Integration of Manufacturing Design Applications in FE–Based Applied Mechanics Courses

Mansur Rastani
North Carolina A&T State University

ABSTRACT

Many mechanical engineering disciplines are implementing numerical methods of designing mechanical and or structural components within junior or senior–level courses utilizing a technique such as finite element analysis (FEA). However, the classical examples and case problems studied in these courses do not usually provide the students with the erudition about the problems encountered in manufacturing of designed components. This manuscript practices a PC–based approach using algor finite element software to incorporate many of the problems which come from actual applications in various industries into the FEA design of mechanical/structural components. Some of the problems come from aftermarket product refinement, or during product development. It is the purpose of this paper to show how to use FEA from model building to obtaining a solution for these problems.

INTRODUCTION

Two rationales could be named for using Finite Element Method (FEM) in product design [1]. First, manufacturing technology is a field that synthesizes the design ideas to produce a valid product using existing tools and/or machinery. However, one has to have the tools and insights to a design in order to effectively synthesize that design into an optimized product. Every bit of a material not necessary for the functioning of a particular product adds to the overall product cost in raw material costs, production costs, shipping costs, and general overhead costs associated with the product. The bottom line for any product is cost and profitability.

Second, Often problems encountered in everyday practical applications do not lend themselves to closed–form or analytic solutions. This complication is found in almost every area of manufacturing design from tool and automotive industries to military, aerospace, consumer medical, and recreational products to mention a few. Consequently, the product designer is forced to make approximations based on similarity to classical, or closed–form soluble problems which result in overdesigned product that may be non–competitive from a cost standpoint. On the other hand, the product designer may rely on physical prototyping which is generally expensive and increases the cost of the design process. With the proliferation of personal computers and the availability of software to aid the design process, it is no longer necessary to rely heavily on intuition and settle for approximate solutions to real–world problems.

Finite Element Method[2] is a numerical analysis procedure to obtain solutions to these many complex problems posed in various areas of the manufacturing environment in order to produce the optimum product: a product that performs as intended, meets all of the specified environmental requirements, and is the least costly to produce. A good FEM software[3] interfaces with a variety of CAD programs. The interface allows a rapid design and analysis cycle of the model.

The engineering curricula commonly practice and implement finite element analysis technique toward academic applications. These applications are usually looking upon the classical type mechanical and/or structural components rather than a practical product part. Students however, need to be exposed to real life forms of elements which utilize manufacturing process to convert the design into the product. Today, the use of FEA in some businesses such as automotive support industry is commonplace. In fact the use of FEA is almost a requirement, since minimizing weight while maintaining strength and overall performance of the automobile is of prime concern. All systems within an automobile can benefit from finite element analysis-everything from engine blocks, transmission housings, and suspensions to wheels, pistons, muffler, and so on. The goals of the automobile manufacturer are to provide safety, durability, and economy in its products. This requires the streamlining of the manufacturing and engineering processes in order to bring advanced technologies to fixed design requirements. Many sport-supported industries can also benefit from finite element analysis in designing their products. As products become more sophisticated and the demands of consumers grow, the application of finite element analysis in product development and aftermarket product refinement increase. This manuscript also looks at an aftermarket application as applied to instrumental product design and its possible improvement, using a FEA software. Most musicians look to gain a performance edge through long, dedicated hours of practice, a new instrument, modifications to their basic instrument, or a combination of the three. Certainly no one can discount the merits of practice. However, there are things that can be done to the instrument to make playing easier and/or to enhance the instrument characteristics. The problem in the following sections addresses the modifications of a trumpet mouthpiece. An algor FEA software[4] has been used to solve these problems. The details of inputting data and using the program efficiently are best left to the ALGOR manuals listed in the reference section.

CLASSIC SHRINK-FIT PROBLEM

The first problem presented herein is a classic shrink–fit problem involving a plain, thin ring gear having an inner diameter of approximately 6 inches and measuring 13/16 inch high with a cross section of 0.18 inch. This gear is a shrink-fitted one–third of the way down from the top of a 1/2"–thick, slightly cone–shaped cylinder. The contact pressure between the inside of the gear and the outside of the cylinder has to be high enough to maintain a friction torque; but if the interference fit is too high, the gear and cylinder will deform. The design goal is to minimize the deformation of the inside cylinder while still producing the required torque transmission between the cylinder and gear. It is also important to know the tolerances and how they would affect the torque transmission capacity of the gear. The materials from which the gear and cylinder will be Fabricated are two types of steel: 4140 and 4150.

