

The Feasibility of Teaching FEA in a Freshman Engineering Graphics Course

Thomas J. Krueger and Ronald E. Barr

Department of Mechanical Engineering
The University of Texas, Austin, TX 78712

ABSTRACT

In the freshman Introduction to Engineering Design and Graphics course, the students are introduced to various solid-modeling techniques, including simple solid modeling, solid model assemblies, dimensioning, and sectioning. Along with these basic graphics fundamentals, the students are also exposed to mass properties reports and simple Finite Element Analysis (FEA). This introductory exposure to the Finite Element Analysis process gives the students some idea of what goes into the design of products and makes them aware of what is necessary for good design. Often the students perceive design as a simple process of function without the concept of material limitations. This unit on Design Analysis can provide the freshman student some concept of how to design a product and to test its integrity. This paper will present the learning module used to introduce the students to this simple design experience.

I. Introduction

Several papers have been presented concerning the possibility of teaching Finite Element Analysis to lower division engineering students ^{1, 4} and whether it can be done effectively. As early as 1992, Steven Howell ³ pointed out that the Finite Element Method (FEM) is typically introduced in the upper level engineering courses. With the ‘integrated design software’ available for the PC, it is now possible to expose freshman level students to the finite element method without overwhelming them in the detailed math required to set up a model.” In the Spring 1999 issue of the *Engineering Design Graphics Journal*, William Cole ² stated, “Solid modeling overcomes this difficulty of complex software with very specific data entry requirements. Once the solid model is created, there are several programs available that can

generate the mesh patterns and perform the FEA analysis on the solid model transparently. Thus FEA can be introduced quickly and easily into an EDG course.”

In 1996 Davor Juricic developed a series of exercises that were designed to expose the freshmen students to the procedure of design analysis⁵. The unit introduced FEA using AutoCAD and ANSYS to introduce the students to a form of analysis without requiring them to know all of the intricacies of the process. The exercise required the student to develop a 2-D slice and then create a small line segment to which the force was to be applied. This in time proved to be too cumbersome. In addition to those confusing steps, the program itself was unreliable and often crashed.

II. Objectives

It is not the intention of this project to demonstrate the power of engineering analysis nor to teach a lower-division engineering student any specific method of engineering analysis. The purpose of extending the Engineering Design Graphics (EDG) labs to engineering analysis is to demonstrate some of the methods of engineering analysis that use the computer description of the model geometry to perform engineering analysis directly, without user interpretation of the geometric data³.

III. Application of the FEA Module

Recently, SolidWorks has offered the COSMOSWorks add-in software to their modeling package. The FEA analysis unit has become very successful in showing the students what happens when items are put under restraints and forces and what changes take place when the design is changed. The newest version of SolidWorks has totally incorporated COSMOSWorks into their software so the interface is seamless. The overall approach to developing an FEA study is depicted in the flowchart diagram of Figure 1. It shows seven sequential steps that must be followed in order to perform an FEA study.

Step 1: Building the Solid Models

To begin the module, the students are required to build a pillow block and a shaft that is positioned at the center of the pillow block. The pillow block design is to be built according to the specifications in Figures 2 and 3. They sketch profiles on a 2-D plane, and add the appropriate dimensions to the sketch. They then extrude the solid part at mid-plane in order to keep the origin in the center of the large hole. Features like holes and slots are cut into the object. A pictorial of the resulting model is shown in Figure 4.

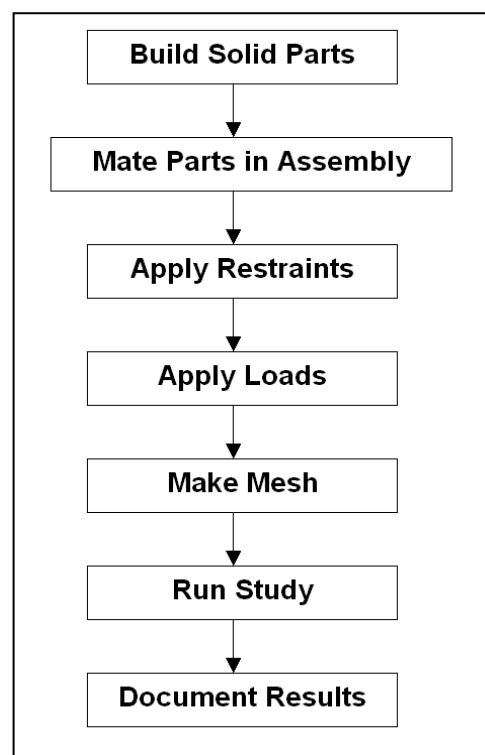


Figure 1. Procedure for FEA Study.

