3D TETRAHEDRAL MESH DEFORMATION FOR SURGERY SIMULATION USING NON-LINEAR FINITE ELEMENT METHOD

A THESIS
SUBMITTED TO THE DEPARTMENT OF COMPUTER ENGINEERING
AND THE GRADUATE SCHOOL OF ENGINEERING AND SCIENCE
OF BİLKENT UNIVERSITY
IN PARTIAL FULFILLMENT OF THE REQUIREMENTS
FOR THE DEGREE OF
MASTER OF SCIENCE

By
Emir Gülümser
October, 2011

I certify that I have read this thesis and that in my opinion it is fully adequate, in scope and in quality, as a thesis for the degree of Master of Science.

Assoc. Prof. Dr. Uğur Güdükbay (Supervisor)

I certify that I have read this thesis and that in my opinion it is fully adequate, in scope and in quality, as a thesis for the degree of Master of Science.

Asst. Prof. Dr. Sinan Filiz (Co-supervisor)

I certify that I have read this thesis and that in my opinion it is fully adequate, in scope and in quality, as a thesis for the degree of Master of Science.

Assoc. Prof. Dr. Y

I certify that I have read this thesis and that in my opinion it is fully adequate, in scope and in quality, as a thesis for the degree of Master of Science.

Asst. Prof. Dr. İlker Temizer

Approved for the Graduate School of Engineering and Science:

Prof. Dr. Levent Onural
Director of the Graduate School

ABSTRACT

3D TETRAHEDRAL MESH DEFORMATION FOR SURGERY SIMULATION USING NON-LINEAR FINITE ELEMENT METHOD

Emir Gülümser
M.S. in Computer Engineering
Supervisors: Assoc. Prof. Dr. Uğur Güdükbay
Asst. Prof. Dr. Sinan Filiz
October, 2011

Finite Element Method is a widely used numerical technique for finding approximate solutions to the complex problems of engineering and mathematical physics that cannot be solved with analytical methods. In most of the applications that requires simulation to be fast, linear FEM is preferred. Linear FEM works highly accurate with small deformations. However, linear FEM fails in accuracy when the large deformations are used. In this thesis, we presented both linear FEM and non-linear FEM in order to examine non-linear FEM's advantage over the linear FEM with using both small and large deformations. In order to make better analysis, linear FEM and non-linear FEM are both implemented with using tetrahedral elements. In addition, we did not use material nonlinearity with non-linear FEM. To state the effect of using the same material with nonlinear geometric properties, we only used geometric nonlinearity (Green-Lagrange strain definitions). In our experiments, it is shown that non-linear FEM gives more accurate results when compared to linear FEM. Moreover, the proposed non-linear solution achieved significant speed-ups for the calculation of stiffness matrices and for the solution of the whole system, compared to a state-of-the-art method. In terms of high accuracy, nonlinear FEM is the suitable method for crucial applications like surgical simulators.

Keywords: tetrahedral mesh, deformation, finite element method, Green-Lagrange strain, surgery simulation.

ÖZET

TÜRKCE BAŞLıK

Emir Gülümser
Bilgisayar Mühendisliği, Yüksek Lisans
Tez Yöneticileri: Doç. Dr. Uğur Güdükbay
Yrd. Doç. Dr. Sinan Filiz
Ekim, 2011

Turkce Ozeti olayin budur.

Anahtar sözcükler: Anahtar Sözlerim.

Acknowledgement

I acknowledge that ...

Contents

List of Figures

List of Tables

List of Algorithms

Chapter 1

Introduction

Mesh is a collection of polygonal facets targeting to constitute an appropriate approximation of a real 3D object [?]. It has vertices, edges and facets. A mesh gives two kinds of information; geometric and connectivity. Geometric information gives the position of its elements, and the connectivity information gives the relationship between its elements. Mesh deformation means modifying the shape of the original object by using control points or external forces. 3D mesh deformation has been a highly active research area in computer graphics. Deformations have widespread usage areas like computer games, computer animations, fluid flow, heat transfer, surgical simulation, cloth simulation, crash test simulations. The major goal in mesh deformations is to establish a good balance between the accuracy of the simulation and the computational cost, which depends on the application.

Biraz daha konuya genis bir girizgah yap.

In order to make deformations realistic and highly accurate, Finite Element Method (FEM) could be used. FEM is a numerical method to find approximate solutions to the problems of engineering and mathematical physics. Typical problems that are solvable by use of the finite element method include structural analysis, heat transfer, fluid flow, mass transport, and electromagnetic potential [?].

