

Numerical Experiments for a Mechanics of Materials Course

SAEED MOAVENI

Department of Mechanical Engineering, Mankato State University, Mankato MN 56002-8400, USA.

E-mail: saeedmoaveni@ms1.mankato.msus.edu

In recent years, the increased exposure to computers have made it possible for students to: 1) solve more complex problems; and 2) gain additional insight into the fundamentals through numerical experimentation and visualization. This paper is devoted to the analysis of strains, stresses and deformation of some structural members through such experimentation. The numerical experiments reported here will introduce students to the importance of boundary conditions, force equilibrium conditions, the stress/strain relationship for a given material, axial loading, stress concentration, pure bending, transverse loading and combined loading. ANSYS and parametric programming are used to set up these experiments.

INTRODUCTION

THE PRIMARY concern of any introductory mechanics of materials course should be to provide the students with a clear and thorough understanding of a few basic concepts [1, 2]. Additionally, it is important that students develop the ability to apply these well-understood concepts to practical problems. In recent years, personal computers and workstations have become ordinary tools of engineering students. Because of this exposure, students can now gain additional insight into the fundamentals by numerical experimentation. Numerical experiments designed for an introductory level mechanics of materials class are reported here. The experiments will allow students to explore analysis of strains, stresses and deformation of some structural members. Students can vary boundary conditions or modify applied loading or change the geometry of a structure and observe the results of these changes. ANSYS [3], a comprehensive finite element package was used to set up these experiments in a form of modules. Each unit begins with introductory materials discussing the basic concepts involved and the objectives associated with the numerical experiments. Each unit ends with a summary of the concepts covered in the exercise. Moreover, the modules incorporate the fact that the structures are commonly subject to four general types of loading namely, axial loading, transverse loading, twisting and bending.

Students must also verify findings of the experiments by hand. This approach will reinforce the importance of the fundamentals. The first two labs must be devoted to providing an overview of the ANSYS program and some very basic ideas about finite element analysis. Other labs may be devoted to axial loading; statically indeterminate problems;

stress concentration; pure bending and combined loading. It is important to note that the basic concepts must be taught in a regular lecture setting and the students must first spend time on theory and homework problems. The purpose of these modules is to enhance students' learning by offering them means of experimenting with concepts they may find troublesome in class. In order to keep this paper at a reasonable length, only overview of these modules will be discussed here. The article includes description of some experiments, discussion of results and concluding remarks regarding the ideal group size and how one may integrate these numerical experiments into a mechanics of materials class.

AXIAL LOADING MODULE

In the axial loading module, an example of a power transmission tower is used. It is emphasized

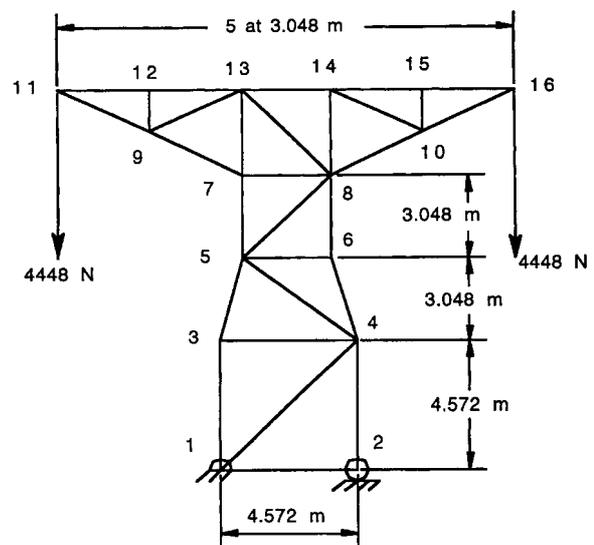


Fig. 1. An example of a power transmission truss.

* Accepted 5 June 1997.

