Semi-Supervised Implicit Augmentation for Data-Scarce VQA

Isabel Novo-Corti

Abstract

In this paper the enhancement of an introductory Finite Element course in the newly established Civil Engineering and Construction Management Department at Georgia Southern University is discussed. Typically in an Introductory Finite Element course offered in many engineering schools, a simple less elaborate FE package is used to deliver the course concepts. In the newly developed course discussed in the paper, a state-of-the-art commercial software package is planned to be utilized to further enhance the marketability of the students upon graduation. Along with this choice comes the challenge of developing suitable tutorials and examples to familiarize the students with various important tools and special features of this sophisticate package in the minimum amount of time possible. The submitted paper explores one possible strategy to accomplish this task. This course is designed for exploring the civil engineering applications focusing specifically on analysis of structural components, rather than solving problems related to other fields such as fluid mechanics or heat transfer. The planned projects in the course explore strategies in analyzing a variety of structural members such as trusses, beams, frames, as well as other solid continuums. The course is expected to complement other structural engineering related courses delivered in the curriculum such as Structural Analysis and Advanced Structural Analysis. Solutions of problems obtained in this course using the FE package can be compared and contrasted against the results from other classical methods discussed in these courses. Some of these methods include: Slope-Deflection Method, Moment Distribution Method, and Matrix Stiffness Method. These comparisons further enhance the knowledge of the students in area of structures and provide them with new perspectives.

Several prepared example problems for the course are included in the paper to further illustrate the value of the project. Each of these examples is selected for a unique purpose, covering specific topics, and illustrating important principles. Collection of the prepared course examples such as the ones included in this paper will provide the students with the strong foundation in the areas of analysis of structures, preparing them well for their future studies and for pursuing rewarding professional careers.

Introduction

In this educational project instructional modules and examples are created for analyzing various structural members using the Abaqus software, a well-known Finite Element Analysis (FEA) package used in many research institutions and prestigious engineering schools in the United States and around the world. Even though a number of electronic tutorial files accompany this comprehensive software packages, these files in their presented format are not suited for direct utilization in a course. In the current project, the investigator has prepared an easier and more practical guide for students to be able to use this sophisticated simulation package. The instructional modules and examples created in this project are developed to complement the CENG 5336 (Introduction to Finite Elements) course, a technical elective in the Civil Engineering program at Georgia Southern University which has never been taught previously in the CE program. These modules provide an overview of some of the most powerful features of this package and aid in introducing the users to various Abaqus special components and commands. Each example is selected to demonstrate the utility of a particular feature of the software in accomplishing a new task, a task which was not required in the previous examples. Developed tutorials essentially provide the details related to performing the following listed activities using the order indicated.

(1) Creating a part
(2) Creating a material
(3) Defining and assigning section properties
(4) Assembling the model
(5) Creating an analysis step
(6) Requesting data output

(7) Applying boundary conditions
(8) Applying loads
(9) Meshing the model
(10) Creating an analysis job
(11) Checking the model
(12) Running an analysis job
(13) Post processing.

Even though the analysis of any structural member using Abaqus mainly involves undertaking the above listed tasks, the intricacies involved in each of these steps can significantly be different depending on the special type of problem analyzed and the analysis method selected. Tutorial examples developed in this project cover the analysis of structural members such as (1) trusses, (2) beams, (3) frames, and (4) solid continuum elements. The project will continue so that other examples can be developed for investigating more complicated problems such as those involving non-linear structures and structures subjected to dynamic loads.

The following two factors can be perceived as possible drawbacks in utilization of this sophisticated tool in a FE course; (a) the cost of the package, and (b) the steep learning curve associated with the software. Each one of these issues has been addressed in the project. It should be noted that even though the commercial cost for purchasing this package is relatively high, the *Dassault Systems* offers substantially discounted educational licenses of the full version of the software for classroom use. Additionally, the student version of Abaqus can be downloaded free of charge from the *Dassault Systems* website. The free student version of the software is the essentially the full commercial version, limited to handling models containing up to one thousand nodes. It is deemed that this number of nodes is sufficient enough for the purpose of modeling a variety of structural components in an introductory finite element course. The student can download this free version on their personal computers and use it as they wish to master their expertise. The second possible drawback associated with the steep learning curve of the software has been addressed by creating a series of tutorials in the course as further discussed in more details in the remaining sections of the paper.

In this course the students are instructed to construct and analyze a variety models for structures such as trusses, beams, frames, and various other solid continuums. Some of these structures are taken from well-known textbooks such as the one listed in the reference section of the paper [1-8]. Several of these textbooks are currently adopted for use in the Civil Engineering program at the authors' institution.

The Finite Element solutions for several of the examples used in the course are provided in the paper to better illustrate the scope of the project. In performing the special tasks needed for each selected sample problem, a series of steps have to be followed in succession in order to yield the correct solution. In most cases, these steps involve selection of various options in specific menus, submenus, and dialog boxes of the software interface. Remembering and following these steps without having a properly prepared guide can be a difficult and confusing task. For the sample problems such as the ones presented in this paper, a step-by-step guide is prepared to outline the needed procedures. In preparing these guides, screenshots of various important stages of the procedures are included to clearly outline the sequence of required steps. The prepared guides will be of utmost importance in completing the assigned exercises.

Once the students are familiarized with the special features of this comprehensive software tool using the provided tutorial examples, they are assigned to develop the solutions for other more complicated problems using their newly acquired skills. Solving these new problems will provide an excellent opportunity for students to test their understanding, gain further experience, and further enhance their competencies in utilizing this tool. Utilization of Abaqus in this course in the manner described will prepare the students for pursuing rewarding professional and academic careers.

Truss Analysis

The first type of structure the students are assigned to analyze in the course is a simple truss such as the one presented in Figure 1. By completing this exercise the students learn how to perform the thirteen essential operations involved in the finite element analysis of a part utilizing Abaqus. These operations were listed in the previous section of the paper. The tools used for initiating each of the thirteen tasks can be seen on the left hand-side of the screen shown in Figure 1. In addition to experimenting with these tools, in this exercise the students specifically learn how to create various cross sectional profiles and assign them to different truss

members, and additionally learn how to generate a tabular report of the computed results. In this problem, students are instructed to use a 2- noded linear 2-D truss element ("T2D2 Element") when meshing the model. A screenshot of the report showing the computed values of the reaction forces at the roller and pin supports, nodal displacements at the joints, as well as the axial stresses in the truss members is provided in Figure 2. The student will have an opportunity to compare the values obtained from their FE analysis to the results from other methods of analysis such as the *Virtual Work Method* or *Castigliano's Method*. The students were exposed to these methods in the Structural Analysis courses they have previously taken.

Beam Analysis

The procedure for analyzing a beam utilizing Abaqus is next discussed in the course. One sample exercise assigned to the students is presented in Figure 3. In this exercise, a statically indeterminate continuous beam subjected to several loads is analyzed. In addition to providing the students with additional expertise in modeling a structural member, the students are further instructed about the techniques in Abaqus by which a variety of loads can be applied to the structures in various combinations and load cases. In this problem, a 2-noded linear beam element in a plane ("B21 Element") is used in modeling the beam. This exercise also outlines a method by which the distribution of the internal reactions along the length of the beam can be plotted. One sample plot showing the moment distribution for the beam is provided in Figure 4. Similar to the previous case, the FE analysis results for this problem can be compared against the calculated values obtained using other methods of analysis such as the *Slope-Deflection Method* or *Moment Distribution Method*.