It is not possible to obtain an analytical solution for problems involving complicated geometries (e.g., 3D organ models), loadings, and material properties. Analytical solutions generally require the solution of ordinary or partial differential equations. However, they are not generally obtainable because of the problems mentioned above [**?**]. When we incorporate these into the analytical solution, the degree of the partial differential equations becomes so high that the solution is not obtainable. Numerical methods that find approximate solutions are used to overcome this problem. Among these, Rayleigh-Ritz, Galerkin, and Finite Difference methods, are the most common ones. Finite difference method approximates the differential equations with equivalent difference quotients using limits over the domain. In other words, the method approximates the solution of differential equations by using approximations to the derivatives. Rayleigh-Ritz method introduces trial functions like FEM's weight functions. However, it only works for conservative systems and it gets very complicated for complex geometries (i.e., 3D organ meshes, crash tests models, etc.). Galerkin method uses weighted residuals to calculate a global stiffness matrix, as FEM does. However, none of these techniques can handle geometrically complex domains. The approximation to the solution becomes very complicated even in simpler domains. In Chapter **??**, we will discuss Galerkin, Rayleigh-Ritz and Weighted-Residual methods, which are called *variational* numerical techniques, in detail.

FEM simplifies these calculations by subdividing the given domain into a finite set of subdomains, called *finite elements* [**?**]. The problem becomes easier and solvable over these domains, such as a set of rectangles, triangles, and in our case, tetrahedra. When using FEM for all types of problems; 1D, 2D, 3D, linear or nonlinear, it follows certain steps that lead to an approximate solution of the given domain.

Surgical simulations require high accuracy, very low error tolerance and real-time interaction that cannot be fully handled with popular deformation techniques, such as regular deformations, Free-form deformation and mass spring system. Numerical solutions that require high accuracy must be used such as finite difference, variational methods and FEM. By far the most popular numerical solution technique is FEM with surgical simulations. Firstly, linear FEM models are used due to faster calculation compared to nonlinear FEM. Cover et al. used surface models to model the system [?]. However, surface models are not sufficient for surgical simulators to perform internal surgical operations. Along with the increase in the computational power of the computers, the work of Bronielsen prompted the usage of volumetric models [?]. Cavusoglu used nonlinear model's reduction to linear model using linear FEM [?]. Most recent surgical simulators uses nonlinear FEM [?, ?]. Because, nonlinear FEM is capable of providing realistic predictions of finite deformations of the tissue. Although, these recent systems provide real-time simulations, precalculation step still takes many hours.

1.1 Research Contributions

In this thesis we

1. Propose a simpler nonlinear FEM by using tetrahedral meshes to speed up the calculation of nonlinear elemental stiffness matrices and residuals.
2. Use geometric nonlinearity in order to compare the results directly with linear FEM.
3. Extend the linear FEM to the nonlinear FEM by extending the linear strains to the Green-Lagrange strains directly.
4. Compare with Pedersen's method to measure the performance and verify the correctness our approach in eight different experiments. We achieve 111% speed-up for the calculation of stiffness matrices and 17% on average for the solution of the whole system, compared to Pedersen's method.

Medical simulation'lar icin kullanilabilecek yontemler. Genel amacli deformasyon teknikleri uygun degil. Daha accurate cozumlere ihtiyac var.

FEM simdiye kadar medical simulation'lar icin nasil kullanilmis?

Sen ne yaptin? Pederson' ....

1.2 Overview of the Thesis

Chapter **??** describes the related work on deformation techniques, Finite Element Method, and the nonlinear analysis. Chapter **??** discusses variational approaches, such as Galerkin, Rayleigh-Ritz and Weighted-Residual methods, and finite difference methods. Chapter **??** explains linear FEM. Chapter **??** describes the nonlinear FEM, including a detailed discussion of the Pedersen's method and the proposed approach. Chapter 2 gives the experimental results obtained using the proposed approach and compares the computational overhead of the proposed approach with that of Pedersen. Chapter **??** gives conclusions and possible future research directions.

Chapter 2

Experimental Results

We conducted eight experiments to compare the linear and nonlinear finite element methods. Moreover, we compare the proposed nonlinear FEM method with the Pedersen's method [**?**].

First, we present how we construct FEM models and continue with error analysis for linear and nonlinear FEM solution with the cube mesh. We make analysis with increasing the mesh's density and comparing the displacements for a selected node.

In the first experiment, our aim is to observe the strain-displacement relationship. The test model is a cube with six elements. We also examine the force-displacement relationship for a selected node to compare the displacements for linear and nonlinear FEMs.

The rest of the experiments are performed with different test models. Our aim in these experiments is to compare the accuracy of the deformations for linear and nonlinear FEMs. The results for these experiments are interpreted by comparing displacement amounts for the force applied nodes and all the nodes. Finally, the computational costs of different methods are compared, including experiments on single-core and multi-cores to assess the parallelization of the methods.

2.1 Construction of the FEM Models

The construction of the FEM models consists of three stages:

1. Reading the surface meshes. The meshes for the cube, beam and the cross surface models are constructed manually, and the liver mesh is taken from 3D Mesh Research Database [**?**].