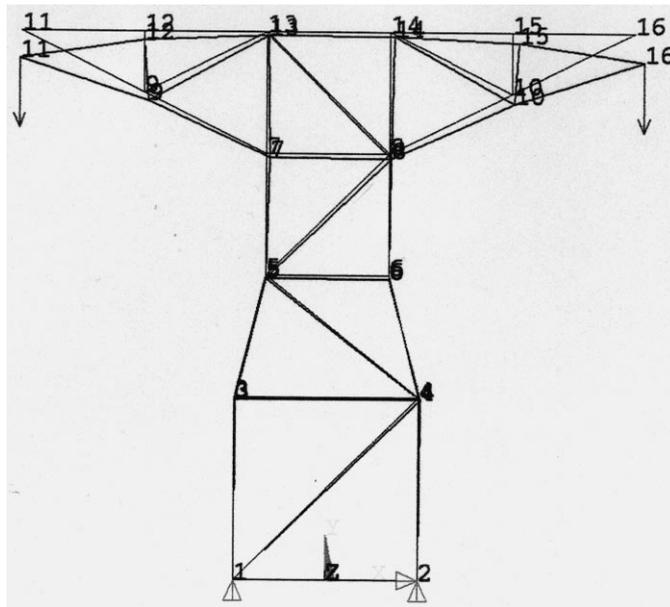


Fig. 2. Deformed shape of the truss.

that trusses play a significant role in offering simple engineering solutions to a number of structural problems. First, a plane truss is defined as a structure whose members lie in a single plane. The forces acting on such a truss must also lie on this plane. Members of a truss are generally considered to be two-force members.

In the numerical experiments that follow it is assumed that the members are connected together by smooth pins. However, it is noted that as long as the center lines of the jointing members intersect at a common point, trusses with bolted or welded joints may be treated as having smooth pins. That is to say, moments at the joints are being neglected. All loads must be applied at the joints. In some

situations, the weight of members are negligible when compared to the applied loads. However, if the weights of the members are to be considered, then half of the weight of each member is applied to the connecting joints. It is brought to the attention of the students that they have studied statically determinant truss problems in their Statics class.

The truss problems were analyzed by the methods of Joints or Sections. However, these methods did not provide information on deflection of the joints or the stresses in each member. Here, they will study deflections of the joints as well as the stresses in each member. The objectives associated with this module are to:

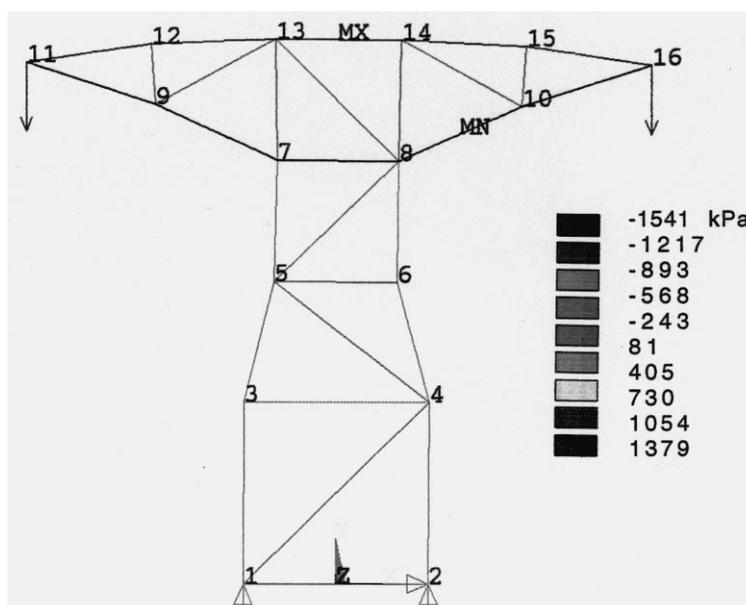


Fig. 3. Stresses in each member.

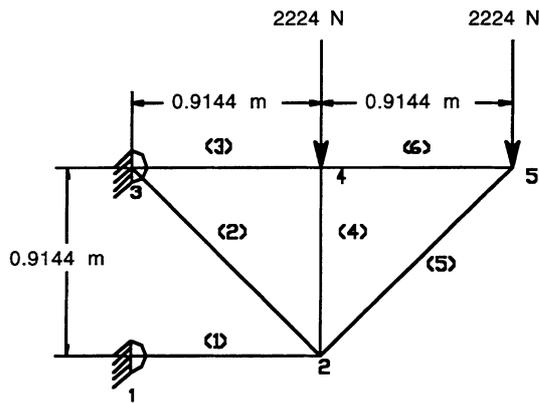


Fig. 4. A balcony truss.