Frame Analysis

The third type of exercise included in the course involves the analysis of a frame utilizing Abaqus. Two sample two-dimensional frame examples that could be assigned to the students are provided in Figures 5 and 8. As stated previously, each assigned exercise in the course is selected to teach the students about a new feature of the package, a feature that was not explored in the previous exercises. For example, when analyzing the frame presented in Figure 5, students learn how to apply an inclined distributed load and also place a roller support on an inclined surface. Each of these specific operations requires following a set of steps. Moreover in this exercise, the students learn how to cut a member to obtain the internal reactions at the cut surface. One sample cut has been presented in Figure 6. In this exercise the students additionally become more familiarized with specific techniques for preparing tabular reports. A sample report showing the computed values of the support reactions as well as the displacements and rotations of the nodes on the frame has been presented in Figure 7. These results are based on utilizing a "B21 Element" and correspond to the case where $E = 200$ GPa and $\nu = 0.32$.

As in the previous examples, the students can be asked to compare the results from their FE analysis against the values obtained utilizing other classical methods of analysis discussed in the Structural Analysis courses.

A sample three-dimensional multi-story frame such as the one presented in Figure 9 is also intended to be used as an additional exercise to further enhance the students' competencies in analyzing a full range of structures. The specific cross sectional profiles of structural members can be inputted in Abaqus utilizing dialog boxes such as the ones depicted in Figure 10. In addition to discovering more about the analysis of three-dimensional structures, in this exercise the students also learn more details related to displaying a variety of results such as the ones presented in Figure 9. Included in Figure 9 is the deflected shape of the frame, along with the Von Mises stress contours produced in the frame members. The results displayed in Figure 9 correspond to the case when a three- dimensional 2-noded beam element ("B31 Element") is used to mesh the part.

In the discussed course, the students further have the opportunity to verify the accuracy of their results obtained using Abaqus to the output generated from other structural engineering software packages such as *SAP2000*. Verification of these results will provide the students with additional confidence in their ability to properly and effectively utilize Abaqus.

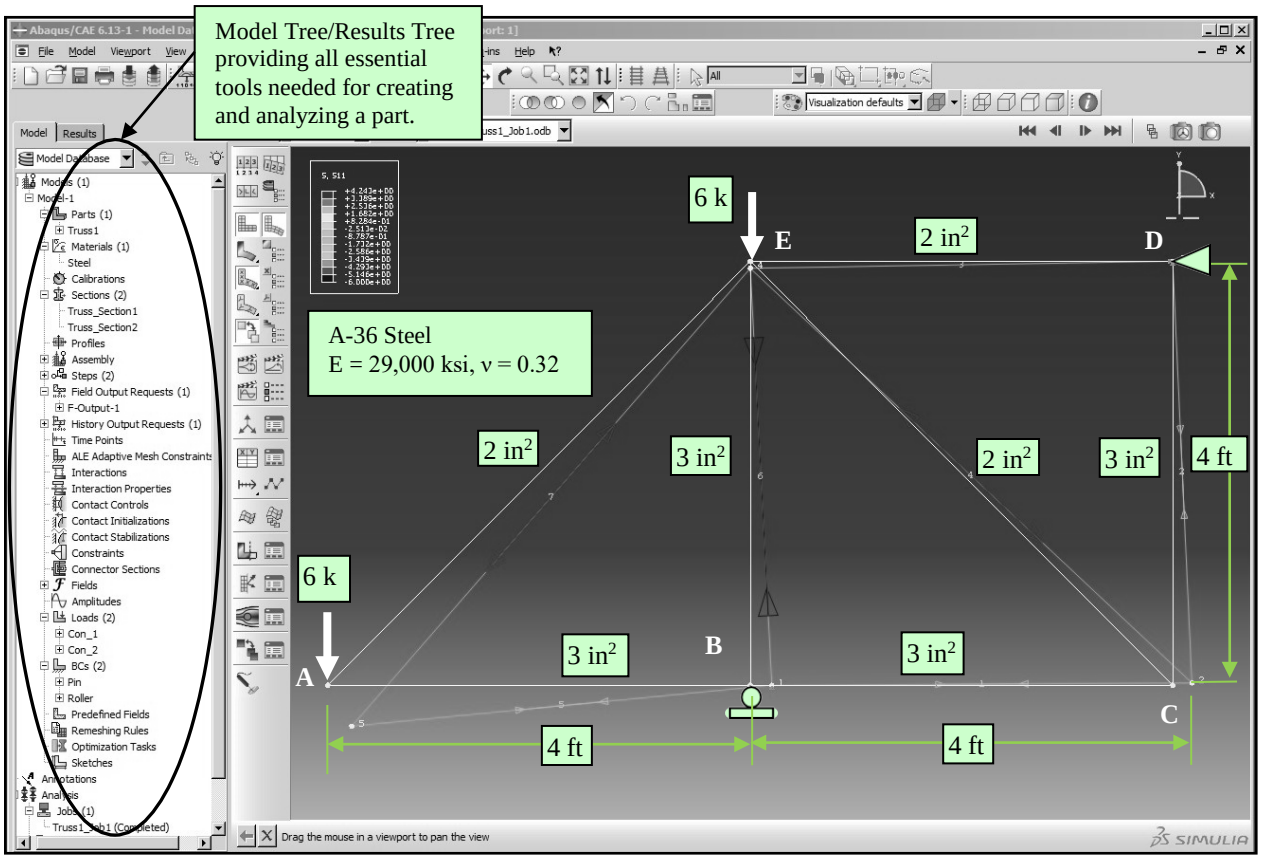
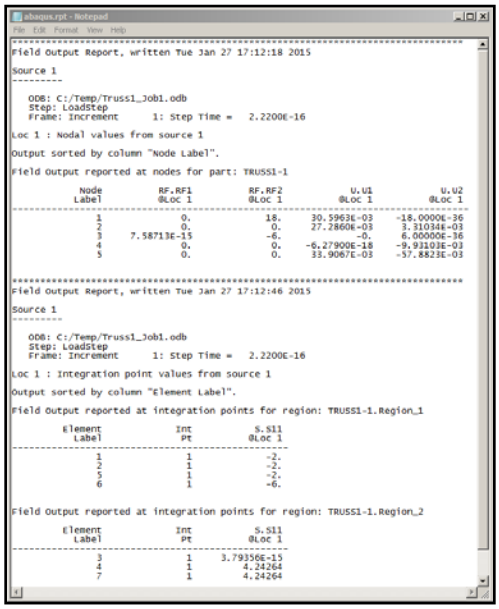


Figure 1. Determinate Truss Subjected to Concentrated Loads Applied at Joints.



```
abaqus.rpt - Notepad
File  Edit  Format  View  Help
*****************************************************************************
Field Output Report, written Tue Jan 27 17:12:18 2015

Source 1
---------

   ODB: C:/Temp/Truss1_Job1.odb
   Step: LoadStep
   Frame: Increment      1: Step Time =   2.2200E-16

Loc 1 : Nodal values from source 1

Output sorted by column "Node Label".

Field Output reported at nodes for part: TRUSS1-1

            Node          RF.RF1          RF.RF2            U.U1            U.U2
           Label          @Loc 1          @Loc 1          @Loc 1          @Loc 1
-----------------------------------------------------------------------------
               1              0.             18.     30.5963E-03    -18.0000E-36
               2              0.              0.     27.2860E-03     3.31034E-03
               3     7.58713E-15             -6.             -0.     6.00000E-36
               4              0.              0.    -6.27900E-18    -9.93103E-03
               5              0.              0.     33.9067E-03    -57.8823E-03


*****************************************************************************
Field Output Report, written Tue Jan 27 17:12:46 2015

Source 1
---------

   ODB: C:/Temp/Truss1_Job1.odb
   Step: LoadStep
   Frame: Increment      1: Step Time =   2.2200E-16

Loc 1 : Integration point values from source 1

Output sorted by column "Element Label".

Field Output reported at integration points for region: TRUSS1-1.Region_1

         Element             Int           S.S11
           Label              Pt          @Loc 1
-----------------------------------------------
               1               1             -2.
               2               1             -2.
               5               1             -2.
               6               1             -6.


Field Output reported at integration points for region: TRUSS1-1.Region_2

         Element             Int           S.S11
           Label              Pt          @Loc 1
-----------------------------------------------
               3               1     3.79356E-15
               4               1         4.24264
               7               1         4.24264
```

Figure 2. Report Tabulating the Reactions at the Truss Supports, Displacement at Joints, and Axial Stress in Members.

