Finite Element Simulation Models for Mechanics of Materials

Dr. Shahnam Navaee, Georgia Southern University

Dr. Navaee is currently a Full Professor in the Civil Engineering and Construction Management Department in the Allen E. Paulson College of Engineering and Information Technology at Georgia Southern University. Dr. Navaee received his B.S. and M.S. degrees in Civil Engineering from Louisiana State University in 1980 and 1983, and his Ph.D. degree from the Department of Civil Engineering at Clemson University in 1989.

Dr. Junsuk Kang, Seoul National University

Dr. Junsuk Kang earned his Ph.D. degree in Structural Engineering in the Department of Civil Engineering at Auburn University, AL, USA in 2007. He obtained his master's degree in Structural Engineering from Korea University, South Korea, in 2000 and his Bachelor's degree was in Civil and Environmental Engineering from Korea University, South Korea, in 1998. Prior to entering PhD study, Dr. Kang worked as a Senior Civil Engineer in Hong Kong site and Seoul Headquarter of Hyundai Engineering and Construction Co., Ltd. during 2000- 2002. After his PhD study, he had taken many projects supported by ALDOT and Air Force Research Laboratory as a research associate at Auburn University during 2007 – 2011. Dr. Kang had been an assistant professor in the Department of Civil Engineering and Construction Management at Georgia Southern University during 2012-2016. Dr. Kang is currently an assistant professor in the Department of Landscape Architecture and Rural Systems Engineering at Seoul National University.

Finite Element Simulation Models for Mechanics of Materials

Abstract

In this paper the creation and utilization of a set of virtual models for complementing a Mechanics of Materials course in the Civil Engineering and Construction Management Department at Georgia Southern University is outlined and discussed. The simulated models are developed utilizing the Abaqus finite elements package. The models can particularly be useful in cases where a physical lab is not accompanying the offered course, as is the case in the authors' institution. Several examples of the developed simulations are provided in the paper to better illustrate the utility and significance of the models. The simulations for example can be used to determine and display the stress and deformation contours at various locations on the solid continuums having different geometries, boundary conditions, material properties, and loading conditions. The models are specifically developed to be used by the course instructor in illustrating and explaining some of the more important mechanics principles and concepts. These visual simulations help students better comprehend the course concepts and more easily understand the limitations and assumptions used in the classical formulation of mechanics problems. Some of the examples explored in the project include the analysis of axially loaded members, torque loaded shafts, bending of beams, combined loading of structural members, and pressurized thin-walled vessels.

As an added measure to further maximize the effect of the project and to creatively enhance the educational effect of the undertaken project for our program as a whole, the developed modules for the mechanics of materials are also planned to be utilized in a newly developed undergraduate-graduate finite element course offered in spring 2017. Obviously, the intent for utilization of these models in the FE course will be different than what is previously described for the mechanics course. In the FE course, the created examples are specifically used to illustrate the actual details and procedures that need to be followed to properly model and analyze a solid continuum. Using these examples, the students will be coached to develop the solution for other similar problems. The newly developed simulations can in turn be used in future offerings of the mechanics of materials course.

I. Introduction

Mechanics of Materials is one of the most important courses the students pursuing a civil engineering, mechanical engineering, and aerospace engineering degrees need to take in preparation to taking other higher level courses in their specific majors. This course mainly covers topics related to stresses and deformations, and discusses the behavior of various solid continuums subjected to a variety of loads. Among the most important topics included in the course are axial loads, torsion, bending, combined loading, deflection, and buckling. Included in the presented paper are six sample Finite Element models developed for the following problems to further complement the course.

(1) Analysis of a rectangular bar with hole and fillet subjected to an axial load
(2) Torsion of a circular shaft subjected to an applied torque
(3) Bending of a curved circular beam
(4) Analysis of a structural member subjected to combined loading

(5) Deflection of a continuous beam
(6) Analysis of pressurized cylindrical and spherical pressure vessels

Developed modules are for problems similar to the ones included in references[1-2]. These are the textbooks adopted for delivering a mechanics of materials and a structural analysis course at Georgia Southern University. The modules created in this project are specifically designed to further complement the mechanics of materials course. These modules can aid the students in better comprehending some of the key fundamental mechanics principles. When utilizing the developed models in a classroom setting, the instructor can utilize some of the special tools available in the ″visualization″ module of Abaqus to generate and display various results such as stresses and displacements at different locations on the analyzed part. Several of these visualization tools are shown and briefly discussed in a few of the sample problems included in the paper. Several other available features in Abaqus, enable the instructor to generate plots of various desired parameters and produce any needed output in a tabular format. The generation of this type of plots and reports can further promote student learning.

The FE models for more complicated problems such as the ones appearing in various structures resources[3-6], advanced mechanics texts[7-12], and finite element analysis resources[13-15] can be developed and utilized in upper level courses to further complement these offerings.