FEA Solution

The full model is created by extruding a 2–D plan view of the cylinder in z–direction using Modify–Copy command. Boundary conditions on the model are set with the ADD:FEA add command. Several nodes are constrained in the direction of the Z–axis to hold the gear fixed in space. All information concerning the elements, their properties, and loading conditions are added to the graphical information in the decoding step[5]. The uniform internal pressure of 284 psi is applied to each of the element faces on the outer diameter of the cylinder. The value of 284 psi for the pressure on the cylinder face and gear ring interface was estimated from the values in the ASME Handbook. The model is viewed in SuperView. The results of the stress analysis of the cylinder, using stress processor[6] (SSAPOH), is shown in figures 1. Given the material, deflections and stress levels are well within requirements. The gear part of the analysis was handled in much the same way as the cylindrical mating part. The gear analysis shows that a radial displacement of approximately 0.00056 inch occurred, figure 2.

The analysis confirmed that this gear could be used on the cylinder and that it would handle the 90 ft–lb of torque to which it would be subjected. That torque will be transmitted by the outside ring to the inner ring and will not slip. The algor program also determined that the stresses, according to the Von Mises criterion, were 5400 psi — very low for the type 4140 steel that would be used. It turned out that the interference fit was 0.0015 inch. This design and the numbers obtained from the analysis mean that we could relax manufacturing tolerances from those that had originally been anticipated, so they wouldn't be unnecessarily high.

SPINDLE PROBLEM

The spindle on a front axle of a vehicle is one of the key components in transferring vehicle weight to the wheels and tires. The particular spindle analyzed in this chapter can be thought of as a circular disk with a hollow cylinder attached at its center. The disk part is rigidly bolted to the steering knuckle, while the cylinder supports the roller bearings for the wheel hub. Load data for the finite element model was estimated from the vehicle weight and length specifications. The spindle is basically a cantilevered cylinder and uses an AISI 4140 steel material. The objective is to check on stress level and deformation contours for safety purposes.

FEA Solution

The basic model was constructed in SuperDraw II. As the full model is developed, the restraints and forces were applied. The model was assembled through 420 elements and 672 nodes. The interesting point is that the drafting part of the model could be done through any other CAD software such as CADKEY, in which case the resulting geometry should be imported to algor SuperDraw using a .CDL file. The model was decoded[5] and processed with algor's linear static stress package[6] (SSAPOH). Figures 3 shows the postprocessed model with stress levels. The deformation contours which are well within the material requirement are not shown because of space limitation. As a further study to this problem could be the FEA analysis of following modification to the spindle which is left to the reader.

SPINDLE MODIFICATION

A vacuum locking hub system design is used to disengage the four–wheel drive cars. This vacuum system requires that a 4 mm hole be drilled through the base of the spindle aligned 90 deg to the loads. Since the spindle is basically a cantilevered cylinder, this will place the hole in the most highly stressed region of the part. The primary purpose of this analysis is to investigate the percentage increase in stress level caused by adding hole.

THE TRUMPET MOUTHPIECE PROBLEM

A standard Schilke 14A4 trumpet mouthpiece is selected for this problem. When the lips buzz to produce the sound, they will vibrate at a certain pitch. For Example, the note A in the staff (for treble clef instruments) has a frequency of 440 Hz. As you approach the natural frequency of the mouthpiece, the forcing function (your vibrating lips) excites the mouthpiece. The once clean vibrations that are carried through the trumpet are now amplified and carry into the horn the overtones of that note and the tone that is modified by the vibrating mouthpiece. The problem solicits a solution by using a tone intensifier ring to damp out the unwanted overtones and other frequencies so that the horn reproduces exactly what is intended by the player.

FEA Solution

To avoid hitting the resonant modes of the mouthpiece it is required that the natural frequencies of the mouthpiece be changed to a higher frequency. This simply means that the mouthpiece cross section should be modified for a heavier–wall design. One way to do this is simply using a tone intensifier ring which is essentially a brass sleeve that fits over a portion of the mouthpiece.

