Scholastic Considerations for Selection and Design of an Honors Project in Structural Engineering

Shahnam Navaee, Gustavo Maldonado, and Kayla Allen
Georgia Southern University / Georgia Southern University / Georgia Institute of Technology

Abstract

In this paper the description of an Honors Project for an undergraduate student enrolled in the Civil Engineering and Construction Department at Georgia Southern University is outlined and discussed. The project involves modeling and analysis of a tensegrity structure using the finite element approach utilizing Abaqus. A tensegrity structure is referred to a special type of assembly consisting of a series of straight solid members, arranged in various geometrical configurations in a three-dimensional space and held together by series of cables. Involvement of a student in this type of project (a) strengthens and extends the student's knowledge in structural analysis, solid mechanics, and finite element analysis, and (b) extends the student's skills in developing the correct three-dimensional models for analyzing structures using advanced tools and features in Abaqus. Included in the paper is a summary of the provided student assessment ratings corresponding to each specific project outcome.

Keywords

Student Educational Project, Engineering Mechanics, Structures, Finite Element Analysis

I. Introduction

The analysis of tensegrity structures in an Honors project utilizing Abaqus is outlined and discussed in the presented paper. As stated previously in the abstract, a tensegrity structure is an assembly consisting of a series of straight solid members arranged in a three-dimensional space and held together by series of cables. A web search for this type of structure yields a variety of structural components ranging from a simple frame in the shape of a table, supporting a flat top platform with three straight members and six cables, to a more innovative architectural bridge structure with large number of members and cables. Tensegrity structures generally have a light self-weight, making them ideal for cases where the weight of the structure must be kept to a minimum. These types of structures have been studied in various forms by a number of researchers[1-4]. Other examples of a tensegrity structure include a variety of load-bearing and non-load bearing sculptures and arches with large number of component parts arranged in various intriguing configurations. Through the course of investigations conducted in this applied research, variety of approaches and techniques were explored to develop a correct model that could accurately determine the behavior of a tensegrity structure under statically applied loads.

The Abaqus software tool selected for this study has sophisticated features which can effectively be used to analyze structures with complex loadings, connections, and supports. Using this software, a variety of computations can be performed to study the behavior of structures subjected to loads. Some of these results include the computation of displacements and rotations

at various locations on the structure, reactions at the structure supports, and stresses and strains induced on the structures. The computed results can be displayed graphically in various ways on the screen and compiled in written reports as desired. When analyzing models in Abaqus, these models can be created using other Computer-Aided Drafting (CAD) packages and then imported in Abaqus, or alternatively the models can directly be created in Abaqus. In the described project, all parts of the studied models were entirely generated using Abaqus.

Some of the specific tasks performed in this project using Abaqus included the following: utilization of various datum points, datum axes, and datum planes to properly create a three-dimensional part, assigning material properties, creating cross section profiles, assigning beam section orientations, generating sections, assigning created sections to various frame components, creating instances, producing interactions, establishing appropriate analyses steps, applying loads and boundary conditions, properly seeding and meshing the part, and generating all desired output in the proper form. A brief description for each of the above listed specific tasks is provided in the next section of the paper to better describe the required tasks related to the analysis of the structures included in the project.

Examples of the developed Finite Element (FE) models for several tensegrity type structures are also included and discussed in the paper to further describe the project, and to clearly identify the special skill sets gained by the student when undertaking the project. The project can further raise the student's interest in the area of structures and play an important role in adequately preparing the involved student for pursuing other more advanced studies.

The procedure developed in undertaking this project can be applied to analyze a variety of other more complex three-dimensional structures, connected in various ways, and subjected to various static and dynamic loads. The effects of material nonlinearity can also be incorporated in the produced models using more advanced features and tools in Abaqus to further enhance the utility of the developed models.

II. Procedure

A listing of the specific steps and procedures used in modeling of the tensegrity structures investigated in this project using Abaqus is included in this section below. The outlined set of steps is essentially needed to analyze any structure and is not limited to the specific models selected in this study. To develop the skills needed to perform each of the listed steps and to learn more about the details involved in each step, a variety of resources had to be consulted by the student. These resources included voluminous Abaqus Documentation Files and Users' Guides developed by *Dassault Syst`emes*[5], various online Abaqus tutorial videos[6-11], and a set of Abaqus tutorial guides developed by the main author of the paper. These guides were discussed in reference[12].

