Paper ID #36576

Simulation Models for Analysis of Structures

Dr. Shahnam Navaee, Georgia Southern University

Dr. Navaee is currently a full professor in the Department of Civil Engineering and Construction in the Allen E. Paulson College of Engineering and Computing at Georgia Southern University. Dr. Navaee received his B.S. and M.S. degrees in Civil Engineering from Louisiana State University, and his Ph.D. in Engineering from Clemson University. Prior to his current faculty position, Dr. Navaee served in several administrative rolls. One year as the Interim Chair of the Dept. of Civil Engineering and Construction Management, and seven years as the Associate Dean of the College of Science and Technology at Georgia Southern University. Dr. Navaee's main research interests are in the area of solid mechanics and structures.

Simulation Models for Analysis of Structures

Shahnam Navaee and Kristin Ackerman
Georgia Southern University

Abstract

In the described research, simulation models for analyzing a variety of structural problems were developed utilizing the *Abaqus* FEA software package. The produced *Abaqus* "*cae*" files and associated *AVI* animation files produced in the study are planned to be used to better clarify some of the more important structural engineering topics and further enhance an introductory Structural Analysis course in the *Civil Engineering and Construction Department* at Georgia Southern University. To establish the significance of the project, sample produced simulation models for some of the studied structures are provided and discussed in the paper. By interacting with the produced models during the lectures, the instructor can effectively enhance the students' understanding of the behavior of structures subject to various loads and further promote student learning. Included in the paper is also the student assessment of the effectiveness of the produced models in complementing the Structural analysis course.

Keywords

Development of Instructional Modules, Analysis of Structures, Finite Element Analysis

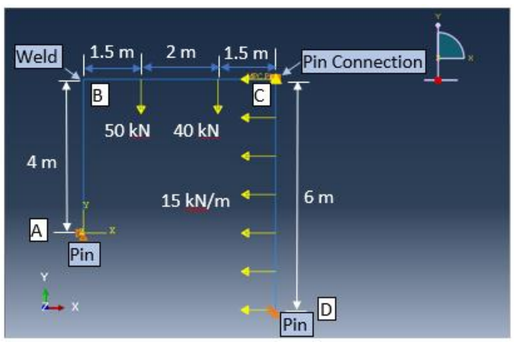
I. Introduction

There is a large volume of published literature on various instructional tools and techniques that could be employed for further enhancement of classroom instructions and elevation of student learning. Four examples of these types of publications, related to complementing a structural analysis course, are included in the list of provided references[1-4]. In these sample publications, the utilization of Mathcad, SolidWorks/Microsoft PowerPoint animations, *SkyCiv* FEA software, and Java Applets was respectively explored. The corresponding author of the paper has also discussed some of his efforts for enhancement of a structural analysis course in several publications[5-8]. In these publications, the application of MATLAB, Flash, Excel, and LabVIEW was respectively discussed. In a later publication of the corresponding author[9], the utility of *Abaqus* in promoting student learning in structural engineering related courses was suggested. In the current presented paper, the development of *Abaqus* FE simulation models for supplementing the classroom instructions in an introductory structural analysis course offered at Georgia Southern University is included and discussed for covering the main topics specifically included in this course. The models were created with assistance from an honor student who is serving as the co-author of the paper. Additional effort was made in the project to create the models in a flexible form so that they could easily be modified during classroom presentations to develop and discuss the solution of variety of problems. In the created models, in addition to the loads, supports and material conditions, various connections used in the structure could also be altered with ease. In the presented paper, sample models for analysis of a statistically determinate frame containing connected members, two truss structures, and two indeterminate building frames subjected to vertical and lateral loads are included to better describe the project. The investigated structures were all selected from R.C. Hibbeler's Structural Analysis text[10], since this source is used to deliver the course at Georgia Southern university.