- calculate the stresses, deflections and support reactions of a truss;
- visualize stresses in each member;
- visualize the deformed shape of the truss;
- experiment with changing the material property, magnitude and location of applied forces, and varying the dimensions of some members.

The teaching begins with a Power Transmission-Line Tower as shown in Fig. 1. The members have a cross-sectional area of 64.52 cm^2 and a modulus of elasticity of 200 GPa . We are interested in determining the deflection of each joint, internal forces and stresses in each member and the reaction forces at the base.

As a part of the result, students will observe the deformed shape of the truss and the average stresses in each member as shown in Fig. 2 and Fig. 3, respectively. The values of nodal displacements, internal forces and stresses in each member can be also obtained in a tabular form. After results are viewed, students are asked to verify their findings by applying the equilibrium conditions and check for buckling by hand. This will be discussed in more detail later under the next module. Once students become comfortable with this problem, they can then begin experimenting by changing the magnitude and location of external forces, changing material property or size of some members.

STATICALLY INDETERMINATE MODULE

The module begins with some basic information about what constitutes a statically indeterminate problem. It is stated that when the number of independent equilibrium equations is not equal to the number of unknowns, then the problem is said to be statically indeterminate. The main difference between a statically determinate problem and an indeterminate one depends on the number of equilibrium equations. The statically indeterminate problems generally consist of two types of unknowns:

- (1) forces or moments;
- (2) deformations or strains.

The unknowns are governed by the following relations:

- (a) force equilibrium relations;
- (b) geometric constraints relations;
- (c) force-deformation relations.

In this teaching module, students will experiment with a statically indeterminate balcony truss. The objectives associated with this numerical experiment are to:

- identify statically indeterminate problems;
- calculate stresses, deformations and support reactions for a statically indeterminate problem;
- visualize stresses in each member;
- visualize the deformed shape of the structure.

Students should also experiment with:

- changing the material property;
- varying magnitude and location of external forces;
- changing boundary conditions and its effects on the final results.

The module begins with a balcony truss as shown in Fig. 4. All members are made from wood (Douglas Fir) with a modulus of elasticity of $E = 13.3 \text{ GPa}$ and a cross-sectional area of 51.62 cm^2 . Along with other results, students will observe the deformed shape of the truss and

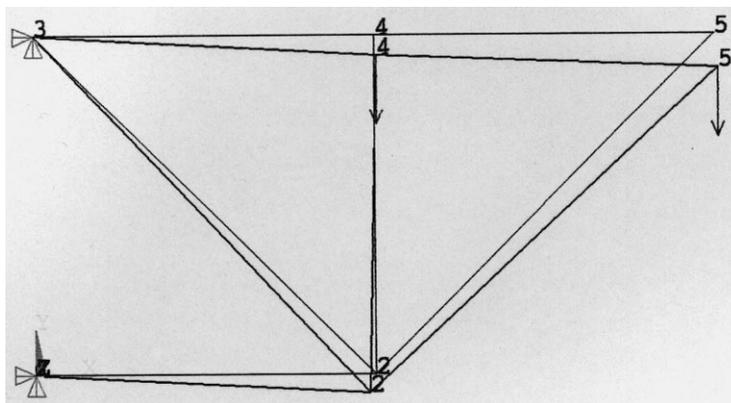


Fig. 5. Deformed shape of the balcony truss.

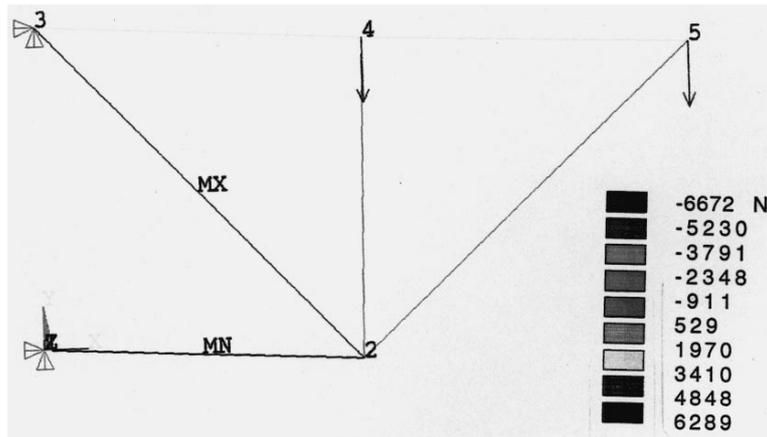


Fig. 6. Internal forces in each member.

