Finite Element Modeling of Bolted Connections for a Steel Sculpture

Saeed Moaveni
Minnesota State University

Supachard Krudtong
Rajamangala University of Technology Lanna

Karen C. Chou
Northwestern University

Abstract

Connections are the glue that holds steel structures together. In practice, a structure is constructed by connecting various members – such as I-beams, columns, channels (C shapes), angles (L shapes), and hollow tubes – using bolts or welds. Depending on the design loading requirements, a steel connection can be classified into three broad categories, *axial, shear, and moment connection*. Moreover, the engineer has to calculate the deformation and stress for every connection to make sure the connection is not subjected to values that are greater than the design limit. The analysis of bolted connection is complex and local stresses are difficult to visualize. In this study, we used ANSYS, a well-known finite element program, to model bolted connections to provide visual aids for the deformation and stresses that could lead to failure modes – such as web local yielding, bolt shear strength, tension rupture, and flexural yielding. The models presented in this paper are intended only as visual tools to enhance the understanding of local deformations and stress build-up in connections; they are not meant for design purposes. The plasticity and the capacity bearing of connections were not considered. For visualization purposes, two approaches were taken: (i) 2-D modeling using non-contact elements, and (ii) 3-D modeling with contact elements. For 3-D models, solid bolted model and contact pairs were created. The advantage of the 2-D modeling is that it is simple, and it takes only a few seconds to solve the problem. On the other hand, the 3-D modeling with contact elements is more complicated and results in a set of nonlinear equations that requires much longer run time.

Introduction

A structure is constructed by connecting various steel members such as beams and columns. The steel structure must carry the design loads which commonly consist of the weight of the floors, columns, beams, equipment, people, snow, and wind to the foundation of the structure safely. Loads are transferred from one structural member to the next and eventually to the foundation through the connections. It is the engineer's responsibility to make sure that each element, including the connection, along the load path is designed properly so that the loads are transferred safely from where they are applied to the foundation. As the result, steel connections are very important and often considered as the glue that holds the members of a structure together. If a connection fails, the entire structure or a section of it could collapse.

Figure 1. An axial connection and an example of axial failure [1].

An introductory steel design course is intended to teach civil engineering students not only about the design of a structure's members but also about the design of different connection types. In this course, it is often difficult for students to visualize the three dimensional nature of the connections. To overcome this shortcoming, a steel connection sculpture was designed by Professor Duane S. Ellifritt at the University of Florida in 1985 [1]. This teaching tool helps students visualize and understand the three dimensional nature of typical steel connections found in standard construction practices.

The next major development in steel connection design was done by Perry S. Green, Thomas Sputo, and Patrick Veltri, who wrote the *Connections Toolkit* – a teaching guide for the steel sculpture [1]. The connections teaching toolkit allows students to better understand the limit states (failure modes) of each connection, and how to analyze the strength of each connection. For example, assuming the connecting members are sized properly to support the load, in their toolkit, Perry S. Green, et al explain failure modes for three types of bolted connections: *axial, shear,* and *moment*, and use the following definitions for limit states.

Figure 1 shows one type of bolted axial connection in which the loading will result in tension along the length of a bolt. Additionally, for this situation the bolt(s) would fail within the threaded portion of the bolt(s), through one of the roots of the threads. One possible failure mode coincides with the least cross-sectional area of the bolts. An example of bolted shear connection is shown in Figure 2. For this situation, the loading will result in shearing along the cross-sectional area of the bolt, and possible failure of shear connection could occur at the bolt-hole region (bearing failure) or within the bolt itself as shown in Figure 2.

A bolted moment connection (Figure 3) is one in which the moment is transferred as a couple at the top and bottom flanges of the supported beam to the supporting member. Moment connections are assumed to have little or no relative rotation between the supporting member and the supported members. Similar to the bolted shear connection, the moment connection could also fail by shearing of the bolts or bearing at the bolt holes.

Although the *Connection Toolkit* is extremely valuable, it does not provide any visual aids about how stresses build up in a connection or how the connection deforms. To provide such visual tools one must resort to finite element modeling. Through a grant from the National Sci-

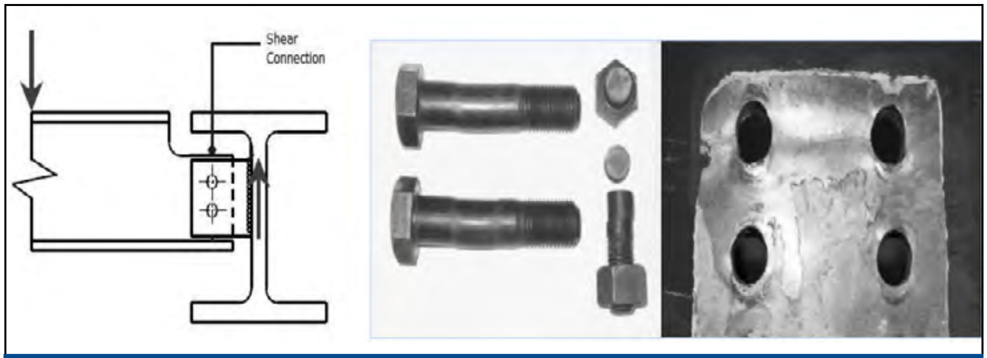

Figure 2. An example of shear connection and shear failure [1]

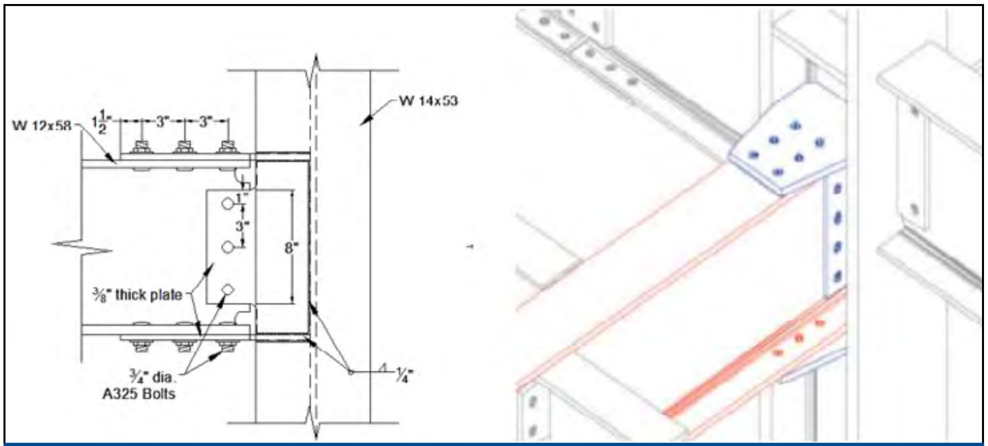

Figure 3. Moment connection [1]

ence Foundation, a three dimensional Virtual Steel Connection Sculpture was developed based on the physical sculpture that is located on the campus of Minnesota State University, Mankato. The Virtual Steel Sculpture is discussed in detail in an article by Moaveni and Chou [2]. This easily maneuverable tool allows the user to zoom, pan, rotate, and spin the Sculpture to view various connections from different directions. Moreover, for a specific connection, one can retrieve additional information such as the blueprint that was used to fabricate the connection, close-up views of the connection, limit states calculations, field example photos, and finite element analysis of stress distribution within a connection. In this paper, the finite element modeling of connections are discussed.

Finite Element Models

In recent years, many attempts have been made to model bolted connections using finite element (FE) analysis. Finite element method [3] is a numerical approach that can be used to solve many engineering problems including those that involve deformation and stress analysis. The FE approach also provides great visual tools to display the results. The basic steps involved in any finite element analysis consist of the following steps [3].

1. We begin by discretizing the solution domain into finite number of elements and nodes.
2. Next, we use a function to represent the physical behavior (solution) for an element.
3. We then develop a set of equations for an element that relates its stiffness to its nodal deformations and applied loads.
4. We assemble all of the elements to present the entire problem.
5. Next, we apply the boundary conditions, initial conditions, and loading.
6. Finally, we solve a set of linear or nonlinear algebraic equations simultaneously to obtain results, such as displacement values and stresses.