II. Axial Loading

A thin A-36 steel plate of 5 mm thickness with a circular hole and a fillet with the dimensions shown in Figure 1(a) is subjected to a uniform distributed tensile load of 20 MPa at the right end and supported by a fixed support on the left. The plate has respectively the modulus of elasticity (E) and Poisson's ratio (ν) of 200 GPa and 0.32. To aid in meshing the part, ten partitions were created on the model in Abaqus as shown in Figure 1(b). The partitions help in creating a finer mesh around the hole and in the vicinity of the fillet where the stress concentrations occur. The meshed model of the part is provided in Figure 1(c) showing the axial stress contour exerted on the plate. A CPS8R type element (An 8-node biquadratic plane stress quadrilateral, reduced integration element) was used in the analysis to produce the displayed results. Various tools in Abaqus allow the users to produce and display the stress distribution along any desired path. One sample created plot showing the distribution of axial stress along a section perpendicular to the loading axis and passing through the center of the hole is provided in Figure 1(d). This plot is used to determine the stress concentration factor K which can be used to yield the maximum normal stress at this section in accordance to Eq. (1). The section is marked in Figure 1(c). Obtained value of K from this analysis can be compared against the theoretical value of stress concentration factor published in various mechanics text[1].

$$\sigma_{\max} = K\sigma_{avg} \tag{1}$$

A similar distribution can be plotted along a vertical section perpendicular to the loading axis close to the location of the fillet. The value of K obtained from this analysis can also be compared against the theoretical values[1]. The distributions of parameter K corresponding to various ratios of r/h and w/h have been documented in various mechanics texts. If desired, more partitions and a further refined mesh can be used to generate more exact results. The developed model can further be used to demonstrate the fact that the normal stress (and normal strain) distribution away from the fixed support and away from the location of the hole and fillets are fairly uniform. This essentially confirms the Saint-Venant's Principle. Using a special tool available in the visualization module of Abaqus, values of stress or displacements at various locations on the part can be probed to further educate the students regarding the true behavior of axially loaded structural members.

