Integrating Fatigue Analysis into a Machine Design Course or Finite Element Course

Josh Coffman, Sachin S. Terdalkar, Joseph J. Rencis/Ashland O. Brown

University of Arkansas, Fayetteville/University of the Pacific

Abstract

Fatigue is a major topic addressed in undergraduate and graduate machine design courses. Practicing engineers today commonly solve fatigue problems by hand coupled with static finite element analysis. More recently fatigue modules have been incorporated into a few commercial finite element codes which are emerging as a powerful numerical tool. A literature review of machine design textbooks, finite element textbooks, engineering educational journals, and engineering educational conference papers reveals that the topics of fatigue and finite elements addressed together are almost non-existent. In this work a simple cantilever beam fatigue example is considered and is solved by hand and the commercial finite element code ANSYS® Academic Teaching Introductory Release 11.0. The hand solution is included to emphasize the importance of verification when solving a problem using the finite element method. The target audience of this paper is an instructor who would like to integrate fatigue into a finite element course or fatigue finite element (FE) analysis into a machine design course.

Introduction

Fatigue is a material based phenomenon that causes failure in machine parts at stress values much lowers than static yield strength of the material. Fatigue failure is due to repeated or cyclic loading and unloading or fluctuating reversal in loading after a large number of cycles. Fatigue failures are estimated to occur in 80-90% of all machine component failures and account for a 4% loss in the gross domestic product of the United States and Europe.[1]

Fatigue failures are commonly found in components used for the automotive and aerospace industries. High cycle fatigue in the automotive industry is common in suspension systems, engine components, and components in the powertrain that include the transmission, drive shafts, and wheel assemblies. A connecting rod is an example of an engine component that experiences large stresses and a high number loading cycles. The connecting rod provides a linkage from the piston head to the crankshaft. The fatigue analysis of a connecting rod can be found in the ANSYS® on-line white paper.[2] Some fatigue failures in automobiles can be life critical, but in aerospace applications any fatigue failure may result in tragic losses of life. Sources of high cycle fatigue in large aircraft include turbo-jet engines, landing gear assemblies, fuselage coverings, and the connection points of wings. In aerospace applications materials may be used that do not have endurance limits due to weight concerns. An example of fatigue failure in the fuse pin connections of the jet engines to the wing of a commercial airliner is studied in Zahavi.[3] Both industries sometime require a full-scale model to verify the fatigue life.

Fatigue is a major topic that is addressed in undergraduate and graduate machine design courses and textbooks by Shigley[4,5] and Norton.[6,7] A machine design course is required most of the time in undergraduate mechanical engineering programs. In academia or industry fatigue problems have traditionally been solved by hand or an in-house computer program specialized for a particular of fatigue application.

The finite element method (FEM) is a computational tool that has been used extensively the past thirty years in industry and is now a standard engineering tool for both analysis and design. When FEM first appeared in the 1960's it was introduced into the engineering curriculum at the graduate level. As the method and computer technology matured, FEM was introduced at the undergraduate level in engineering and engineering technology programs, even in some two-year engineering technology programs. FEM is today primarily offered as an elective undergraduate course in mechanical, civil, and aeronautical engineering programs.[8]

Fatigue analysis that once was carried out by hand and/or in-house computer programs is now done using commercial FEM software. Fatigue modules have recently been integrated into commercial FEM codes that include ABAQUS®[9], ALGOR®[10], ANSYS®[11], COMSOL®[12], COSMOSWorks®[13], and Pro/ENGINEER®.[14] The usage of FEM in fatigue analysis does not go without limitations. An absence of actual loading data throughout the life of the components will not allow for the accurate results for life prediction. A second limitation of FE fatigue analysis is the random variance in material performance even in materials of the same type.

This paper will first review educational literature that considers both fatigue and FEM. A simple cantilever beam example is then solved by hand and the FEM commercial code ANSYS®. The target audience of this paper is an instructor who wants to integrate fatigue into a finite element course or fatigue finite element analysis into a machine design course.

Literature Review

A literature review of machine design textbooks, FEM textbooks, engineering educational journals, and engineering education conference papers revealed that fatigue and FEM addressed together are almost non-existent and have only appeared recently. This causes a knowledge gap between fatigue analysis and FE analysis.

A machine design course typically relies on a textbook that contain one or more chapters on fatigue theory and design. Early machine design textbooks did not provide any background in FEM and commonly just mention FEM. For example, the popular machine design textbook by Shigley[4,5] (1977-2006), did not mention FEM until the eighth edition in 2008.[15] Other textbooks briefly mention how FE analysis is a powerful engineering tool.[16,17,18] Newer and applied approaches in textbooks, such as Juvinall[19] (2000), Norton[7,8] (2000 and 2006), Shigley[15] (2008), and Ugural[20] (2004) provide an introduction to FEM in sections or entire chapters. The textbook by Edwards and McKee[21] (1991) discusses fatigue and FEM together. At the end of chapter nine the need for computer-aided fatigue design is described; however, no examples are considered. The authors' discussion also includes analysis types available in software and commercial FE codes.

Two FEM textbooks mention fatigue and discuss its importance for designing machine components. The textbook by Adams and Askenazi[22] (1999) provides a review of fundamental fatigue analysis principles. In the chapter on nonlinear analysis both authors state that accurate stresses are required to estimate fatigue life or damage. Also stated is that the stresses are highly dependent on how accurately the material properties are defined. They also state that future FEM codes will employ stochastic methods to allow "automated" fatigue life analysis.[22] The second FEM textbook by Zahavi[23] (1992) discusses that reducing the geometric stress concentration factor will increase fatigue life. Zahavi mentions fatigue a few other times, but only to state the importance of fatigue design, never actually using FE to predict fatigue life.[23] These two textbooks[22,23] never apply FEM to a fatigue example.

A literature review of fatigue textbooks reveals FEM as an analysis tool is addressed on a very limited basis. Fatigue textbooks that mentioned FEM usually discuss how it is used to determine stresses and some other discussions include the use of FEM to study fracture mechanics and the analysis of plasticity in crack propagation. Zahavi[3] has a fatigue design textbook that clearly ties fatigue with FEM as a tool for determining static stresses in three-dimensional machine components. Several examples are considered using static stresses to determine the fatigue life of machine components.[3]

The consideration of fatigue and FEM together in educational journals and conference papers is very limited and has only appeared recently. A review of educational journals yielded no papers that consider both fatigue and FEM. A conference paper by Hagigat[24] (2005) explains the general concept of fatigue and also emphasizes that a major contributor to high cycle fatigue failures is vibration. Hagigat[24] states that using mode shapes and S-N curves will yield an accurate fatigue analysis. However, no fatigue analysis is presented, nor is any actual FE analysis used for determining fatigue life. In regard to the use of commercial FE software with fatigue capabilities, Hagigat[24] states, "…from an educational point of view, it is recommended that these capabilities not be used initially. After a student understands the concepts by going through the steps in this article, he/she can then use the additional capabilities of the software correctly. A lack of knowledge of the theory behind the more advanced capabilities of the software can lead to the incorrect use of the software." Still no direct computation of fatigue life was carried out using FE software.

Educational Goals and Objectives

This work is part of a larger scale project to develop FE learning modules for undergraduate engineering courses[8] that will be available 24/7 to the world-wide community on the internet. The project goals and project objectives have been divided into developmental, educational, and assessment.

The *project developmental goal* is to develop FE learning modules in different engineering areas that are easily accessible and require minimal instructor effort. The *project developmental objectives* to accomplish this goal are as follows:

1. *Integrate into Different Courses.* Develop FE learning modules that can be integrated into different types of undergraduate engineering and introductory finite element courses.

2. *Time and Accessibility.* Develop FE learning modules that require minimal classroom time to be integrated into a course with minimal instructor preparation, and are easily accessible.