2. Tetrahedralizing the surface mesh by using TetGen [**?**]. We improve the quality of the models using TetGen.

3. Interactively specifying the constrained (fixed) nodes and the nodes to which the forces to be applied.

2.2 Material Properties

We did not use material nonlinearity with nonlinear FEM. To state the effect of using the same material with nonlinear geometric properties, we only used geometric nonlinearity (Green-Lagrange strain definitions), and we used linear material properties for the models in the experiments. We used 1 for Young's modulus (ϵ), and 0.25 for poisson's ratio (ν).

2.3 Error Analysis

Error analysis is one of the crucial steps of the finite element method to assess the quality of the computed results. We need to make error analysis using approximate results when the exact solution is not available. Error analysis is done by comparing the displacements of the two approximate results by increasing the number of elements in meshes uniformly. We chose a cube mesh ($10 \times 10 \times 10$) to work with because uniformly increasing the number of elements of the cube is much easier than using a complex mesh.

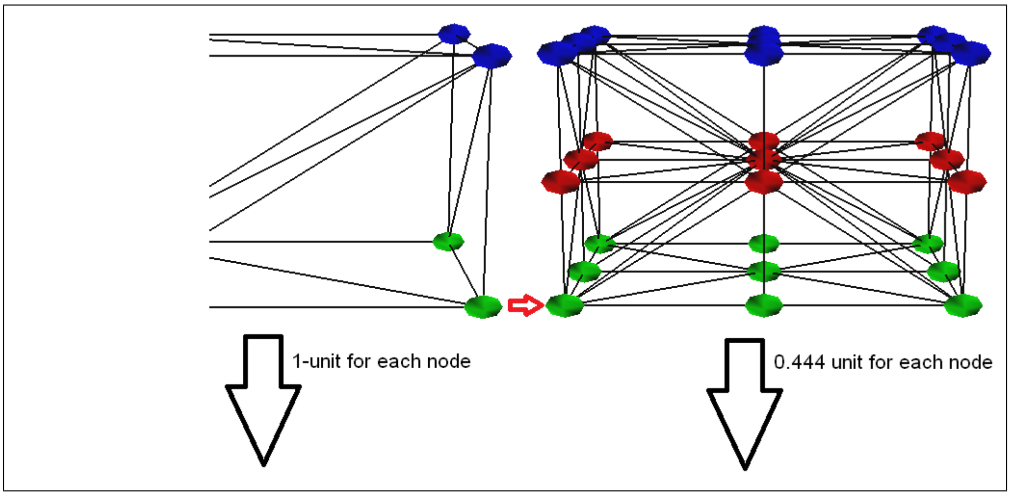

Figure 2.1: The cube mesh with six elements (left) and 48 elements (right).

$$\| u^d - u^{\frac{d}{2}} \| = C \| (u - u^{\frac{d}{2}}) \| \tag{2.1}$$

The error analysis is achieved by comparing the displacements with mesh density d and $\frac{d}{2}$ in 1D (Equation 2.1). If we adapt the 1D formula to 3D, we need to increase the density by 8-times (for every dimension by d to $\frac{d}{2}$) for error analysis. Figure 2.1 shows that number of elements are increased from 6 to 48 for the first step.

The force amount must be the same for each step to observe the displacement errors. Hence, the cube is constrained from the bottom face and pulled towards the direction of the black arrow with same amount of force uniformly distributed among the green nodes (4 units for both the 6 and 48 element meshes) for each step. We choose the node that is highlighted by red arrow to observe the displacements. Moreover, we limited our analysis with 1536 elements because of the high computational cost of nonlinear FEM.

The results in Table 2.1 and 2.2 show that the difference $u^d - u^{\frac{d}{8}}$ decreases with mesh refinement in each step. Using Equation **??**, we can state that the solutions of the linear and nonlinear FEM are valid and converge to the exact solution for this experiment.

Element	**Displacement - z**	**Element**	**Displacement - z**	**Error (%)**
6	0.3831	48	0.3995	4.105
48	0.3995	384	0.4027	0.794
384	0.4027	1536	0.4025	0.049

Table 2.1: Element displacements along the z-axis for node 4 and their corresponding error ratios for linear FEM

Element	**Displacement - z**	**Element**	**Displacement - z**	**Error (%)**
6	0.3622	48	0.3795	4.558
48	0.3795	384	0.3751	1.159
384	0.375162	1536	0.375101	0.016

Table 2.2: Element displacements along the z-axis for node 4 and their corresponding error ratios for nonlinear FEM

Error norms are required to compute the error for the whole solution. L_2 and *Energy* norms are the most frequent used norms to compute the errors. These are defined as

$$\begin{aligned} L_2 &= \sqrt{\int\int\int e^2 dx\ dy\ dz} \quad \text{and} \\ Energy &= \sqrt{\tfrac{1}{2}\int\int\int \tfrac{\partial e}{\partial x} + \tfrac{\partial e}{\partial y} + \tfrac{\partial e}{\partial z} dx\ dy\ dz}. \end{aligned} \tag{2.2}$$

where e is the error. Error is computed by deducting u (the actual solution; d) from u_N (approximate solution; $\frac{d}{2}$). Figures 2.2 and 2.3 show that error decreases linearly decreases and converges with mesh refinement in each step.