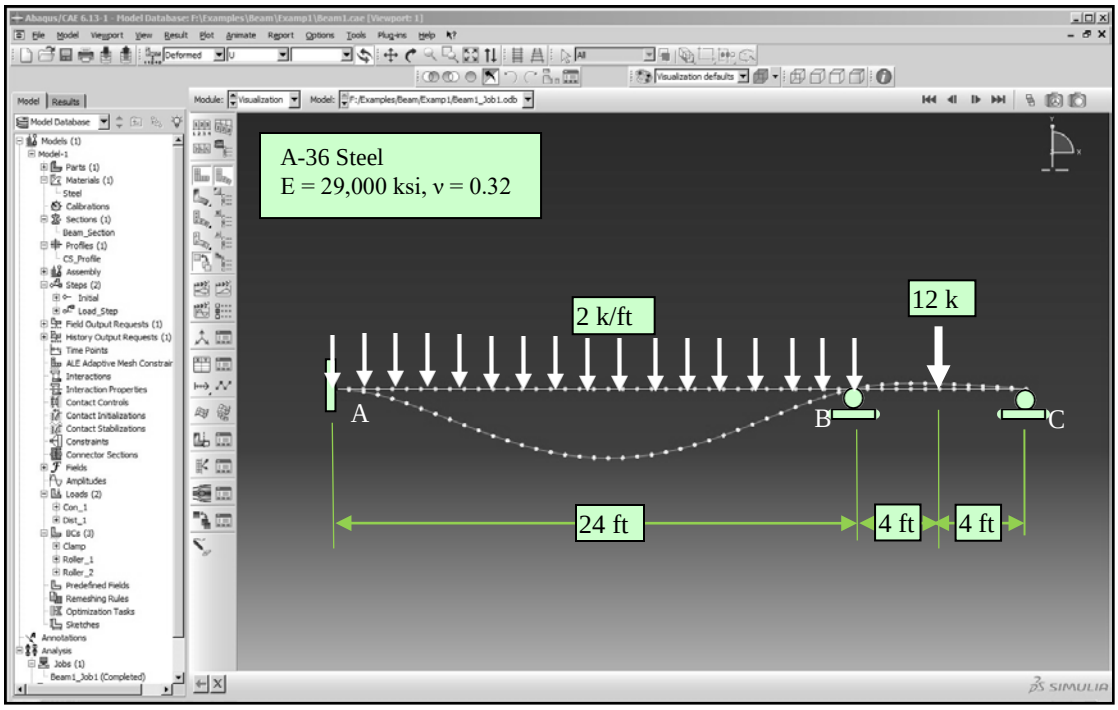


Figure 3. Continuous Beam Subjected to a Concentrated and a Distributed Load.

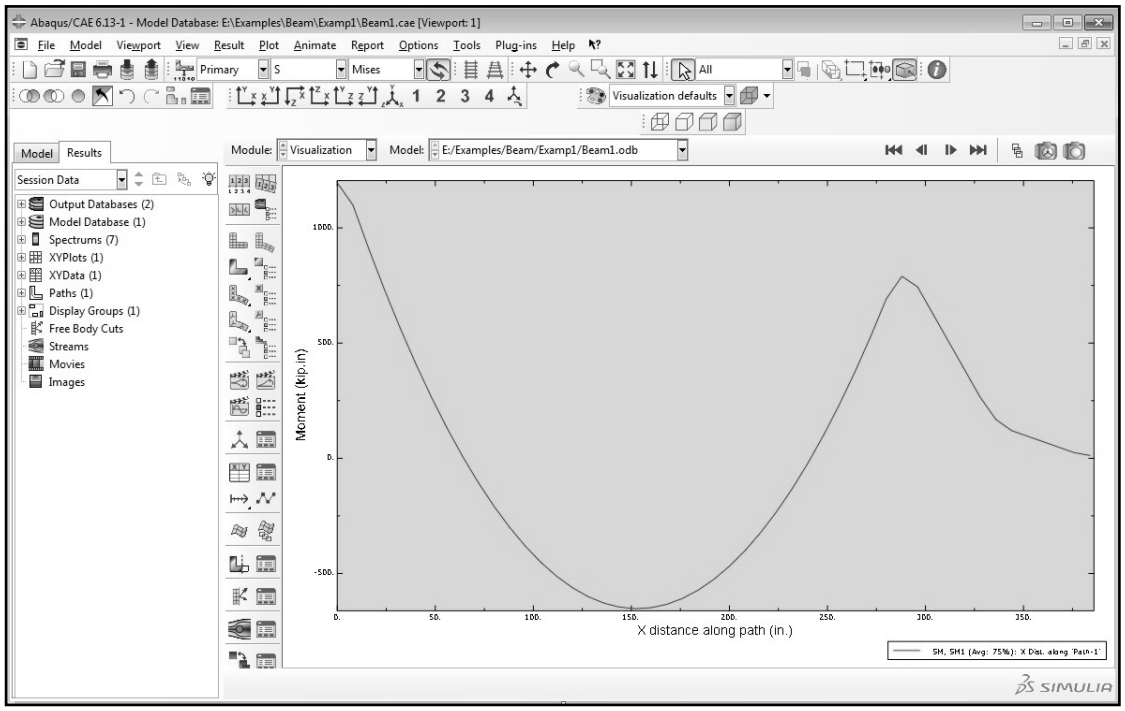


Figure 4. Sample Plot Showing the Distribution of the Moment along the Beam.