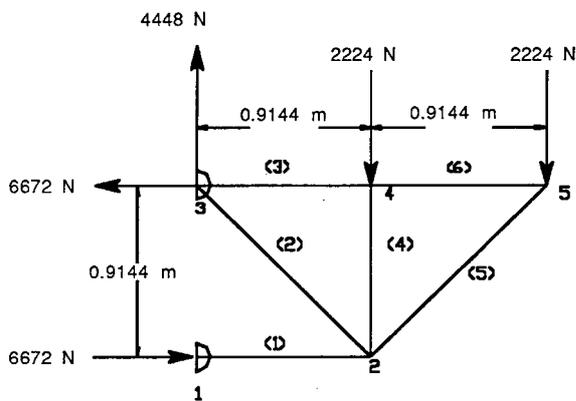


Fig. 7. A free body diagram of the truss.

internal forces in each member as shown in Fig. 5 and Fig. 6, respectively. After the results are viewed, students are asked to verify their findings in various ways. Some examples of how they may go about this follow.

Students are asked to use the computed reaction forces and the external forces to check for static equilibrium. That is:

$$\sum F_X = 0 \quad (1)$$

$$\sum F_Y = 0 \quad (2)$$

$$\sum M_{about\ a\ point} = 0 \quad (3)$$

The reaction forces computed by ANSYS are: $F_{1X} = 6672\text{ N}$; $F_{1Y} = 0$ and $F_{3X} = -6672\text{ N}$; $F_{3Y} = 4448\text{ N}$. As an example, using the Free Body Diagram shown in Fig. 7 and applying the static equilibrium conditions as given by equations (1)–(3), we have:

$$\sum F_X = 0 \quad 6672 - 6672 = 0$$

$$\sum F_Y = 0 \quad 4448 - 2224 - 2224 = 0$$

$$\sum M_{node\ 1} = 0 \quad (6672)(0.9144) - (2224)(0.9144) - (2224)(1.8288) = 0$$

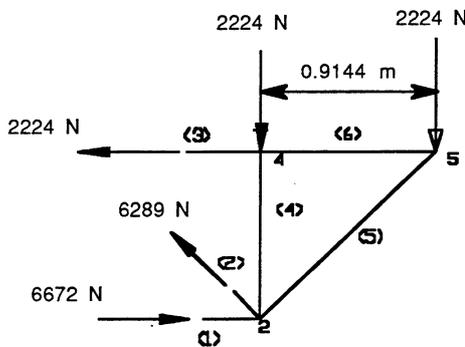


Fig. 8. Cutting a section through the truss.

Another way of checking for the goodness of

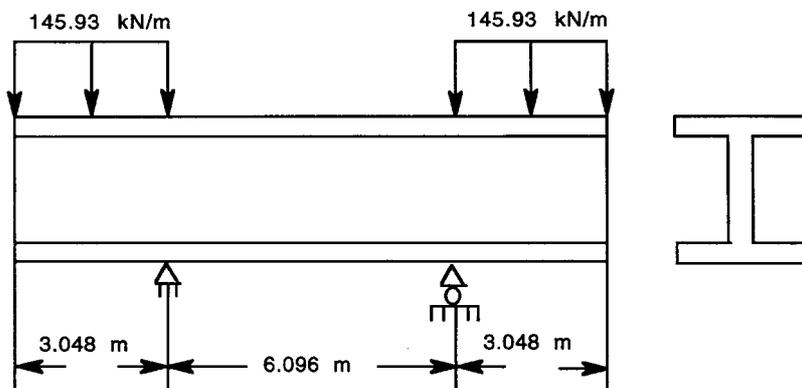


Fig. 9. A beam subjected to distributed load.

