BME 676 - Computational Biomechanics

References: Ansys Help and Tutorial Guide
Finite Element Simulations Using Ansys, Esam M. Alawadhi

Links: http://ansys.com/

Instructor:
Dr. M. Roman
Fenster 696, x5270
max.roman@njit.edu
Office Hours: TBA

Grading:
Exam 1: 35%
Final Project: 35%
HW: 30%

Course Objectives:
This is an introductory course on the use of computational methods for solving design problems related to the biomedical engineering industry. The Finite Element Method will be introduced. Examples and problems will be solved both analytically and with the use of ANSYS. By the completion of the course students will be proficient with the use of ANSYS and familiar with the ANSYS help guide. The course will cover examples in structural analysis, fluid mechanics, dynamical problems (modal and harmonic analysis), and contact analysis. By the completion of the course students will know how to verify and interpret the computational results.

Course Structure
The course will be a combination of lecture and lab. Typically, the first half of the class will be a lecture to introduce new modeling concepts. The remainder of the class time students will work individually on the assigned exercise at a computer terminal or using their computer. Exercises will be due two weeks from the date assigned. ANSYS is available to use in the classrooms. However, it is highly recommended that students purchase a student license (cost: $25/year available through ANSYS Student Portal) for use on their personal computer. Use of a laptop is recommended but not required. In addition to ANSYS, some basic knowledge of MATLAB or other computer programming language is recommended. Instructor notes and tutorials will be given out each class and for each exercise.

Prerequisites
A background in engineering is required. Students should be familiar with concepts in BME 302 Introduction to Biomedical engineering II and/or BME 670 Introduction to Biomechanical Engineering or equivalent. This course is recommended for students in the biomechanics or biomaterials concentration and is suitable for BS/MS, MS, and PhD students.

Week 1:
Introduction to FEM method and structural analysis: Nodes, Elements, Global Stiffness Matrix. Using Matlab to solve for displacements, Verifying your results
Example Problem: Analysis of a thin plate subject to tensile load

Week 2:
Introduction to using Ansys, types of structural analysis with Ansys
Element types, CAD Modeling, Meshing, loads, solving, postprocessing,
ANSYS Workbench and APDL
Exercise 1: Analysis of a thin plate subject to tensile load (Workbench and APDL)

Week 3:
ANSYS LAB
Meshing (mapped vs free), *Element types*, selecting nodes, applying loads, verifying results
Exercise 2: Hex Key analysis with loads on free end.
.
Week 4:
Introduction to a composite structure. Gluing volumes, setting mesh attributes, global stiffness matrix for composite structures.
Exercise 3A: muscle-tendon complex. Composite structure subject to a tensile load.
Exercise 3B: Acetabular cup composite structure.

Week 5:
Torque/torsion, rotational displacements, shear stress. Global stiffness matrix, master node. Introduction to beam elements using Beam 189 and MPC 184 elements.
Example problem: composite beam/shaft fixed on both ends subject to a torsional load
Exercise problem 4: Analysis of thin beam with bending moment and force load using Beam 189 and MPC 184 elements

Week 6:
Introduction to shell elements using Shell 63 elements. Applying loads to shell elements, stress concentration.
Exercise 5A: Stress concentration, analysis of bone fixing plate with holes.

Week 7:
Introduction to modal and harmonic analysis. Dynamic analysis and fatigue. Time history post-processor
Exercise 5B: Modal and harmonic analysis of bone fixing plate with holes.
Exercise 5C: Dynamic analysis of circular membrane subject to shaking

Week 8:
Introduction to contact analysis.
Exercise 6: Attaching bone fixing plate to bone. Bending and torsion analysis of bone with and without bone fixing plate

Week 9:

First half of class: Exam 1-Midterm
Second half of class: Ansys Lab (use time to complete assignments from previous weeks)

Week 10:
Shell elements continued. Applying pressure load over curved surfaces
Exercise 7: Analysis of a test tube rotating in a centrifuge. Calculation of fluid force on test tube. Comparison of analytical and computational results.

Week 12:
Introduction to computational fluid dynamics (CFD) 2D and 3D, boundary conditions (pressure, velocity, walls), path operations, drag and lift force.
Example: 2D flow of microfluidic channel with wells for BIOMEMS device.
Exercise 8: Analysis of a catheter fluid check valve

Week 13:
Introduction to advanced topics for future study: fluid-structure interaction, unsteady/time dependent problems, boundary conditions,
Example: Artery Model with Bifurcating Y-channel
Complete all exercise problems. Discuss final projects

Week 14:
Ansys Lab: Projects

Week 15:
Ansys Lab: Projects

Final Exam Period:
Project presentations and project final report due.