It is important to note here that the finite element models presented in the proceeding sections are intended only as visual tools to show how local deformations and stresses build up in the connections over the elastic region. The models are not meant for design purposes; the plasticity and the capacity bearing of connections were not considered, as they are structure specific and are beyond the scope of this study. Moreover, most undergraduate engineering students have no background in finite element modeling or in nonlinear mechanics.

As mentioned previously, a structure is constructed by connecting various members such as I-beams, columns, channels (C shapes), angles (L shapes), and hollow tubes. In practice, to connect these structural members bolts or welds are used. Depending on the design loading requirements, a steel connection can be classified into three broad categories: *axial, shear*, and *moment* connection. The engineer has to calculate the deformation and stress of every connection to make sure the connection is not subjected to conditions that are greater than the design limits. In general, the analysis of bolted connection is complex. In this study, we have used ANSYS, a well-known finite element program, to analyze bolted connections. Two approaches were taken: (i) modeling using non-contact elements, and (ii) modeling with contact elements. The advantage of modeling without the use of contact elements is the ease of modeling and quick computational time. It takes only a few seconds to solve these types of models, and most students with limited background in finite element analysis can create a model. On the other hand, the 3-D modeling with contact elements is more complicated and results in a set of nonlinear equations that requires much longer run time. In this presentation, the focus is placed on bolted connections and the stress build up that could lead to their failure modes – such as web local yielding, bolt shear strength, tension rupture, and flexural yielding. To keep the length of this paper manageable, only a few models are presented. For all of the models in the proceeding sections, the following material properties were used: modulus of elasticity = 29000 ksi, Poisson's ratio = 0.32, and when applicable the coefficient of friction between contact surfaces was set at 0.57. For each model, detailed information about the element type, number of elements, nodes, and degrees of freedoms associated with each node and the model run-time are given in the appendix. This approach was taken to avoid overwhelming those readers not familiar with finite element modeling with information that they may not find beneficial; yet, those with extensive finite element modeling background, can obtain additional modeling information if they so desire.

Finite Element Modeling Using Non-Contact Elements

Since I-beams are used in many branches of the Steel Sculpture [2], we study it first. Figure 4 shows the finite element (FE) model of cross sectional area of an I-beam for which the lower flange is fixed, and the load is applied to the upper portion of the flange. Depending on the way the load is applied and cross-sectional characteristics of the beam, several modes of failure could result. They include *web local yielding, local web buckling, web compression buckling*, or *web crippling*.

Web local yielding (Figure 5 (left)) is caused by compressive force acting on the beam perpendicular to the beam flange. This compressive force causes the web to develop a stress greater than or equal to the yield limit of the material and results in compressive crushing of the beam's web. Local web buckling (Figure 5 (right)) occurs when a member is slender and not stable enough to properly support the loading, and as the result it will buckle. Because of this failure mode, the slenderness ratio must be checked. Web compression buckling occurs when a concentrated force, distributed through a bearing plate to lower the applied stress, becomes too large for the web of the beam. This causes the beam to buckle out-of-plane

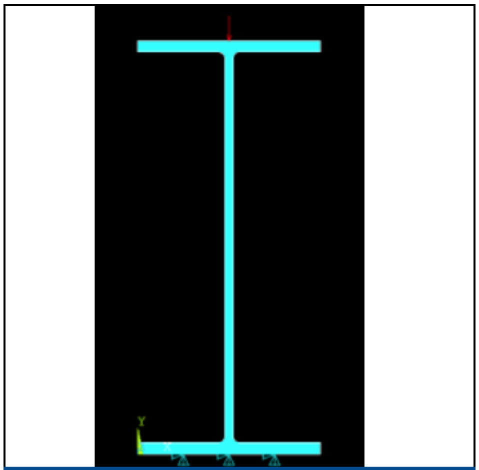

Figure 4. A two-dimensional FE model of an I-beam, note the restraints at the bottom flange

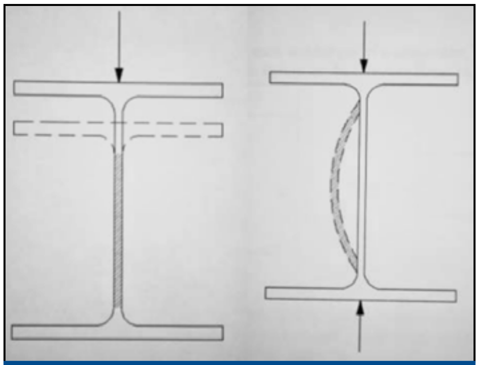

Figure 5. Web local yielding (left) and local web buckling (right) [1]

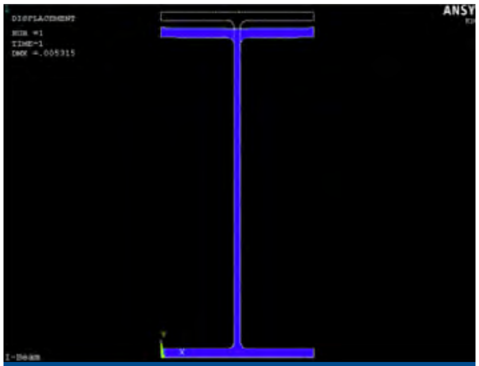

Figure 6. Deflection of the beam

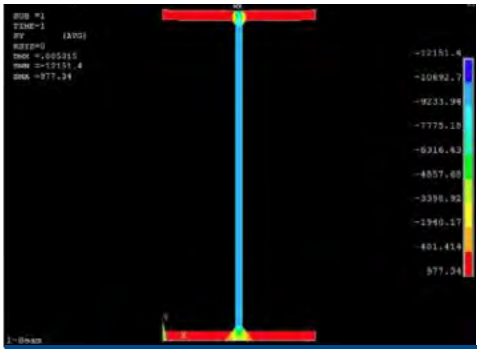

Figure 7. Y-component of the stresses

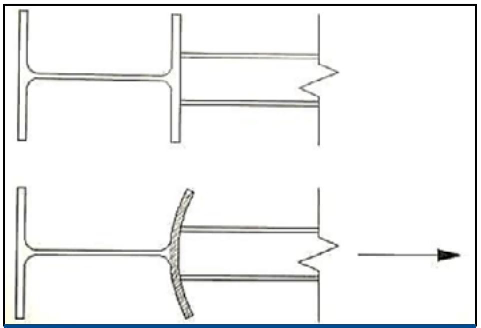

Figure 8. Flange local bending [1]

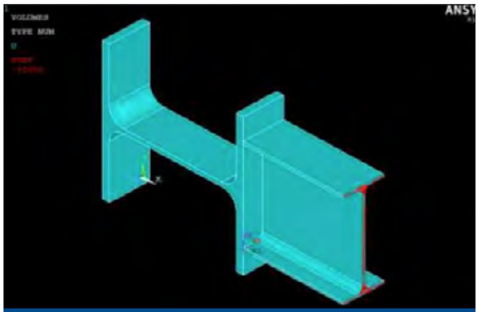

Figure 9. FE modeling of Flange local bending

similar to local web buckling. Web crippling occurs due to concentrated compressive force acting on both flanges in line with the web. When the compressive force is large enough, the web of the beam will buckle similar to local web buckling.

Web Local Yielding – We model the stress build-ups that would lead to web local yielding of a W 16 x 36 wide flange subjected to a compressive load of 10,000 psi. The results for this model are shown in Figures 6 and 7. The displacement solution had a maximum value of 0.005 in. (Figure 6), and the stress in elements along the web portion of the beam have value of 10,000 psi (SY = -10,000), which are consistent with the applied compressive load.

Flange Local Bending is caused by a tensile force acting perpendicular to a beam's flange, resulting in an increased stress in the flange (Figure 8). Two different sizes of wide flange beams were used to model this situation (see Figure 9). The beam on the left-hand-side is a W14 x 90, whereas, the beam on the right-hand-side is W10 x 15. A load of 10,000 psi was applied to the second beam (W10 x 15) on the right-hand-side on its cross-sectional area. The results for this model are shown in Figures 10 and 11.