1. *Creating Parts*
   The first step in analyzing a structure using Abaqus is to create a geometric model that accurately represents the physical size and type of structure which is being analyzed. A variety of two-dimensional and three-dimensional structures and solid continuums can be analyzed using this powerful tool. To model the tensegrity structures in this project a

three-dimensional deformable ″wire structure″ was created to model each of the individual components of the structure. For example, for the first created model in the project depicted in Fig. 1, twelve separate parts had to be created to represent the entire structure. To create each of the individual parts, an initial *Reference Point* was generated in Abaqus, so that other *Datum Points* could be identified relative to the selected *Reference Point*. These *Datum Points* were the points located at the extremities of each of the components and the locations where the cables were attached to the solid straight members. Once all structure components were created, these parts were then assembled using step 7 outlined below.

2. *Defining Material*
   Once the members of the structures were created, the material properties of each component piece were specified. For the tensegrity structures selected for this project, structural steel (A-36 Steel) with Modulus of Elasticity of $E = 29 \times 10^6$ psi and Poisson Ratio of $\nu = 0.32$ were selected.

3. *Creating Cross Section Profiles*
   In this step, the geometric shape and dimensions of the cross section for each component of the structure were defined. In this project, a hollow circular tube with an outside radius of $r = 5/16$ in. and thickness of $t = 1/8$ in. were selected for all compression members. For the tension cables, a solid circular cross section with a radius of $r = 1/16$ in. was chosen.

4. *Creating Sections*
   Next, the created material properties and member profiles were assigned to each member of the structures. This assignment was done through creating two specific *sections* in Abaqus, one for the compression tube members, and the other for tension cables.

5. *Assigning Sections to Members*
   Once the *"sections"* were generated as described in the previous step, these sections had to be assigned to various members of the structures, so that Abaqus could accurately predict the behavior of these structures.

6. *Assigning Beam Section Orientations*
   In this step, the orientation of the members of the structure should be defined to enable Abaqus to position the members properly in the structure. For example, to position a rectangular cross section in the proper orientation in the assembly, the direction of the short and long sides of the cross section should be specified with relation to the geometry of the structure. Since in this project, circular and annular cross sections were used, the orientation of the cross sections could have been assigned arbitrarily. Note that in spite of this, the step related to specifying beam section orientation is still a necessary step for the analysis of a structure using Abaqus.

7. *Creating Instances*
   Once all the individual components of the tensegrity structure were created and defined, these parts were then assembled to create the structure in its entirety by creating an *Instance* of each of the created parts.

8. *Creating an Analysis Step*

    After the creation of the *instances* of the structures, the type of analysis that had to be performed was specified next. There are various specific analyses that could be prescribed in this step. In the described project, a *General Static* analysis step was selected to analyze the behavior of the tensegrity structures subjected to static loads.

9. *Creating Interactions*

    In this phase of the process, specific details regarding the connections (″*interactions*″) among various members of the structure were specified. Powerful inherit tools in Abaqus allow the creation of a variety of connections for connecting structural members. In the described project, the cables were connected to the tubular compression members of the tensegrity structure using ″pin″ connections. Use of a pin connection, enforces that the connected joint of the members has the same linear displacements in the three-dimensional space, with no moment transfer from the continuous tubular compression members to the attached cables. These pin connections were modeled using the *Coincidence Builder* feature in Abaqus. This process was repeated at each connection to properly predict the behavior of the tensegrity structures under applied loadings.

10. *Applying Loads and Specifying Boundary Conditions*

    Once the models were completely defined and assembled, the applied loads and support conditions were specified. In this project, concentrated forces were placed at the top of structures and the base points of the structures were defined either as ″pinned″ or ″fixed″ supports.