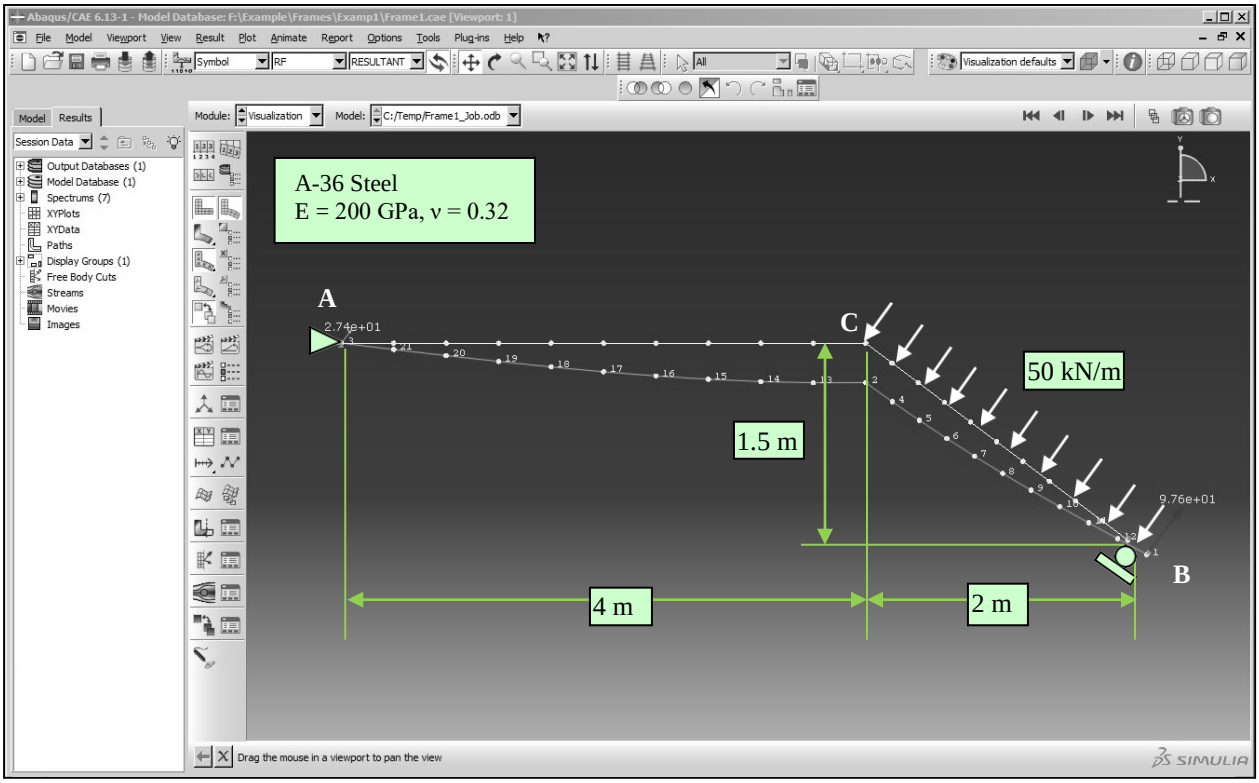


Figure 5. 2D Frame with an Inclined Roller Support Subjected to an Inclined Distributed Load.

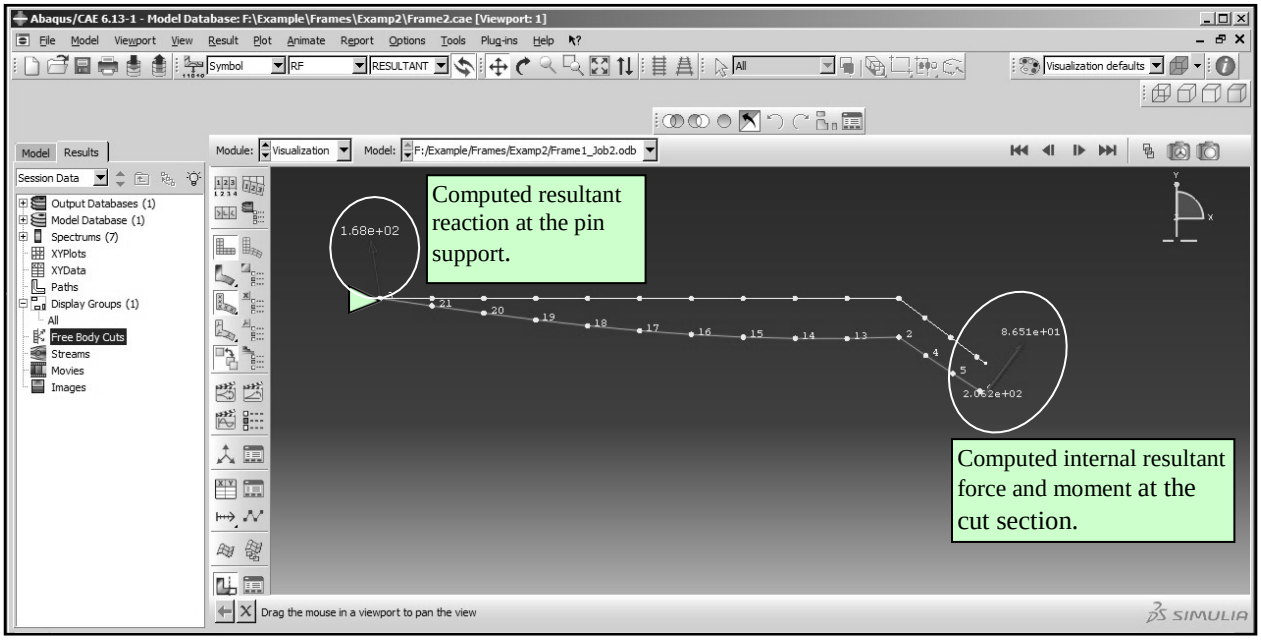


Figure 6. A free Body Diagram of the Cut Frame Showing the Resultant Internal Force and Moment at the Cut Section.

abaqus.rpt - Notepad

File Edit Format View Help

Field Output Report, written Tue Jan 20 21:50:22 2015

Source 1

ODB: C:/Temp/Frame1_Job.odb
Step: Load_Step
Frame: Increment 1: Step Time = 2.2200E-16

Loc 1 : Nodal values from source 1

Output sorted by column "Node Label".

Field Output reported at nodes for part: F1-1

Node Label	RF.RF1 @Loc 1	RF.RF2 @Loc 1	RM3 @Loc 1	U.U1 @Loc 1	U.U2 @Loc 1	UR3 @Loc 1
1	58.5526	78.0702	0.	14.4066E-03	-10.8050E-03	14.8635E-03
2	0.	0.	0.	-26.3158E-06	-30.0489E-03	1.49246E-03
3	16.4474	21.9298	0.	-16.4474E-36	-21.9298E-36	-11.9812E-03
4	0.	0.	0.	322.749E-06	-29.5835E-03	3.22746E-03
5	0.	0.	0.	940.344E-06	-28.7600E-03	5.03404E-03
6	0.	0.	0.	1.83271E-03	-27.5702E-03	6.85220E-03
7	0.	0.	0.	2.99707E-03	-26.0177E-03	8.62194E-03
8	0.	0.	0.	4.42168E-03	-24.1182E-03	10.2833E-03
9	0.	0.	0.	6.08577E-03	-21.8994E-03	11.7762E-03
10	0.	0.	0.	7.95956E-03	-19.4010E-03	13.0406E-03
11	0.	0.	0.	10.0043E-03	-16.6747E-03	14.0167E-03
12	0.	0.	0.	12.1723E-03	-13.7841E-03	14.6443E-03
13	0.	0.	0.	-23.6842E-06	-30.1160E-03	-1.06754E-03
14	0.	0.	0.	-21.0526E-06	-29.2130E-03	-3.35806E-03
15	0.	0.	0.	-18.4210E-06	-27.4477E-03	-5.37912E-03
16	0.	0.	0.	-15.7895E-06	-24.9279E-03	-7.13070E-03
17	0.	0.	0.	-13.1579E-06	-21.7613E-03	-8.61280E-03
18	0.	0.	0.	-10.5263E-06	-18.0558E-03	-9.82543E-03
19	0.	0.	0.	-7.89473E-06	-13.9191E-03	-10.7686E-03
20	0.	0.	0.	-5.26316E-06	-9.45904E-03	-11.4423E-03
21	0.	0.	0.	-2.63158E-06	-4.78341E-03	-11.8465E-03
Minimum	0.	0.	0.	-26.3158E-06	-30.1160E-03	-11.9812E-03
At Node	21	21	21	2	13	3
Maximum	58.5526	78.0702	0.	14.4066E-03	-21.9298E-36	14.8635E-03
At Node	1	1	21	1	3	1
Total	75.0000	100.000	0.	60.9983E-03	-428.346E-03	22.4404E-03

Figure 7. Generated Tabular Report Showing the Reaction Forces at the Frame Supports and the Displacements and Rotations of Nodes.