II. Analysis of Structural Frames Subjected to Static Loads

A two-dimensional frame is composed of two columns and a beam as shown in Fig. 1 (a). The right column is subjected to a lateral uniform distributed load, and two vertical concentrated loads are applied to the beam at the top. At point B, column AB is rigidly connected to beam BC, allowing moment transfer between the beam and the column members. At point C, column CD is pinned to the beam, so no moment is transferred. A special "Connector Section" had to be created in *Abaqus* to establish the proper connection between column CD and beam BC as depicted in Fig. 1 (b). The resulting shear and bending moment diagrams for each member of the frame, generated from the developed model, are shown in Fig. 2. When generating the distributions in this figure, the shear and moment variations were plotted along the length of each member; for columns respectively from points A, D, and for the beam from point B. The produced results are almost identical to the expected analytical plots[10].

(a)



(b)

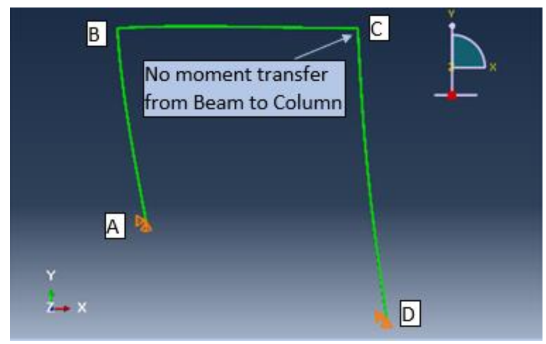
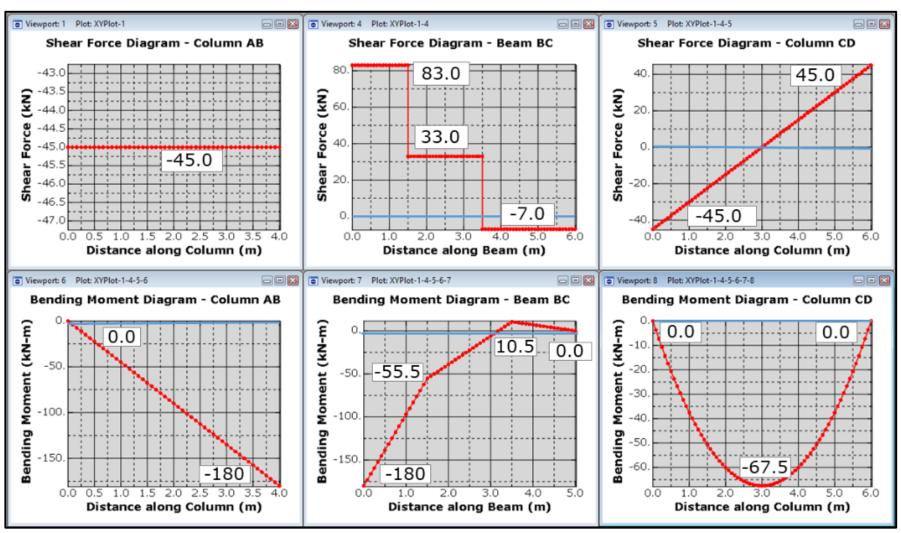


Fig. 1. (a) Two-Dimensional Structural Frame, and (b) Deformed Shape Under Load Application



(a) (b) (c)

Fig. 2. Shear and Bending Moment Diagrams Along the Three Segments of the Frame

Using the developed model, the instructor can apply the loads gradually using animation in *Abaqus* and fully discuss the specifics related to the deformation of the structure subjected to applied loads. During the animation phase, the zooming tool can effectively be used to show the details of the deformations of each part. Students often have a difficult time grasping the effect of various connections of the structure in the overall response of the structure. The problem was assigned as homework in the course and the students were asked to develop the hand-solution for this statically determinate structure on paper. The task required the students to draw the free-body diagrams of the detached parts, solve for the support reactions, determine the applied internals forces and moment at the end of each detached part, and finally construct the shear and bending moment diagrams for all members. Using the created model, the students can check the validity of their solution and clearly see the details related to the deflected shape of the frame. During the presentation, the instructor also has an option to change the load and support conditions in the model and create other scenarios for discussion. This will further enhance the students' understanding of the internal reactions developed in various parts of the structures due to applied loads.