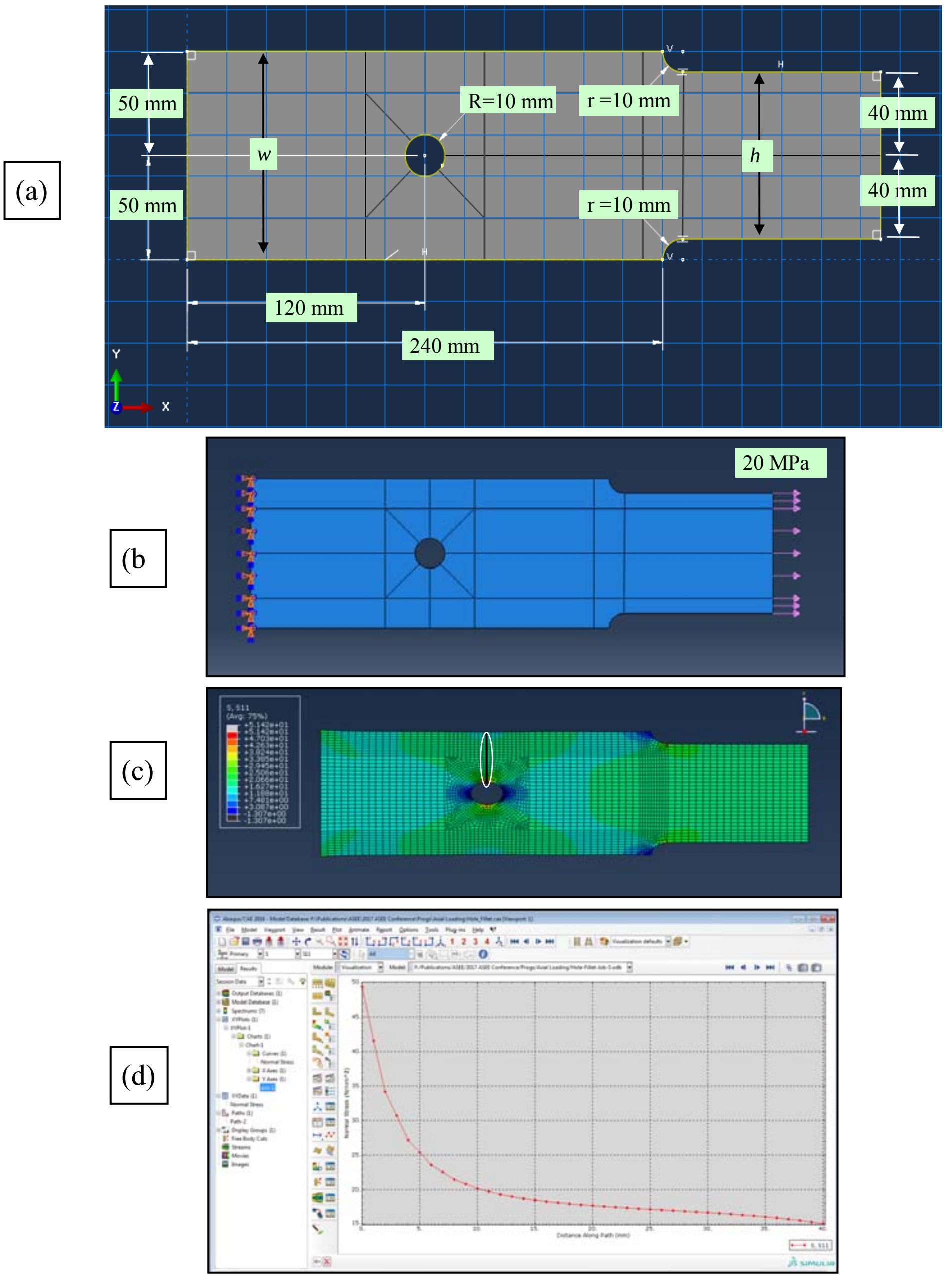

Figure 1. Model of Plate with Hole & Fillet Subjected to an Axial Load: (a) Plate Constructed in Abaqus Sketcher, (b) Created Partitions for Meshing, (c) Axial Normal Stress Contour Plot, (d) Axial Normal Stress Distribution Across the Plate Width at the Hole Location

III. Torsion

To investigate the behavior of torque loaded members, a model of straight A-36 steel shaft with a circular cross section subjected to an end torque was developed in Abaqus. This model has been provided in Figure 2(a). The applied torque was emulated by subjecting the shaft to a couple at its end. The radius (*r*), length (*L*), shear modulus (*G*) of the shaft, and the applied couple moment (*T*) were respectively: 1.25 in, 20 in, 11×10^3 ksi, and 8.75 kip.in. Observed in Figures 2(a) are also the six partitions that were created on the model using six previously created datum planes. These partitions help in meshing the model and obtaining the results in the desired manner. When developing the model for this example, a C3D8R type element (an 8-node linear brick, reduced integration element) was used to produce the results. The contour for the shear stress developed in the shaft is provided in Figure 2(b). Again, the stress value at any particular location on the member can be obtained using the probing tool in Abaqus. One sample probed element on the boundary of the shaft can be seen in Figure 2(b). The stress and angle of twist values obtained from the developed model can be compared against the theoretical values calculated using the following fundamental equations.

$$\tau = \frac{Tr}{J} \qquad \phi = \frac{TL}{Gj} \tag{2}$$

Where in these equation, *T*, *r*, *J*, *L*, and *G* are respectively: the applied torque, shaft radius, polar moment of inertia of shaft's cross section, shaft length, and shear modulus. This confirms for the students that the distribution of shear stress along any radial line on the cross section of a circular shaft is linear, being zero at the shaft's center and maximum at the outside boundary of the shaft. Also that the angle of twist varies linearly along the length of the shaft.

Other similar models can be developed for non-circular members such as squares, triangles, and elliptical cross sections to verify the documented values for each case. For example, when dealing with a square cross section, the model can be used to verify the following theoretical formulas[1].

$$\tau_{\max} = \frac{4.81T}{a^3} \qquad \phi = \frac{7.10TL}{a^4 G} \tag{3}$$

Where in these equation, *a* is the length of the square side. The model can further be used to establish that the maximum shear stress occurs in the middle of each side of the square cross section, and also that the shear stress is zero at the four corners. The students can additionally verify that the square cross section also actually warps due to the applied torque.

IV. Bending

A model of an initially curved beam subjected to a bending moment was also developed in this project to investigate the behavior of this structural member subjected to this type of loading. The normal stress results acting on a cross section of the model can be compared against the theoretical values obtained using the following theoretical equation[1].

$$\sigma = \frac{M(R-r)}{Ar(\bar{r}-R)} \qquad \text{, where } R = \frac{A}{\int_A \frac{dA}{r}} \tag{4}$$

The terms in the above equation are defined as shown below.

σ: Normal stress acting on the beam's cross section
M: Internal moment about the neutral axis of the cross section (positive if it increases beam's radius of curvature)
R: Distance from the center of curvature to the neutral axis
$\bar{r}$: Distance from the center of curvature to the centroid of the cross section
r: Distance from the center of curvature to the point where the normal stress is to be computed
A: Cross-sectional area of the beam

Note that for a composite beam consisting of rectangular component parts, the equation for R can be defined as shown below[1].

$$R = \frac{\sum A}{\sum b \ln \frac{r_2}{r_1}} \tag{5}$$

In Eq. (5); b, r_1, and r_2 are respectively the width of the rectangle, and the minimum and maximum perpendicular distances measured from the nearest and farthest parallel edges of the rectangular cross section to the center of curvature of the curved beam.