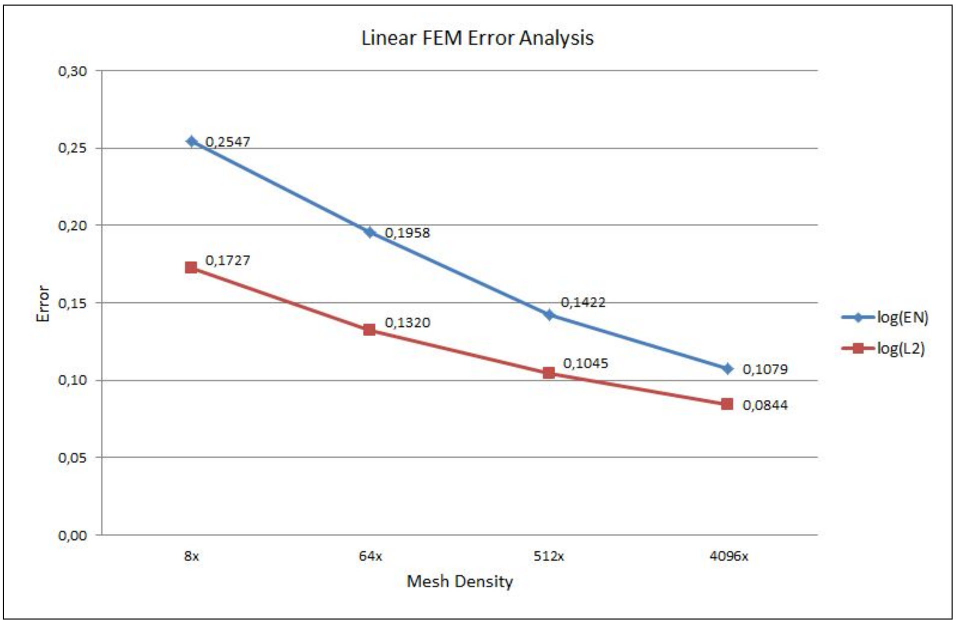

Figure 2.2: Linear FEM error analysis showing in Energy and L2 Norms.

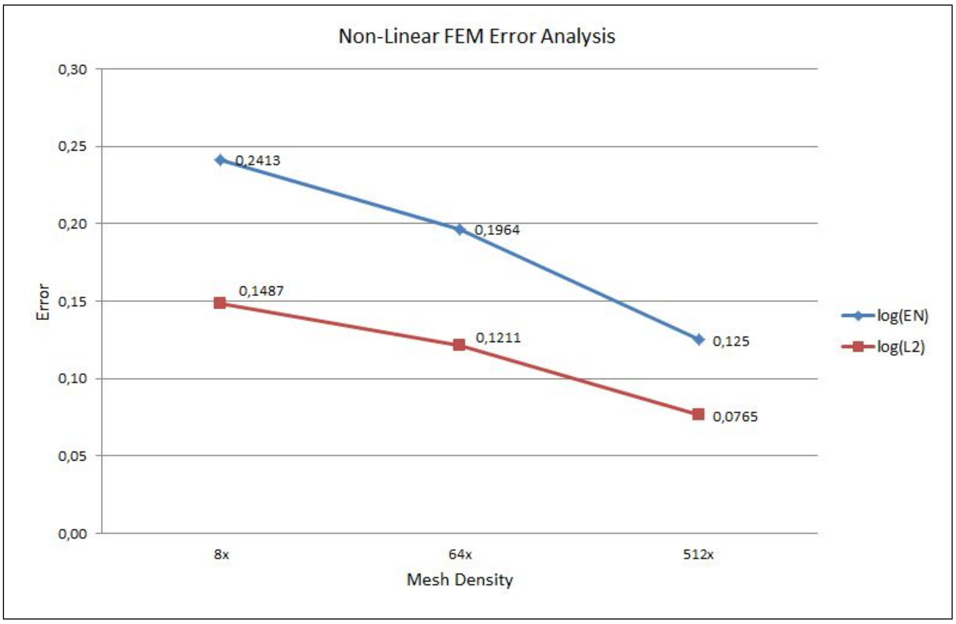

Figure 2.3: Nonlinear FEM error analysis showing in Energy and L2 Norms.