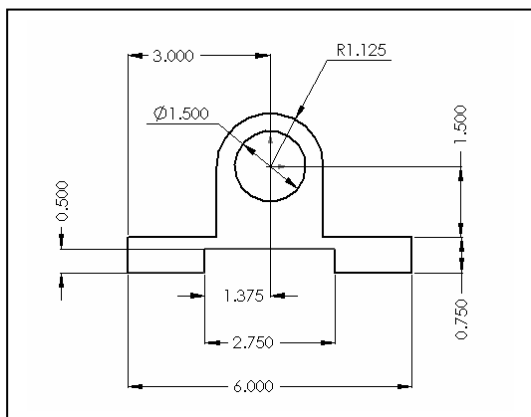


Figure 2. Front View Sketch.

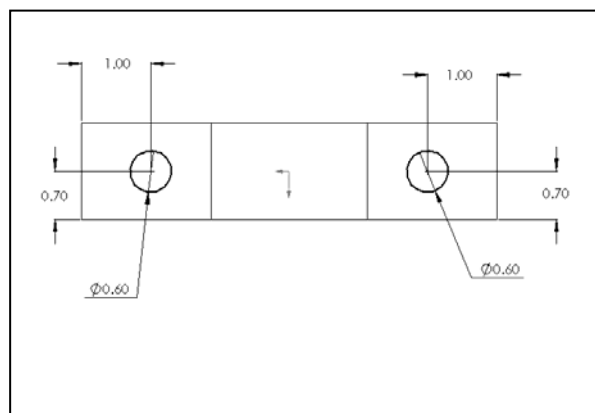


Figure 3. Top View of Sketch

Similarly the shaft is built with a 1.50 diameter thickness that is 7 inches long. It too is extruded at Mid-plane to maintain its origin at a convenient location for assembly. There also is a 0.125in. chamfer on each end. The finished shaft model is shown in Figure 5.

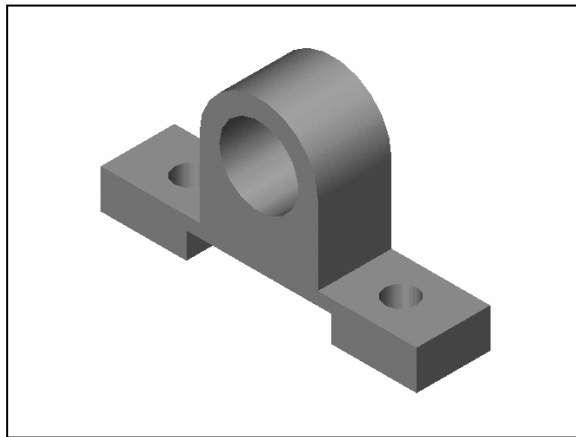


Figure 4. The Completed Pillow Block.

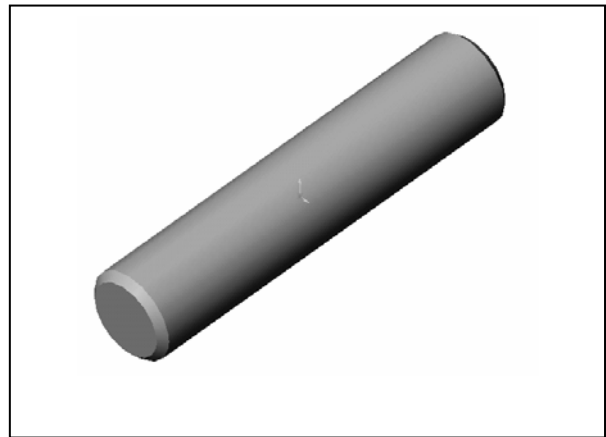


Figure 5. The Completed Shaft.