III. Analysis of Portal Trusses

A portal truss is subjected to two lateral loads as illustrated in Fig. 3. The two vertical side columns are continuous members, pin-connected to the truss members. To properly model the structure, two column sections and a truss section (containing all two-force members) had to be created and joined using a special "Connector Section" in *Abaqus*. This connector will prevent moment transfer from the truss members to the columns. Using the developed simulation model, the resulting forces in the truss members were determined and found to be very close to the theoretical values[10]. Included in Fig. 3 are the axial load values in each truss member. Provided in the figure is also the deformed shape of the portal frame, illustrating that the truss members used in the structure help in keeping the top section of the columns in nearly vertical position under the applied loads. Using the developed model, the shear and bending moments of the columns can also be produced if desired.

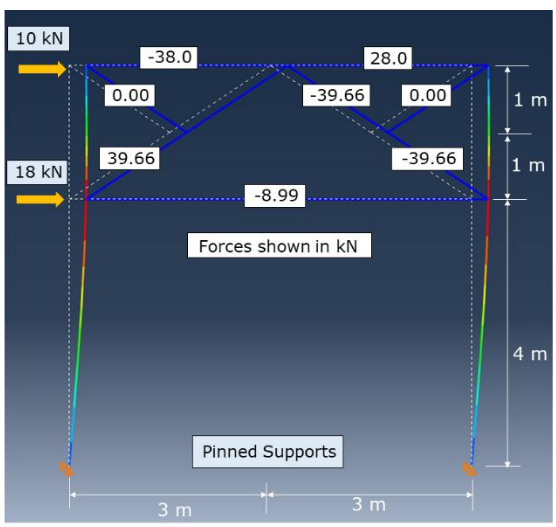


Fig. 3. Portal Truss Subjected to Lateral Loads

Analysis of truss portals with pinned and fixed supports is also another topic covered in the offered Structural Analysis course. In the course, the students are expected to solve this problem using a classical method of analysis, by assuming that the horizontal reactions at the base supports are equal and proceed to determine the internal forces in each truss member using either the *Method of Joints* or *Method of Sections*. The determination of the truss forces will assist the students to calculate the applied forces acting on the vertical column members and consequently be able to construct the shear and moment diagram for the columns. Using the produced model, the students can verify the validity of the assumption made in the theoretical solution. The effect of the lateral forces on the created model can further be investigated by changing the pinned supports to fixed supports.

IV. Analysis of Trusses Subjected to Mechanical Loads and Temperature loads

In the offered course, the determination of the displacement of various points on the structure is also discussed using several approaches including the energy method. One sample model created in the project for analysis of a truss is provided in Fig. 4. One of the members of this truss (the vertical member AD) is subjected to a temperature increase of 120˚F and the truss is further acted upon by a horizontal load at C and a vertical force at joint B. Two different cross-sections were used in the truss as indicated in Fig. 4. Using the developed simulation model, the joint displacements of the truss can be determined and compared against the theoretical values obtained using the classical methods of analysis. The vertical displacement values of truss joints are shown in Fig. 4, along with the deformed shape of the truss. The joint displacements computed using the developed model were all almost identical to the displacements calculated using the *Virtual Work* method. The general equation for determination of the truss joint displacements using this method is provided in Eq. 1.

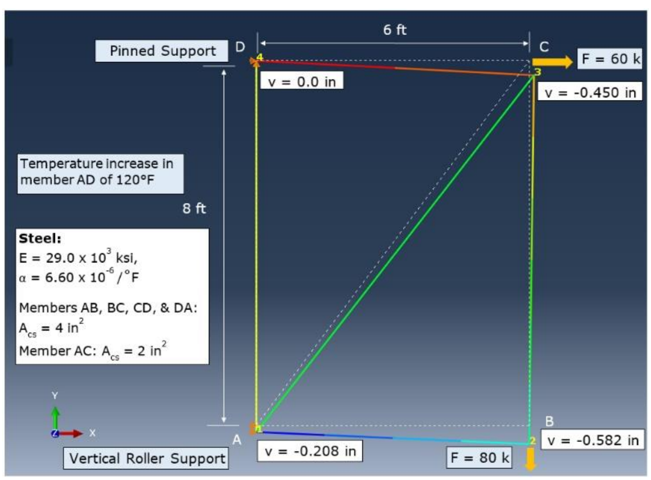


Fig. 4. Truss Experiencing a Temperature Increase

$$1 * \Delta = \Sigma \frac{nNL}{AE} + \Sigma\, n\, \alpha\, \Delta T L \quad (1)$$