The model of a curved beam which initially has a circular shape is presented in Figure 3. A screenshot of the beam's cross section created in Abaqus sketcher is provided in Figure 3(a) to show various dimensions for the cross section. The solid model of the beam generated through a revolving action has been presented in Figure 3(b). Also seen in this Figure are the fixed boundary condition for the beam, along with the applied loading, and the seven partitions which were used to mesh the part. A C3D20R type element (A 20-node quadratic brick, reduced integration element) was used in this example to mesh the model. The normal stress contour for a cut section of the model is provided in Figure 3(c). The cut was made to clearly show the stress values acting on a cross-section of the beam along the x-direction perpendicular the y-z plane. These stresses can be compared against the values obtained using the theoretical equation provided above (Eq. 4). Also shown in Figure 3(c) is the stress result for one sample probed element on the cross section obtained using a special tool in the visualization module of Abaqus. Models similar to this can be developed for other curved members with various other cross sections. The equations for bending stress of curved beams with triangular, circular, and elliptical cross sections can be found in various mechanics textbooks[1].

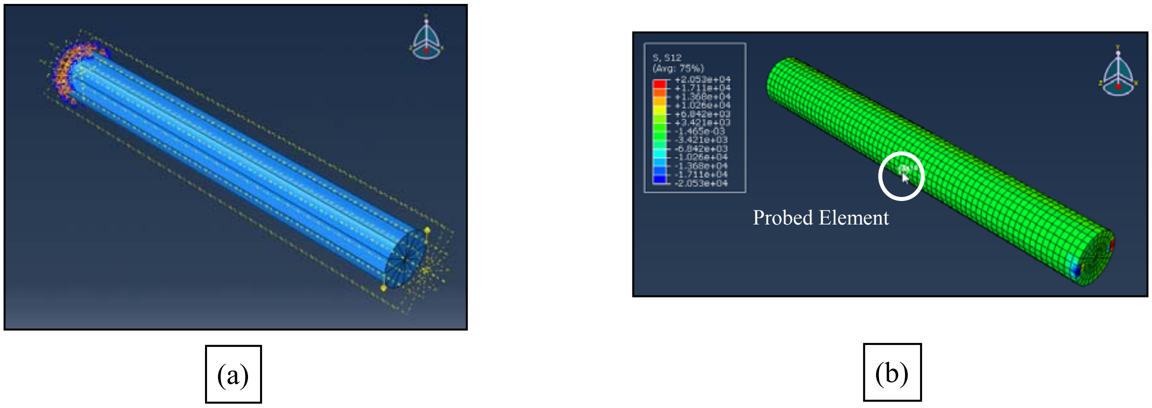

Figure 2. Model of Shaft with Circular Cross Section Subjected to an End Torque: (a) Created Partitions for Meshing, (b) Shearing Stress Contour Plot