Baseline Model: To model the mouthpiece, a somewhat common mouthpiece was used. All data were taken from instrument specifications. The baseline model was constructed, meshed and the restraints were applied in Super Draw II, figure 4. The model is next preprocessed by the algor decoder, using brass material and an eight–node brick element. The model was analyzed using the modal anal ysis processor[7] (S SAP1H) to determine the natural frequencies up to the third mode. The first, second, and third mode frequencies areat681 Hz, 681 Hz, and 2582 Hz respectively.
Modified Model: The modified mouthpiece consists of a brass ring approximately 0.75 inch in diameter by 0.98 inch in length. The inner diameter fits the taper of the shank of the mouth piece. Use is made of SuperView to render the model in solid perspective to check for any modeling defects. Figure 5 shows the solid rendering of the model. After processing the natural frequencies for this model were 1162 Hz, 1162 Hz, and 3461 Hz. As can be seen, the additional piece of brass increased the natural frequencies by a factor of 1.71 for the first two modes and 1.34 for the third mode. As a result of this change the resonant modes are prevented from occurrence during playing the instrument.

BICYCLE PROBLEM

A bicycle frame is made of aluminum tubing welded together. The properties for this material are:
Young's modulus E = 1.0E7 psi
Shear modulus G = 3.8E6 psi
Yield Strength SY = 70 ksi
Weight density WDEN = 194 lb/ft^3 = 0.112 lb/in^3
Four groups of tubes with different sectional properties as follows are used:

ID#	Area(in^2)	J 1 (in^4)	12 and I3(in^4)	S2 and S3(in^3)
1	0.4418	0.1415	0.0708	0.1132
2	0.2827	0.0580	0.0290	0.0580
3	0.1590	0.0183	().()092	0.0245
4	0.0707	0.()()36	0.0018	0.0072

Where I, J, and S are moment of inertia, torsional resistance, and section modulus of the tube section, and subscript 1, 2, and 3 are referred to x, y, and z directions respectively. Assume that a reasonable worst condition is when the bicycle rider is standing up and pedaling hard. Find the point with the highest stress relative to yield.

FEA Solution

The bicycle frame consists of 8 elements as follows: element 1 use tube group #l, elements 2, 3, and 4 use tube group #2, elements 5 and 6 use tube group 3, and elements 7 and 8 use tube group #4.
The model is created, using Superdraw II of algor software. Then transfer the model to BEdit[8], and
a) apply the boundary conditions. The rear wheel axis is assumed to be fixed in space and the front wheel axis can not move up or down,
b) The nodal forces and moments are then applied. When the bicycle rider is standing up and pedaling hard, the forces and moments applied by the handle bars and the pedal shaft could be as follows:
At node 2: Fz = 200 lb, Mx = –2400 in.lb
At node 4: Fx = 800 lb, Fz = –400 lb, Mx = 2400" in.lb, Mz = 1600 in.lb
c) The material and geometry properties are inputted.
The model is then analyzed using SSAPO processor. Using Superview and Postprocessing the model is dithered and stress levels are viewed, figure 6.

CONCLUSION

These were just few practical FEA applications. The real-life applications are numerous, The integration of manufacturing design applications in FE–based applied mechanics courses allows the student to face the issues encountered in manufacturing of the designed component and help him with a better understanding of the problems associated with fabrication of the product. So that he could recycle the issues back into the design process for an optimal solution. It also provide the student the opportunity to interact with the actual problems meet during a product development. These practical FEA applications could be used as a step toward streamlining of the manufacturing and the engineering design process. The application of enhancing the characteristics of an aftermarket product is another example to promote students' engineering design consensuses.