The deformed shape for this model is shown in Figure 10 with a maximum deformation of 0.02 in. The stress solutions have a maximum value of 47,974 psi (within the yield strength) and a minimum stress of 0.1829 psi at the positions shown in Figure 11. Gupta [4] in his book entitled *Principles of structural design, wood, steel, and concrete* discusses non-compact flange sections and reports results that are similar to the solution given here. Also note from the deformation solution shown in Figure 10 that the right-hand-side flange of the W14 x 90 beam (on left side) is bent.

Finite Element Modeling with Contact Elements

Next, a FE model for Connection 1 of the Steel Sculpture [2], which consists of C (channel) and L (angle) shape members with three 3/4 inch bolts, was created. Note that for this connection, the C channel will carry the design load. After each piece for Connection 1 was modeled (Figures 12-14), they were assembled as shown in Figure 15.

The base of the L shape member was fixed to the main beam, and a load of 100 psi was applied to the top surface of the C channel. For the given model, the maximum deformation occurred at the front tip of the C channel (see Figures 16 and 17). Also note that the C channel experienced torsion as the result of the applied load.

For this model the Von Mises stresses were also computed and are shown in Figures 18 and 19.

The X, Y, and Z components of the stresses are shown in Figures 20 through 25.

Next, we will examine *Prying Action* using a finite element model.

Prying action (Figure 26) is a phenomenon in which additional tension forces are induced in the bolts due to the deformation of the connection near the bolt. Flexibility of the connected parts within the grip of the bolts creates these additional tension forces. An example from *Principles of Structural design, Wood, Steel, and Concrete* by Gupta [4] was chosen to develop a finite element model for the prying action. A non-compact section, the W12 x 65 wide flange beam was chosen for this example. The beam has a total height of 12⅛ inch, and the lower and upper flanges are both 12 inches wide, with flange and web thicknesses of 5/8 inch each. A full cross section for a W12 x 65 beam was modeled to be the main part (the upper part). The T- portion of the cross sectional area was modeled to be the part that carries the applied load (the lower part). These two parts were then connected by two 3/4 inch-nominal-diameter bolts (Figure 27). A deformation similar to Figure 26 was sought. Therefore, we fixed the upper beam and applied a load of 10,000 psi to the web of lower beam. The results for this model are shown in Figures 28 and 29. This model had a maximum deformation of 0.08 in. Next, we will examine *Bolt Shear Strength* phenomenon using a FE mode.

Bolt Shear Strength (Figure 30) is applicable to each bolted ply of a connection that is subjected to shear. The

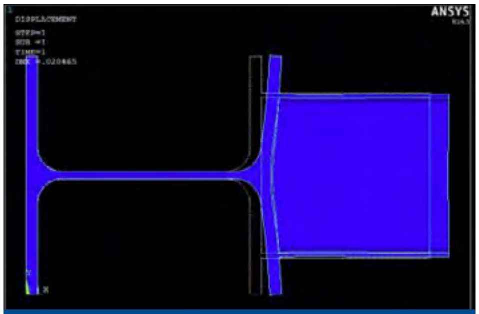

Figure 10. Deflection for flange local bending

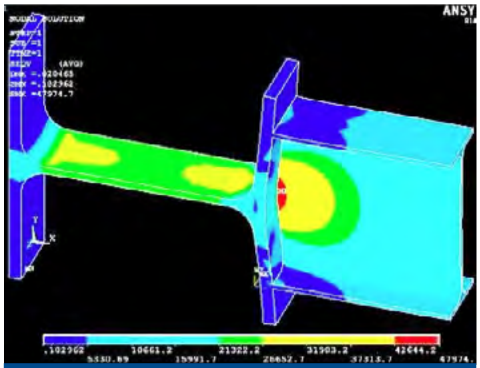

Figure 11. Von Mises Stresses for flange local bending

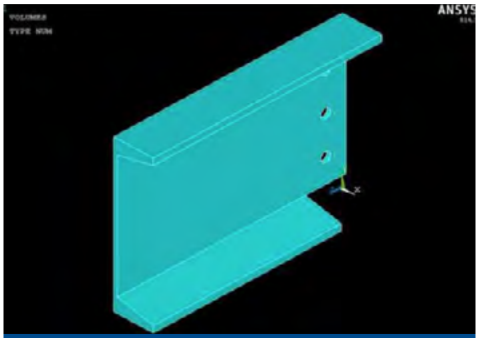

Figure 12. The solid model of C channel in ANSYS

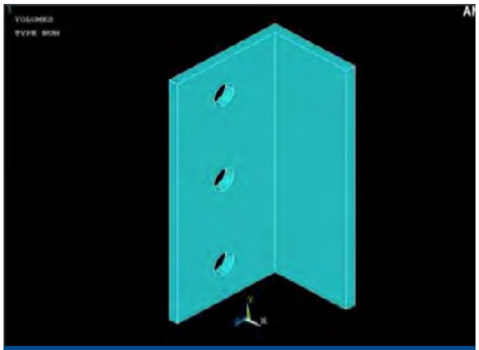

Figure 13. The solid model of L shape member in ANSYS

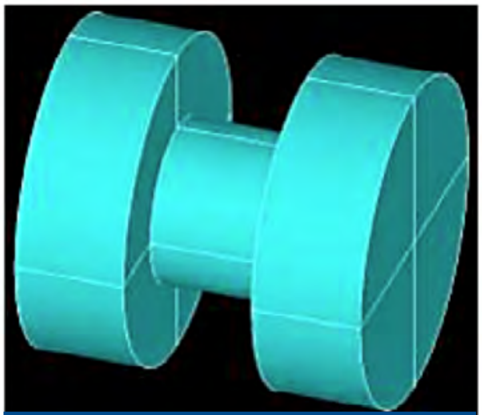

Figure 14. The solid model of a bolt in ANSYS

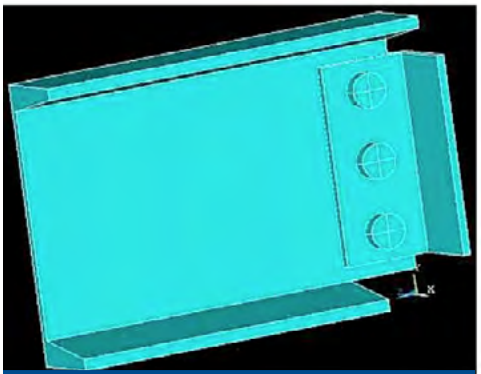

Figure 15. The solid model of the Connection 1 in ANSYS

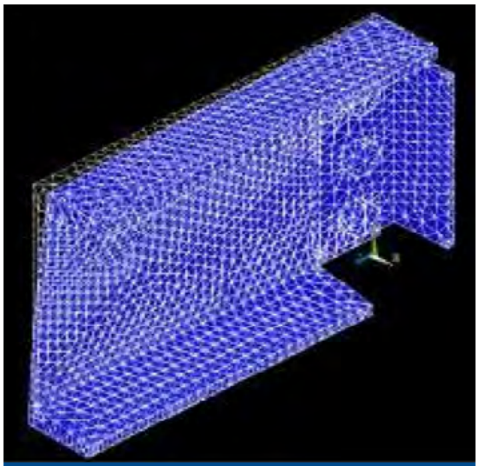

Figure 16. Deflection of the Connection 1 (isometric view)

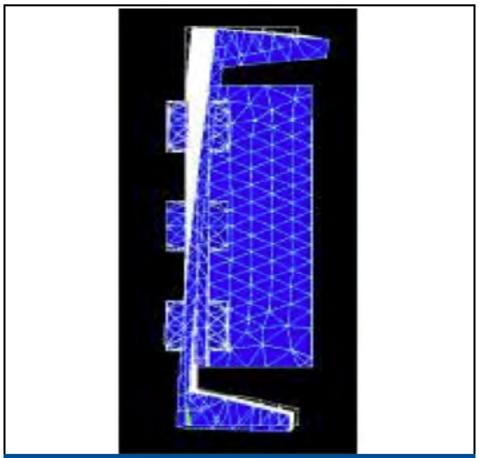

Figure 17. Deflection of the Connection 1 (side view)

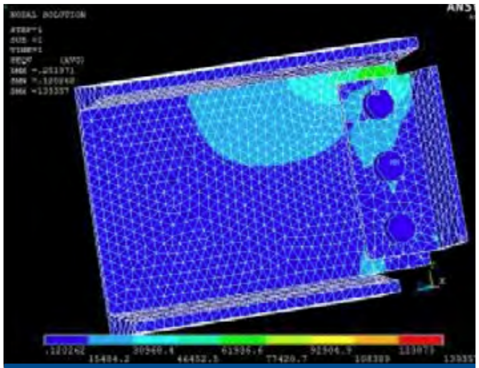

Figure 18. Von Mises Stresses (front view)

shear strength of a bolt is directly proportional to the number of interfaces (shear planes) between the plies within the grip of the bolt that a single shear force is transmitted through. Single shear occurs when the individual shear force is transmitted through bolts that have two plies within the grip of the bolt. Additional plies further distribute the shear force. Three plies of material represent two shear planes with the bolt or bolt group in double shear and with effectively twice the strength as single shear.