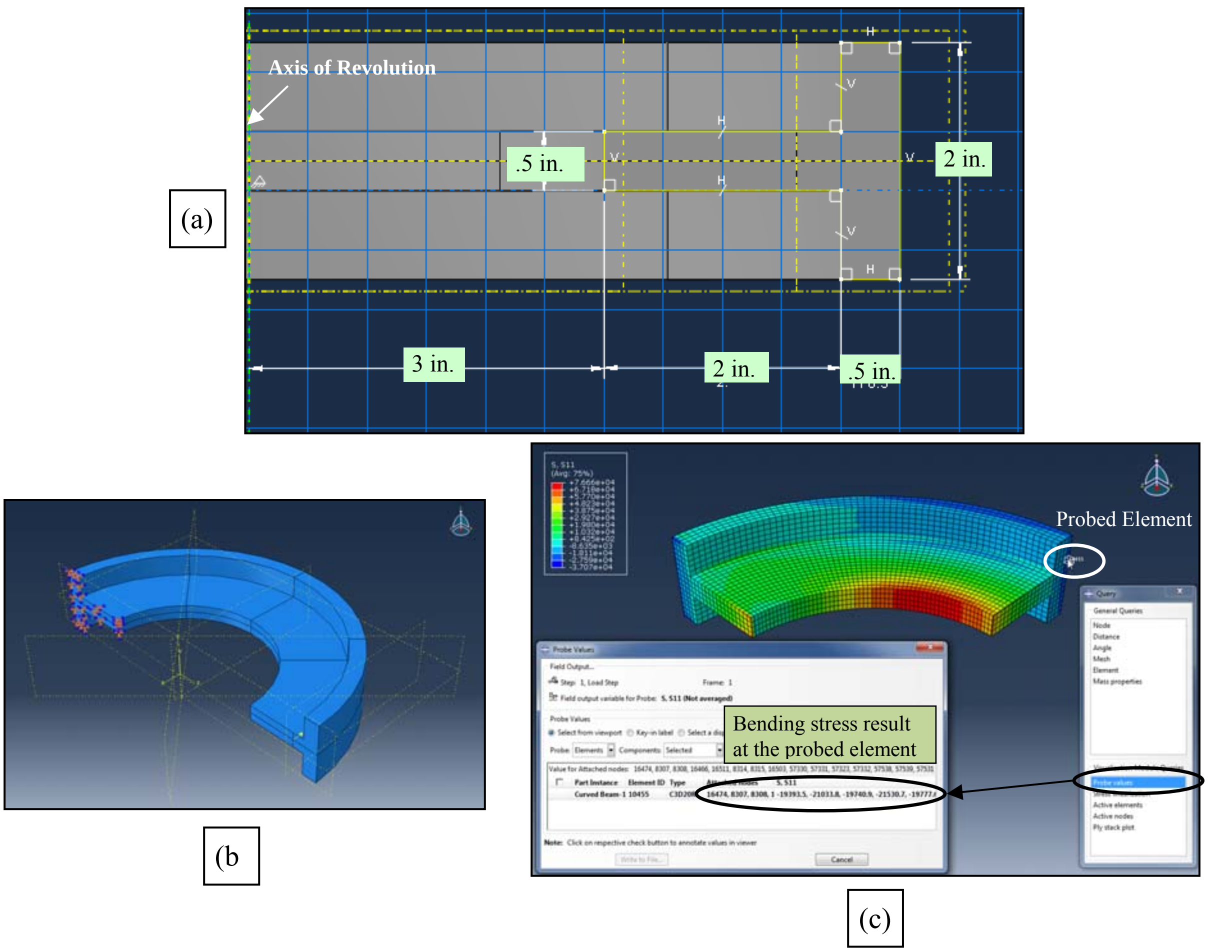

Figure 3. Model of a Curved Beam Subjected to Bending: (a) Beam's Cross Section Constructed in Abaqus Sketcher, (b) Loading, Boundary Condition, and Meshing Partitions, (c) Normal Stress Contour Plot

V. Combined Loading

A sample model for a bent A-36 steel rod with a circular cross section of radius 0.5 in. subjected to three concentrated end loads along the x, y, and z directions was also developed in the project, so that it can be analyzed. The model is provided in Figure 4(a). Also seen in this Figure are the five partitions used in meshing the part. Two sample contour plots for the stresses are also provided in Figure 4(b) and 4(c). The first contour plot shows the variation of the normal stress along the x-axis, while the second displays the variation of the Von-Misses stress. The stresses in the model were obtained using a C3D8R type element. The parts of the model which are more severely stressed can be seen on the stress contour presented in Figure 4(c). The Von-Mises stress contour can be used to make sure that the yield stress in the material is not exceeded. Using this type of analysis the normal and shear stresses at any cross section on the structural member can be determined and compared against the theoretical stress values obtained using Eq. (6).

$$\sigma_x = -\frac{M_y z}{I_y} + \frac{M_z y}{I_z} + \frac{N}{A} \qquad \tau_{xy} = \frac{V_y Q_z}{I_z b_z} \pm \frac{Tc}{j} \qquad \tau_{xz} = \frac{V_z Q_y}{I_y b_y} \pm \frac{Tc}{j} \tag{6}$$

The terms used in the above equation set are defined as shown below.

M_y and M_z: Moments about the y and z axes at the cross section
N and T: Normal force and torque acting on the cross section
V_y and V_z: Shear force in the y and z-directions at the cross section
c and A: Rod's radius and cross-sectional area
y and z: Perpendicular distance from the point where the stress is to be calculated to the z and y axes
I_y, I_z, and J: Moments of inertia about the y and z axes, and the polar moment of inertia of the cross section
Q_y and Q_z: First moment of area at the section where the shear stress is to be calculated with respect to the y and z axes
b_y and b_z: Width at the section where the shear stress is to be calculated along the y and z-axes

The loads acting on the rod can easily be altered and applied in various load combinations to generate the results for other cases. Additionally, models for analyzing other structural components subjected to variety of other combined loading cases can also be developed and studied.

VI. Beam Deflection

Beam deflection is another main topic traditionally covered in an elementary mechanics of materials course. One sample developed model for analyzing the deflection of a continuous beam is presented in Figure 5. The steel beam considered is supported by a pin and two rollers and subjected to a uniform distributed and a concentrated load. Included in this figure is the deflected configuration of the beam which has been generated using a B22 type element (a 3-node quadratic beam element). The deflection values determined using Abaqus can be compared against the results obtained using a variety of available classical approaches. In the mechanics of materials course offered at Georgia Southern University, the double-integration method is mainly discussed in detail. When using this approach the differential equation of the beam (Eq. 7) is utilized.