Step 2: Mating the Parts in an Assembly

After the two parts are built the students create an assembly where the shaft is centered in the hole of the pillow block. The assembly will look like the model in Figure 6. At this point the students are asked to save their assemblies in preparation to use COSMOSWorks.

The actual analysis of the pillow block could be a complex procedure even with COSMOS/Works, since analysis of the pillow block would have to include the effects of the rotating shaft and thermal stress on the shaft and the pillow block. In this case only a simple static analysis of the pillow block will be carried out.

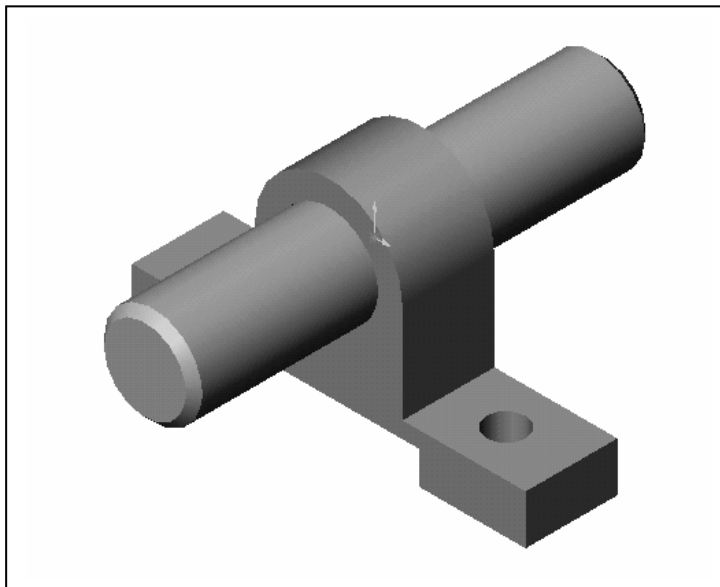


Figure 6. The Completed Assembly

Step 3: Applying Restraints

The students are instructed to carry out a series of tasks to prepare the assembly for analysis. They must first name the study, indicate the type of study to be conducted (static), and identify the mesh type (solid mesh). Following these activities they will assign the type of material to each part of the assembly. The assembly is now ready to have restraints applied to the bottom surfaces of the pillow block (Figure 7).

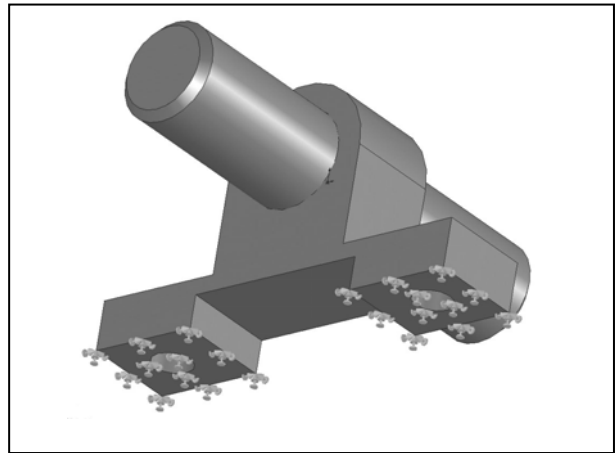


Figure 7. The Applied Restraints.

Step 4: Applying Forces

After the restraints are applied to the pillow block, a downward force is applied to the shaft to indicate a load on the shaft (see Figure 8). The force applied to the shaft will test the integrity of the pillow block.