The finite element model used to demonstrate this phenomenon consisted of two plates and two bolts. Let the left-hand-side plate be denoted as plate number one, and the right-hand-side plate be plate number two (Figure 31). Also, the left-hand-side bolt is designated as bolt number one, and the bolt on the right side is bolt number two. Plates are 8 inch long and 4 inch wide, with a thickness of 0.5 inch. Two holes were drilled through both plates and the 3/4 inch-diameter bolts were added. A load of 10,000 psi was applied to the area on the right edge of the second plate as shown in Figure 31. The results for this model are shown in Figures 32 through 37.

As shown in Figure 32, note the plate is bent downward in the Y-direction and the deflection along the X-axis is not visible. The solution to this problem yielded a maximum deformation of 0.294 inch and the maximum and minimum Von Mises stresses shown in Figure 33.

The solutions for each plate and bolt are shown in Figures 34, 35, 36, and 37. Note that even though the plates are subjected to axial loads, bending would also occur, because the loading creates a couple. Also, note, from stress solutions, the lower plate experiences a maximum stress around hole number one as shown.

Another example of bolted connection is one in which a *set of bolts* connect two members. Figure 38 is the example of two plates that are connected by a set of bolts. In the situation shown in Figure 38, six bolts, connect two

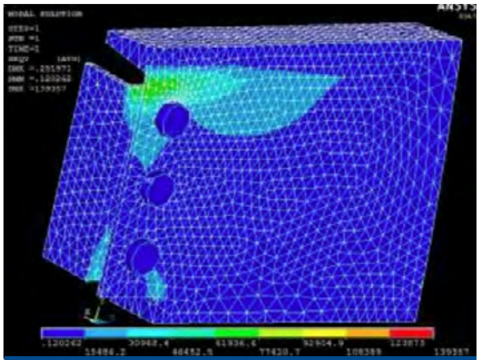

Figure 19. Von Mises Stresses (back view)

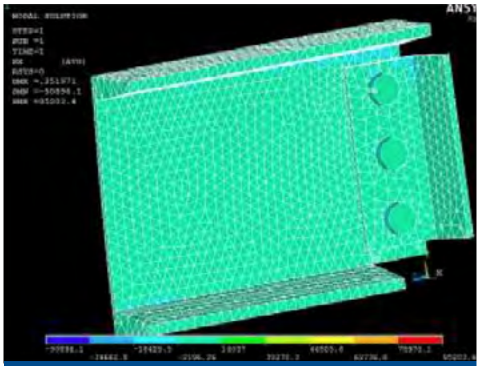

Figure 20. The X-component of stresses (front view)

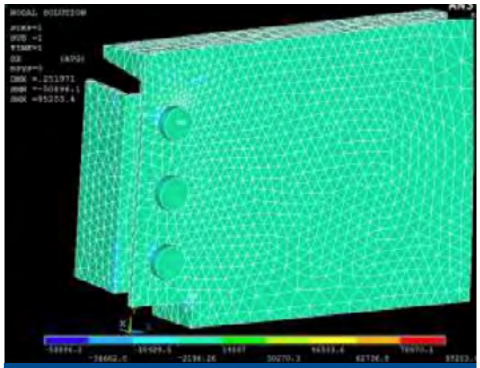

Figure 21. The X-component of stresses (back view)

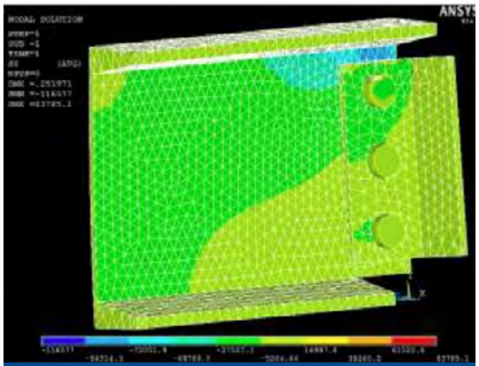

Figure 22. The Y-component of stresses (front view)

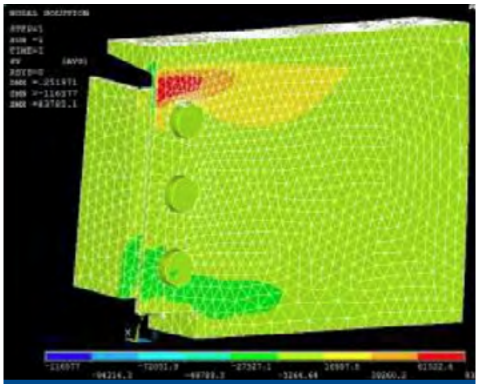

Figure 23. The Y-component of stresses (back view)

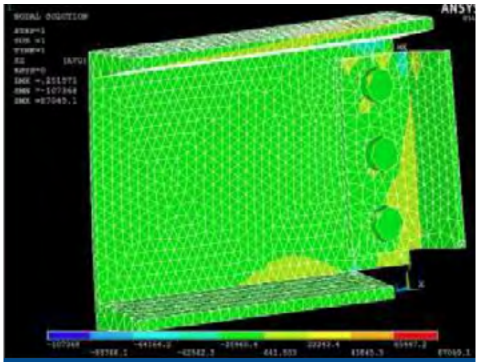

Figure 24. The z-component of stresses (front view)

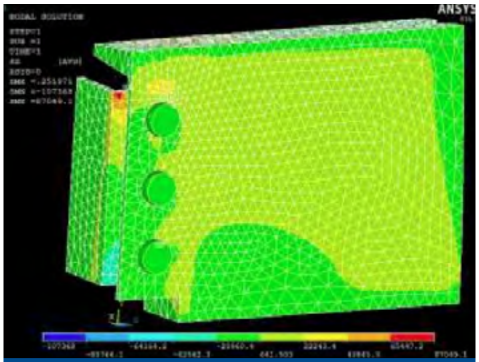

Figure 25. The z-component of stresses (back view)

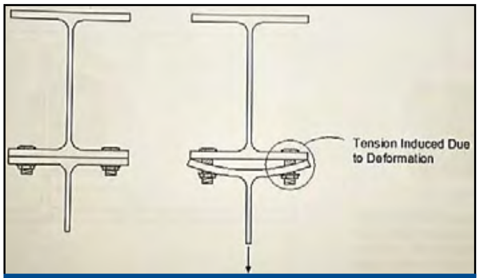

Figure 26. Prying Action [1]