$$EI \frac{d^2 v}{dx^2} = M(x) \tag{7}$$

In the above equation, v is the transverse deflection and $M(x)$ is the bending moment. The E and I terms are as defined previously. Using the visualization module of Abaqus, the students can see the shape of the elastic curve and better understand the behavior of beams when subjected to various loads and boundary conditions. The elastic curve produced in Figure 5(a) specifically illustrates how the moment (or the curvature) is changing sign along the length of the beam, indicating the portions of the beam which are subjected to tensile and compressive bending stresses. The distribution of the moment along the length of the beam obtained using Abaqus is provided in Figure 5(b). This distribution can be used to determine the location of the inflection points of the beam. Several other FE models designed for analyzing a variety of other structural members such as trusses, beams, two and three-dimensional frames, as well as, several other solid structural members were included in a separate publication of the authors[16]. The models provided in that publication were specifically designed for enhancement of a finite element course offered in the civil engineering curriculum.

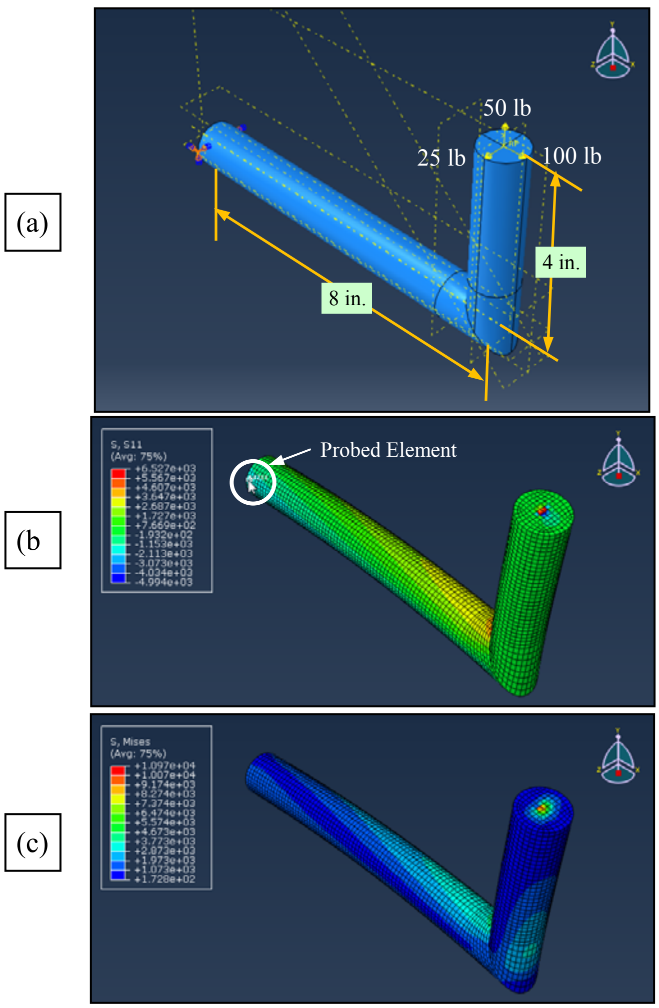

Figure 4. Model of a Bent Structural Component with Circular Cross Section: (a) Loading, Boundary Conditions, and Meshing Partitions, (b) Normal Stress Contour Plot, (c) Von-Mises Stress Contour Plot

(a)

(b)