11. *Seeding the Model*

    To perform the finite element analyses of the considered structures, each member of the structure was seeded for the purpose of meshing the model in the next step. In general, smaller size elements could be used to produce more detailed or more accurate results.

12. *Meshing the Model*

    Once the structures were seeded, they had been meshed by selecting an appropriate element type. There is a collection of appropriate element types that could be selected for various structures and solids. For the structures analyzed in this project, a linear beam element (*B31 Element: A 2-node linear beam in space*) was used.

13. *Creating and Running the Program*

    After performing the steps outlined above, analyses jobs were created to produce the needed results. Sample partial screenshots of the Abaqus' viewport showing the shapes of the analyzed models in this project are provided in Figs. 1 & 2. Provided in Tables 1-3 are sample produced reports showing the analyses results for one of the studied models (Model 1). These results include the reaction forces at the structure supports, displacements at the upper connection points of the structure, and the axial stress in the members.

III. Produced Models

A total of five tensegrity type structures were modeled in this project. All structures were assumed to be structural steel (A-36 Steel) with a Modulus of Elasticity of $E = 29 \times 10^6$ psi and a Poisson Ratio of $\nu = 0.32$. All compression members used in the models were selected as tubes

with an outside radius of 5/16 in. and a thickness of 1/8 in. The tension cables incorporated assumed to have a circular cross section with a radius of 1/16 in. To analyze the behavior of each component of the models studied in the project, six nodal points were used to divide each segment of the structure into five elements. As previously stated, more modal points (or elements) can be used if desired.

Model 1

The model of the first tensegrity structure developed in the project is presented in Fig.1. Fig. 1(a) shows the undeformed structure and the loading, while Fig. 1(b) illustrates the deformed configuration of the structure and the nodal points used in the analysis of the problem. This model had twelve members, 3 tubular rods (compression members), and nine cables (tension members). The structure was pinned at the bottom three base points and was loaded by three vertical 300 lb. forces (P_1 = 300 lbs.) applied at the top of the structure. The total length of each rod used in the structure was 24 in. and the cables were connected to the rods at their end points. The length of each side of the equilateral triangles at the top and bottom of the structure was 12 in., and the relative rotation of the top triangle base with respect to the bottom base was 160°. To calculate the specific coordinates of the end points of the rods in this model, an Excel spreadsheet was initially created as an aid in creating the model. The data generated in this spreadsheet was used in Abaqus to produce the parts of Model.

Model 1 (12 Parts):