In this equation “1” represents the unit virtual load applied to the truss joint in the direction of the sought joint displacement Δ. The terms n, N, L, A, E are respectively the virtual axial member load, real axial member load, member length, member cross-sectional area, and Young's Modulus. The terms $\alpha, \Delta T$ are respectively the coefficient of thermal expansion, and change in temperature. The model for this truss is developed in a flexible form in which the applied temperature loads, and other applied mechanical loads can easily be modified to generate the solution for a variety of other problems. Additionally, if desired the tensile and compressive stresses in the truss members can also be computed for any prescribed cross-sectional area assigned to the members.

V. Analysis of Multistory Frames Subjected to Distributed Vertical Loads

The two-dimensional analysis of building frames subjected to uniform vertical loads was also included in the project. In the offered introductory Structural Analysis course, a simplified approach for analysis of this type of frame is discussed using an approximate solution by assuming that the inflection points in the frame girders are located at predetermined locations on that girder. In one common accepted practice, the inflection points are assumed to be positioned at a set distance of one tenth of the girder length measured from each girder end, where this member is connected to the two side columns. A sample model for analysis of a building frame developed in the project is provided in Fig. 5, along with the bending moment distribution for the top girder of the structure in Fig. 6 (a). Using this distribution, a more accurate location for the inflection points can be obtained. The variation of this moment distribution is also provided in Fig. 6 (b) and compared against the distributions obtained using the simplified approach used in the course. Obviously, the bottom lower girders and the columns of the structure can also easily be analyzed using the produced model, if desired. Using this model, the instructor can more effectively explain the effect of the common assumptions made in the real solution of the discussed problem.

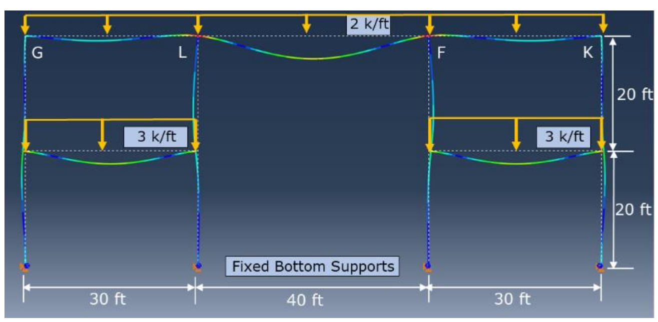


Fig. 5. Multistory Frame Subjected to Vertical Distributed Loads on its Girders

(a) (b)

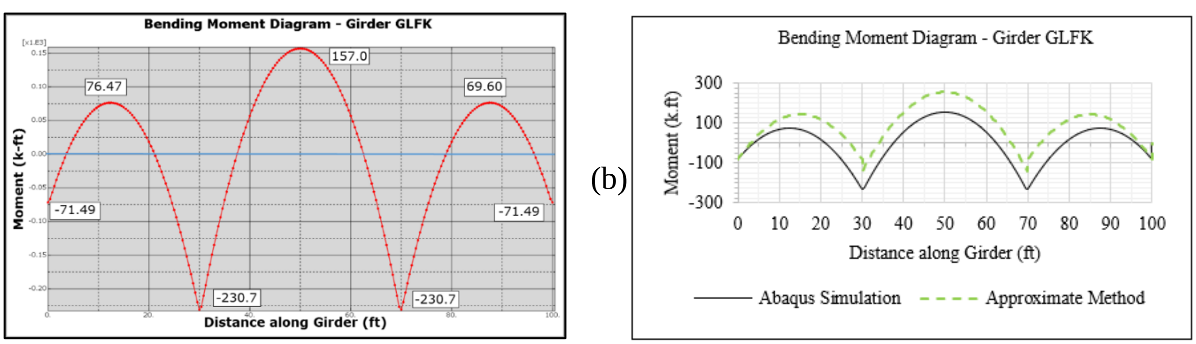


Fig. 6. (a) Bending Moment Diagram of the Top Girder Obtained using FE Analysis, and (b) Bending Moment Comparison Plots for the Simulated and Approximate Method Results