The *project educational goal* is to provide undergraduate engineering students with understanding of a specific engineering topic and FE theory, along with an ability to apply commercial FE software to typical engineering problems. The educational goal will be accomplished through four *project educational objectives* based on Bloom's Taxonomy[25] and ABET Criterion 3 for Engineering Programs[26] as follows:

1. *Engineering Topics (Comprehension; 3a, 3k).* Understand the fundamental basis of engineering topics through the use of finite element computer models.
2. *FE Theory (Comprehension; 3a).* Understand the fundamental basis of FE theory.
3. *FE Modeling Practice (Application; 3a, 3e, 3k).* Be able to implement a suitable finite element model and construct a correct computer model using commercial FE software – integrates objectives #1 and #2 above.
4. *FE Solution Interpretation and Verification (Comprehension and Evaluation; 3a, 3e).* Be able to interpret and evaluate finite element solution quality, including the importance of verification – integrates objectives #2 and #3 above.

The *project educational objectives* address three of six Bloom's Taxonomy levels, i.e., *comprehension, applications, and evaluation,* but a future follow up project will address all six. The educational outcomes above were mapped to ABET Criterion 3 Program Outcomes for Engineering Programs so that instructors can integrate an exercise into their in-house ABET assessment process.

The *project assessment goal* is to accurately and comprehensively assess each educational objective. The assessment goal will be accomplished through two *project assessment objectives* as follows:

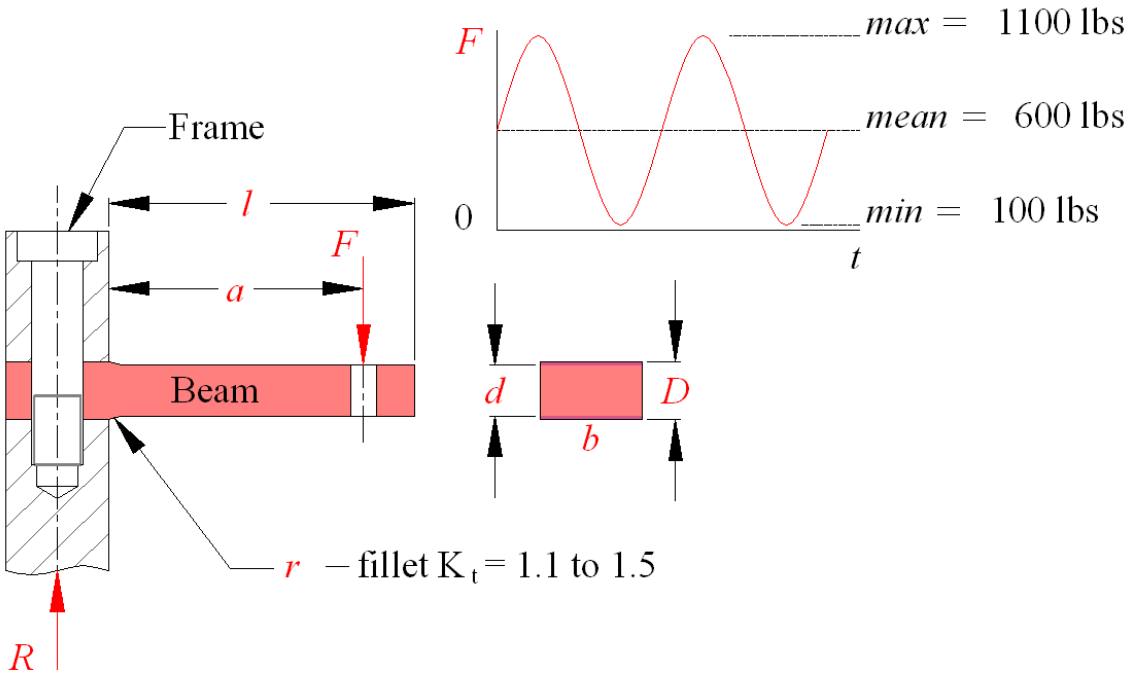
1. *Assessment System.* Develop and implement a closed loop (iterative) assessment system.
2. *Learning Styles.* Gain insight into the effectiveness of the FE learning modules across various personality types and Learning Styles.

The assessment program for the fatigue FE learning module will be carried out in the future and is discussed in the Future Work section at the end of this paper.

Example Problem Overview

The fatigue example is shown in Figure 1 and can be found in the machine design textbook by Norton.[6,7] Both the second[6] and third editions[7] contain this example problem. This example problem was selected since it is in a commonly used machine design textbook and has a hand solution. This example will be analyzed using the version of ANSYS® Academic Teaching Introductory Release 11.0. The authors have also developed a FE fatigue module based on a simply supported beam in the machine design textbook by Nisbett and Budynas.[15]

The problem states that a feed roll assembly is supported on both ends by cantilever brackets. This assembly is subjected to an applied fluctuating load of 200 lbs at a minimum and 2200 lbs at a maximum. For analysis purposes, this means that a single bracket is modeled using half of the applied fluctuating load. The schematic of the bracket, geometric properties, applied fluctuating load, and material properties are shown in Figure 1. An additional design requirement is that the maximum vertical deflection does not exceed 0.02 in. Other design criteria include an operating environment of 120°F, maximum cantilever length of 6 in, and only ten brackets will be manufactured. Norton[6,7] assumes that the parts are machined due to the low volume that will be manufactured.



Geometric Properties	Material Properties
l = 6.0 in	SAE 1040 Normalized Carbon Steel
a = 5.0 in	E = 30 x 10^6 psi
r = 0.5 in	ρ = 0.2834 lb/in^3
d = 1.0 in	υ = 0.28 (Poisson's Ratio)
b = 2.0 in	S_{ut} = 80 kpsi
D = 1.125 in	S_y = 60 kpsi

Figure 1. Cantilever beam subjected to a fluctuating load.[6,7]

Norton[6,7] applies some assumptions in this example. First, the bracket will be clamped between what is assumed to be rigid plates. The load is applied in a small hole near the tip of the beam. Following the example explicitly, the hole's stress concentration effects will be neglected for the hand and FEM analyses because the bending stresses near the free end of the beam are very low. The bracket will have a selected material that will allow for 10^9 loading cycles or an infinite fatigue life.

The analyses will include the following: frequency/modal analysis, static displacement analysis, static stress analysis, and fatigue life analysis. Each analysis will be carried out first by

hand based on Norton[6,7] and then by the commerical FEM code ANSYS®. The hand solution is included to emphasize the importance of verification when solving a problem using FEM.

Finite Element Model

The cantilever beam was modeled with the commercial FE code ANSYS® and used the plane stress, PLANE42, a four node quadrilateral element. The geometry, material properties and loading are shown in Figure 1. The same FE mesh was used for the modal/frequency, static displacement, static stress, and fatigue analyses. The mesh size was determined based on a convergence study of stresses since a finer mesh is required to obtain accurate stresses compared to deflections and frequencies. The FE mesh consists of 1,329 nodes and 1,224 elements as shown in Figure 2. Each node has two degrees of freedom (DOF) and the mesh has 2,658 DOFs. The bracket mounts are located at the vertical left-hand side of the beam in Figure 2 and these DOF were fixed in the horizontal and vertical directions.