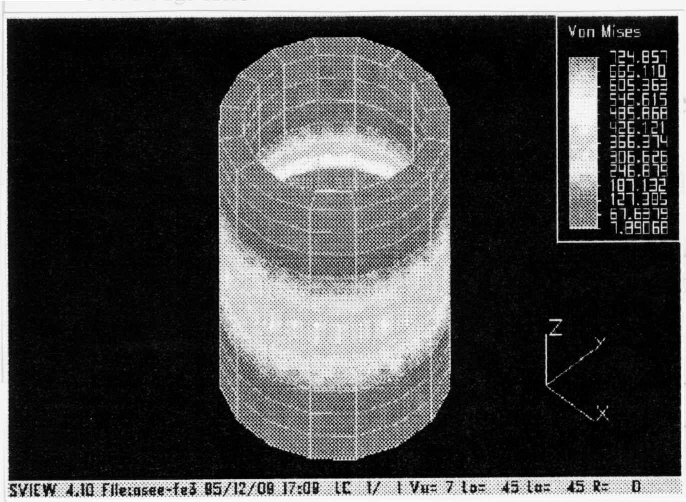
These examples could form either a part of existing 2nd level finite-element-bused applied mechanic course or could be taught at the applied mechanic project level courses. In latter case, a prerequisite of introduction to FEA is a necessity. Full participation of the finite element analysis of the examples are expected from students' role in the project.

REFERENCES

1. Woodard, K. S., and L.B. Gray, Computers in Mechanical Engineering, ASME, 1990, Vol.2, pp. 51 –56.
2. Segerlind, Larry J., Applied Finite Element Method, Vol 1, Fourth Edition, McGraw–Hill, New York, 1989.
3. Kardestuncer, H, Finite Element Handbook, McGraw–Hill, New York, 1987.
4. Algor Publication, Education Seminars, No. 950.700, Algor Inc., P. A., 1991.
5. Algor Publication, Finite Element Analysis System, Stress Decoder Reference Manual, April 1990.
6. Algor Publication, Finite Element Analysis System, Accupak Ref. Manual., No. 110.401, Rev. 3, Dec. 1993.
7. Algor Publication, Finte Element Modal Analysis, APAK 1 Package, Reference Manual, 1993.
8. Algor Publication, Finite Element Analysis, BEdit Reference Manual, 1992.

Mansur Rastani

Dr. Mansur Rastani received his B.S. and M.S. degrees in Mechanical Engineering from Aryamer University in 1978. He earned his Ph.d. degree in Mechanical Engineering from NC State University in 1986. Currently he is an Associate Professor in the Department of Manufacturing Systems at NC A&T SU. He has been a registered Professional Engineer (P. E.) by the board of NC state and also certified as Manufacturing Engineer by MECI since 1989. His teaching interests include courses in mechanics of solids, machine design, finite element stress analysis, and materials. He has been involved with NASA-SSF research project and worked as a consultant for US ARDEC. His recent research interests are optimal design of structures, smart structure, and advanced materials. He is a past president of SME organization and a senior member of ASME and ASEE.



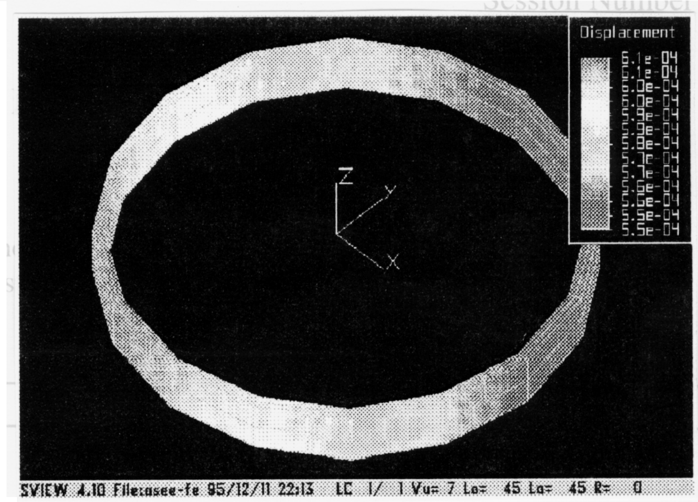
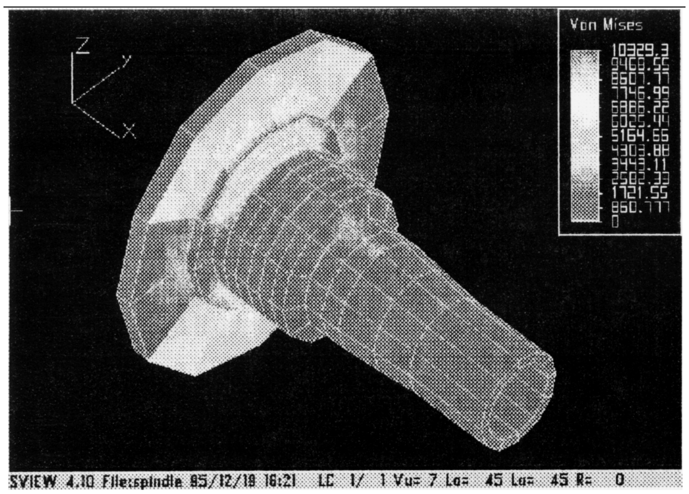