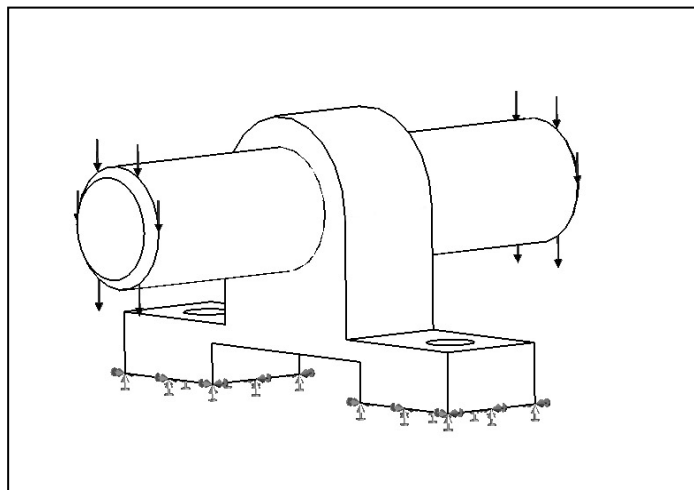


Figure 8. The Applied Force on the Shaft

Step 5: Creating the Mesh

The next step of preparation before the study can be run is creating the mesh. The student opens the mesh dialog box, which allows the user to define the mesh size for the analysis. A smaller mesh yields better results, but takes longer to mesh and longer to run the analysis. Hence it is a tradeoff between mesh size and computation time. At the end of the meshing process COSMOSWorks displays a mesh similar to the object shown in Figure 9.

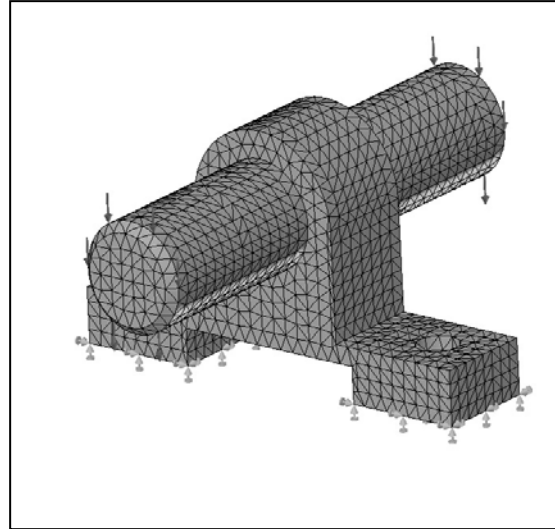


Figure 9. The Meshed Model

Step 6: Running the Study

With the “Run” command the program computes the outcomes according to the specified criteria. When the program run is complete, the results will show Stress, Displacement, Strain, Deformation, and Design Check. Each of these results will display in color to indicate where problems in the design might arise

Step 7: Documenting the Study

At this point, COSMOSWorks has made all the calculations needed to display the results (Stress, Displacement, Strain, Deformation, and Design Check). All displays except for the Design Check can also be animated to show how they react to the force. These display and animation capabilities of COSMOSWorks are very appealing to the student and serve as intuitive insight into the FEA process of analysis.

➤ **Stress** - The Stress factor (von Mises stress) is shown in Figure 10. By the light colors it is obvious that the indentation at the bottom of the pillow block has some weaknesses.

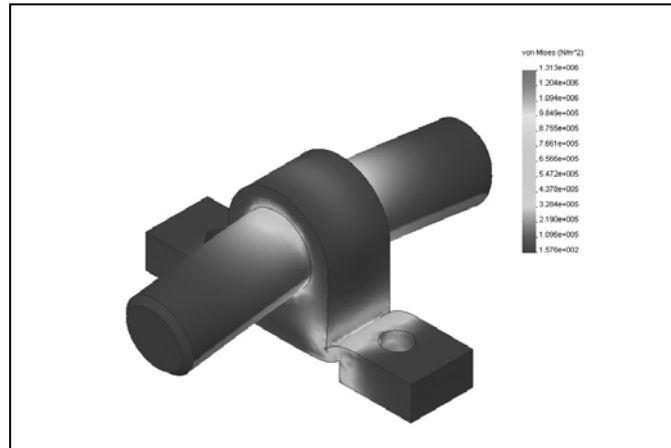


Figure 10. Stress Display.

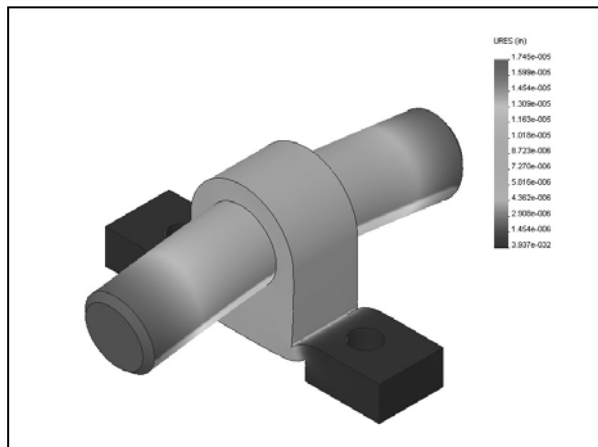


Figure 11. Displacement Display.