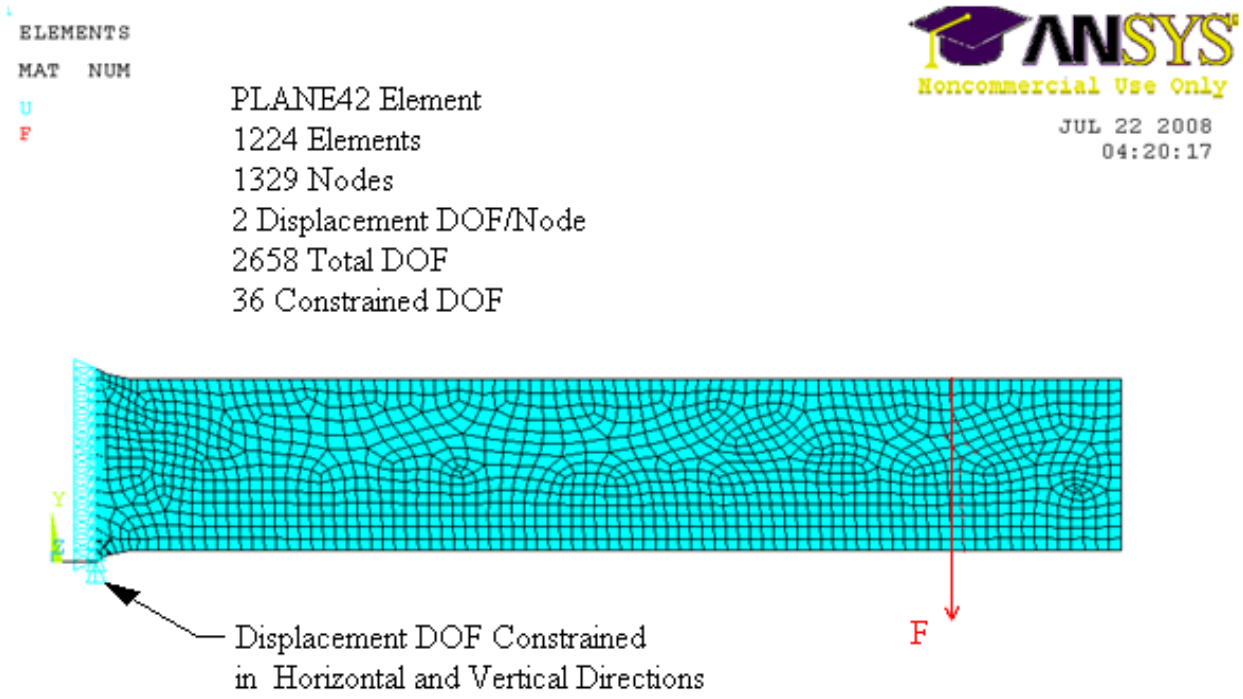


Figure 2. Plane stress FE mesh of cantilever beam.

Frequency/Modal Analysis

A modal analysis was carried out since a major contributor of high cycle fatigue loading is due to vibration. If the frequency of the loading reaches a resonance condition, large amplitudes of vibration will occur in a machine component. If the component is subjected to large vibrational amplitudes, the applied cyclic stresses may cause fatigue failure depending on geometry, material, loading type, and number of cycles.[24] The modal analysis can provide insight on where to locate a larger mass and/or where to increase component stiffness. The modal analysis was not carried out in the machine design textbook by Norton.[6,7]

The cantilever beam has a fixed boundary on the left-hand side and all other DOFs in the FE mesh are free throughout the beam in Figure 2. A hand solution to determine the frequencies (eigenvalues) and mode shapes (eigenvectors) are well documented in vibrations and structural dynamics textbooks for the long cantilever beams.[27] However, the geometry of the cantilever beam in Figure 1 classifies the beam as short due to the length to depth ratio (ten to one or less). The frequency of a short beam is obtained by multiplying the long beam frequency by a correction factor found in the handbook by Harris.[27] When a beam is short then the effects of rotary motion and shearing forces must be taken into account in the long beam hand frequency analysis.[27] These effects are based on Timoshenko beam theory and are not commonly found in undergraduate machine design textbooks or most vibrations and structural dynamics textbooks. The frequencies for the first five modes based on the hand analysis are shown in the third column of Table 1.

The commercial FEM code ANSYS® was used to calculate the natural frequencies and mode shapes of the beam. The FE model is shown in Figure 2. The FE results for the first five frequencies are shown in fourth column of Table 1. There is very good agreement between the hand and FE analyses. One should note that since the ANSYS® model was formulated based on theory of elasticity, therefore, the effects of rotary motion and shearing forces are included.

Table 1. Natural frequencies of the cantilever beam for hand and ANSYS® analyses.

Mode	**Mode Type**	**Frequency (Hz)**		**% Difference of Solutions**
		Short Beam Hand Analysis*	**ANSYS® Analysis (PLANE42 Elements)**	
1	Bending	898.92	898	0.10%
2	Bending	5008	5051	0.86 %
3	Axial	8426	8457	0.36 %
4	Bending	12270	12442	1.40%
5	Bending	20923	21234	1.49%

*Hand analysis frequencies are shown as corrected using short beam correction factors for modes 1 through 5, 0.99, 0.88, 1.0, 0.77, and 0.67, respectively.[27]

The FE model was verified with a hand analysis to ensure that the total mass and mass center is correct. If the total mass and mass center of the FE mesh is incorrect, then the frequencies and mode shapes will be incorrect. Based on past experience the authors have found that students, and even practitioners, do not carry out these two simple checks. The mass and the mass center for the cantilever beam are shown for the hand and ANSYS® analyses in Table 2. The hand analysis was based on the theory in statics textbooks.[28-30]

Table 2. Total mass and mass center locations for hand and ANSYS® analyses.

Analysis Method	Total Mass lbm.	% Difference in Total Mass	Center of Mass Location (X, Y) in.	% Difference in Center of Mass Locations	
				X	Y
Hand	3.4094	0.08%	(2.9931, 0.5)	0.07%	0.0%
ANSYS®	3.4065		(2.9952, 0.5)		

Deflection Analysis

The design requirement is that the vertical deflection of the beam is less than 0.02 in. A maximum load of 1,100 lbs ($F = F_{max} = 1{,}100$ *lbs*) was applied at the right end of the cantilever beam as shown in Figure 1. A hand analysis using mechanics of materials principles in Norton[6,7] yielded a vertical deflection at the end of the cantilever beam of 0.0119 in. ≈ 0.012 in. as displayed in the textbook. The actual magnitude of this value is important when considering the accuracy of the solution. This calculation ignores the effects of transverse shear deflection since it assumed a long uniform beam. If the transverse shear deflection is considered using Castigliano's energy method for a short beam (not considered in Norton), the maximum vertical deflection increases to 0.01226 in., a 3.03% increase.

The maximum vertical deflection, shown in Figure 3, was determined by ANSYS® to be 0.011975 in., a 0.63% difference in the hand (long uniform beam) and FEM solutions. When compared to Castigliano's method for short beams, the ANSYS® solution is 2.32% different. The hand and ANSYS® analyses show that the design requirement for the vertical deflection is satisfied since it is less than 0.02 in at the free end.

You might be asking why is there a difference between the long beam hand solution, short beam hand solution, and ANSYS® solution. First, both hand solutions are based on a uniform cross-section, i.e., no fillet radii. A long or short beam containing two fillet radii has a greater stiffness than a uniform beam and the result is a smaller vertical deflection. Carrying out an ANSYS® analysis using PLANE42 elements for a uniform beam (no fillet radii) yields a vertical deflection that corresponds to short beam theory, not long beam theory considered in Norton.[6,7] Since the PLANE42 ANSYS® element was formulated based on theory of elasticity, shear deformations are accounted, therefore, the vertical deflection corresponds to short beam theory. Second, another reason for a difference between the hand solutions and ANSYS® solution is due to how the force is applied. Applying the concentrated force in Figure 1 as a parabolic shear stress distribution throughout the beam depth will result in an ANSYS® deflection that corresponds to the short beam hand solution.

Static Stress Analysis

A hand stress analysis for the maximum loading case of 1,100 lbs ($F = F_{max} = 1{,}100$ *lbs*) ensures that the maximum bending stresses are far below the nominal value required for yielding on the first loading cycle. Two static stress analyses are required to carry out a fatigue analysis. The

first static analysis is where the mean load of $F = F_m = 600$ *lbs* is applied one in. from the right end as shown in Figure 1. The second static analysis is where the alternating load of $F = F_a = 500$ *lbs* is applied one in. from the right end as shown in Figure 1.

