122nd ASEE Annual Conference & Exposition
June 14 - 17, 2015
Seattle, WA
Making Value for Society
Paper ID #11664

Using 3D Printing and Physical Tsesting to Make Finite-Element Analyis More Real in a Computer-Aided Simulation and Design Course

Dr. Robert T. Bailey P.E., Loyola University Maryland

Dr. Robert T. Bailey is currently associate professor and chair of the Department of Engineering at Loyola University Maryland. He received his B.S., M.S., and Ph.D. degrees in Mechanical Engineering from the University of Florida, the latter in 1991. He worked in industry for Westinghouse and Science Applications International Corporation, served as a senior program officer at the National Research Council, and taught previously at the University of Tennessee at Chattanooga. His research interests include mechanistic engineering analyses to support risk and safety assessment of industrial processes, application of computational fluid dynamics to microscale flows involving mixing and chemical reaction, and improvements in engineering education. Dr. Bailey is a member of the American Society of Mechanical Engineers and the American Society of Engineering Education and is a registered professional engineer in the state of Maryland.

Using 3D Printing and Physical Testing to Make Finite-Element Analysis More Real in a Computer-Aided Simulation and Design Course

Abstract

The mechanical engineering curriculum at Loyola University Maryland includes a junior-level course in computer-aided simulation and design (EG426). In this course, students use SolidWorks® to create computer models of three-dimensional parts and assemblies and learn how to generate engineering-quality design drawings. The class also covers the use of finite-element analysis (FEA) to evaluate stresses and deflections of parts under load. Ultimately, the course culminates in a professional project where each student designs a mechanical part to meet a set of specific performance requirements, utilizing a design process that includes the iterative application of FEA. The course is generally very popular with students and alumni who comment favorably on its practicality and applicability in industry. Unfortunately, until recently, the Engineering Department's manufacturing equipment made it impractical to have the students actually fabricate their designs.

In the spring of 2013, a Stratasys Objet Model 30 3D printer was purchased to expand our rapid prototyping capabilities. This technology was a natural fit for incorporation into EG426, and changes were subsequently made to do so. Instead of being a purely digital and paper exercise, the students were asked to fabricate their final designs using the 3D printer and to subject their parts to physical testing to verify that certain performance requirements had been met. This also allowed them to compare the deflection predictions (made using FEA) with the actual deflections under load. The overall goal of these changes was for the students to come to a clearer understanding of how the successful and informed execution of FEA in the design process could positively affect the ultimate performance of their designs.

The effectiveness of this active, project-based learning approach was assessed through instructor evaluation of student performance, student questionnaires, and solicitation of oral comments. Written student evaluations of the experience indicated that the students (1) enjoyed actually seeing their products come to life, and (2) developed a better understanding of how FEA could be used to guide and enhance their designs. Suggestions for improving the incorporation of 3D printing in the course, based on both student comments and instructor reflections, are discussed.

Introduction

Finite-element analysis (FEA) has become firmly integrated into today's product design and development process[1]. Engineers now routinely subject virtual models of their evolving designs to simulated loadings to determine stresses and deflections and subsequently adjust their designs to better satisfy performance requirements and constraints. (Additional types of numerical simulations involving thermal, vibrational, fluid flow, and other physical phenomena are also conducted[2], but the focus of this paper is on stress and displacement.) Previously, FEA was performed almost exclusively by analysts with advanced scientific or engineering degrees and/or multiple years of experience, but commercial software has evolved to the point where relatively inexperienced, bachelor's level engineers are expected to conduct reasonably straightforward finite-element simulations with confidence and proficiency. To meet this expectation, most universities now include some instruction on FEA in their undergraduate curricula[3-5]. Because these curricula are packed with other material that is considered vital to the successful practice of engineering, it is challenging to give FEA instruction the time and depth necessary to ensure students are prepared to apply it effectively upon graduation.

At Loyola University Maryland, FEA is covered initially in EG426—a junior-level course in computer-aided simulation and design. During the first part of the course, students use SolidWorks®[6] to create computer models of three-dimensional parts and assemblies and learn how to generate engineering-quality design drawings. Because these students have already had an introductory course in solid mechanics, as well as three semesters of calculus and a course in differential equations, this class subsequently includes the use of FEA to evaluate stresses and deformation of parts under static loads using SolidWorks® Simulation—an "add-in" module available with the premium educational edition of SolidWorks®[1]. The five specific learning outcomes for this course are listed below.

At the completion of the course, students will have demonstrated the ability to

I. create professional engineering drawings of mechanical components using computer-aided design (CAD) software; (a, e, k)

II. use CAD software to create three-dimensional (3D; solid) computer models of mechanical components that can be used as input to commercial engineering analysis programs [*i.e.*, finite element analysis (FEA) software]; (a, e, k)

III. use commercial FEA software to generate numerical solutions for stress, strain, and displacement associate with mechanical components for realistic boundary conditions; (a, e, k)

IV. assess the accuracy of computer-generated solutions by comparing them with experimental data and/or analytical predictions; (a, b, e, k)

V. design a mechanical component to meet specific performance requirements, create associated CAD drawings to document the design in a professional format, and support the design by appropriate application of engineering analysis software and traditional engineering analyses. (a, e, g, k)

(Letters in parentheses after each outcome above represent the ABET student outcomes that are supported by that particular course learning outcome[7].)