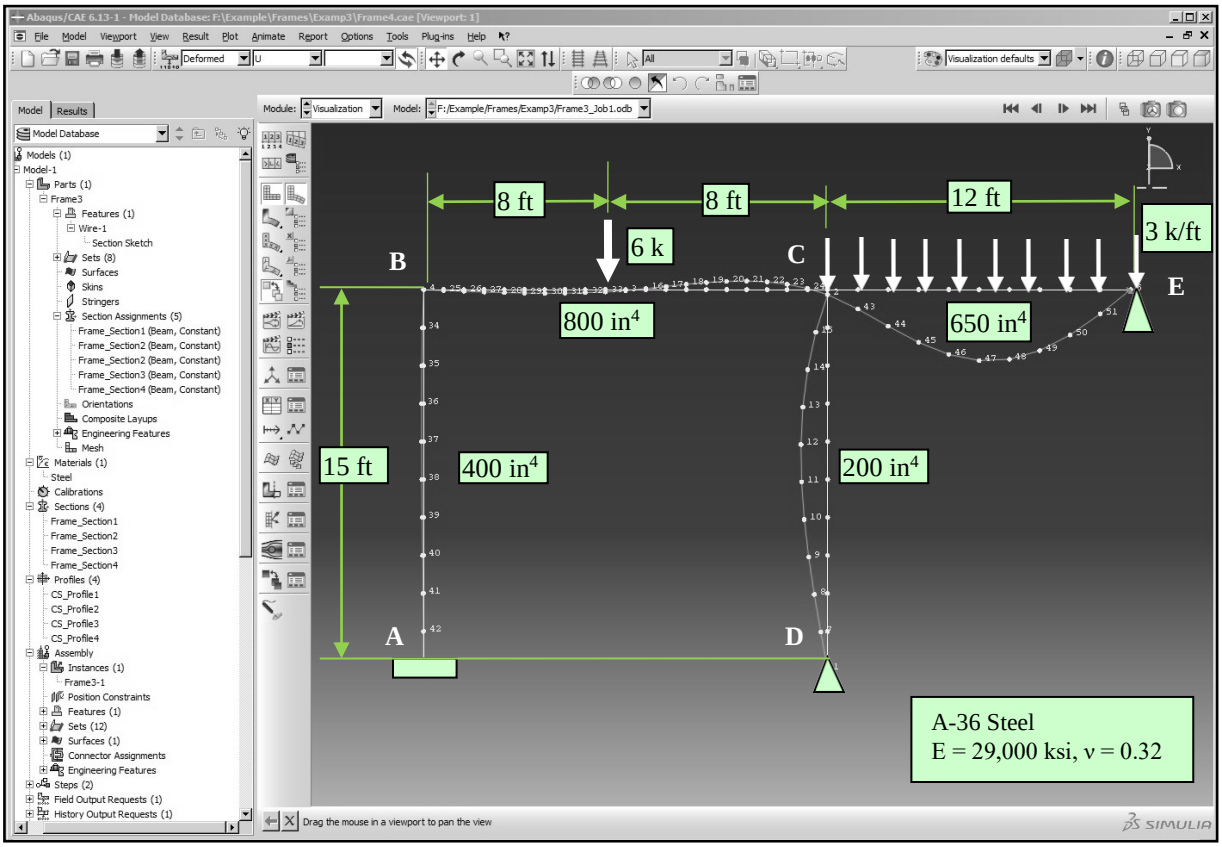


Figure 8. Example of a 2D Frame Containing Various Cross Section Profiles.

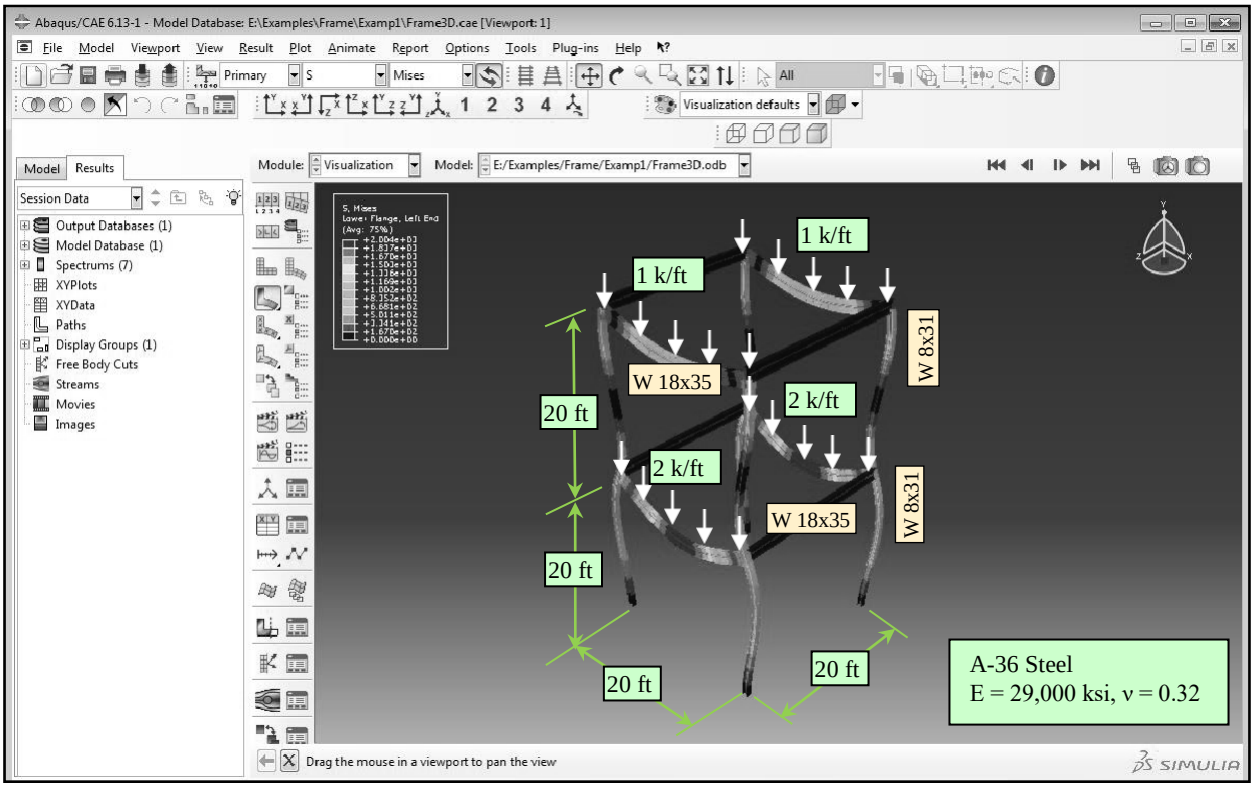
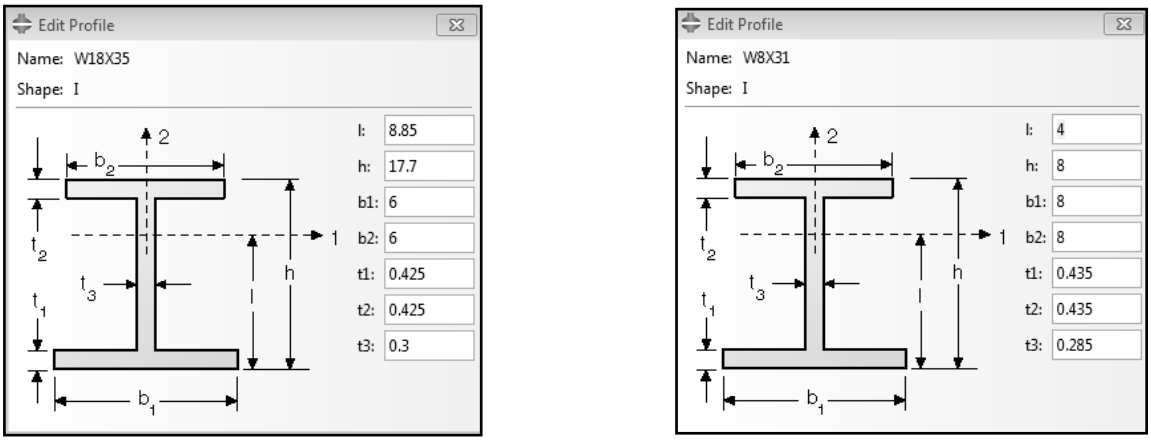


Figure 9. Sample 3D Frame Subjected to Four Distributed Loads.