VI. Analysis of Multistory Portal Frames Subjected to Lateral Loads

Analysis of building frames subjected to lateral forces is also investigated in the course using the *Portal Method* and *Cantilever Method*. There are several assumptions that are made in each of these two simplified approximate methods of analyses. When using the *Portal Method*, it is commonly assumed that the inflection points (points of zero moments) are located in the middle of all columns and girders of the structure. It is further assumed that the shear force in each interior column is twice as large as the shear in each exterior column. In the *Cantilever Method* of Analysis, aside from assuming that there are inflection points in the middle of columns and girders (similar to the *Portal Method*), it also assumed that the normal stress in the columns is linearly proportional to the distance measured from the centroid of the cross-sectional areas of the columns at any given floor level. In the described project, the building frame presented in Fig. 7 was analyzed using a FE model. The investigated structure had four fixed supports at the bottom and was subjected to three lateral loads as depicted. The distributions of the bending moment along the top girder and the left column, obtained using the performed FE analysis are provided respectively in Figs. 8 (a) and 8 (b). Using these distributions, a more accurate location for the infection points of the girder and column and moments in each of the members can be obtained. The variations of these moment distributions are provided in Fig. 9 and compared against the distributions obtained using the two approximate methods of analyses discussed in the course. It should be stated that other results such as support reactions can also easily be obtained during the classroom presentation using the probing tool in *Abaqus* to show how these values compare against those obtained using the approximate methods. Utilizing the developed FE model, the instructor can better discuss the deformation of the portal building and the effect of the common approximate methods of analyses on the problem solution. The produced model can additionally be used to familiarize the students with other available classical methods of analyses such as the *Slope Deflection Method* and *Moment Distribution Method*.

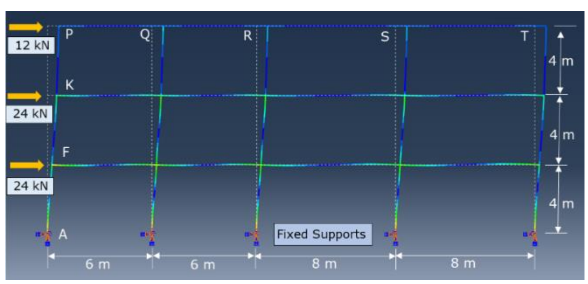
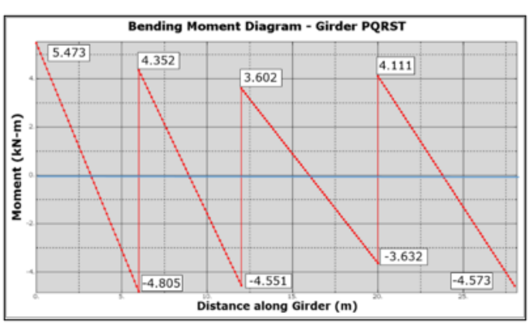


Fig. 7. Multistory Frame Subjected to Lateral Loads

(a)



(b)

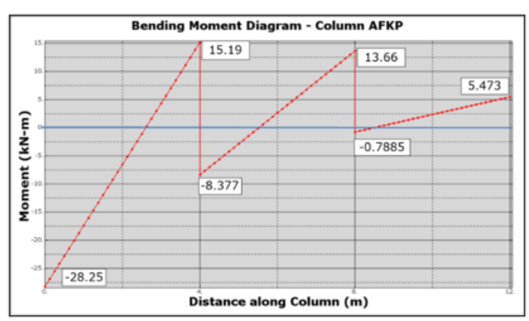
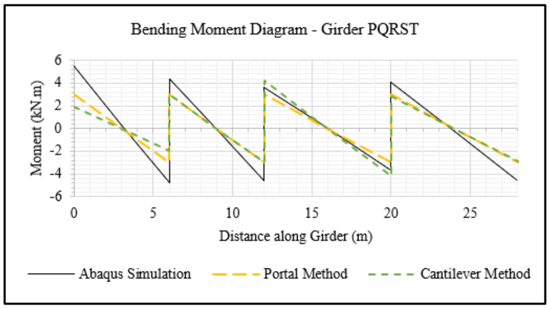


Fig. 8. Bending Moment Distributions of the Top Girder (a) and Left Column (b)

(a)



(b)