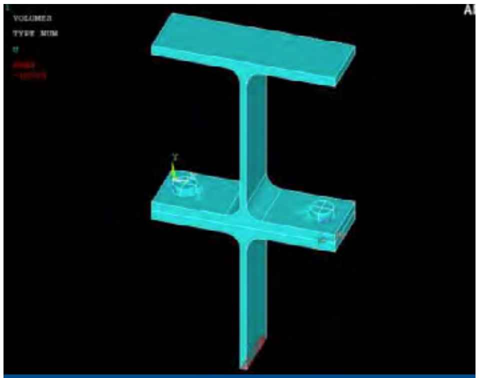

Figure 27. A FE model for Prying Action

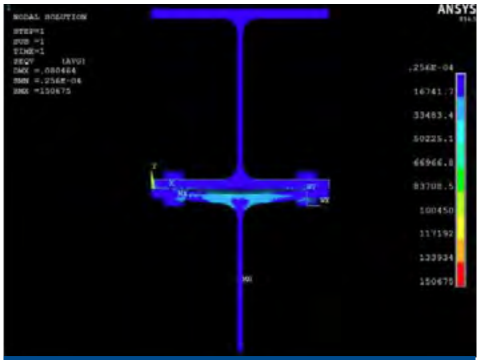

Figure 28. FE solutions for Prying Action

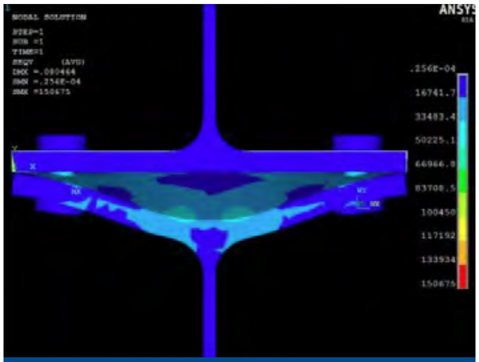

Figure 29. FE solutions for Prying Action (close-up view)

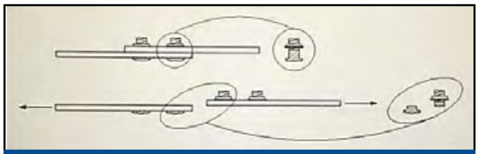

Figure 30. Bolt shear strength [1]

plates. The bigger plate on the left-hand-side (plate 1) is fixed on its left edge. The smaller rectangle plate on the right-hand-side (plate 2) is loaded by a tension load on its right edge. This type of bolted connection may experience several types of failures.

Block Shear Rupture (Figure 39) is a limit state in which the failure path includes an area subject to shear and an area subject to tension. This limit state is so named because the associated failure path tears out a *block* of material [1]

Bolt Bearing Strength (Figure 40) is concerned with the deformation of material at the loaded edge of the bolt

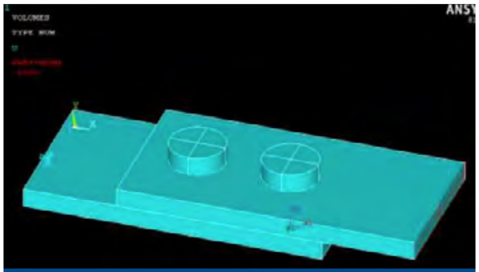

Figure 31. A FE model of bolt shear modeling

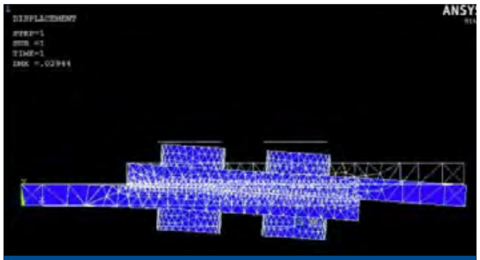

Figure 32. Deflection of bolt shear strength model

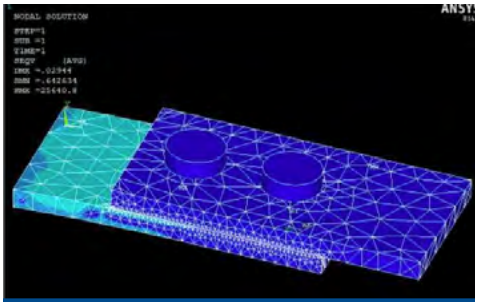

Figure 33. Von Mises stresses for the FE model of bolt shear

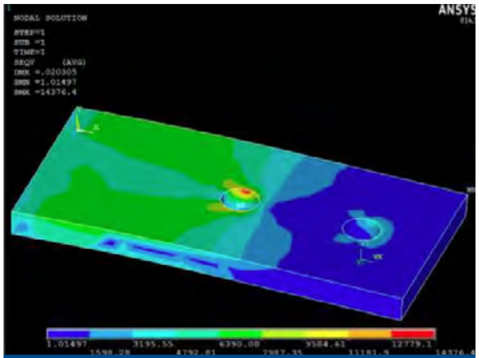

Figure 34. Von Mises stresses in lower plate

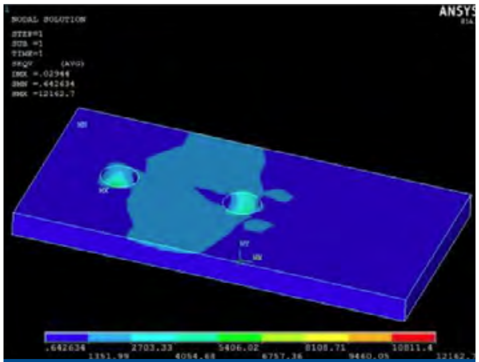

Figure 35. Von Mises stresses in the upper plate

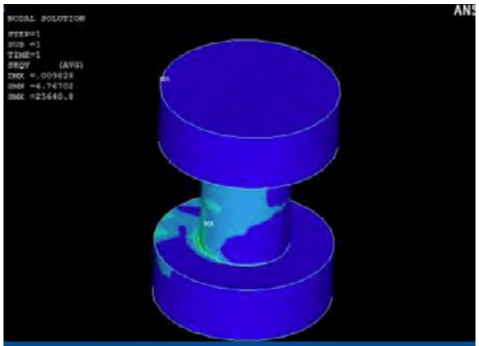

Figure 36. Von Mises stresses in the first bolt

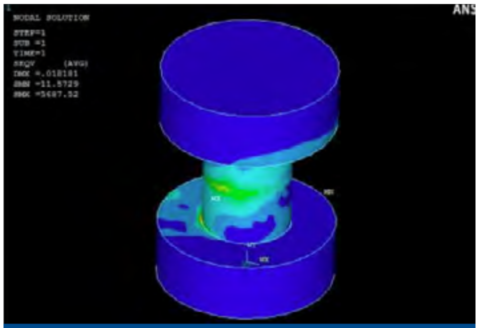

Figure 37. Von Mises stresses in the second bolt

holes. Bearing capacity of the connection is influenced by the proximity of the bolt to the loaded edge or the spacing between two bolt holes [1].

Tension Rupture (Figure 41) is a function of the effective net area. The net area is the reduced gross area due to bolt holes or notches. This net area is further reduced to account for the effects of shear lag. Shear lag occurs when the tension force is not evenly distributed through the cross sectional area of a member. When the section has a stress greater than or equal to the ultimate stress of the material, tension rupture is said to have occurred [1].

Tension Yielding (Figure 42) is a function of the gross cross-sectional area of the member subjected to tension load. When the section has a stress greater than or equal to the yield stress of the material, tension yielding is said to have occurred [1].

The finite element model used to demonstrate the above phenomena consists of 2 plates and 6 bolts (see Figure 43). A load of 10,000 psi was applied on the right-hand-side edge of the second plate. The results for this model are shown in Figures 44 through 48.

The deformation result shows a maximum deformation of 0.185 inch. The front view does not show much deformation, however when viewed from the bottom direction, the model is bent as shown in Figure 44. The Von Mises stresses are shown in Figures 45, 46, 47, and 48, with the maximum stress occurring at the holes on the left-hand side. The von Mises stress for plate number one (the left plate) is shown in Figure 46. The maximum stress occurs at the upper and lower portion of the holes. The Von Mises stress for plate number two (the right plate) is shown in Figure 47. The Von Mises stress for the bolt on the upper left-hand-side connection is shown in Figure 48.

Next, using a FE model (Figure 49), we demonstrate *shear rupture* and *shear yielding* phenomena. In this example an L shape (angle) bracket is used to connect two wide flange beams by four bolts as shown in Figure 49. This connection's primary function is to transfer shear from one beam to another. Due to the double coping (removal of the flanges and part of the web at both ends), the moment of inertia of the coped beam is reduced. Hence, it is necessary to verify that the coped beam can support the moment and shear in the reduced cross section region.

Shear Rupture (Figure 50) is a function of the effective net area. The net area is the reduced gross area due to bolt holes or notches. When the section has a stress greater than or equal to the ultimate stress of the material, shear rupture would occur.

Shear Yielding (Figure 51) is a function of the gross cross-sectional area of the member subjected to a shear load. When the section has a stress greater than or equal to the yield stress of the material, shear yielding could occur.