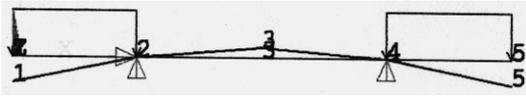


Fig. 10 Deflection of the beam.

their findings is by arbitrarily cutting a section through the truss and applying the static equilibrium conditions. As an example, consider cutting a section through elements (1), (2), and (3) as shown in Fig. 8.

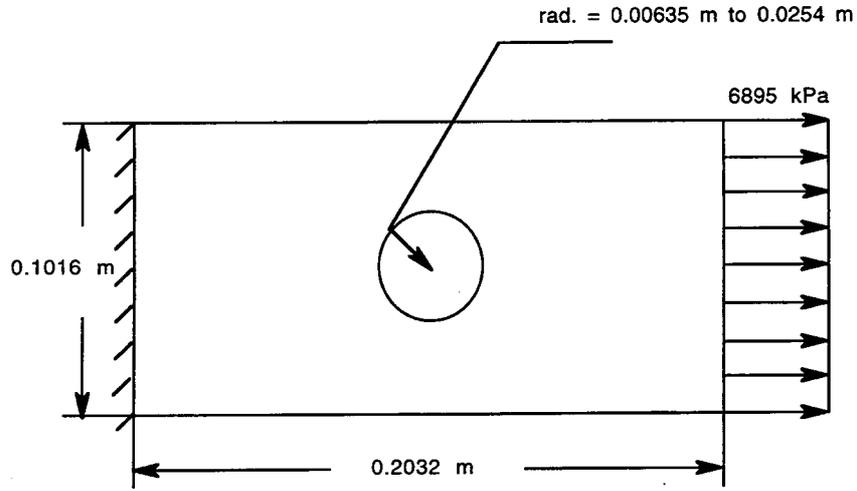


Fig. 11. A flat bar with a circular hole under axial loading.

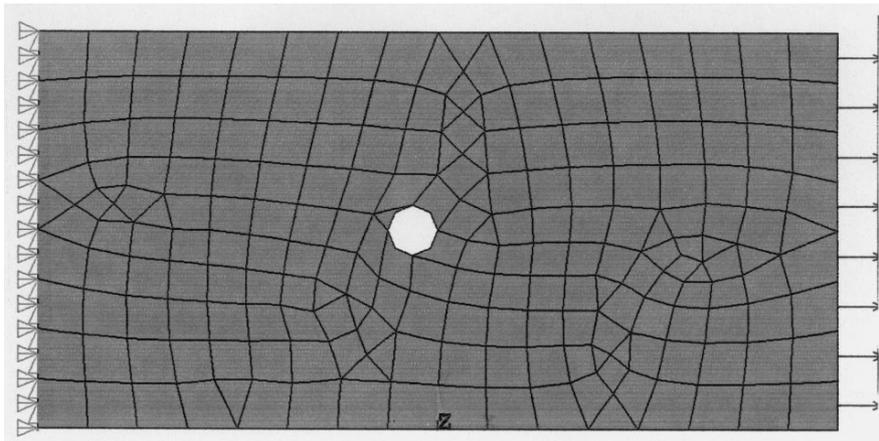


Fig. 12. A finite element model of the flat bar.