➤ **Displacement** – This item of the analysis report is the displacement as shown in Figure 11. Though it can not be seen in the black and white illustration, the ends of the shaft which are colored red show the portion of the assembly that has moved the furthest from its original position.

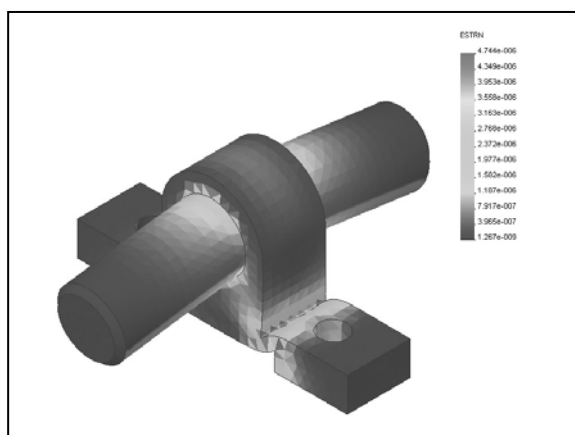


Figure 12. Strain Display.

➤ **Strain** - The next item on the FEA report is the Strain, which can be seen in Figure 12. The strain factor illustrates those areas that may eventually deform and change properties due to the forces applied to the bodies.

➤ **Design Check** - The last item on the analysis report is the Design Check which is shown in Figure 13. This illustration shows areas that may need to be strengthened to produce a more viable design.

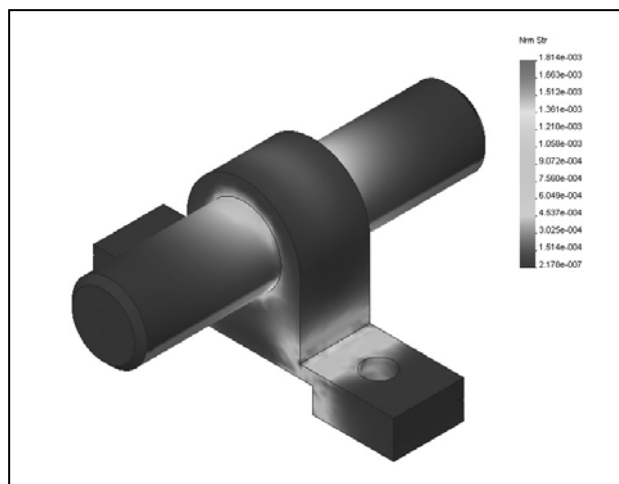


Figure 13. The Design Check

IV. Design Modification

Upon completion of the initial FEA study the students are directed to make some fundamental design changes to strengthen the original pillow block. Those changes include adding fillets between the vertical and horizontal parts of the pillow block, reducing the size of the lower cut-out and increasing the wall thickness parts around the shaft hole. They are given some suggested values for these design revisions, but are also encouraged to come up with their own ideas. The revised pillow block is shown in Figure 14.

After they have finished these operations, they are asked to reapply the restraints, the force and create a new mesh in order to run the study the second time. In so doing, they can verify that the new design has better design performance in the areas of stress concentration and displacement than the original design.

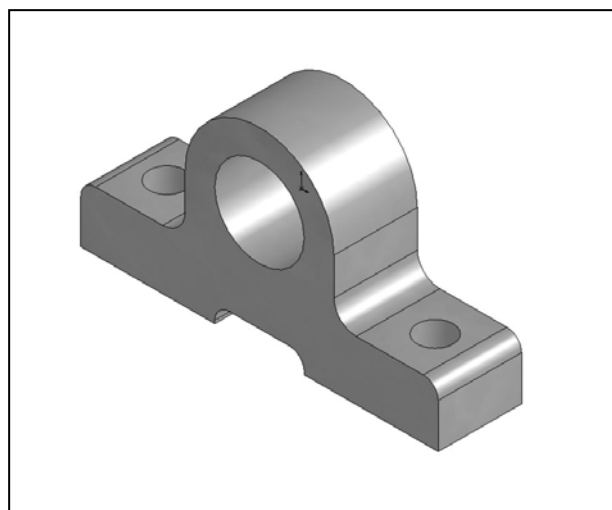


Figure 14. Revised Pillow Block.