Figure 2. Deformation of ring gear.



m

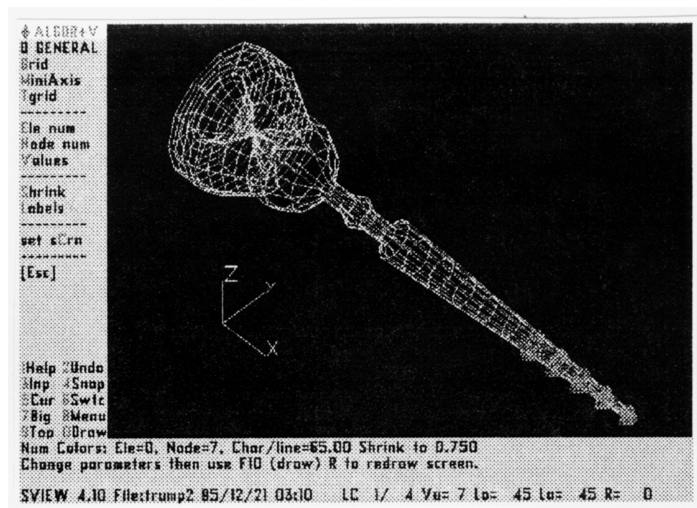
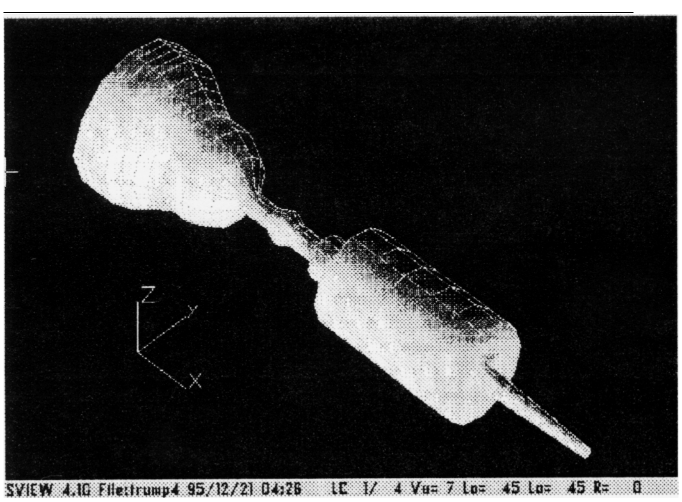


Figure 4. Wireframe, baseline mouthpiece model.



m th

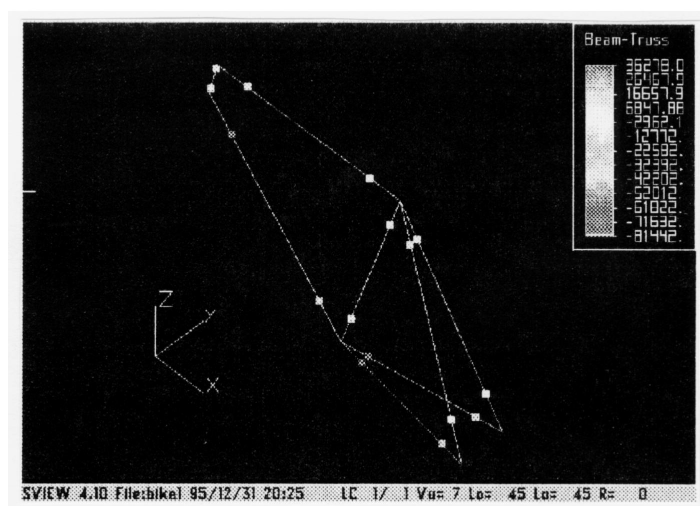


Figure 6. FE model of the bicycle frame,