FEA PROFESSIONAL (FEGN)

FEGN525. ADVANCED FEA THEORY & PRACTICE. 3.0 Semester Hrs.
This course examines the theory and practice of finite element analysis. Direct methods of deriving the FEA governing equations are addressed as well as more advanced techniques based on virtual work and variational methods. Common 1D, 2D, and 3D element formulations are derived, and key limitations examined. Matlab is used extensively to build intuition for FEA solution methods and students will create their own 2D FEA code by the end of the course. The commercial FEA software Abaqus is introduced with hands-on examples and Matlab solutions are compared to Abaqus for model validation.
Course Learning Outcomes

- Define DOF.
- Recall three different approaches for developing governing equations in FEA and list typical applications for each.
- Apply FEA governing equations to solve2D structural analysis by hand using symbolic math in Matlab.
- Explain and execute a mesh convergence study.
- Define the isoparametric element formulation and use shape functions to derive isoparametric elements for 2D and 3D applications.
- Recall numbers and locations of integration points for different element types.
- List and explain limitations of common 2D and 3D elements.

FEGN526. STATIC AND DYNAMIC APPLICATIONS IN FEA. 3.0 Semester Hrs.
This course emphasizes proficiency with commercial FEA software for solution of practical static, quasistatic, and dynamic structural problems. Common 1D, 2D, and 3D elements are examined in the context of linear solution techniques. Students will explore efficient methods for model construction and solution with commercial tools (the Abaqus FEA software). Emphasis will also be placed on verification, validation, and reporting standards for effective application of FEA software tools. Online course. Prerequisite: FEGN525.
Course Learning Outcomes

- Explain the difference between implicit and explicit solvers for static, quasi-static, and dynamic analyses.
- Compare the pros and cons of solutions obtained using implicit and explicit solvers for static, quasi-static, and dynamic analyses.
- Perform a 1D, 2D, or 3D structural analysis with or without symmetry (axi, cyclic).
- Request desired outputs from commercial FEA software and recall the difference between field and history output data types.
- Setup an FEA analysis to request desired output variables defined spatially and temporally.
- Use commercial FEA software pre-processor to visualize results from an FEA solution.

FEGN527. NONLINEAR APPLICATIONS IN FEA. 3.0 Semester Hrs.
This course explores common nonlinearities frequently encountered in structural applications of FEA. Students will gain proficiency in modeling geometric nonlinearity (large strains), boundary nonlinearity due to contact, and material nonlinearity (creep, rate dependence, plasticity, temperature effects, residual stress). The commercial FEA software Abaqus is used for hands-on experience. Online course. Prerequisite: FEGN526.
Course Learning Outcomes

- Recall and explain the three most common sources of nonlinearity in an FEA simulation.
- Perform an FEA simulation including large strains and finite rotations.
- Execute an FEA simulation including contact and compare several strategies for modeling contact interactions.
- Develop and apply nonlinear models for hyperelastic, viscoelastic, and elastic-plastic materials.
- Use an FEA simulation to compute residual stresses in a part following plastic deformation.
- Construct a clear report to communicate work performed for an FEA simulation.

FEGN528. FEA FOR ADVANCED DESIGN APPLICATIONS. 3.0 Semester Hrs.
In this course students will learn the automation tools and methods necessary for effective application of FEA on advanced design problems. Strategies for parametric analysis, performance optimization, and consideration of statistical uncertainty will be examined using Python scripting and commercial automation software. Online course. Prerequisite: FEGN526.
Course Learning Outcomes

- Apply Python scripting to automate parametric analysis of a part or assembly using commercial FEA software.
- Apply Abaqus Isight to automate parametric analysis of a part or assembly using commercial FEA software.
- Use Python scripting or other software tools to automate extraction and post-processing of results from commercial FEA software.
- Apply automation tools to perform optimization and probabilistic analysis using commercial FEA software.
- Construct a clear report to communicate work performed for an FEA simulation.