With the redesign of the pillow block, there were apparent changes in the study results. The design of the pillow block was significantly improved, as the illustrations of the stress analysis and the design check indicate in Figures 15 and 16.

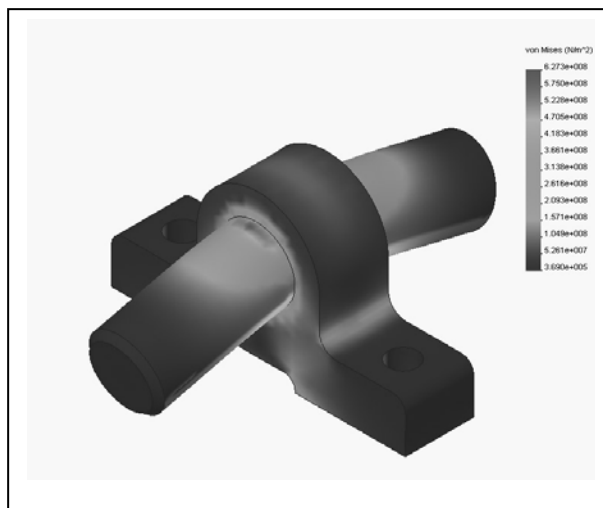


Figure 15 – Stress of the Revised Pillow Block

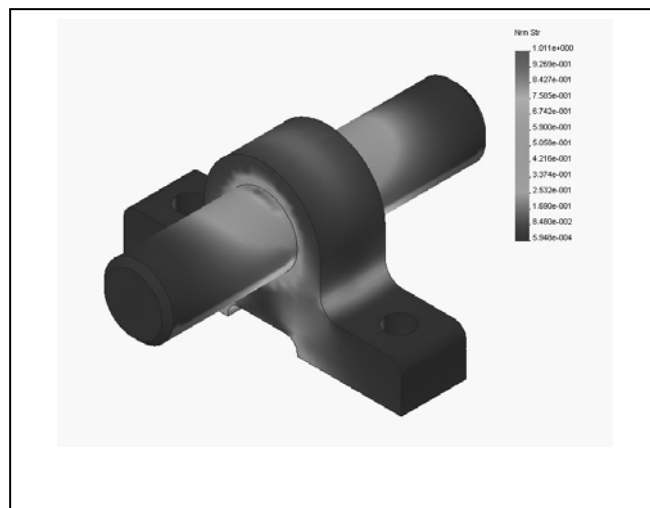


Figure 16. Design Check for Revised Pillow Block.

V. Classroom Testing

A survey was administered to 75 students in the freshman Engineering Graphics course in the fall of 2004 to determine if there was a significant improvement of their knowledge of finite element analysis. Three questions were posed to them:

- (1) At this stage in the course, what is your understanding of Finite Element Analysis (FEA) of a 3-D solid model?
- (2) At this stage in the course, what is your understanding of applying constraints, loads, and meshes in Finite Element Analysis of a 3-D solid model?
- (3) At this stage in the course, what is your understanding of visualizing results of Finite Element Analysis of a 3-D solid model?

The same three questions were asked as a Pre-exercise survey and as a Post-exercise survey. As can be seen in the Figure 17, there was a significant increase in the perceived knowledge of what finite element analysis is and what it does for the designer.

