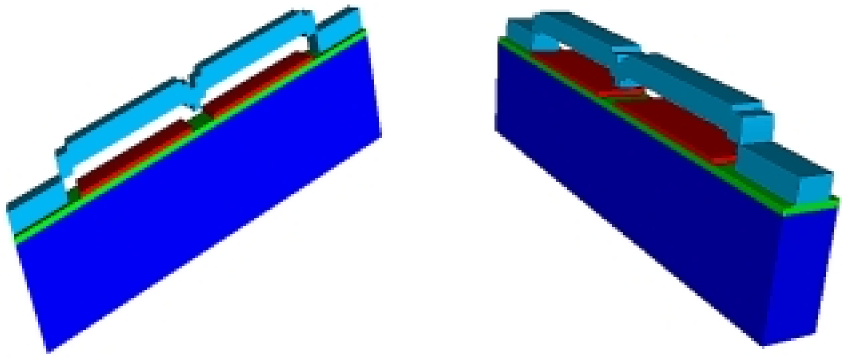
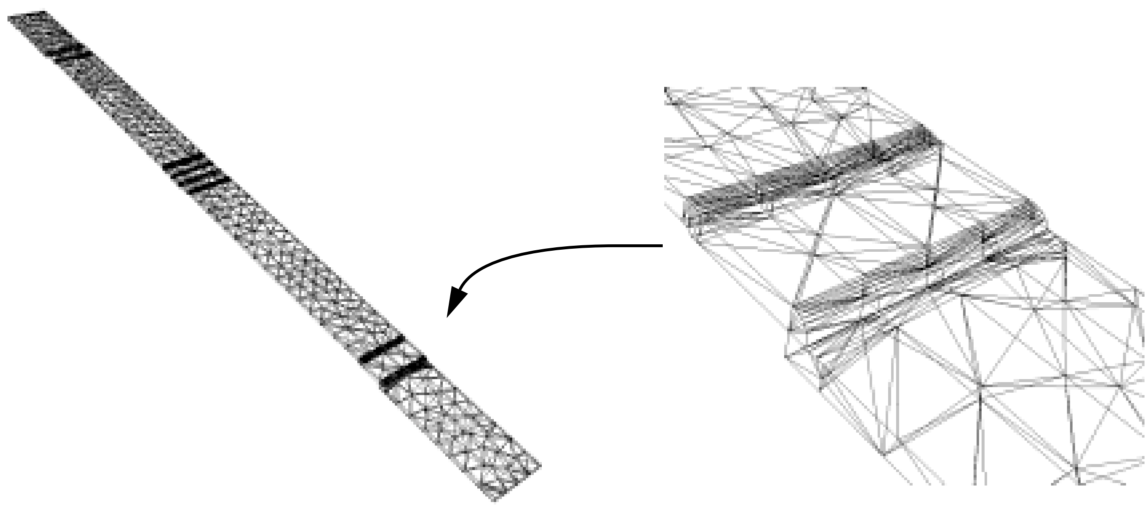


Fig. 4: The convergence of the maximum displacement of the plate. Notice that for the converged solution (assumed to be 0.99 of maximum displacement), the cubic tetrahedrons and quadratic bricks perform approximately the same.



(a) (b)

Fig. 1: A two electrode switch which has complicated geometry due to conformal deposition and etches causing angled sidewalls. (a) & (b) Two views of the RF switch.



Dihedral angles	:	# elements	percent.	cumul.
<= 145	:	2326	99.57 %	99.57 %
]145..160]	:	10	0.43 %	100.00 %
> 170	:	0	0.00 %	100.00 %

Aspect ratios	:	# elements	percent.	cumul.
< 2	:	0	0.00 %	0.00 %
[2.. 4[	:	98	4.20 %	4.20 %
[4.. 6[	:	1062	45.46 %	49.66 %
[6.. 10[	:	376	16.10 %	65.75 %
[10.. 20[	:	636	27.23 %	92.98 %
[20.. 40[	:	135	5.78 %	98.76 %
> 40	:	29	1.24 %	100.00 %

Number of vertices	:	964	
Number of elements	:	2336	
Smallest dihedral angle	:	Element = 2284	0.01147
Largest dihedral angle	:	Element = 2309	54.36747
Worst r/R Ratio	:	Element = 2284	0.00031
Worst Edge Length Ratio	:	Element = 1971	5724.26847
Worst Aspect Ratio	:	Element = 2284	6718.54616
Worst Shape	:	Element = 240	3.34889e-14

Fig. 2: Mesh of switch (actuation layer) generated using MEGA. (a) Full view of the mesh. (b) Enlarged view of the corner. (c) Mesh statistics.

where:

$\mathbf{u}$ = exact solution (displacements)

$\mathbf{u}_h$ = finite element solution (displacements)

h = characteristic element dimension

k = degree of complete polynomials

c = a constant independent of h

Although the given expression is for a uniformly refined mesh, it is qualitatively the same in the case of general meshes. Optimally, a combination of *h* (element size) and *k* (polynomial order) refinement of the mesh can lead to exponential convergence. However, in practice obtaining these results are difficult. For simplicity, the numerical results presented in this paper will be limited to uniform *h* or *k* refinement of the mesh. In addition, for the test problem we term the solution converged when the maximum displacements converge.