(a) **(b)**

Figure 10. Dialog box in Abaqus for Specifying the Cross Sectional Profiles of Members.

Continuum Member Analysis

Once the students have been familiarized with the essential features of Abaqus for analyzing structural members such as the ones presented in the previous sections, they are next instructed to utilize this package to perform a FE analysis for investigating the behavior of continuum members using several other planned exercises. Two sample exercises are included in this section to further illustrate.

Lever Arm Example

In the presented exercise a lever arm containing two small cylindrical holes is subjected to two concentered load at one end and is supported by a fixed hexagonal shaped prism inclusion at the other end. The inclusion has a different material property than the lever arm. A generated three- dimensional model for this problem is provided in Figure 12. Included in this figure are the material specifications

for each of the two material types used. To create the three-dimensional model of the problem, students are first coached to develop a two-dimensional sketch that they can later extrude. The sketching and modifying tools in Abaqus along with the two-dimensional sketch of the lever arm can be seen in Figure 11. Using these tools, any combination of geometric shapes can be created, modified, dimensioned, and further edited. Introduction of students to these tools enables the students to generate the model for most any structural component they mayencounter.

Once the three-dimensional model is created, the students are further instructed to partition the model using several portioning techniques and tools available in Abaqus. This partitioning provides a guide for effective meshing of the model. The specific partitions created for this problem can be seen in Figure 12.

The procedure for meshing of a part including topics related to various meshing techniques, and selecting appropriate element shapes, element types, and seed numbers are explored in detail at this stage of the exercise, so that students fully understand this essential stage in the FE analysis of a part. A 10-noded quadratic tetrahedron element ("C3D10 Element") is used in this exercise for the main and the sub part. Figure 13 shows the meshing used along with the Von Mises stress contours for each of the two parts of the model.

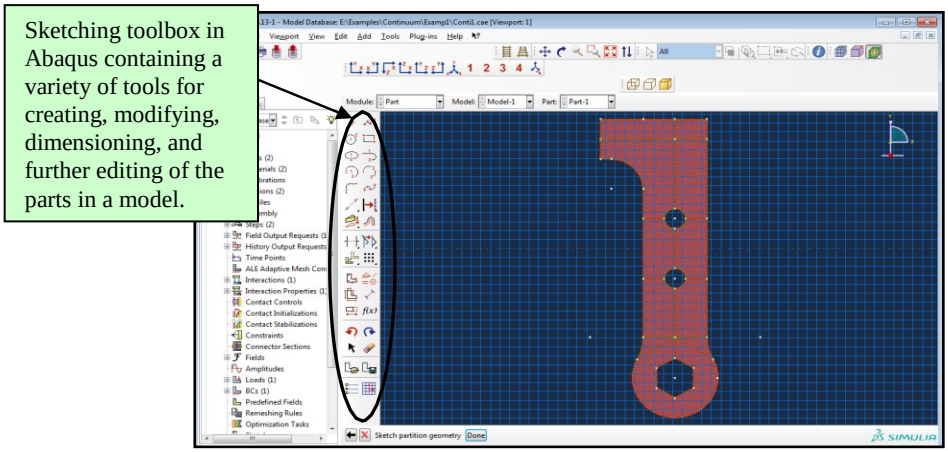


Figure 11. 2D Sketch of the Part Created for Construction of the 3D Model.

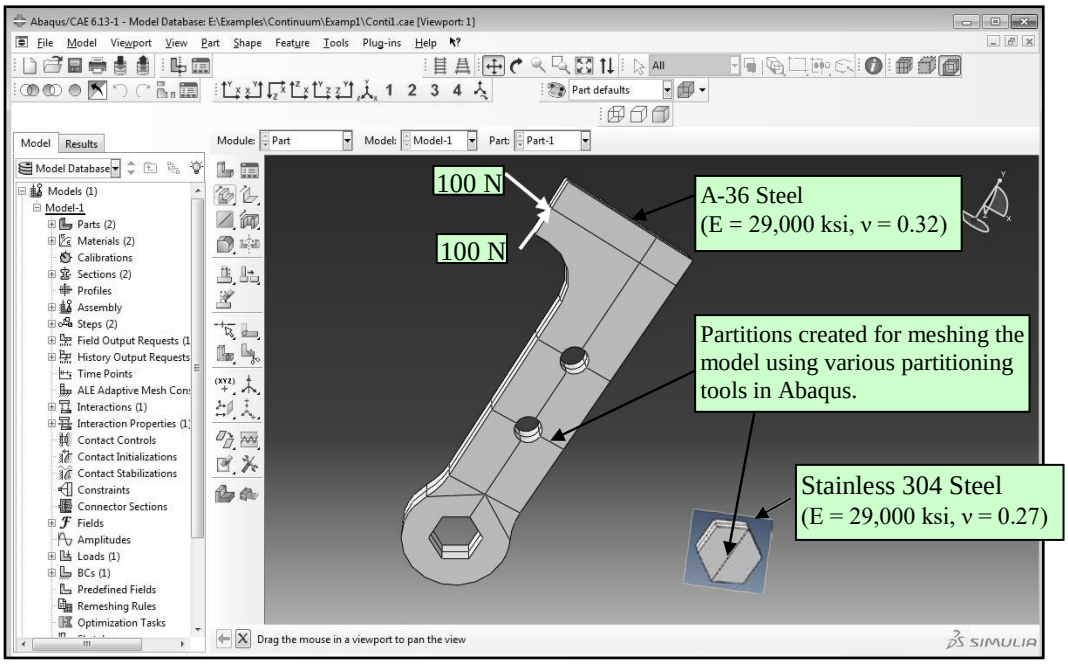


Figure 12. Screenshot of the Created Abaqus Model showing the Partitions Used for the Lever Arm and the Inclusion.

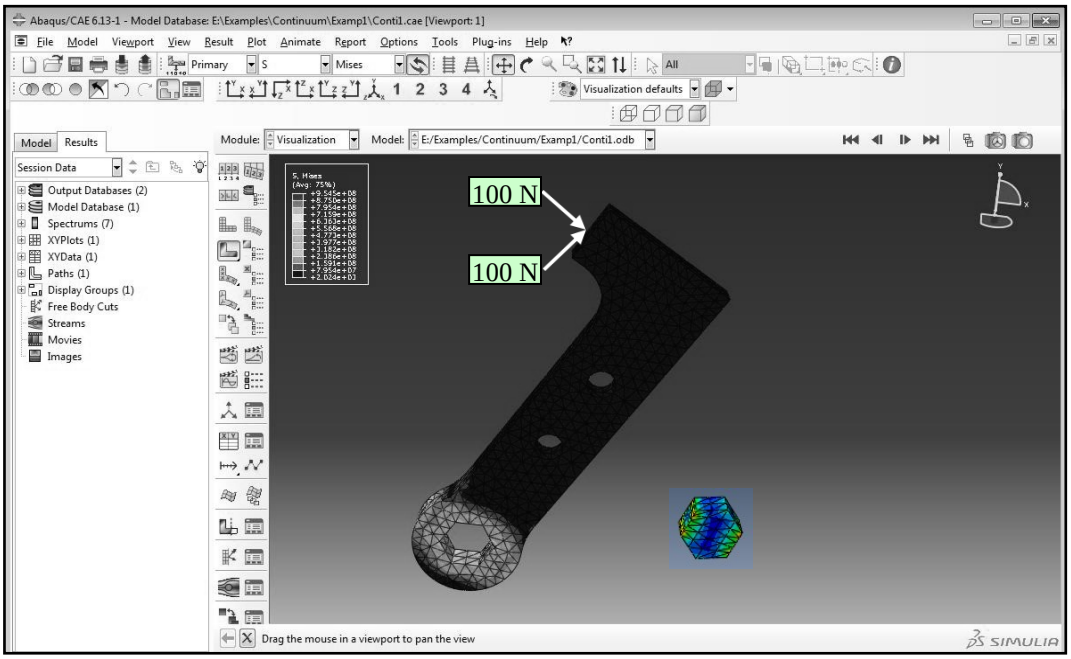


Figure 13. Von Mises Stress Contours for the Separated Parts of the Model.