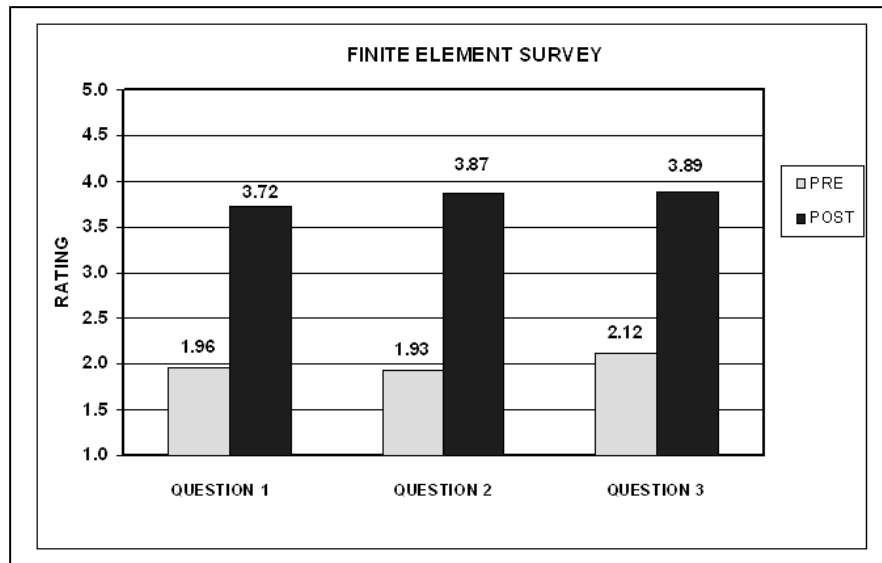


Figure 17. Pre- and Post- FEA Study Results for Three Questions Posed.

VI. Discussion and Conclusions

What are the benefits of FEA in a freshman Engineering Graphics course? For one, they get a visual picture of stress, strain, deformation, displacement and design check. Also, they see a modern application of the solid model that takes the lab experience beyond geometry and into analysis.

Several of the students were not able to get this course as a freshman. One of those students, a senior, gave high knowledge ratings in the Pre survey, but commented that those initial ratings should be ignored when he filled out the Post survey. He commented: “I learned that application is a better teacher than conversation.”

Though the students may not comprehend the total workings and the theory of the analysis, they can visualize what happens when forces are applied and also visualize what might be done to strengthen and improve a design. This early exposure to analysis will help the students later in their more rigorous design courses.

The success that we have had in introducing this unit in our freshman engineering design graphics course shows that an introduction to Finite Element Analysis is feasible and effective for our students.

References

1. Boronkay, T. G., and Janak, D. "Introduction of Finite Element Methods in the Lower Division Mechanical Engineering Technology Curriculum." *Proceedings of the ASEE Annual Conference*, Milwaukee, WI, 1997. Session 2238.
2. Cole, W.: "Graphical Applications: Analysis and Manufacturing". *Engineering Design Graphics Journal*, Spring, 1999, pp 43-49.
3. Howell, S.: "Finite Element Analysis in a Freshman Graphics Course?" *Engineering Design Graphics Journal*, Winter, 1993, pp 29-32.
4. Juricic, D., Howell, S., Jenison, R., and Barr, R. "Extending Engineering Design Graphics Laboratories to have a CAD/CAM Component – Part II: Engineering Analysis". *Proceedings of the ASEE/EDGD 1995 Mid-Year Conference*, Ames Iowa, 1995, pp 73-82.
5. Barr, R. E., Juricic, D., Krueger, T. J., Wood, B. H., and Miller, L. S. *Engineering Design Graphics Workbook*. Schroff Development Corporation, Mission, KS. 1997.

Biographical Sketches

Thomas J. Krueger

Dr. Krueger is a Teaching Specialist in the Mechanical Engineering Department at the University of Texas at Austin, where he has taught since 1994. He received his B. S. from Concordia Teachers College in 1966 and his M. Ed and Ph.D. from Texas A&M University in 1971 and 1975 respectively. Before coming to the University of Texas at Austin, Krueger taught at Texas A&M University, Brazosport College, and Southwest Texas State University.

Ronald E. Barr

Dr. Barr is a Professor of Mechanical Engineering at the University of Texas at Austin, where he has taught since 1978. He received both his B.S. and Ph.D. degrees from Marquette University in 1969 and 1975, respectively. Barr is the 1999 recipient of the Distinguished Service Award (DSA) of the Engineering Design Graphics Division of ASEE. Barr is a Fellow of ASEE and a registered Professional Engineer (PE) in the state of Texas.