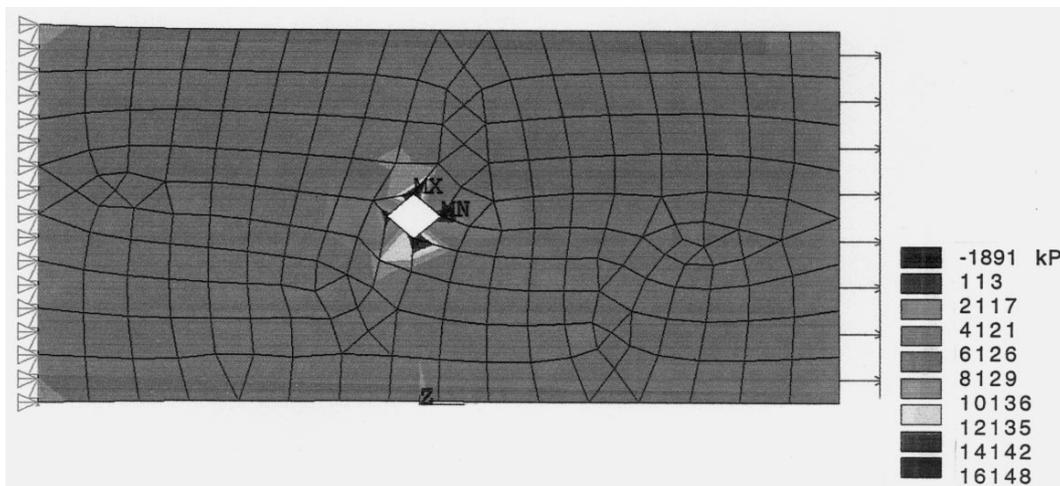


Fig. 13. Stresses in the x-direction.

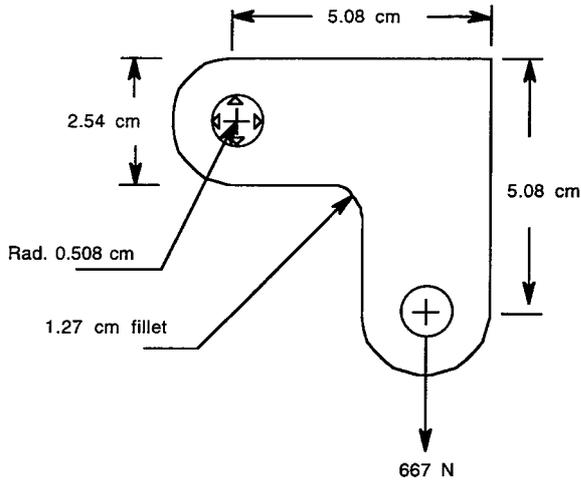


Fig. 14. A right angle bracket.

$$\begin{aligned} \sum F_X = 0 & \quad -2224 + 6672 - 6289 \cos 45 = 0 \\ \sum F_Y = 0 & \quad -2224 - 2224 + 6289 \sin 45 = 0 \\ \sum M_{node2} = 0 & \quad -(2224)(3) + (2224)(3) = 0 \end{aligned}$$

Again, the validity of the computed internal forces is verified. Moreover, it is important to emphasize to the students that static equilibrium conditions must always be satisfied. Students are also asked to apply equilibrium conditions for an arbitrary node as well. Once students become comfortable with this problem, they can begin experimenting by changing the magnitude and location of external forces and/or changing the boundary conditions.

PURE BENDING MODULE

In this module, a beam is defined as a structure member whose cross-sectional dimensions are much smaller than its length. It is emphasized that beams are generally subjected to transverse loading. Typical beam supports are discussed. The objectives associated with this module are:

- to identify beam problems and typical beam supports;
- to calculate and observe the longitudinal stresses and how they are related to bending moment;
- to calculate and observe deformation of a beam and how it is related to applied loads.

This module starts with a standard wide-flanged rolled steel beam (W30 × 173) with uniformly distributed load as shown in Fig. 9. Again, students must verify computed support reaction, bending stresses, deflections at specific points, and the slope along the beam. Figure 10 shows the deflection of the beam under the given load. Nodal deflections, slope along the beam and stresses can also be obtained in a tabular form. Students can then experiment with changing the geometric properties of the beam, for example, using a bigger beam, such as W33 × 201 or change the loads or boundary conditions.

STRESS CONCENTRATION MODULE

In this module the concept of stress concentration is introduced. It is noted that, in many