A special powerful feature of Abaqus allows the users to be able to cut a model with the planar, cylindrical, or spherical cut surface to reveal and examine the stress distribution in the interior of a part. Once this surface cut has been generated, it can also easily be translated along specified directions to conveniently allow the user to inspect the stress distribution at any desired location on the part. In a case where a planar cut has been used, the students can also rotate this plane surface cut around specified axes, if necessary.

Another useful feature in Abaqus allows the users to create a "free body cut" to compute the resultant force and moment at any cut surface on the continuum member. One such free body cut is shown in Figure 14 for the discussed model. This cut is generated by cutting the lever arm by a plane perpendicular to the axis of the lever arm.

Once the full set of results for the created model is successfully generated, students are further instructed on how to create a tabular report of the variables needed for any subset of the model, in the format desired.

Bolted Connection Example

In this exercise a three-dimensional FE model of a bolted joint for a full-scale two-story aluminum frame is developed and discussed. The frame assembly shown in Figure 15 is constructed in the Structures laboratory in the Civil Engineering and Construction Management Department at Georgia Southern University.

The FE model for one of the connections of the frame depicted in Figure 15(b) is provided in Figure 17. This model was constructed using the details obtained from the shop drawing of this particular connection provided in Fig. 16. Figure 17(a) shows the beam and loading conditions, while Figures 17(b) and 17(C) respectively show the generated mesh and the corresponding Von-Mises stress contours. In producing the FE model of this connection, various connecting parts consisting of two I-beam girders, four gusset plates, and thirty bolts & nuts were created and assembled to accurately describe the details of the bolted connection.

Material properties of various components of the analyzed connection are provided in Table 1. Similar to the previous exercise a 10-noded tetrahedron element ("C3D10 Element") is used in this study for all girders, plates, bolts, and nuts. Von-Mises stress contours for the bolt and nut assemblies are provided in Figure 18 along with a truncated section of the produced report showing the numerical values of some the computed stresses.

Once the students are fully familiarized with the more advanced features of Abaqus related to

analyzing connected and assembled parts using the example such as the one presented above, they will be assigned to develop the FE solution for various other connections used in the discussed frame. Calculated FE analysis results can be compared against the theoretical values obtained using the AISC design codes. The assigned exercises offer the following two main advantages: (a) they enhance the competencies of students in properly applying the finite element tools and techniques in analyzing continuum members, and (b) they elevate the students' understanding of the behavior of structural connections under loads covered in other courses such as the Structural Steel Design. To make the discussed course more challenging, the students can additionally be assigned to perform the FE analysis to determine the response of the frame connections subjected to a variety of dynamically applied loads.

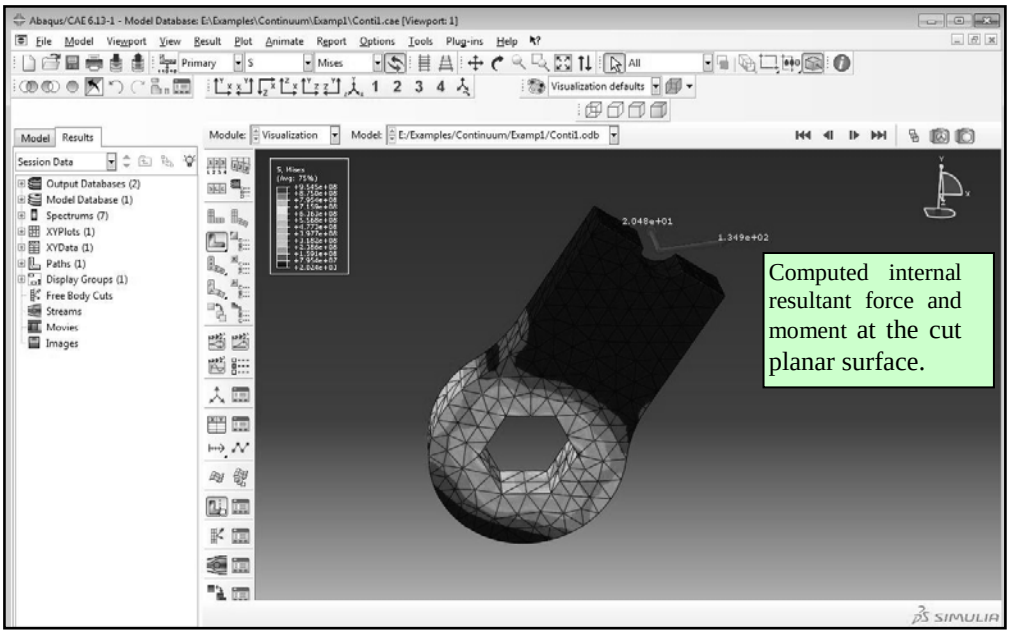
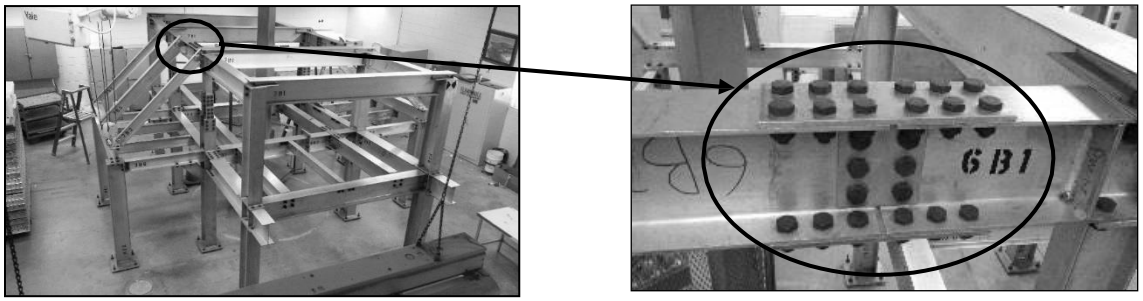


Figure 14. A Free Body Cut Generated by a Planar Cut Perpendicular to Axis of the Lever Arm.



(a) **(b)**

Figure 15. Two Story Aluminum Frame under Investigation:
(a) Entire Structure, and (b) Shear-Moment Connection at 6B1-6B2 Location.