Element distortion and effects on accuracy

Convergence studies indicating the superiority of quadrilateral and hexahedral elements over triangular or tetrahedral elements for regular geometries and meshes can be found in nearly all introductory texts on finite elements methods. However, when angular distortion occurs, solution accuracy can be significantly impacted depending on the type of element used. There are two distinct causes of angular distortion encountered in the simulation of the elasticity problem of the switch. First, it can occur in the initial mesh due to complex geometry and/or the algorithms used to automatically generate the mesh. Second, the large deformation that the switches undergo during actuation can cause elements to become significantly distorted. The behavior of distorted elements has been recognized as one of the most important aspects in the element selection process [7,8]. Two-dimensional isoparametric quadrilaterals have been studied extensively for their poor behavior in the distorted configuration, and the results are assumed to extrapolate to 3-D. The convergence of the method is affected both by the obvious change in the characteristic element length (*h*) and also by the subtle lose of polynomial completeness (*k*) in the serendipity elements. The effective polynomial completeness of an angularly distorted cubic serendipity isoparametric quadrilateral element is only equal to that of the linear element, and therefore, the element of choice for poorly shaped quadrilateral elements should be the complete Lagrangian-type element [9]. As discussed in [10], certain classes (e.g. the integrated Legendre polynomials) of *p*-finite element basis functions retain a high level of accuracy even with high aspect ratios and significant element distortion. Thus hierarchical *p*-finite element basis functions may prove particularly beneficial in MEMS simulation due to the geometry of the devices.

NUMERICAL RESULTS

The numerical results for a plate model subjected to a uniform pressure loading (Fig. 3) is shown in Fig. 4. The width of the plate is such that only refinement along the length is required to converge the solution. An *hp*-adaptive commercial finite-element code [11], which utilizes hierarchical shape functions [10], was used to simulate the finite-deformation of the plate. The convergence of displacements is plotted for the solutions using both quadratic and cubic elements. As predicted by theory, the convergence of displacement for the brick elements is slightly better that tetrahedral elements. However, it is interesting to point out that in the test case cubic tetrahedrals perform as well as quadratic hexahedral elements to obtain a fully converged solution (0.99 of maximum displacement). Since state-of-the-art MEMS simulation tools rely on the convergence rate of quadratic brick elements [12,13], further investigation is needed to see if cubic tetrahedral elements can offer equivalent convergence performance without significant computational penalty. The trade-offs of the cubic shape functions will be discussed including the need for more Gauss quadrature points and the possible impact on the condition number of the stiffness matrix.

AKNOWLEDGEMENTS

This work was supported under the DARPA Composite CAD program contract #F30602-96-2-0308-P001.

REFERENCES

[1] Owen, S. J., 1998, “A Survey of Unstructured Mesh Generation Technology”, 7th International Meshing Roundtable, Dearborn, Michigan, Oct. 26-28.

[2] Lee, C. K., Lo, S. H., 1997, “Automatic Adaptive 3-D Finite Element Refinement Using Different-Order Tetrahedral Elements”, Int. Jour. Num. Meth. Eng., vol. 40, pp. 2195-2226.

[3] MEGA, http://scorec.rpi.edu/software/Software.html

[4] Joe, B., Liu, A., 1994, “Relationship Between Tetrahedron Shape Measures”, BIT, vol. 34, pp. 268-287.

[5] Berzins, M., 1998, “A Solution-Based Triangular and Tetrahedral Mesh Quality Indicator”, SIAM Jour. Sci. Comp., vol. 19, pp. 2051-2060.

[6] Bathe, K.J., 1996, *Finite Element Procedures*, Prentice-Hall, Upper Saddle River, New Jersey.

[7] MacNeal, R. H., Harder, R. L., 1985, “A Proposed Standard Set of Problems to Test Finite Element Accuracy”, Fin. Elem. in Anal. Des., vol. 1, pp. 3-20.

[8] Robinson, J., 1976, “A Single Element Test”, Comp. Meth. Appl. Mech. Eng., vol. 7, pp. 191-200.