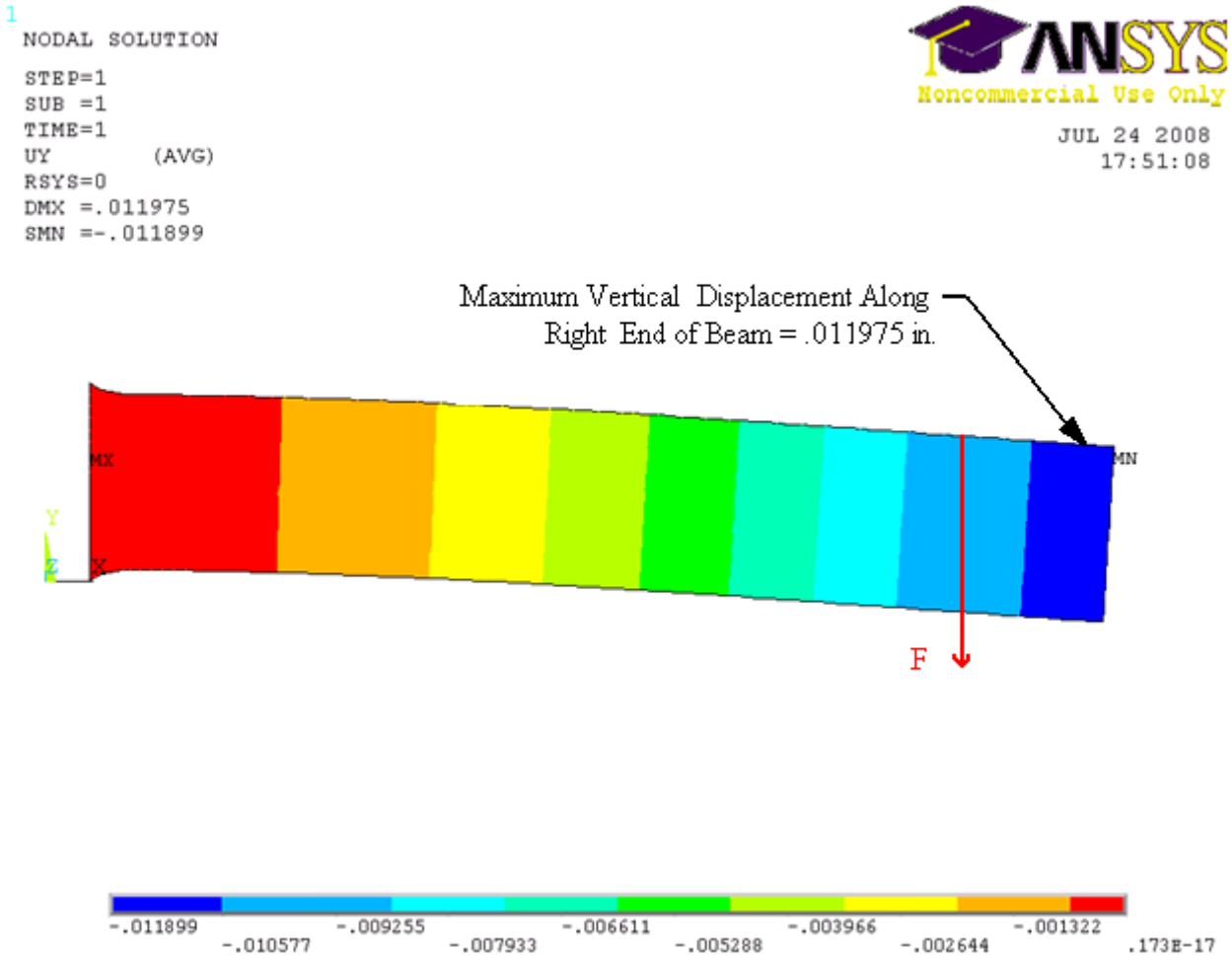


Figure 3. Maximum vertical deflection (in.) and deflected shape of the beam due to a maximum applied load of $F = F_{max} = 1{,}100$ *lbs.*

Mean Load Case

A mean load of $F = F_m = 600$ *lbs* is applied on the right side of the cantilever beam as shown in Figure 1. A hand static stress analysis determined that the maximum bending stresses at the top and bottom fibers of the cantilever beam wall[6,7] is 9,000 psi. By knowing that the fillet radii at the left end is the location of the highest localized bending stresses, the geometric stress concentration factor, K_t shown in Figure 1, is used to determine the maximum stress at the fillet. Using the figure for geometric stress concentration factors and functions for a stepped beam in pure bending and the modifications for the ultimate strength and notch sensitivity from Chapter 4 of Norton[6,7], the corrected geometric stress concentration factor is 1.16. The actual bending stress at the fillet radius is 10,454 psi. The shear stresses near the outer fibers of the cantilever

beam at the left end are very small in magnitude such that Norton[6,7] neglected their contribution when determining the von-Mises stress.

The ANSYS® using the PLANE42 four node quadrilateral element includes the stress concentration effect since the element was formulated based on theory of elasticity. The shear stresses are included in the von-Mises stress since the element was formulated based on the theory of elasticity. This is why the von-Mises stress is slightly lower for FEM compared to the hand analysis. The FEM approach calculates the von-Mises stress to be 9,865 psi as shown in Figure 4. This value is slightly lower and is why there is a 5.63% difference in hand and FEM solutions.

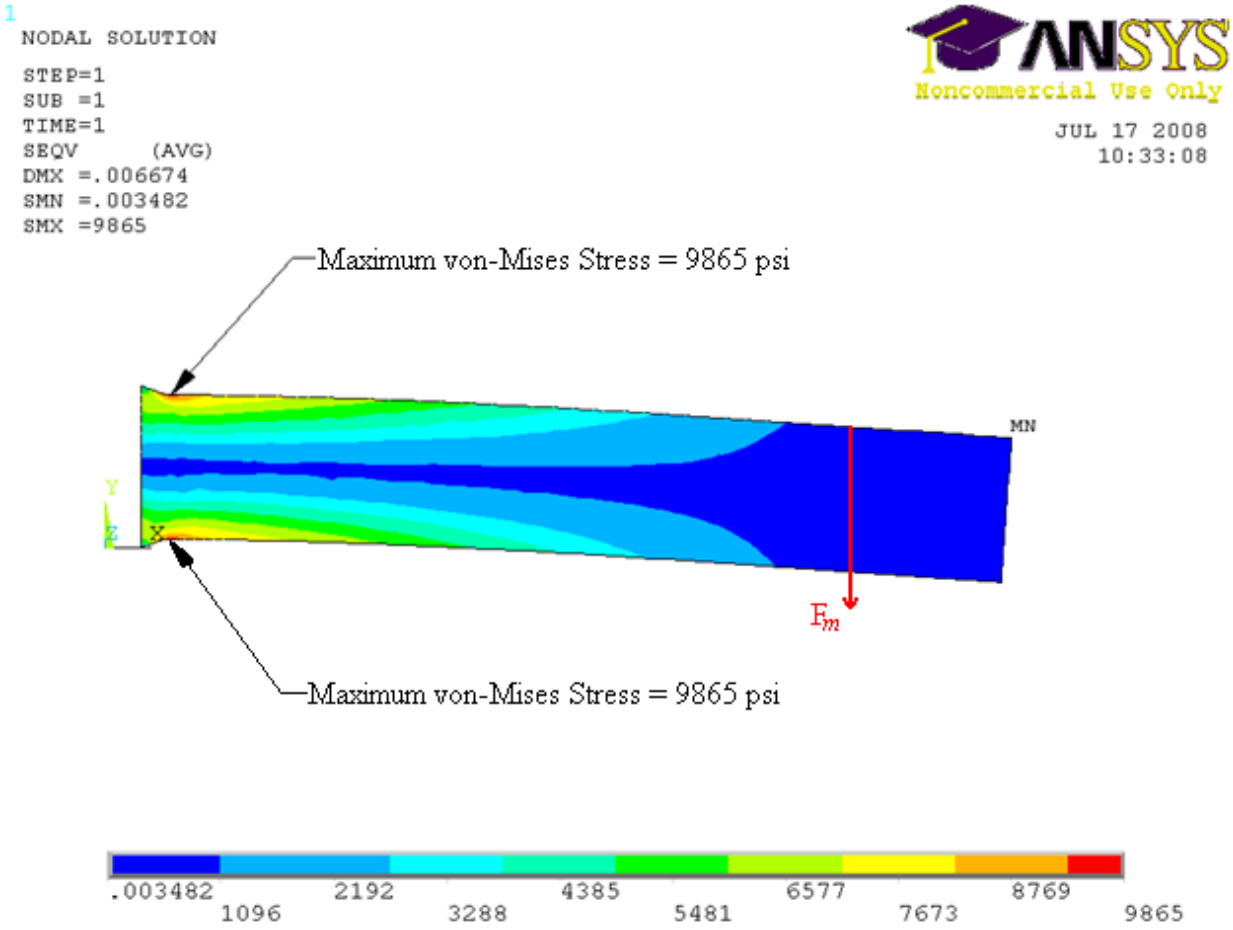
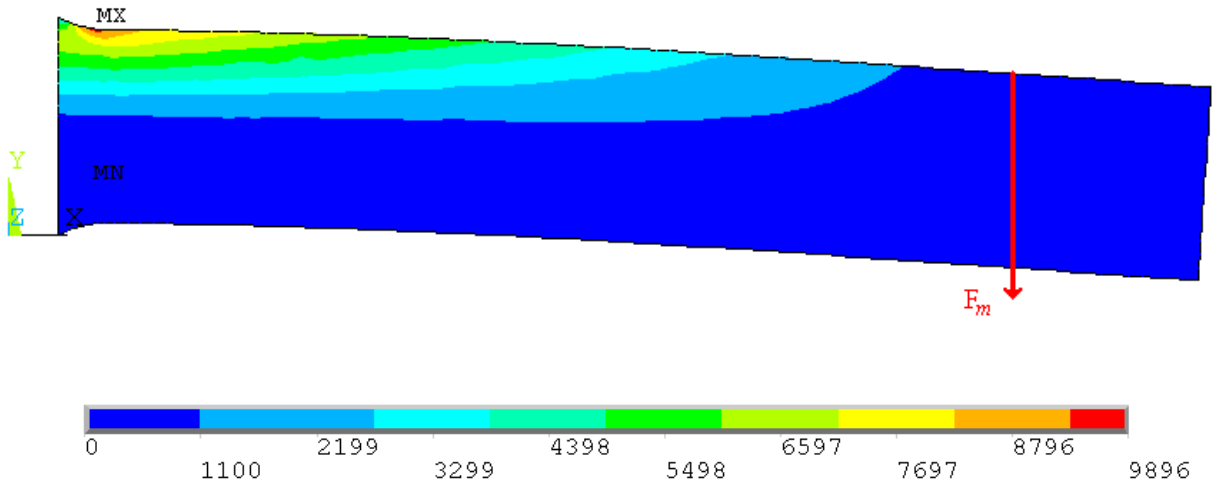