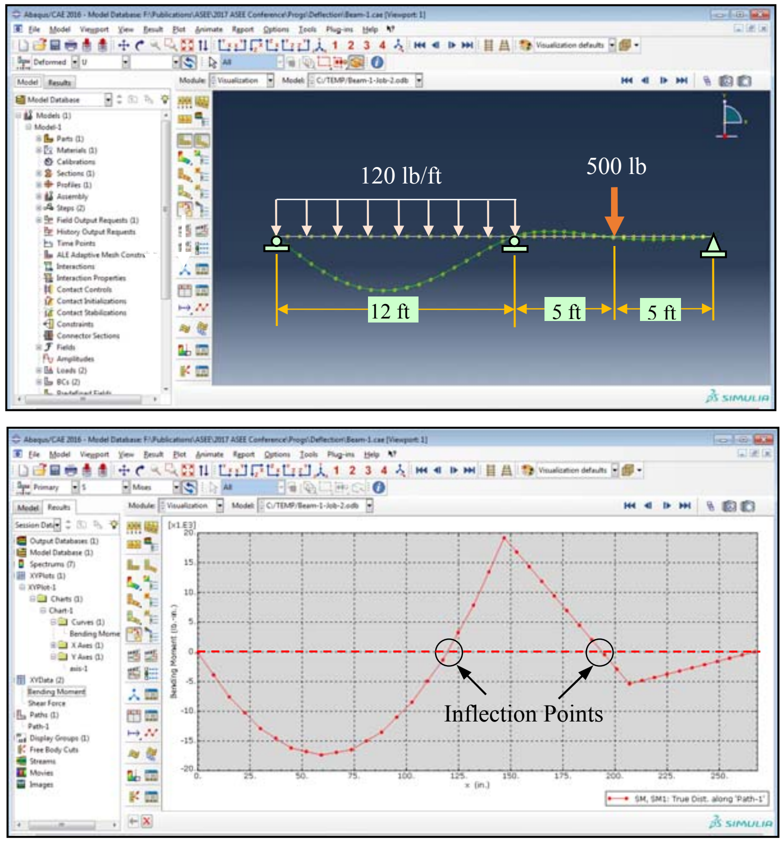

Figure 5. Model of a Deflected Continuous Beam Subjected to a Distributed and Concentrated Load: (a) Deflected Shape of the Beam, (b) Bending Moment Distribution Diagram

VII. Pressure Vessels

A sample model of a thin-walled steel cylindrical and a spherical pressure vessel were also developed in this study for determining the principal stresses in such vessels. The generated models are provided Figure 6. The vessels both have an inside radius of 3 ft and a thickness of 0.5 in. The height of the cylindrical model is 8 ft. Depicted in Figures 6(a) and 6(d) are the partitions that were created for meshing the two parts. To generate the results for these examples, a S8R element (an 8-node quadratic doubly curved reduced integration shell element) was utilized. The stress contours for circumferential and longitudinal normal stresses are shown in Figure 6(b-c) and Figure 6(e-f), along with a sample probed element for each case. The horizontal and vertical planar cuts in these figures are used to better visualize the stress results acting on the cut surfaces. The values for the circumferential and longitudinal stresses can be compared against the results obtained using the theoretical equations[1].