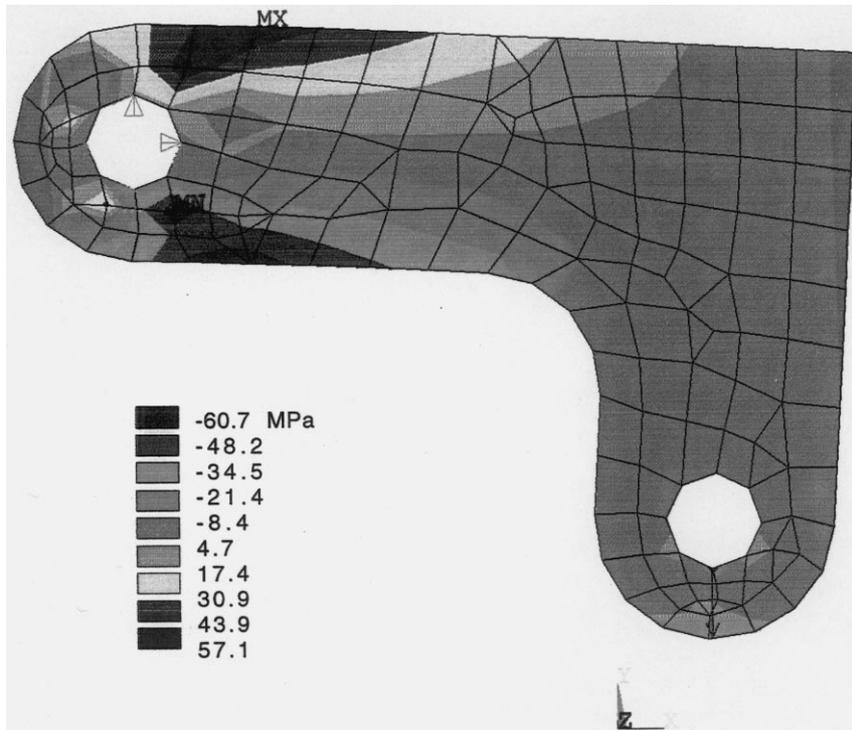


Fig. 15. Stresses in the x-direction.

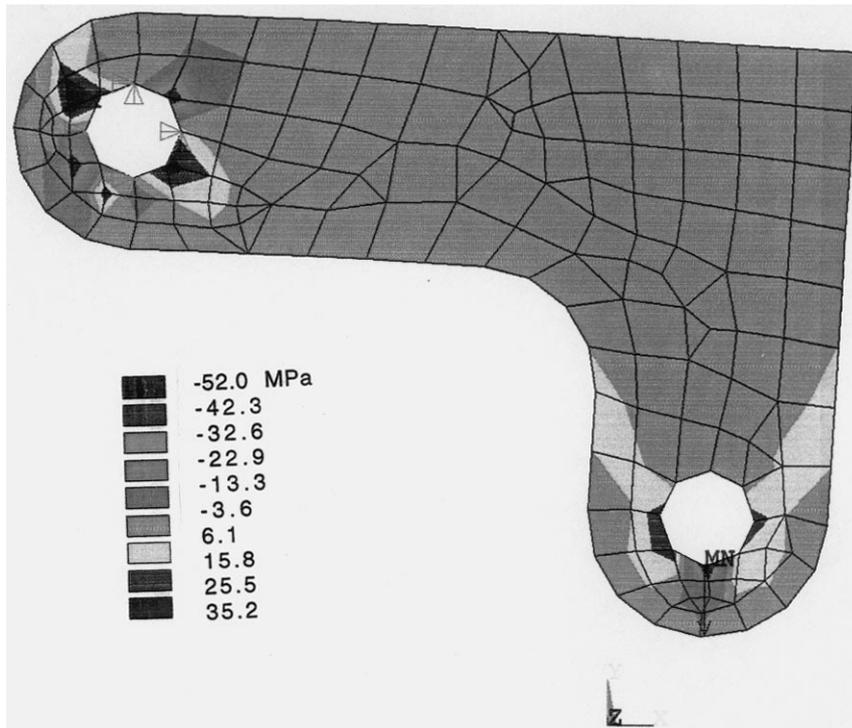


Fig. 16. Stresses in the y-direction.

engineering applications we are concerned about the magnitude of maximum stresses and not the detailed stress distributions. Wherever there are holes, notches, grooves, or sharp corners in a structure then one needs to be concerned about failure, because stresses increase near the discontinuity in the material at points of these stress

concentrations. A flat bar with a circular hole under axial loading, as shown in Fig. 11, is used as an example in this module. The objectives associated with this experiment are to:

- identify solid shapes that lead to stress concentration;