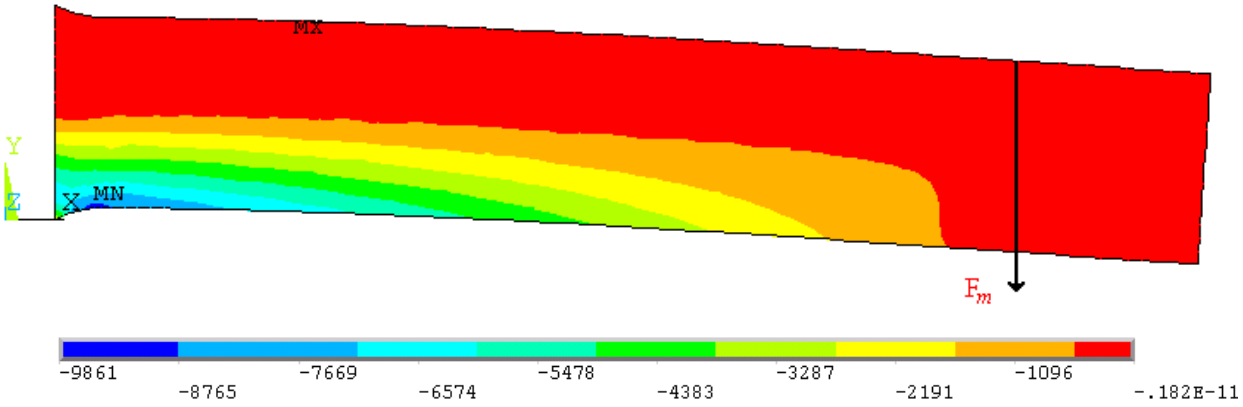
Figure 4. von-Mises stress (psi) distribution for a mean load of $F = F_m = 600$ *lbs.*

Apart from von-Mises stresses, a closer look at the maximum and minimum principal stresses is taken. An advantage of the principal stresses over von-Mises stresses is the ability to describe the nature of the load. The maximum and minimum principal stresses are shown in Figure 5a and 5b, respectively, for the mean loading case ($F = F_m = 600$ *lbs*). The maximum principal stresses shown in Figure 5a are all tensile. The maximum tensile stress of 9,896 psi is located at the fillet radius on the top left-hand side of the beam. The location of maximum tensile stress will be located at the fillet radius on the bottom as the applied direction of the cyclic load changes. Knowing the location of highest areas of tensile stresses will allow an

engineer to predict the possible location of crack intiation, the main cause of fatigue failure. Figure 5b displays the areas of compressive stresses located in the bottom half of the beam. The maximum compressive stress is -9,861 psi. The presence of compressive stresses is assumed to only increase the fatigue strength. As previously discussed, as the cyclic load changes direction the location of tensile and compressive stresses will switch. One should note that the magnitudes of the maximum principal stresses are slightly more conservative than the von-Mises stresses, while the minimum principal stresses are slightly reduced when compared to the von-Mises stresses. Norton[6,7] did not consider maximum and minimum principal stresses.



a.) Maximum principal stress (psi).



b.) Minimum principal stress (psi).

Figure 5. Principal stresses for the mean loading case $F_m = 600$ *lbs.*

Alternating Load Case

An alternating load of $F = F_a = 500$ *lbs* is applied at the right-hand side of the cantilever beam as shown in Figure 1. The bending stress at the top and bottom fibers at the left end of the beam was determined by hand as 7,500 psi. The geometric stress concentration must be

accounted for at the fillet locations as discussed for the mean load case. The corrected geometric stress concentration factor is the same one used for the mean load case, a value of 1.16. The value for the maximum bending stress using the stress concentration factor was 8,711 psi at the top and bottom fillets. The shear stresses were once again neglected due to their low magnitude and for hand calculation simplicity. The von-Mises stress is 8,711 psi, and is the same value as the bending stress. The FEM approach calculated the von-Mises stress to be 8,239 psi, as shown in Figure 6, a difference of 5.42% in solution types.