- Total rod length: **24 in**.
- Side length of top & bottom triangular bases: **12 in.**
- Relative rotation of top triangular base with w.r.t. bottom base in the undeformed state: **160°**
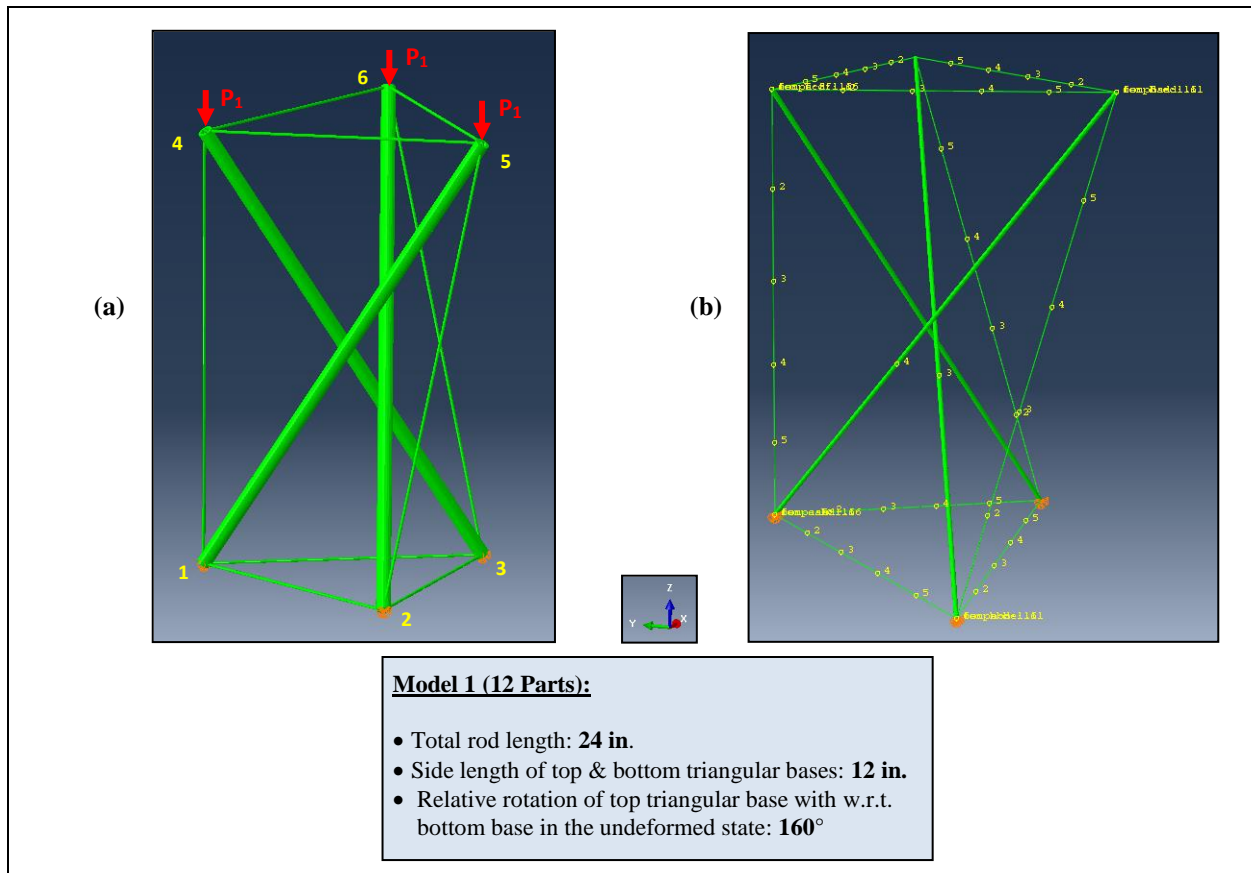
Figure 1. Model of the First Developed Tensegrity Structure:
(a) Undeformed Shape, (b) Meshed Deformed Shape

IV. Sample Results for Model 1

The computed results of the analyses performed in Abaqus can be tabulated by using the *Field Output Request* feature of Abaqus in any desired form. Sample partial results for the studied tensegrity structure presented in Figure 1 are provided in Tables 1-3. These results are the reaction forces at the structure supports, the displacements at the top three points of the structure, and axial stress in each of the members.

Note that by selecting the proper "Analysis Step" described in stage 8 of the procedure provided previously, the nonlinear deformations of the studied structures can be determined. When solving any nonlinear problem, an iterative procedure is employed in Abaqus through which the loads are applied to the structure in small increments. Convergence of results is needed in each load increment, before the next increment of load is applied. The process is continued until the total load is applied in full. Abaqus contains powerful features which can handle nonlinear problems with relative ease.

Table 1. Reaction Forces at Bottom Base Points of Model 1

Node Label	Reaction Force in x-direction (lbs.)	Reaction Force in y-direction (lbs.)	Reaction Force in z-direction (lbs.)
1	-0.01	-408.44	300.00
2	353.74	204.22	300.00
3	-353.73	204.22	300.00

Table 2. Displacements of the Top Base Points of Model 1

Node Label	Displacement in x-direction (in.)	Displacement in y-direction (in.)	Displacement in z-direction (in.)
4	-201.350E-03	-162.958E-03	-38.2039E-03
5	241.819E-03	-92.9135E-03	-38.2065E-03
6	-40.4468E-03	255.885E-03	-38.2120E-03

Table 3. Axial Stress in the Structure Components

Structure Component	Axial Stress (psi.)
Tensile stress in the three cables at the top base of the structure:	+ 19,217
Tensile stress in the three cables connecting the top & bottom bases along the structure height:	+ 28,596
Compressive stress in the three tubes connecting the top & bottom bases along the structure height:	- 3,970

V. Other Sample Structures

The models of the other four structures developed in this study are provided in Fig. 2. The details regarding the material properties, cross section dimensions, as well as other specific structure geometries are also included in this Figure. For each of the studied cases, various checks were

performed to ensure the validity of the developed models. The complete numerical analysis results for the studied models were included in the final report of the student.