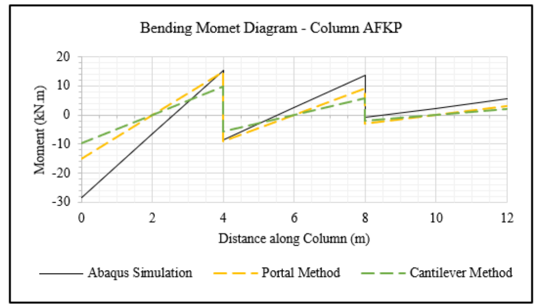


Fig. 9. Comparison of the FE Results Against the Results Obtained Using the Approximate Methods for the Top Girder (a) and Left Column (b)

VII. Project Assessment

As previously stated, the models in the described project were mainly developed for classroom demonstration purpose to further complement and enhance an introductory Structural Analysis course at the authors' university. To obtain a measure for the effectiveness of these models, an anonymous student survey was developed and administered in the fall of 2022. In this survey the students were asked to provide their assessment rating scores relative to the effect of the models on enhancement of their abilities in analyzing structures in six main areas on a scale of 1-5, as outlined in Table 1. The result of the conducted survey is provided in Table 2. Included in Table 2 are the average assessment score for each of the six assessed areas, as well as the overall ratings for the entire project. Reviewing these scores, the authors believe that the project has been moderately successful in terms of helping students better understand the specific topics discussed in the course.

VIII. Summary and Conclusion

The development of finite element simulation models for further enhancement of an introductory Structural Analysis course at Georgia Southern University was included and discussed in the presented paper. The models were essentially created to cover some of the main topics and concepts covered in the course. The results obtained from the simulations were all almost identical to the theoretical values, except in cases where approximate methods of analyses were used.

As discussed, the models could be used by the instructor during classroom presentations to more effectively discuss the behavior of structures subjected to a variety of geometric, support, connection type, and loading conditions. The student assessment of the project was conducted through administering a student survey in the fall of 2022 and the results were included in the paper. The authors of the paper believe that the assessment scores can further be improved if more time could be dedicated to fully explaining the details of each developed model at a slower pace. Expertise gained and techniques developed through undertaking this project serve as an invaluable tool to foster other more in-depth investigations for analyzing a variety of other more complicated structures and solids.

Anonymous Student Assessment of the FE Simulation Models
for Enhancement of the Structural Analysis Course – Fall 2022

You are asked to evaluate the educational value of the demonstrated Finite Element simulation models in enhancement of the Structural Analysis course. Please provide your assessment score on a **scale of 1-5** according to the following rating guide.

(1) Strongly Disagree **(2) Disagree** **(3) Neither Disagree or Agree** **(4) Agree** **(5) Strongly Agree**

The FE simulation models demonstrated in the course have contributed to the enhancement of my ability to solve the following class of problems:

1. **Analysis of Structural Frames Subjected to Static Loads**
   - The effect of support and connection reactions on the deformation of the frames.
   - The distribution of internal shear and bending moment along the length of the frame members.

2. **Analysis of Portal Trusses**
   - Deformation characteristics of portal trusses subjected to applied lateral loads.
   - Determination of the axial forces in the portal truss members.
   - Determination of internal shear force and bending moment variations for the column members.

3. **Analysis of Trusses Subjected to Mechanical and Temperature loads**
   - Determination of the axial forces in the truss members (using method of joints & method sections).
   - Determination of the displacements of truss joints (using energy methods).

4. **Analysis of Multistory Two-dimensional Frames Subjected to Distributed Vertical Loads**
   - Deformation of these frames subjected to indicated loads.
   - Variation of shear force and bending moment along the frame girders.
   - The effect on the actual deformation behavior of this type of structure due to the common assumptions made in the simplified classical method of analysis.

5. **Analysis of Multistory Portal Frames Subjected to Lateral Loads**
   - Deformation of portal frames subjected to horizontal loads.
   - Variation of shear force and bending moment along the frame girders.
   - The effect on the actual deformation behavior of this type of frame due to the common assumptions made in the simplified classical methods of analysis (i.e., *Portal Method* & *Cantilever Method*).