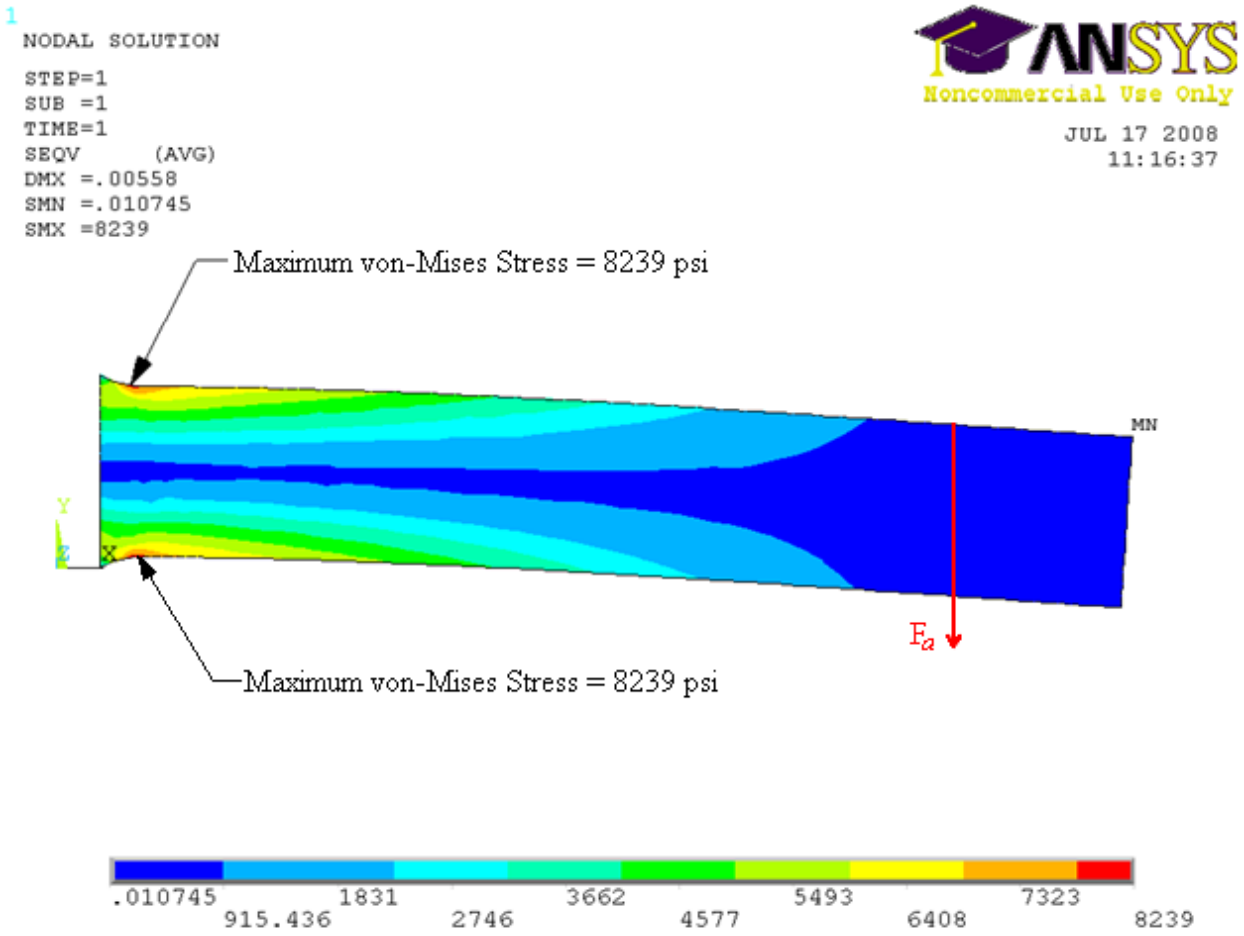
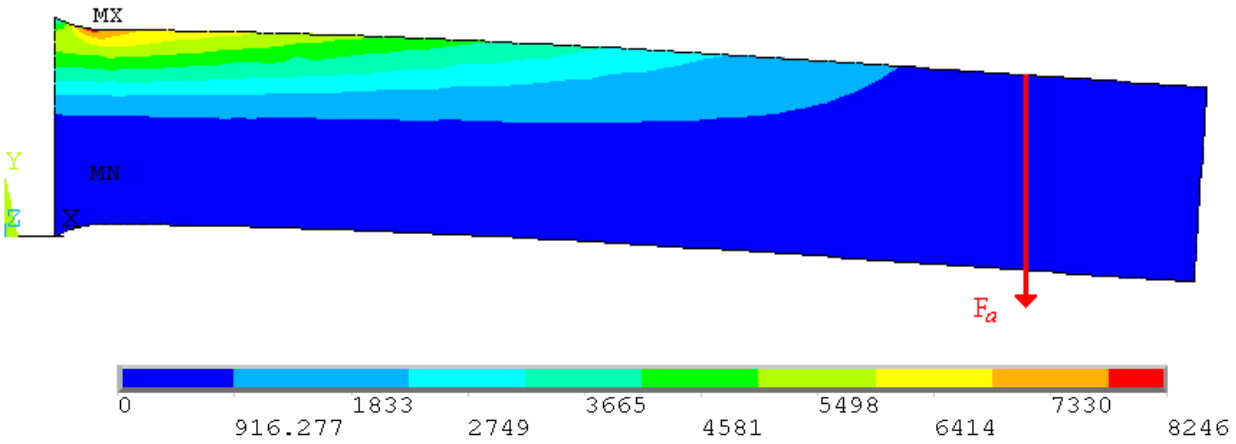
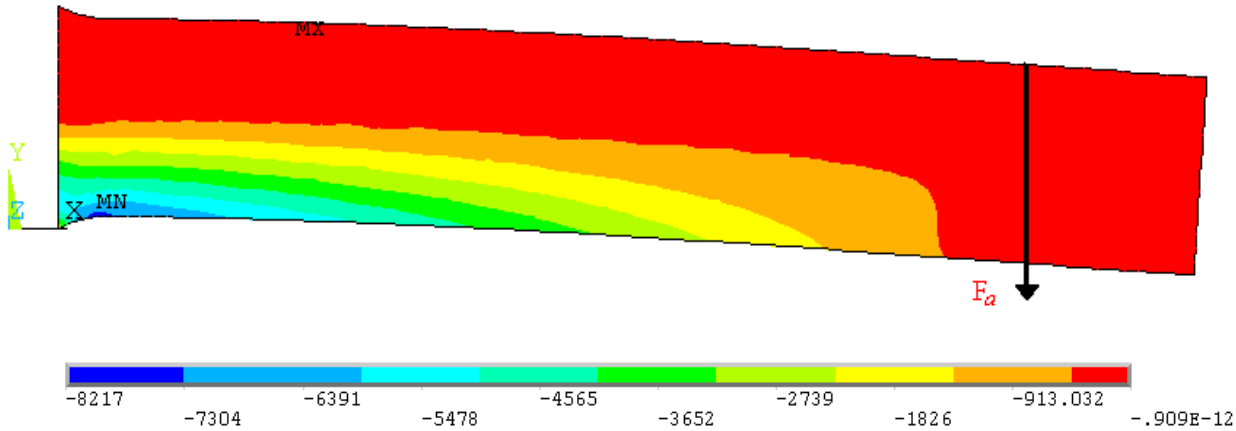


Figure 6. von-Mises stress (psi) distribution for a alternating load of $F = F_a = 500$ *lbs.*

The maximum and minimum principal stresses for the alternating loading case ($F = F_a = 500$ *lbs*) are shown in Figure 7a and 7b, respectively. The maximum principal stresses shown in Figure 7a are all tensile. The maximum tensile stress is 8,246 psi. Figure 7b shows the variation of compressive stress throughout the beam. The maximum compressive stress is -8,217 psi. As mentioned in the previous discussion, the maximum and minimum principal stresses can be used to predict areas of the highest tensile stresses. The tensile stresses are of importance because these areas tend to be locations of crack initiation and growth over cyclic stresses.



a.) Maximum principal stress (psi).



b.) Minimum principal stress (psi).

Figure 7. Principal stresses for the alternating loading case F_a = *500 lbs.*

Fatigue Analysis

The beam is designed to withstand 10^9 loading cycles, which is considered high cycle fatigue. A stress-life approach was used as for this example to carry out the fatigue analysis since it is valid for high cycle fatigue, and it is commonly found in undergraduate and graduate machine design courses.

Knowing the ultimate tensile strength of the SAE 1040 normalized carbon steel to be S_{ut} = 80 kpsi (Figure 1), the estimated endurance limit is 40 kpsi.[6,7] This estimated endurance limit must be corrected for the following factors: loading type, surface finish, temperature of operating environment, component size compared to test samples, and desired reliability. The corrected endurance limit is 21.833 kpsi. This means that for SAE 1040 normalized carbon steel the stress values are well below the limit that is required for an infinite fatigue life or 10^9 loading cycles. This corrected endurance limit is also required to find the safety factors.