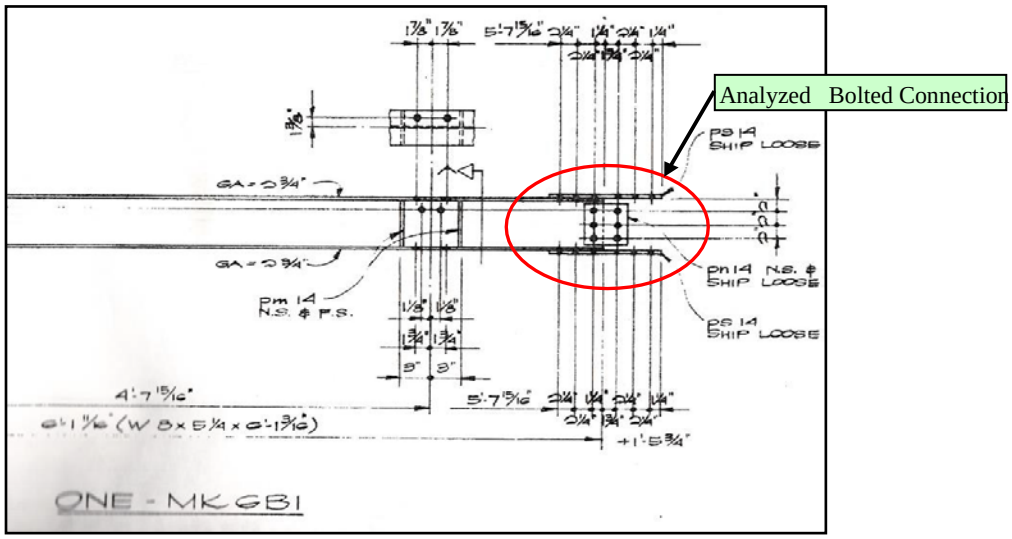
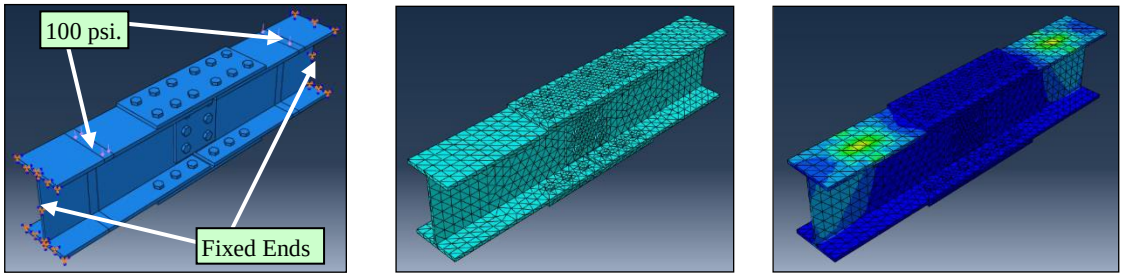


Figure 16. Shop Drawing of Bolt Connection Depicted in Figure 16(b).



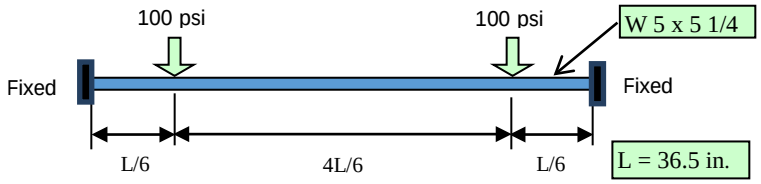
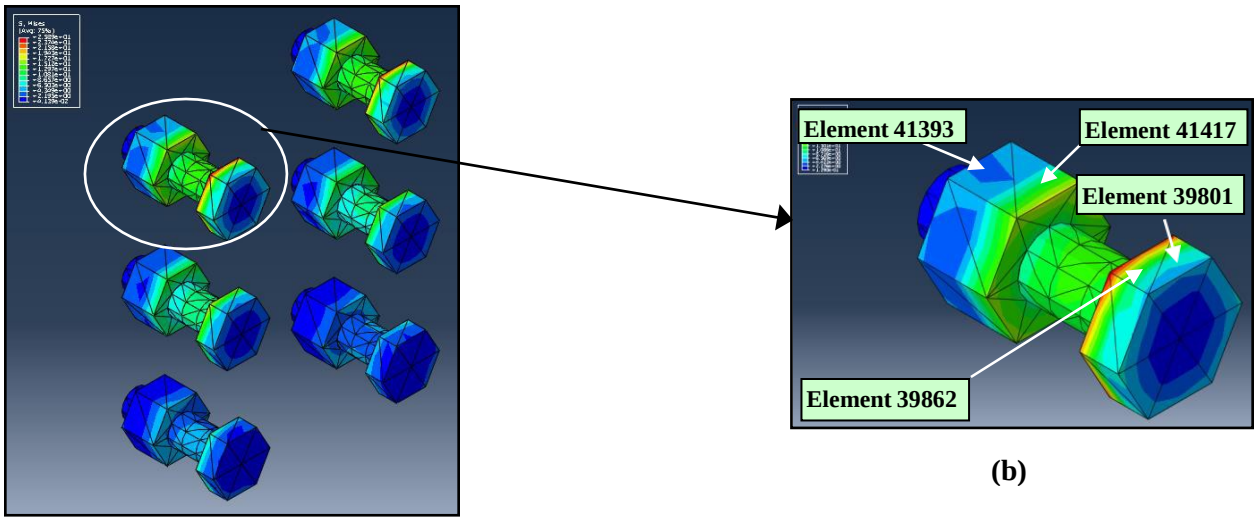


Figure 17. Developed Finite Element Model: (a) Load and Boundary Conditions, (b) Generated Mesh, and (c) Von Mises Stress Contours.

Table 1. Material Properties for the Bolted Connection.

Components	Material	Elastic Modulus (psi)	Poisson's Ratio
Girders & Plates	Aluminum	10 (10^6)	0.33
Bolts & Nuts	A-36 Steel	29 (10^6)	0.32



(a)

(b)

	Element Label	Int Pt	S.Mises @Loc 1
Bolt Head Elements	41417	1	11.5672
	41417	2	2.87295
	41417	3	12.2044
	41417	4	4.78902
	41393	1	1.68289
	41393	2	3.13512
	41393	3	2.83655
	41393	4	7.09506
	(In-between results omitted)		
Nut Elements	39801	1	4.27337
	39801	2	8.70055
	39801	3	3.58308
	39801	4	12.8905
	39862	1	2.98997
	39862	2	18.1548
	39862	3	16.2448
	39862	4	11.8663
Minimum			155.813E-03
At Element			39669
Int Pt			3
Maximum			18.7395
At Element			39804
Int Pt			2

(c)

Figure 18. (a) Von Mises Stress Contours for the Entire Bolt-Nut Assemblies, (b) Sample Element Designations for a Bolt-Nut Assembly, and (c) Sample Report Showing the Von Mises Stress Results.

Summary & Conclusion

In the presented paper a procedure for delivering a finite element analysis course in a civil engineering program using Abaqus is outlined and discussed. The newly designed course guides the students to effectively utilize various powerful features of this comprehensive package to analyze a variety of structural components using a series of easy-to-follow tutorials. Some of the special procedures included in these tutorial guides may not even be known to more experienced Abaqus users. As previously stated, the electronic documentation files that accompany the software, even though well-prepared, are not suitable for use in a course in the form presented.

After mastering the essential finite element tools and techniques through performing the discussed tutorial exercises, the students are assigned additional more complicated problems to further enhance their knowledge, expertise, and competencies in using this powerful tool. Since the newly designed course has not yet been delivered, no assessment data is provided in the paper. Once the course is offered, the collected assessment results will be used to further tweak and enhance the course.

Introducing the students to the powerful features of this comprehensive package will provide them with the foundation and expertise they need to be able to pursue high level research in an academic or professional setting. This perhaps is one of the most significant outcomes of the proposed project. At the authors' institution, this tool can specifically be utilized for preparing students to conduct undergraduate research, or graduate research in the *Master of Science in Applied Engineering (MSAE)* program.

The authors hope to be able expand this project further and possibly develop more advanced tutorial examples for other advanced topics such as analysis of nonlinear problems and dynamic analysis of structures. These new guides can be incorporated in more advanced courses such as Advanced Finite Elements and Structural Dynamics.

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, 2015.

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. Logan, D.L, A First Course in the Finite Element Method, Fifth Edition, Cengage Learning, 2012.

7. 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.

8. Reddy, J.N., An Introduction to the Finite Element Method, Third Edition, McGraw Hill, 2006.