All Models:
- Material Property (A-36 Structural Steel): **E = 29x10⁶ psi**, $υ = 0.32$
- Rod Radius: **5/16 in**., Thickness: **1/8 in**., Wire Radius: **1/16 in.**

(a)

Model 2 (12 Parts):

- Total rod length: **28 in**.
- Cable attachment points from rod ends: **2 in.**
- Side length of top & bottom equilateral triangular bases: **12 in.**
- Relative rotation of top triangular base with w.r.t. bottom base in the undeformed state: **160°**

(b)

Model 3 (12 Parts):

- Side length of bottom equilateral triangular base: **12 in.**
- Dimensions of top equilateral triangular base: **6 in.**
- Relative rotation of top triangular base w.r.t. bottom: **0°**

(c)

Model 4 (16 Parts):

- Dimensions of top & bottom square bases: **12 in.** x **12 in.** Length of diagonal members: **24 in.**

(d)

Model 5 (17 Parts):

- Dimensions of top rectangular base: **48 in.** x **72 in.**
- Dimensions of bottom rectangular base: **96 in. x 144 in.**
- Height of structure: 144 in.
- Top rectangular base is located symmetrically above the bottom base with no rotation.
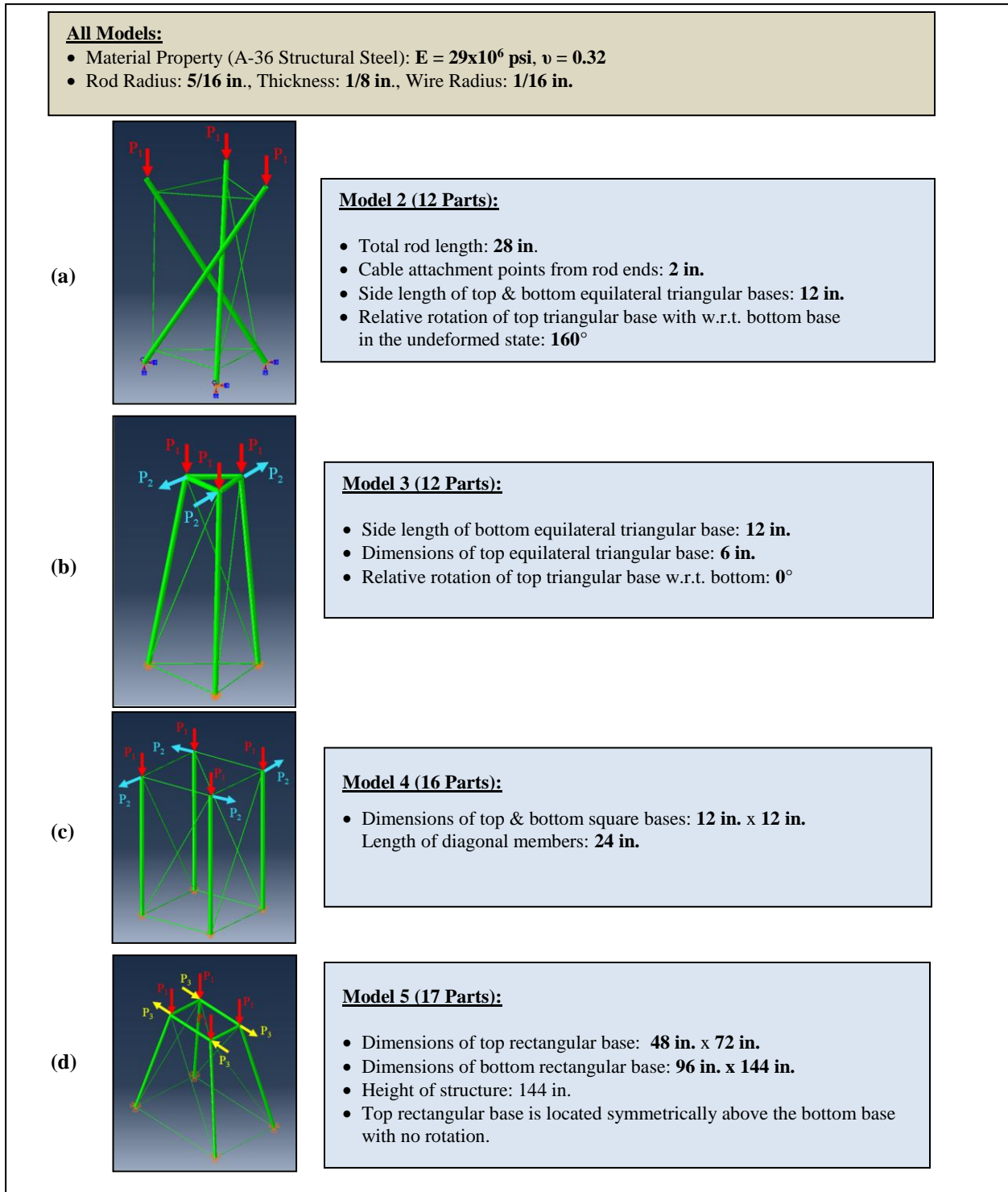
Figure 2. Developed Models of the Other Four Studied Structures