For the finite element model, two wide flange beams W16 x 36 were connected with an L shape (angle) bracket (see Figure 52). The beam on the right-hand-side is designated as beam number one, and the beam on the left-hand-side is beam number two. Four holes with diameter of 3/4 in. were drilled and bolts were added. The areas of upper and lower flanges of the beam number two were fixed and a load of 1,000 psi was applied to beam number one. The results for this model are shown in Figures 53 through 55.

The deformed shape for this model is shown in Figure 53 with the maximum deformation of 0.03 in. Also, the maximum Von Mises stress occurs at the position shown in Figure 54, and the von Mises stress distribution for the L-shape bracket is shown in Figure 55.

Next, a load of 100 psi was applied at the lower flange of beam number one (Figure 56) to simulate a moment loading to show flexural rupture and flexural yielding.

Flexural Rupture (Figure 57) occurs in moment connections where the connection must be designed to carry an applied moment. Rupture occurs when the stress caused by the applied moment is greater than or equal to the rupture strength of the material [1].

Flexural Yielding (Figure 58) needs to be checked for situations wherein a beam is coped. This is necessary because the reduced section modulus of the remaining beam cross section may significantly reduce the flexural strength of the member. Flexural yielding can also occur in

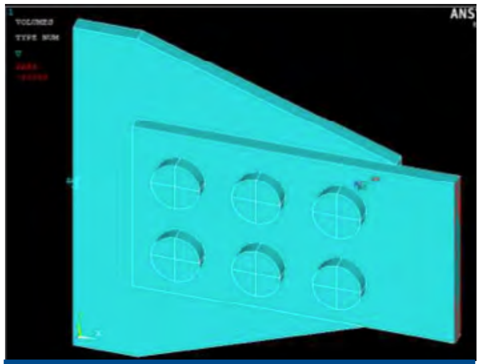

Figure 38. A set of bolts connecting two plates

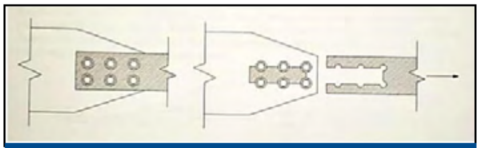

Figure 39. Block Shear Rupture [1]

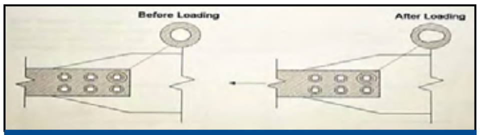

Figure 40. Bolt Bearing Strength [1]

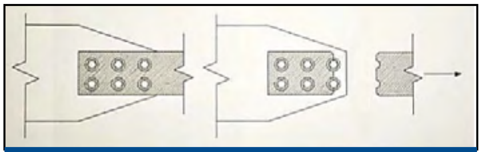

Figure 41. Tension Rupture [1]

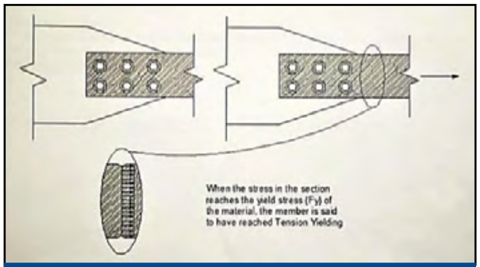

Figure 42. Tension Yielding [1]

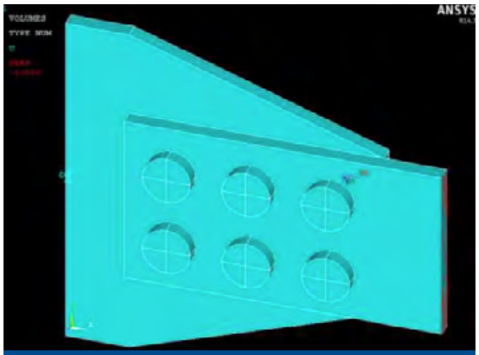

Figure 43. The FE model of the plates and the bolts

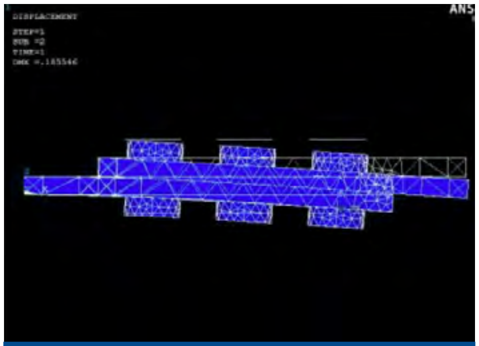

Figure 44. The deflection of the plates and the bolts

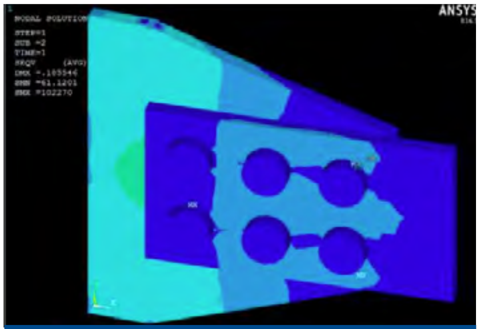

Figure 45. Von Mises stresses

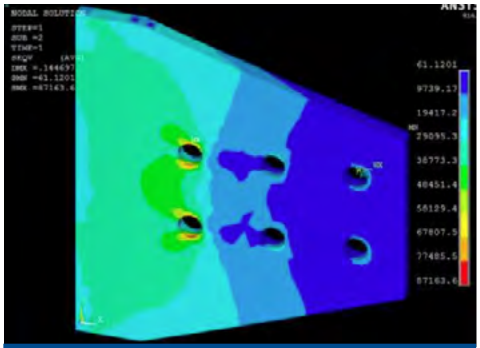

Figure 46. Von Mises stresses for plate number one

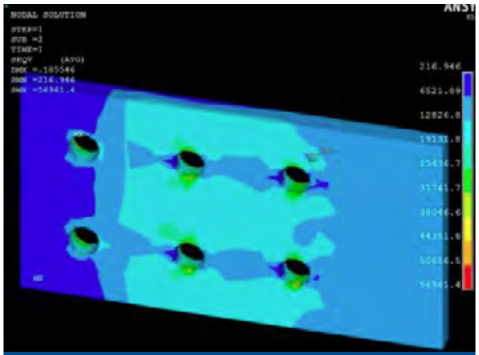

Figure 47. Von Mises stresses for plate number 2

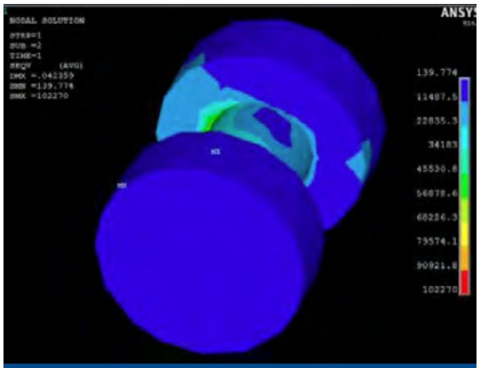

Figure 48. Von Mises stresses for the bolt on the upper left connection

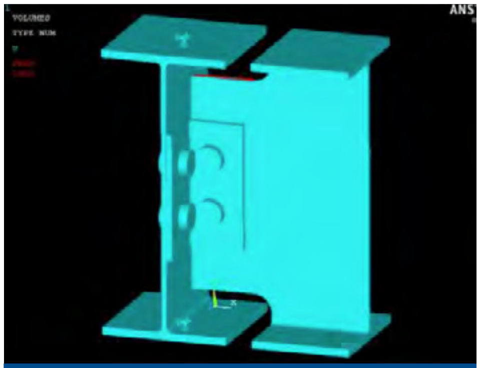

Figure 49. A FE model for shear load connection

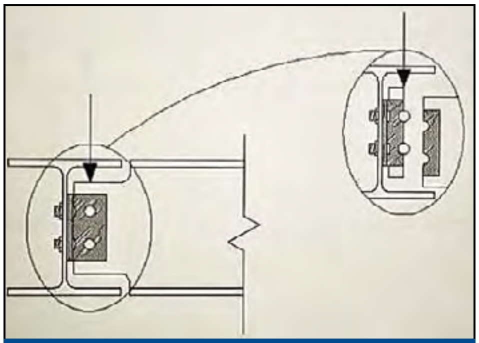

Figure 50. Shear rupture [1]