Figure 8 shows the safety factors based from a hand analysis based on the Modified-Goodman diagram. There are four methods described in Norton[6,7] to determine the lowest safety factor. Each safety factor is calculated by varying the mean and alternating stresses. The first safety factor (N_{f1}) is based on assuming that the alternating stress value is held constant. For this loading configuration the value of the first safety factor is relatively large compared to the other three safety factors shown in Figure 8. The second safety factor (N_{f2}) assumes a constant mean stress value. The third safety factor (N_{f3}) is calculated using a proportional amount of both alternating and mean stress values. The fourth safety factor (N_{f4}) is a random set of values for the mean and alternating stresses. This is the most conservative safety factor. Depending on the state of loading, any of the four mentioned cases could become minimum calculated value for the safety factor. The fourth case provides the minimum safety factor for the fatigue design as shown in Figure 8, i.e., $N_{f14} = 1.7$.

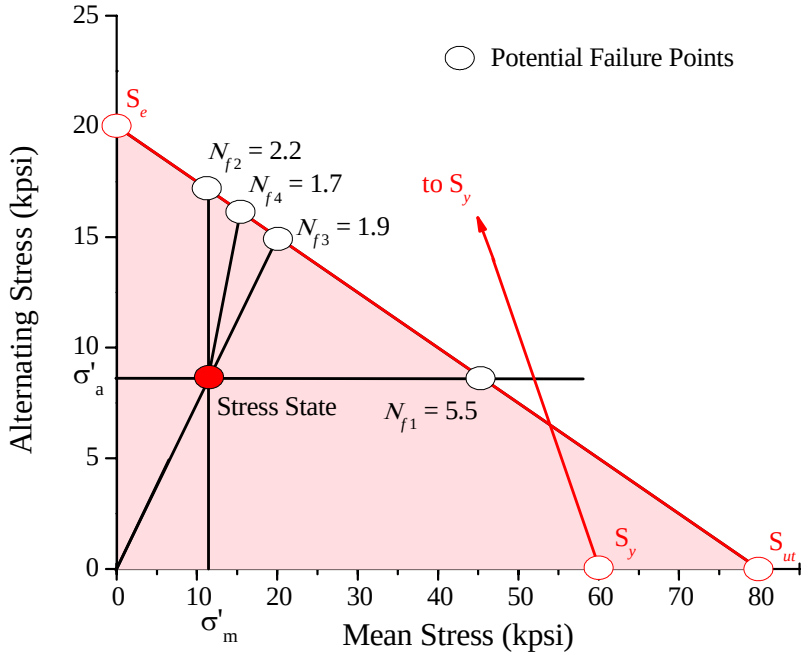


Figure 8. Modified-Goodman diagram displaying fatigue design safety factors for hand static stress analysis.

The safety factors are calculated by using the von-Mises stress values from ANSYS® as shown in Figure 9. The safety factors are calculated using the same four methods as previously described for the hand analysis. The values shown in Figure 9 indicate that the safety factors slightly increased. The inclusion of shear stresses in the FE analysis reduces the von-Mises stresses by approximately 5% in the beam. The minimum safety factor for the ANSYS® stress analysis is 1.8. The increased safety factor provides a difference in the two solution methods of only 5.88%. The hand analysis is found to be more conservative than the FE analysis.

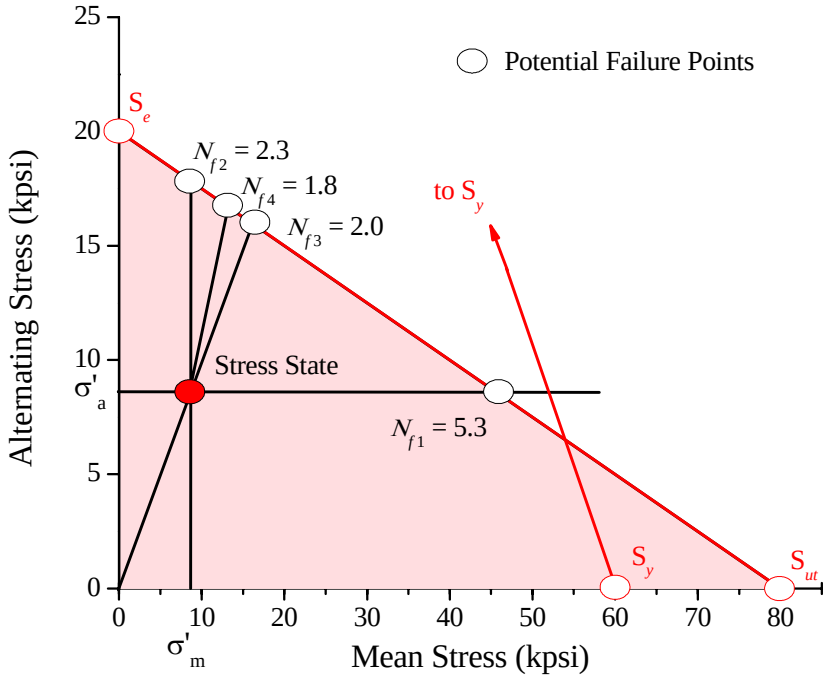


Figure 9. Modified-Goodman diagram displaying fatigue design safety factors for ANSYS® static stress analysis.

Conclusion

The use of commercial FE codes in the workplace is rapidly impacting the field of fatigue analysis and design. Engineering students and practitioners must have a basic understanding of the fatigue theory before being able to carry out a fatigue FE analysis. Based on a literature review by the authors, the integration of fatigue into a finite element course or finite elements into a machine design course has not been done in the past. This paper considered a simple example of a cantilever beam that is analyzed by hand and using the commercial FE code ANSYS®. This paper is a resource for both instructors and practitioners who want to consider both fatigue and FEM.

Future Work

This work is part of a larger scale project to develop FE learning modules for undergraduate engineering courses[8] that will be available 24/7 to the world-wide community on the internet. The project goals are as follows:

1. *Developmental.* Develop FE Learning Modules in different engineering areas that are easily accessible and require minimal instructor effort.
2. *Educational.* Provide undergraduate engineering students with an understanding of a specific engineering topic and FE theory, along with an ability to apply commercial FE software to typical engineering problems.

3. *Assessment.* Accurately and comprehensively assess each educational objective and the effectiveness of the FE Learning Modules.

This module will be integrated into an undergraduate machine design course or undergraduate finite element course at one of the six participating universities associated with this project. An assessment program will be carried out for the fatigue FE learning module that will include the following four assessment tools: post student survey, pre-course and post-course quizzes, learning styles (Felder-Soloman), and personality types (Myer-Briggs). The student survey and quizzes will indicate what the student liked and disliked about the FE fatigue learning module and if the student has improved learning using the module when compared to a traditional classroom approach. The learning styles and personality types of each student are identified through a survey and are used to determine whether the fatigue FE learning module is biased towards a particular learning style or personality type. The goal is to have a FE learning module that does not have a bias towards particular learning styles and personality types. The assessment results will be used for continuous improvement of the fatigue FE learning module over the next year. An in-depth discussion of the assessment program that will be carried out for this module can be found in Brown.[8]