VI. Project Assessment

As stated previously, involvement of a student in a project similar to the one described in this paper can have a positive effect in terms of the academic development of the student, and his/her future professional success. After the conclusion of the project, the involved student was asked to assess the value of the project by rating each specific project outcome on a scale of 1-5. Summary of the provided student ratings corresponding to each of the ten listed specific outcomes of the project is provided in Fig. 3. The overall average score for the measured project outcomes was 4.7.
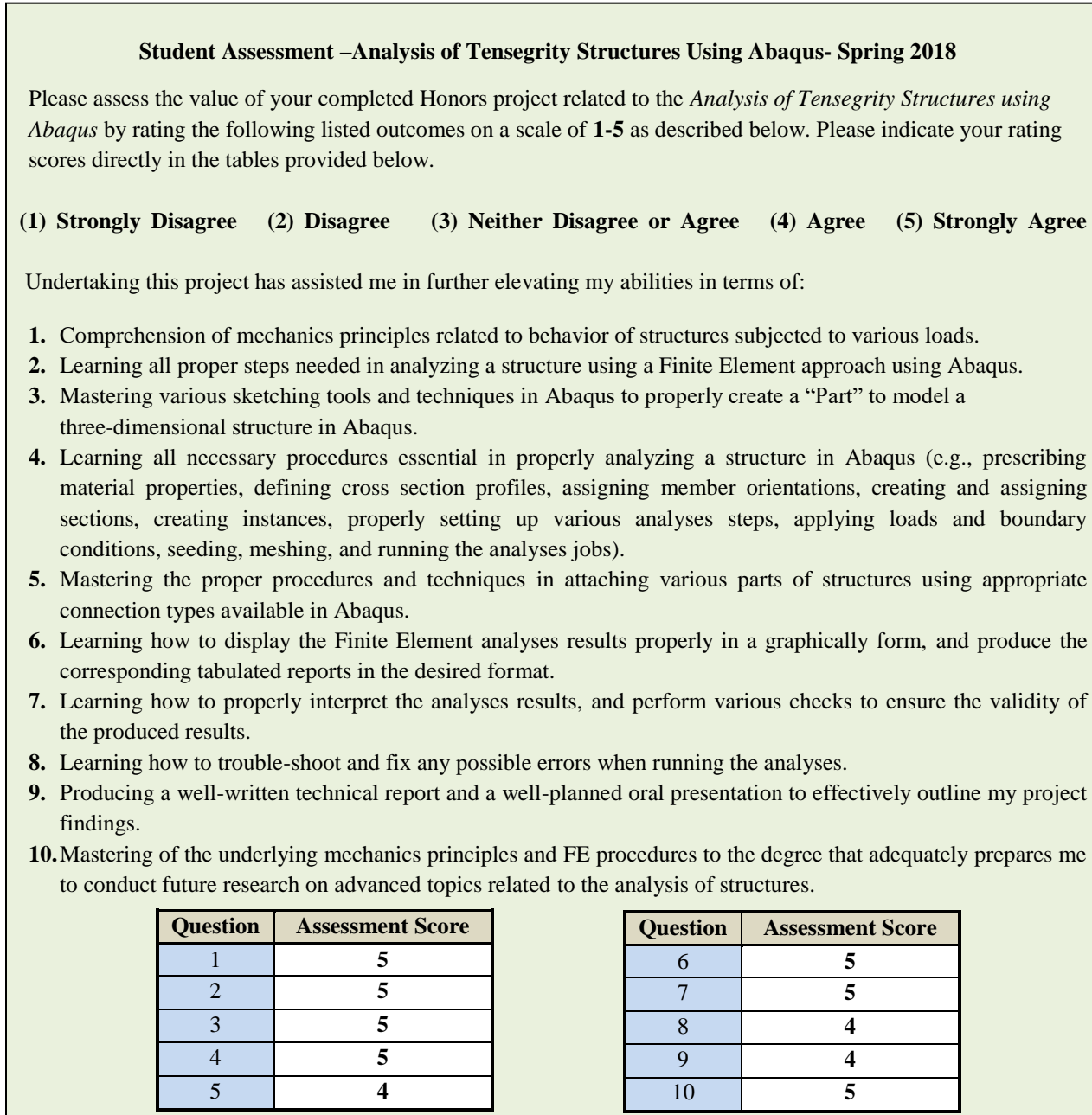