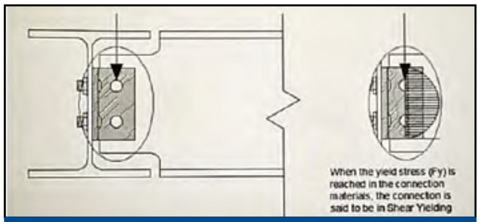

Figure 51. Shear yielding [1]

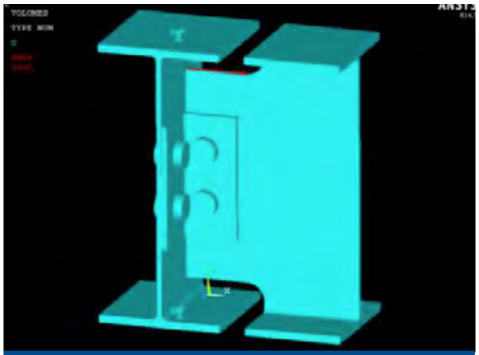

Figure 52. A FE model for shear rupture

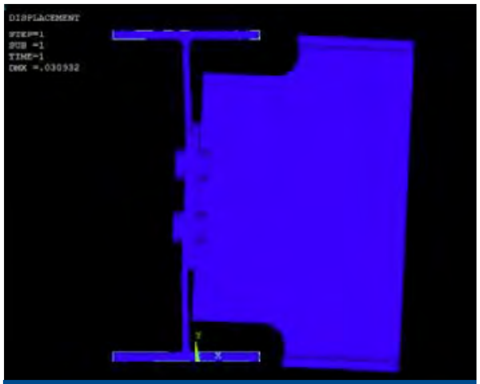

Figure 53. The deflection of beams and the connecting bracket

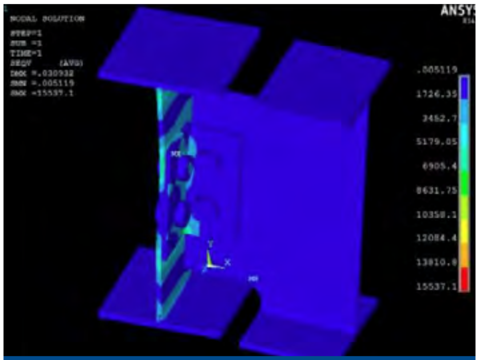

Figure 54. The Von Mises stresses

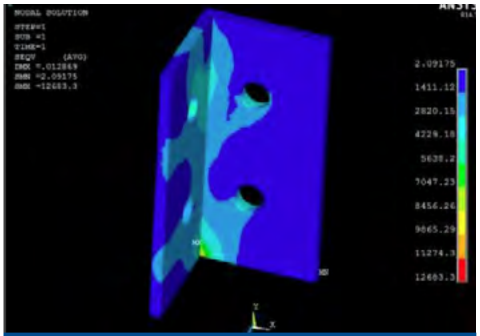

Figure 55. The Von Mises stresses for the L-shaped bracket

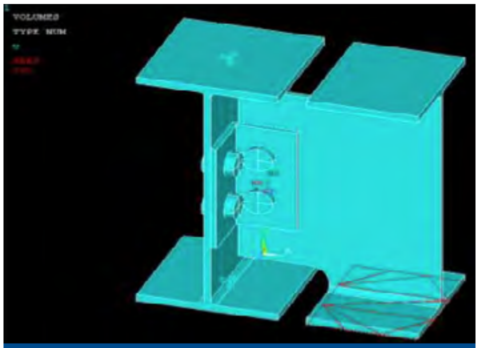

Figure 56. A FE model for moment load connection

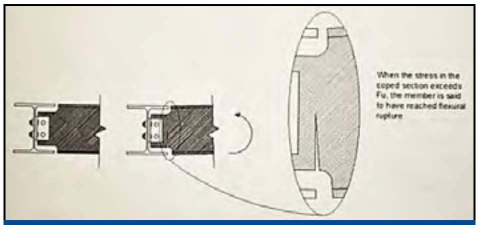

Figure 57. Flexure rupture [1]

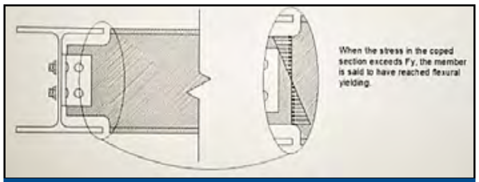

Figure 58. Flexural yielding [1]

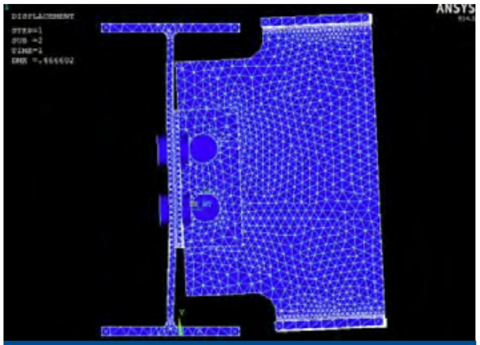

Figure 59. A FE model for moment load connection

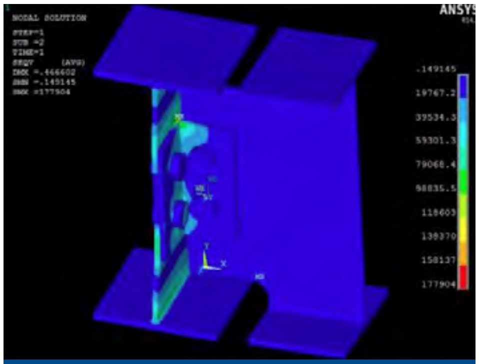

Figure 60. The Von Mises stresses for the entire system

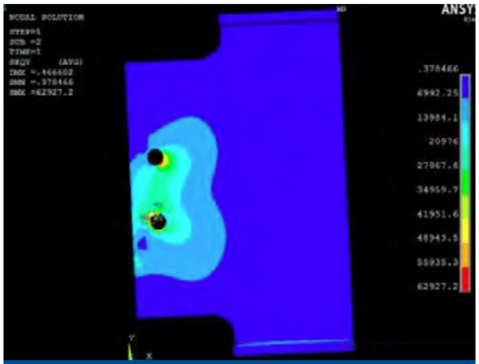

Figure 61. The Von Mises stresses for I-beam number 1

moment connections where the connection must be designed to carry an applied moment. Yielding occurs when the stress caused by the applied moment is greater than or equal to the yield strength of the material [1].

The finite element model for this situation was created as discussed previously. The results for this model are shown in Figures 59 through 61. As expected, the load created a counter-clockwise moment that made the model bent upward (Figure 59) with a maximum deformation of 0.466 inch as shown. The Von Mises stresses are shown in Figures 60 and 61.

Closing Remarks

The finite element models used in this study are intended as visual tools to enhance understanding of local deformations and stress distribution in connections. For example, referring to Figure 32, one would note that even though the plates are subjected to axial loads, bending would also occur, because the loading creates a couple. As one can see the tendency of the top plate is to move the bolts to the right and bend them whereas the bottom plate tends to move the bolts to the left. These are important observations that are often missed when studying connections. For example in Figure 30 [1] the bending of the plates are not demonstrated when each bolted ply of the connection is subjected to shear. Each of the finite element cases presented here could also be analyzed further in a similar fashion to offer additional insight into local deformations and stress distributions in connections. Finally, the models were kept simple, so undergraduate students with some experience in finite element analysis background could follow and generate them on their own and perform additional simulations if they so desire.