Bibliography

1. Draper, J., "New Ideas in Fatigue Analysis," *Machine Design*, December 12, 2002, http://www.machinedesign.com/.
2. Browell, R., and Hancq, A., "Calculating and Displaying Fatigue Results," ANSYS® on-line white papers, March 29, 2006, http://www.ansys.com/assets/white-papers/wp-fatigue.pdf/.
3. Zahavi, E. and Torbillo, V., *Fatigue Design, Life Expectancy of Machine Parts*, CRC Press Boca Raton, FL, 1996, pp. 29-38, 65-140, 260-267, 272-296.
4. Shigley, J., *Mechanical Engineering Design*, Third Edition, McGraw-Hill, New York, 1977.
5. Shigley, J., *Mechanical Engineering Design*, Sixth Edition, McGraw-Hill, New York, 2001.
6. Norton, R.L., *Machine Design: An Integrated Approach*, Second Edition, Pearson Prentice Hall, Upper Saddle River, NJ, 2000, pp. 371-397.
7. Norton, R.L., *Machine Design: An Integrated Approach*, Third Edition, Pearson Prentice Hall, Upper Saddle River, NJ, 2006, pp. 300-372.
8. Brown, A., Rencis, J.J., Jensen, D., Chen, C-C., Ibrahim, E., Labay, V. and Schimpf, P., "Finite Element Learning Modules for Undergraduate Engineering Topics using Commercial Software," Mechanical Engineering Division, *CD-ROM Proceedings of the 2008 American Society of Engineering Education (ASEE) Annual Conference and Exposition*, Pittsburg, PA, June 22-25, 2008.
9. *ABAQUS*, Dassault Systèmes of America Corp., 6320 Canoga Avenue-Trillium East Tower, Woodlands Hills, CA, http://www.simulia.com/.
10. *ALGOR,Inc.*, Corporate Headquarters, 150 Beta Drive, Pittsburgh, PA, http://www.algor.com/.
11. *ANSYS, Inc.*, Southpointe, 275 Technology Drive, Canonsburg, PA, http://www.ansys.com/.
12. *COMSOL, Inc.*, 1 New England Executive Park, Suite 350, Burlington, MA, http://www.comsol.com/.
13. COSMOSWorks, SolidWorks Corporation, 300 Baker Avenue, Concord, MA, http://www.solidworks.com/.
14. *Pro/Engineer*, Parametric Technology Corp., 140 Kendrick Street, Needham, MA, http://www.proengineer.com/.
15. Nisbett, J., and Budynas, R., *Shigley's Mechanical Engineering Design*, Eighth Edition, McGraw-Hill, New York, 2008 pp 277-310, 933-953.
16. Hamrock, B.J., Jacobson, B.O. and Schmid, S.R., *Fundamentals of Machine Elements*, McGraw Hill, Boston, MA, 1999.

17. Mott, R.L., *Machine Elements in Mechanical Design*, Third Edition, Prentice Hall, Upper Saddle River, NJ, 1999.
18. Dimarogonas, A., *Machine Design: A CAD Approach*, John Wiley and Sons, Inc., New York, 2001.
19. Juvinall, R.C. and Marshek, K.M., *Fundamentals of Machine Component Design*, Third Edition, John Wiley and Sons, Inc., Englewood Cliffs, NJ, 2000.
20. Ugural, A.C., *Mechanical Design An Integrated Approach*, McGraw Hill, Boston, MA, 2004.
21. Edwards, Jr., K. and McKee, R.B, *Fundamentals of Mechanical Component Design*, McGraw Hill, New York, 1991, pp. 272-273.
22. Adams, V. and Askenazi, A., *Building Better Products with Finite Element Analysis*, Onward Press, Sante Fe, NM, 1999, pp. 56-57, 62-64, 65-66, 58-62.
23. Zahavi, E., *The Finite Element Method in Machine Design*, Prentice Hall, Englewood Cliffs, NJ, 1992, pp. 122-126, 180, 182, 256.
24. Hagigat, C.K., "Using Commercially Available Finite Element Software for Fatigue Analysis" *CD-ROM Proceedings of the 2005 American Society of Engineering Education (ASEE) Annual Conference and Exposition*, Portland, OR, June 20-23, 2005.
25. Bloom, B.S., *Taxonomy of Educational Objectives*, David McKay, New York, NY, 1956.
26. "Criteria for Accrediting Engineering Programs: 2008-2009 Accreditation Cycle," ABET Inc., Baltimore, MD, http://www.abet.org/forms.shtml.
27. Harris, C.M. (editor) and Piersol, A.G. (editor), *Harris' Shock and Vibration Handbook*, Fifth Edition, McGraw Hill, New York, NY, 2002.
28. Hibbler, R.C., *Engineering Mechanics: Statics*, Eleventh Edition, Prentice Hall, Upper Saddle River, NJ, 2006.
29. Beer, F.P., Johnston, E.R., Eisenberg, E.R., and Mazurek, D., *Vector Mechanics for Engineers: Statics and Dynamics*, Eighth Edition, McGraw Hill, New York, 2007.
30. Meriam, J.L., and Kraige, L.G., *Engineering Mechanics Statics*, Sixth Edition, John Wiley and Sons, Inc., Englewood Cliffs, NJ, 2006.

JOSH COFFMAN
Josh Coffman is a M.S student in the Department of Mechanical Engineering at the University of Arkansas, Fayetteville. He has worked as a civil design technician for Crafton, Tull, Sparks, and Associates in Russellville, Arkansas. Responsibilities included design of residential subdivisions, commercial properties, and municipal water and sewer systems. He received a B.S. in Mechanical Engineering from Arkansas Tech University in 2006. V-mail: 479-970-7359; E-mail: jacoffma@uark.edu.

SACHIN S. TERDALKAR
Sachin S. Terdalkar is a Ph.D. candidate in the Department of Mechanical Engineering at the University of Arkansas, Fayetteville. His current research is mainly in using computational methods to study the mechanics of nano structures like carbon nanotubes, graphene sheets. He has worked on molecular dynamic simulation of ion deposition induced curvature in thin films. Currently he is working on using nudged elastic band method and molecular mechanics to study the brittle to ductile transition in graphene sheet fracture. He has worked as senior engineer in John Deere Technology Center, Pune INDIA. His responsibilities at John Deere included finite element analysis and fatigue analysis to determine life of the newly designed components for new generation tractors. He received a M.S. in Mechanical Engineering from the Worcester Polytechnic Institute in 2003. His M.S. research formulated and developed a new algorithm for interactive stress reanalysis in early stages of design using ANSYS®. He received his B.S. from the College of Engineering, Pune INDIA in 1999. V-mail:479-575-6821; E-mail: sterdal@uark.edu.

JOSEPH J. RENCIS
Joseph J. Rencis has been professor and Head of the Department of Mechanical Engineering at the University of Arkansas, Fayetteville since 2004. He has held the endowed Twenty-first Century Leadership Chair in Mechanical Engineering since 2007. From 1985 to 2004 he was professor in the Mechanical Engineering Department at

Worcester Polytechnic Institute. His research focuses on boundary element methods, finite element methods, atomistic modeling, and engineering education. He currently serves on the editorial board of Engineering Analysis with Boundary Elements and is associate editor of the international Series on Advances in Boundary Elements. Currently he serves as Chair of the ASME Mechanical Engineering Department Heads Committee, Program Chair of the ASEE Mechanical Engineering Division, and an ABET program evaluator. He currently serves on the Academic Advisory Board of the College of Engineering at United Arab Emirates University. He received the 2002 ASEE New England Section Teacher of Year Award, 2004 ASEE New England Section Outstanding Leader Award, and 2006 ASEE Mechanics Division James L. Meriam Service Award. Dr. Rencis is a fellow of the ASME. He received a B.S. from Milwaukee School of Engineering in 1980, a M.S. from Northwestern University in 1982, and a Ph.D. from Case Western Reserve University in 1985. V-mail: 479-575-4153; E-mail: jjrencis@uark.edu.

ASHLAND O. BROWN

Ashland O. Brown is a professor of mechanical engineering at the University of the Pacific in Stockton, CA. He has held numerous administrative, management and research positions including Program Director, Engineering Directorate, National Science Foundation, Dean of Engineering at the University of the Pacific; Dean of Engineering Technology at South Carolina State University; Engineering Group Manager at General Motors Corporation: and Principal Engineering Supervisor, Ford Motor Company and Research Engineer, Eastman Kodak Company. He received his B.S. in Mechanical Engineering from Purdue University and M.S. and Ph.D. in Mechanical Engineering from the University of Connecticut. He has authored over 40 referred and propriety publications in automotive design, finite element modeling of automobile body structures, and photographic film emulsion coating instabilities. His most recent research includes development of innovative finite element tutorials for undergraduate engineering students and vibrational analysis and measurement of human skeletal muscles under stress using laser holography. V-mail: 209-946-3091; E-mail: abrown@pacific.edu.