6. **Overall Educational Value of the Demonstrated FE Simulation Models**
   - The presented simulation models helped me better understand the behavior of the specific structures discussed in the course due to various loading conditions.
   - The presented simulation models helped me better understand the enormous power and utility of the finite element method in analysis and design of structures.

Table 1. Assessment Instrument Developed for Evaluation of the Produced Structural Engineering Simulation Models

Total Students = 17						
Question #	Strongly Disagree (1)	Disagree (2)	Neither Agree or Disagree (3)	Agree (4)	Strongly Agree (5)	Average Rating Score/5.0
1	0 (0%)	1 (5.9%)	1 (5.9%)	10 (58.8%)	5 (29.4%)	4.1
2	0 (0%)	0 (0.0%)	1 (5.9%)	11 (64.7%)	5 (29.4%)	4.2
3	0 (0%)	1 (5.9%)	1 (5.9%)	11 (64.7%)	4 (23.5%)	4.1
4	0 (0%)	0 (0.0%)	1 (5.9%)	12 (70.6%)	4 (23.5%)	4.2
5	0 (0%)	0 (0.0%)	1 (5.9%)	10 (58.8%)	6 (35.3%)	4.3
6	0 (0%)	0 (0.0%)	3 (17.7%)	10 (58.8%)	4 (23.5%)	4.1
Total	0 (0%)	2 (2.0%)	8 (7.8%)	64 (62.7%)	28 (27.5%)	4.2

Table 2. Anonymous Student Assessment Results for
Evaluation of the Produced FE Simulation Models

References

1. Das, N. K., "Teaching and Learning Structural Analysis Using Mathcad," Proceedings of the *2002 ASEE Annual Conference & Exposition*, Montreal, Canada, June 16-19, 2002.
2. Martino, N., & Ghanem, A., "Innovative Approach to Teaching Applied Structures Courses," Proceedings of the *2016 ASEE Annual Conference & Exposition*, New Orleans, LA, June 26-29, 2016.
3. Sitaram, P., & Manyama, A. M, "Computer Analysis of Structures for Architectural Technology Students," Proceedings of the *2020 ASEE North Central Section Conference*, Morgantown, WV, March 27, 2020.
4. Rojiani, K.B., & Schottler, R., "Java Applets for Structural Analysis," Proceedings of the *2000 ASEE Annual Conference & Exposition*, St. Louis, MO, June 18-21, 2000.
5. Navaee, S., & Das, N., "Utilization of MATLAB in Structural Analysis," Proceedings of the *2002 ASEE Annual Conference & Exposition*, Montreal, Canada, June 16-19, 2002.
6. Navaee, S., "Developing Instructional Modules for Analyzing Structures," Proceedings of the *2003 ASEE Annual Conference & Exposition*, Nashville, TN, June 22-25, 2003.
7. Navaee, S., "Utilization of Excel in Solving Structural Analysis Problems," Proceedings of the *2003 ASEE Annual Conference & Exposition*, Nashville, TN, June 22-25, 2003.
8. Navaee, S., "Alternate Approach in Analyzing Structures Utilizing LabVIEW," Proceedings of the *2004 ASEE Annual Conference & Exposition*, Salt Lake City, Utah, June 20-23, 2004.
9. 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, pp. 11-24.
10. Hibbeler, R.C., Structural Analysis, Tenth Edition, Pearson Prentice Hall, 2014.

Shahnam Navaee

Dr. Navaee is currently a full professor in the Department of Civil Engineering and Construction in the Allen E. Paulson College of Engineering and Computing at Georgia Southern University. Dr. Navaee received his B.S. and M.S. degrees in Civil Engineering from Louisiana State University, and his Ph.D. in Engineering from Clemson University. Prior to his current faculty position, Dr. Navaee served in several administrative rolls. One year as the Interim Chair of the Dept. of Civil Engineering and Construction Management, and seven years as the Associate Dean of the College of Science and Technology at Georgia Southern University. Dr. Navaee's main research interests are in the areas of solid mechanics and structures.

Kristin Ackerman
Kristin Ackerman completed her B.S. degree in the Civil Engineering and Construction Department at Georgia Southern University in Dec. of 2022. Ms. Ackerman is currently working in a construction engineering firm in Savannah, Georgia.