References

1. Green, Perry S., Sputo, Thomas, and Veltri, Patrick *Connections Teaching Toolkit, A Teaching Guide for Structural Steel Connections*, American Institute of Steel Construction, Inc.
2. Moaveni, S. and Chou, K. (2015) *Teaching Steel Connections Using an Interactive Virtual Steel Sculpture*, Journal of STEM Education: Innovations and Research, Vol. 16, No.4, pp 61-68, 2015.
3. Moaveni, Saeed (2008), *Finite Element Analysis, Theory and Application with ANSYS*, Third edition, Pearson Prentice Hall.
4. Gupta, Ram S. (2011), *Principles of Structural Design, Wood, Steel, and Concrete*, CRC Press.

Saeed Moaveni is currently Professor of Mechanical Engineering at Minnesota State University, Mankato. Dr. Moaveni has nearly 30 years of experience in teaching and research in mechanical and civil engineering related areas. Professor Moaveni also has served as Dean of the David Crawford School of Engineering at Norwich University and Chair of the Mechanical and Civil Engineering Department at Minnesota State University. He is a licensed Professional Engineer (PE) in the State of New York, and a member of various engineering societies.

Dr. Karen C. Chou is Assistant Chair and Clinical Professor of Civil and Environmental Engineering at Northwestern University. She is a Fellow of ASCE and a registered Professional Engineer in seven states. Dr. Chou has 35 years of academic and engineering experience. She was recognized numerous times by the American Society of Civil Engineers (ASCE) for her excellent service to the student chapters. Dr. Chou received the Charles W. Britzius Distinguished Engineer Award from the Minnesota Federation of Engineering, Science, and Technology Societies and the Civil Engineer of the Year from the Illinois Section ASCE.

Mr. Supachard Krudtong is an instructor at the Rajamangala University of Technology Lanna in Chiang Mai, Thailand.

APPENDIX

DETAILED INFORMATION FOR EACH MODEL

Figures 4, 6, and 7 –The PLANE183 element of ANSYS, which is a higher order 2-D, 8 node plane element with quadratic displacement behavior, was used to model a W16 x 36 wide flange. The W16 x 36 beam is considered as a compact cross-sectional shape with web local yielding as a failure mode. The lower flange of the W-beam was fixed, and a uniform load of 10,000 psi was applied along the thickness of the web. The mesh size was set at level 2 (fine mesh) and using the smart size control of ANSYS, the model resulted in 915 elements and 3,280 nodes.

Figures 9 through 11 – Two different sizes of wide flange beams were used to model this situation. The beam on the left-hand-side is a W14 x 90, whereas, the beam on the right-hand-side is W10 x 15. The SOLID187 element was used in this model. Element SOLID187 is a higher order 3-D, 10-node element with a quadratic displacement behavior. The element is defined by 10 nodes having three degrees of freedom at each node: translations in the nodal X, Y, and Z directions; only elastic behavior was modeled. The elements on the interface were connected using the "glue" option of ANSYS. The boundary condition for the beam on left-hand-side (W14 x 90) was defined by setting the displacement field equal to zero at its fixed area. A load of 10,000 psi was then applied to the second beam (W10 x 15) on the right-hand-side. The load was applied on the cross section area of the W10 x 15 beam. Again, the mesh size was set at level 2 (fine mesh) using the smart size control of ANSYS. This resulted in a model with 15,210 elements and 28,704 nodes.

Figures 12 through 25 – The FE model for Connection 1 of the Steel Sculpture [2], which consists of C (channel) and L (angle) shape members with three 3/4 inch bolts, was created. Note that for this connection, the C channel will carry the design load. The bolted connections were modeled with contact elements between the bolts and the flange. The solid model of the C channel was created first (Figure 12), followed by three holes that were drilled for the bolts. Next, the L shape member was created (Figure 13) next to C channel and finally the three bolts were created to connect the two members. For this model we used a C12 x 20.7 designation channel. To simplify the model and to focus on the contact surface areas between structural members the threaded sections of bolt and nut were ignored. Furthermore, when bolt and nut are fastened, the bolts and nuts were modeled as if they fit together like they are the same piece (see Figure 14). Also, note that there are 3 bolts and nuts (3 rivets) in Connection number 1 of the Steel Sculpture. After each piece for Connection 1 was modeled, they were assembled as shown in Figure 15.

The SOLID187 of ANSYS was used to model this connection. The SOLID187 element is defined by 10 nodes having three degrees of freedom at each node: translations in the nodal X, Y, and Z directions. The base of the L shape member was fixed to the main beam. A load of 100 psi was applied to the top surface of the C channel. We used smaller load value and the level 10 of the size control (coarse) to keep the model size reasonable so that it could be run on the computer and yield results in reasonable amount of time.

Contact Pairs – Connection number 1 consists of five solid pieces (a C channel, an L shape member, and three 3/4 inch bolts). To avoid the penetration between solid pieces when they are deformed, contact pairs were set up between the C channel and the 3 bolts (rivets in this model) and the L shape member and the bolts. TARGET 170 and CONTA 174 elements were used in this analysis. In ANSYS, TARGE 170 is used to represent various 3-D "target" surfaces for the associated contact element such as CONTA 174. This model resulted in 123,954 nonlinear equations and took approximately 10 minutes for the solution to converge.

Figures 27 through 29 – An example from *Principles of Structural design, Wood, Steel, and Concrete* by Gupta [4] was chosen to develop a finite element model for prying action. A non-compact section, the W12 x 65 wide flange beam was chosen for this example. The beam has a total height of 12 inch, and the lower and upper flanges are both 12 inches wide, with flange and web thicknesses of 5/8 inch each. A full cross section for a W12 x 65 beam was modeled to be the main part (the upper part). The T-portion of the cross sectional area was modeled to be the part that carries the applied load (the lower part). These two parts were then connected by two 3/4 inch-nominal-diameter bolts (Figure 27). The SOLID187 element was used for this model. A deformation similar to Figure 26 was sought. Therefore, we fixed the upper beam and applied a load of 10,000 psi to the web of lower beam. The model was meshed into 44,342 elements. Three pairs of contacts were included in the analysis: contact between the lower beam and the bolts, the upper beam and the bolts, and the lower and the upper beams.

Figures 31 through 37 – The finite element model used to demonstrate this phenomenon consisted of two plates and two bolts. Let the left-hand-side plate be denoted as plate number one, and the right-hand-side plate be plate number two (Figure 31). Also, the left-hand-side bolt is designated as bolt number one, and the right bolt is bolt number two. Plates are 8 inch long and 4 inch wide, with a thickness of 0.5 inch. Two holes were drilled through both plates and 3/4 inch diameter bolts were added. The SOLID187 element was used in this model. The fixed boundary condition was defined at the left end of first plate by setting the displacement field to be zero. The load was applied to the right end of the second plate as shown in Figure 31. The area on the right edge of the second plate was subjected to a load of 1,000 psi. The meshing resulted in a model with 48,033 elements. Three pairs of contacts are used for this model. They are the contact pairs between plate number two and both bolts, both bolts and plate number one, and plate number two and plate number one.

Figures 38, 43 through 48 – The finite element model used for this situation consists of 2 plates and 6 bolts (Figure 43). The SOLID187 element was used to model this problem. The left edge of the plate number one was fixed as shown in Figure 43. A load of 10,000 psi was applied on the right edge of the second plate. The model was then meshed into 40,126 elements. Contact pairs were created for three pairs. They were the contact pairs of the second plate (plate on right) and all bolts, the contact pair of all bolts and the first plate (plate on left), and the contact pair of right plate and left plate. The run time for this model was approximately 15 minutes.

Figures 49, 52 through 56, and 59 through 61 – Two wide flange beams W16 x 36 were connected to an L shape (angle) bracket. The beam on the right is designated as beam number one, and the beam on the left is beam number two. Four holes with diameter of 3/4 in. were drilled and bolts were added. The SOLID187 element was used to model this situation. The areas of upper and lower flanges of the beam number two were fixed and a pressure load of 1,000 psi was applied to beam number one. The model was meshed into 71,443 elements. Contact pairs were created for 6 pairs: the beam number one and two bolts on right side, the two bolts on the right side and L-shape bracket, the L-shape bracket and beam number one, the beam number two and the L-shape bracket, the L-shape bracket and the two bolts on left side, the two bolts on the left side and beam number one. The run time for this model was approximately 20 minutes.

Next, a load of 100 psi was applied at the lower flange of beam number one (Figure 56). The run time for this model was also approximately 20 minutes.