Student Assessment –Analysis of Tensegrity Structures Using Abaqus- Spring 2018

Please assess the value of your completed Honors project related to the *Analysis of Tensegrity Structures using Abaqus* by rating the following listed outcomes on a scale of **1-5** as described below. Please indicate your rating scores directly in the tables provided below.

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

Undertaking this project has assisted me in further elevating my abilities in terms of:

1. Comprehension of mechanics principles related to behavior of structures subjected to various loads.
2. Learning all proper steps needed in analyzing a structure using a Finite Element approach using Abaqus.
3. Mastering various sketching tools and techniques in Abaqus to properly create a "Part" to model a three-dimensional structure in Abaqus.
4. Learning all necessary procedures essential in properly analyzing a structure in Abaqus (e.g., prescribing material properties, defining cross section profiles, assigning member orientations, creating and assigning sections, creating instances, properly setting up various analyses steps, applying loads and boundary conditions, seeding, meshing, and running the analyses jobs).
5. Mastering the proper procedures and techniques in attaching various parts of structures using appropriate connection types available in Abaqus.
6. Learning how to display the Finite Element analyses results properly in a graphically form, and produce the corresponding tabulated reports in the desired format.
7. Learning how to properly interpret the analyses results, and perform various checks to ensure the validity of the produced results.
8. Learning how to trouble-shoot and fix any possible errors when running the analyses.
9. Producing a well-written technical report and a well-planned oral presentation to effectively outline my project findings.
10. Mastering of the underlying mechanics principles and FE procedures to the degree that adequately prepares me to conduct future research on advanced topics related to the analysis of structures.

Question	Assessment Score
1	5
2	5
3	5
4	5
5	4

Question	Assessment Score
6	5
7	5
8	4
9	4
10	5

Figure 3. Student Assessment Summary of the Project Outcomes

VII. Conclusions and Future Research Plans

In the described student project, a procedure for analyzing a tensegrity structure using Abaqus was outlined and discussed. Five developed structure models and a sample partial analysis results for one of the models were included in the report to further illustrate the details related to project and the procedures followed. The models are capable of determining small as well as large elastic deformations of the structures. In developing the models, each component of the structure was created as a separate part. This was intentional in order to make the developed models more flexible, so that in the generated models the material and geometric properties of each part and each connection could easily be modified to obtain the solutions for a variety of other similar structures.