$$\sigma_c = \frac{pr}{t} \qquad \sigma_l = \frac{pr}{2t} \tag{8}$$

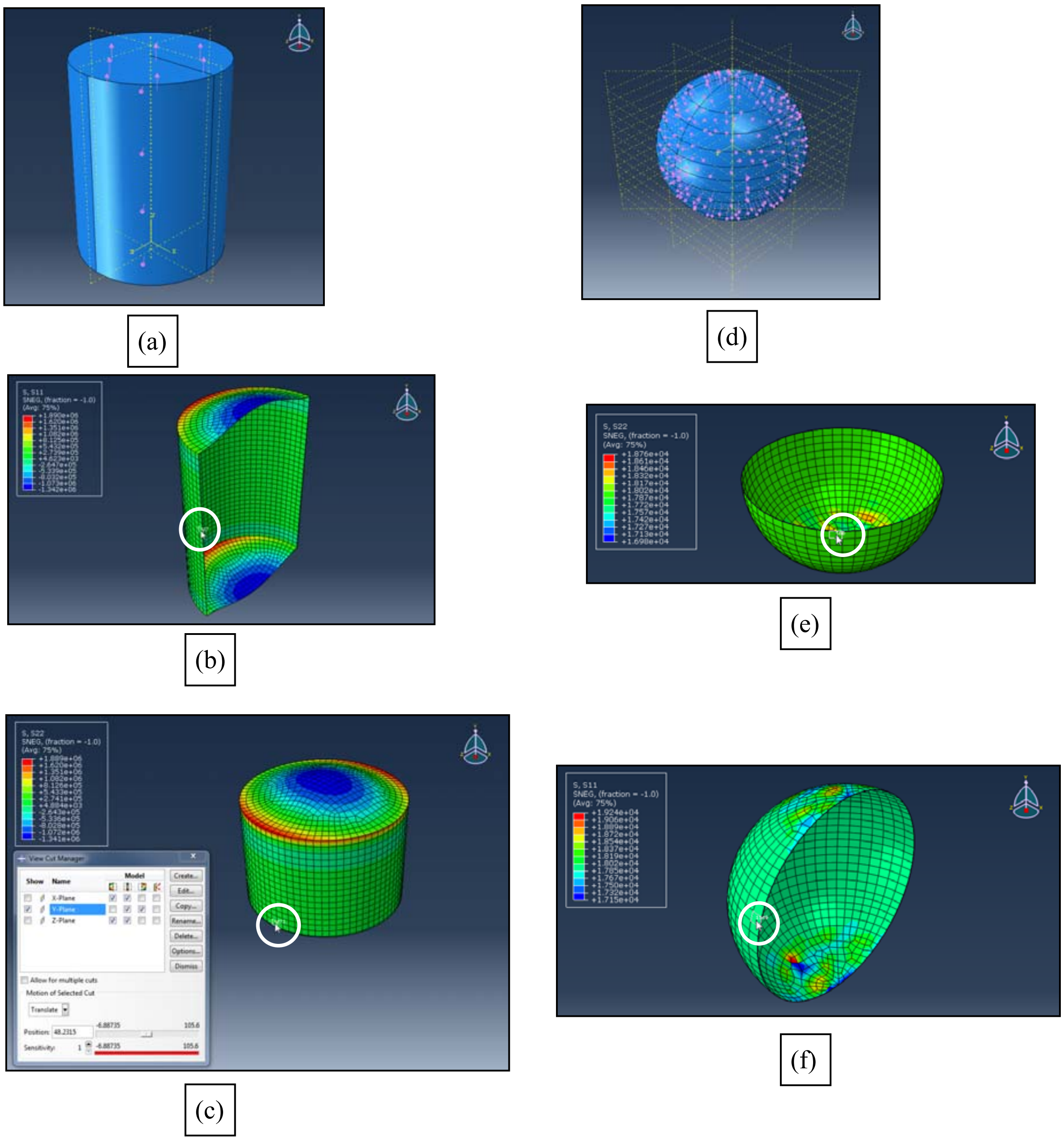

Figure 6. Pressurized Cylindrical and Spherical Vessels: (a) Developed Meshing Partitions for a Cylindrical Vessel, (b-c) Contour Plot of Circumferential and Longitudinal Stresses in a Vertical and Horizontal Cut Section of a Cylindrical Vessel, (d) Constructed Meshing Partitions for a Spherical Vessel, (e-f) Contour Plot of Circumferential Stress in a Vertical and Horizontal Cut Section of a Spherical Vessel

VIII. Summary & Conclusion

In the presented paper, the development of models produced utilizing Abaqus for complementing a mechanics of materials course was included and discussed. The created modules can be used in a classroom setting to further enhance the students' understanding of the course topics and provide a better learning environment for students. Included in the paper were six examples to clearly establish the value and utility of the developed models. The simulated models cover some of the more important topics typically covered in an introductory mechanics of materials courses. In the created instructional modules, the geometric details and size of various features of structural members, including details regarding the boundary conditions and loading conditions, as well as the material properties of components can easily be adjusted to create the solutions for a wide range of other problems. This adjustments may not be possible through using other canned programs. This is perhaps one of the main advantages of the created modules. The models will be used by the main author to complement a section of the mechanics of materials course offered in fall 2017. These models will additionally be made available to other instructors who may be interested in using them.

Models similar to the ones discussed in this paper can be developed to analyze other more complicated structures discussed in advanced courses. Some examples can include the analysis of structural components subjected to various dynamic loading conditions, investigation of the behavior of various composite materials when used in different settings, and analysis of contact problems involving solid bodies.

As previously stated in the paper, to further enhance the impact of the project, the main author is also planning to use the developed modules in a finite element course offered at Georgia Southern University in spring 2017. The purpose for this utilization is to further enhance the students' knowledge regarding the utility of various element types and meshing techniques that can be incorporated when analyzing solid components. The students in the course can additionally be coached to develop FE models for other more advanced topics.

Bibliography

1. Hibbeler, R.C., Mechanics of Materials, Ninth Edition, Pearson Prentice Hall, 2014.
2. Hibbeler, R.C., Structural Analysis, Ninth Edition, Pearson Prentice Hall, 2012.
3. McCormac, J., Elling, R.E., Structural Analysis (A Classical and Matrix Approach), Harper Collins, 1988.
4. Nelson, J.K., McCormac, J.C., Structural Analysis: Using Classical and Matrix Methods, Third Edition, John Wiley & Sons, 2002.
5. Hsieh, Y., Mau, S.T., Elementary Theory of Structures, Fourth Edition, Prentice Hall, 1995.
6. Wang, C.K., Statically Indeterminate Structures, McGraw Hill, 1953.
7. Ugural, A.C., Fenster, S.K., Advanced Strength and Applied Elasticity, Elsevier, 1975.
8. Wang, C.T., Applied Elasticity, McGraw Hill, 1953.
9. Timoshenko, S. Goodier, J.N., Theory of Elasticity, Third Edition, McGraw-Hill, 1934.
10. Timoshenko, S. Gere, M.G., Theory of Elastic Stability, Second Edition, McGraw-Hill, 1961.
11. Timoshenko, S., Woinowsky-Krieger, S., Theory of Plates and Shells, Second Edition, McGraw-Hill, 1959.
12. Jones, R.M., Mechanics of Composite Materials, McGraw-Hill, 1975.
13. Logan, D.L, A First Course in the Finite Element Method, Fifth Edition, Cengage Learning, 2016.
14. Cook, R.D., Malkus, D.S., Plesha, M.E., Witt, R.J., Concepts and Applications of Finite Element Analysis, Fourth Edition, John Wiley & Sons, 2002.
15. Reddy, J.N., An Introduction to the Finite Element Method, Third Edition, McGraw Hill, 2006.
16. Navaee, S., Kang, J., Enhancement of a Finite Element Analysis Course for Structural Engineering, Computers in Education Journal, Vol. 7, No. 4, Oct.-Dec. 2016.