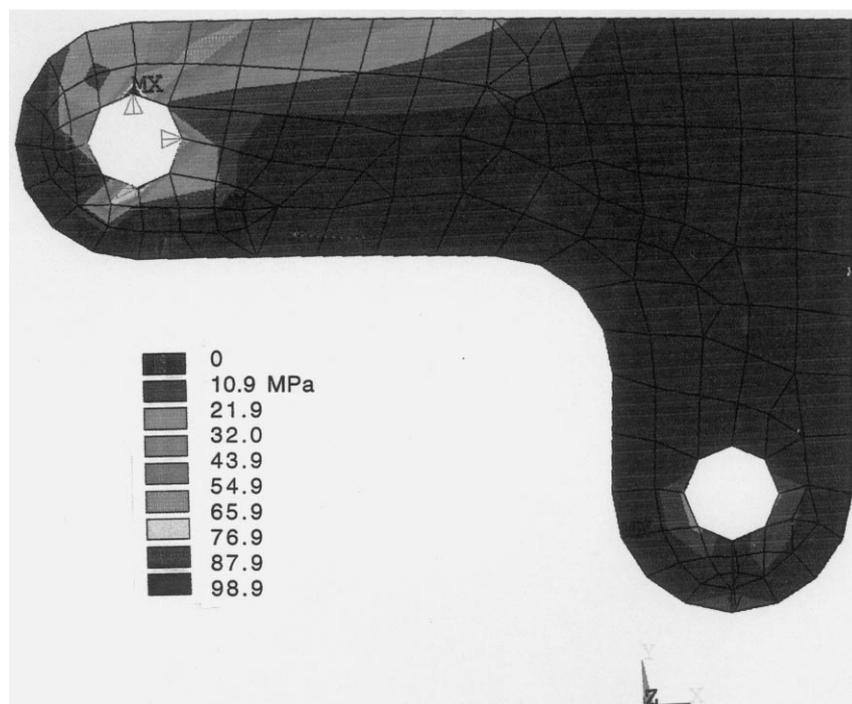


Fig. 17. A principal stress distribution.

- to introduce changes necessary to reduce stress concentration;
- observe the changes in the magnitude of the maximum stresses as you change the hole size, in the flat bar, from 0.635 cm to 2.54 cm;
- calculate stress concentration factors and compare them to the established results.

A finite element model of the flat bar with 0.635 cm hole is shown in Fig. 12. The corresponding stress distribution in the x-direction for this flat bar under axial loading is shown in Fig. 13.

COMBINED LOADING MODULE

In the previous teaching modules, students were introduced to normal stresses, bending stresses in beams, stress concentrations, statically indeterminate problems. Here, in this module, it is noted that generally members are subjected to loads that can create both normal and shear stresses on an element of a member. The objectives associated with this module are:

- to recognize what type of loading will create both normal and shear stresses;
- to determine stresses in different directions;
- to determine the principal stresses.

The right-angle bracket shown in Fig. 14 is used in this module. The bracket has a thickness

of 6.35 mm and has a modulus of elasticity, $E = 200 \text{ GPa}$. The upper left hole is fixed. The students will experiment with increasing the length from 5.08 cm to 10.16 cm and observing the changes in the stresses in different direction as well as the principal stresses. Examples of the results are shown in Figs 15 to 17.

CONCLUSIONS

The numerical experiments described in this paper provide students with a better understanding of a few basic concepts in mechanics of materials. Important topics including axial loading, stress concentration, transverse loading and combined loading are covered in these modules. The numerical experiments allow students to visualize stresses, and calculate deformations and reaction forces. They can change boundary conditions, vary certain dimensions, and observe the results. The experiments may be integrated into an introductory mechanics of materials class as two-hour lab sessions. The first two labs should be devoted to provide a brief overview of ANSYS and finite element analysis. Other labs are introduced at the appropriate times after relevant theories are discussed in class first. The ideal group size, to conduct these experiments, should include two students to allow for discussion between students.

REFERENCES

1. R. C. Hibbeler, *Mechanics of Materials*, Macmillan (1991).
2. F. P. Beer and E. R. Johnston, *Mechanics of Materials*, 2nd Edition, McGraw-Hill (1992).
3. ANSYS Documents, Swanson Analysis Systems, Inc., Houston PA, USA.

Saeed Moaveni is an Associate Professor in the Mechanical Engineering Department at the Mankato State University (Minnesota State University—Mankato). His current interests include innovative teaching, geophysical problems, fluid flow through porous media and inverse problems in engineering. Dr. Moaveni is a member of ASEE, ASME, ASHRAE, AGU.