[9] Bathe, K. J., Lee, N.S., 1993, “Effects of Element Distortions on the Performance of Isoparametric Elements”, Int. Jour. Num. Meth. Eng., vol. 36, pp. 3553-3576.

[10] Woo, K. S., 1993, “Robustness of Hierarchical Elements Formulated By Integrals of Legendre Polynomials”, Computers & Structures, vol. 49, pp. 421-426.

[11] ProPHLEX, http://www.comco.com

[12] Intellicad, http://www.intellisense.com.

[13] MEMCAD, http://www.memcad.com.

Investigation of Tetrahedral Automatic Mesh Generation for Finite-Element Simulation of Micro-Electro-Mechanical Switches

Nathan M. Wilson[1], Peter M. Pinsky[1], Robert W. Dutton[2]
Departments of Mechanical[1] and Electrical[2] Engineering
Division of Mechanics and Computation[1] and TCAD group[2]

CISX 332, Stanford University, Stanford, CA 94305-4075
email: nwilson@leland.stanford.edu

ABSTRACT

Simulation of micro-electro-mechanical systems (MEMS) typically involves the requirement to mesh complex 3-D representations of the geometry. Since proper manual or semi-automated mesh generation requires significant user time and knowledge, there is great interest in automating this tedious and error prone process. There are two key technical issues. First, a software package must be capable of automatically generating a mesh. Secondly, the solver must be able to "adjust" the mesh according to the error in the solution to achieve a result which is useful to the designer. This paper starts with a survey of the current state-of-the-art in meshing software, and then it discusses the connection between element choice, mesh quality, and the accuracy of the simulation results.

keywords: unstructured meshes, micro-electro-mechanical systems (MEMS), tetrahedral elements, finite element analysis.

INTRODUCTION

The geometry of micro-electro-mechanical switches is typically complex (see Fig. 1). If accurate finite-element simulations of this switch are desired, for example to utilize constitutive models which can incorporate physical effects such as fatigue, automatic mesh generation may be required. One of the major "bottlenecks" in currently available MEMS design systems is the lack of automatic and robust mesh generation. The purpose of this paper is to discuss the use of unstructured, automatic mesh generation and use a simple test problem to gain insight into possible strategies in utilizing the state-of-the-art in meshing software. As the numerical results tend to indicate, promising finite-element technology may be able to maintain the current convergence rates obtained by commercial MEMS simulation tools, while allowing fully-automated mesh generation.

AUTOMATIC MESHING TECHNOLOGY

Automatic mesh generation has been an area of intense research for decades, and a tremendous amount of literature and numerous algorithms have been developed. There are three fundamental challenges in the field: robustness, mesh quality, and computational efficiency in generating the mesh. A recent survey indicated there were over 80 commercial and academic meshing products available, of which 39 automatically generated tetrahedral ("tet") elements compared to 20 that performed unstructured hexahedral ("brick") mesh generation [1]. The current dominance of tetrahedral meshing can be attributed most notably to its ability to robustly mesh arbitrary, complex geometries. In addition, the use of tetrahedral elements often simplifies the process of adapting the mesh during simulation [2]. Most commercially available 3-D unstructured hexahedra meshing codes (e.g. CUBIT and ANSYS) rely on "hex-dominant" meshing, that is, the structure is meshed with a combination of hexahedral and tetrahedral elements. Even in this case, knowledge of the performance of tetrahedral elements is critical since they often appear in the areas with the most geometric complexity which normally correspond to areas of interest (e.g. a stepup). Due to the geometric complexity of the switch in Fig. 1, an octree-based tetrahedral mesh generator (MEGA [3]) was used to mesh the switch (Fig. 2).

MESH QUALITY STATISTICS

There are numerous geometric quality measures used by automatic meshing programs to iteratively improve the quality of the mesh. The two most important are [4]:

1) Minimum solid angle, θ_{min}. (Note: since the solid angles of a tetrahedron sum to 2π, a large solid angle implies a small solid angle, so they are equivalent measures).

2) Radius Ratio, $\rho=3*r_{in}/r_{out}$, where r_{in} and r_{circ} are the inradius and circumradius, respectively.

An example mesh of the switch is given in Fig. 2, along with the mesh quality measures for the given mesh. As is common practice in the mesh generation community, the global min solid (dihedral) angle and radius ratio are listed along with the percentage of elements in a given range. It should be noted that there is current work in generating errors estimators that include both solution and geometric information [5].

CONVERGENCE OF THE FINITE-ELEMENT METHOD

The convergence of displacements in the finite-element method for linear elasticity is given by [6]:

$$\left\|\boldsymbol{u}-\boldsymbol{u}_h\right\|_0 \le c h^{k+1}\left\|\boldsymbol{u}\right\|_{k+1}$$