The project described in the presented study has been very successful, since it has not only contributed to the academic development of the involved student, but also due to the fact that it has laid the foundation for conducting other future studies related to the analysis of structures subjected to more complicated loadings, connections, and material conditions. In the spring of 2019, the students enrolled in a newly created finite element course developed by the faculty investigator were coached to pursue several projects related to the analysis of structures and present their results at the *2019 Student Research Symposium* on campus. Each performed research was conducted utilizing the Abaqus software. The conducted projects were related to the following topics: (1) Rehabilitation of Bridge Piles, (2) Analysis of Seismic Soil-Structure Interactions, (3) Seismic Analysis of Multistory Structures, (4) Crash Simulation of Vehicles with Roadside Barriers, and (5) Analysis of Composites. These projects resulted in five student presentations and two college awards at the *2019 Student Research Symposium*.

References

1   Caluwaerts, K., & Carbajal, J.P. (2015). Energy Conserving Constant Shape Optimization of Tensegrity Structures. *Journal of Solids and Structures*, 58, 117-127, Elsevier.
2   Do, D.T., Lee, S., & Lee, J. (2016). A Modified Differential Evolution Algorithm for Tensegrity Structures. *Composite Structures*, 158, 11–19, Elsevier.
3   Lee, S., & Lee, J. (2016). A Novel Method for Topology Design of Tensegrity Structures. *Composite Structures*, 152, 11–19, Elsevier.
4   Bohm, V., Sumi, S., Kaufhold, T., & Zimmermann, K. (2017). Compliant Multistable Tensegrity Structures. Mechanism and Machine Theory, 115, 130-148, Elsevier.
5   SIMULIA User Assistance (2018). *Dassault Syst`emes*. [Online].
    http://catsithub:2080/English/DSSIMULIA_Established.htm
6   AbaqusPython. (2014). Static Analysis of a 3D I-Beam Frame - Part 1 of 3. *YouTube*. [Online].
    https://www.youtube.com/watch?v=dBPL6pYO-4s.
7   AbaqusPython. (2014). Static Analysis of a 3D I-Beam Frame - Part 2 of 3. *YouTube*. [Online].
    https://www.youtube.com/watch?v=av2zjZbRXZ4.
8   AbaqusPython. (2014). Static Analysis of a 3D I-Beam Frame - Part 3 of 3. *YouTube*. [Online].
    https://www.youtube.com/watch?v=N-kS050IrkE.
9   Advanced FEA (2016). Abaqus Geometric Nonlinear Example. *YouTube*. [Online].
    https://www.youtube.com/watch?v=Urh37Ec-YiU.

10   Advanced FEA (2017). "Abaqus Plasticity Bar with Plots," *YouTube*. [Online].
     https://www.youtube.com/watch?v=opDuBmTJA_w.
11   PSU DrZ (2016). "Abaqus/CAE Plasticity Tutorial," *YouTube*. [Online].
     https://www.youtube.com/watch?v=d4T0MAz3nc0.
12   Navaee, S., & Kang, J. (2016). Enhancement of a Finite Element Analysis Course for Structural Engineering. *Computers in Education Journal*, Vol. 7, No. 4, pp. 11-24.

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 area of structural mechanics and educational research.

Gustavo Maldonado

Dr. Maldonado obtained his B.S. in Civil Engineering from National University of Cordoba, Argentina. Initially, he worked in several consulting firms and later obtained his M.S. and Ph.D. degrees in Engineering Science & Mechanics from Virginia Tech. He worked as an Earthquake Engineer in California and was an Associate Professor at the University of Puerto Rico, Mayagüez. Later, he served as the Chairperson of Engineering at Miami Dade College. He is a California licensed professional engineer and a member of ASCE. His research interests include structural analysis and design, vibrations, earthquake and wind engineering, and remote sensing.

Kayla Allen

Kayla Allen is currently enrolled at Georgia Institute of Technology obtaining her M.S. in Civil Engineering, while working at a structural engineering firm in Atlanta. Ms. Allen obtained her B.S. in Civil Engineering from Georgia Southern University. At both universities, Ms. Allen has specialized in structural engineering.