A key instructional concern regarding FEA is that commercially available software has, in general, become so user-friendly and robust that stress and displacement solutions can almost always be obtained, even when such solutions do not accurately reflect physical reality. Errors can be due to the incorrect specification of boundary conditions (loads and restraints) in a simulation, or they can also occur as a result of inappropriate modeling assumptions (*e.g.*, performing a linear analysis for a nonlinear problem). It is, therefore, important that the treatment of FEA in undergraduate courses equip students to critically evaluate the results of their simulations and to identify when these results do not make physical sense.

Another concern about the coverage of FEA in our program was that the curriculum did not include an opportunity for students to exercise FEA as it is used in industry in terms of guiding the development of prototypes which are then fabricated and physically tested. Our department has a machine shop with versatile manufacturing and fabrication equipment, but we lacked genuine, cost-effective, rapid prototyping capabilities. To address this issue, which was broader than just achieving effective instruction in FEA, the department purchased a Stratasys Object Model 30 three-dimensional (3D) printer in the spring of 2013. 3D printing is an additive manufacturing technology where physical parts are built up a layer at a time in a computer-controlled process[8,9]. This particular device uses the PolyJet method, whereby a liquid photopolymer is deposited layer by layer and cured using ultraviolet light. The process generally produces smooth, detailed parts with precision down to 0.1 mm[10]. A removable, gel-like material is used where overhangs or complex shapes require support during fabrication. Overall, this process yields higher-fidelity parts than the more-common fused deposition modeling (FDM) approach in which polymers such as acrylonitrile butadiene styrene (ABS) or polylactic acid (PLA) are heated and extruded to form the 3D part. However, PolyJet machines are generally more expensive, as are the parts produced using them[11].

The addition of a 3D printer afforded the opportunity to improve our approach for teaching FEA. With this in mind, an FEA-based final project from EG426 was modified to include the creation and testing of a physical prototype. This project was conceptually linked to a prior, mid-term project in the same course. The underlying theme was to use these two projects to tie FEA to (1) analytical mechanics concepts presented in a prerequisite solid mechanics course and (2) physical confirmation testing of student-designed product prototypes. Both projects represent *active-learning* activities that require the students to dynamically participate in developing their own skills and understanding[5,12]. They contain aspects of *problem-based learning*[12], but they are individually-based, rather than team-based. The structures of the projects also provide *scaffolding*[13] to guide the students as they learn about and apply FEA, first to a tightly-specified analysis problem (Project I) and later to a more open-ended, performance-based, design task (Project II). The concept of building on prior knowledge to create mental models for categorizing and internalizing new knowledge—a key component of the *constructivist* point of view—is integral to the overall two-project approach[14].

Each of these projects is now described.

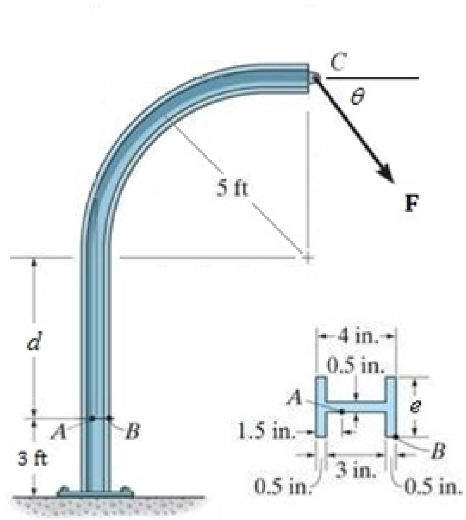
Project I – Finite-Element Stress Evaluation

Project Description

Evolving versions of this first (mid-term) project have been developed over the course of five years. It does not contain a 3D printing component, but is sets the stage for the second (final) project, which does. The first project was assigned following two 75-minute class periods dedicated to exploring the fundamentals of FEA and the steps required to execute a static stress analysis using SolidWorks® Simulation. Instruction was provided about the mathematical foundation for the method, and students worked through two example problems from Reference 1, which allowed them to apply the method within the SolidWorks® environment. A brief review of analytical approaches from solid mechanics for determining combined, principle, and von Mises stresses under plane stress conditions was also performed. After this, the students were assigned the project summarized in Figure 1.

A crane boom is formed from a curved I-beam as shown in the figure below. The boom is subjected to an in-plane load of F = ____________ lbf. Determine the principle stresses, maximum shear stresses, and von Mises stresses at points A and B using the analytical approach from your solid mechanics course. Also, calculate the von Mises stresses and displacements for the boom using SolidWorks Simulation. Dimensions d and e and angle θ are as follows:

d = ________________ in e = ________________ in θ = ________________ °



Crane boom with load (not to scale)

Figure 1. Core Elements of Project I

This project represents a modified version of a geometry taken from Reference 15. The students were provided with the following additional specifications and instructions.

1. Assume that a linear relationship exists between stresses and loads and that the boom geometry does not undergo significant changes when the load is applied.
2. The beam is made of AISI 4340 steel, annealed [yield strength: 68.2 ksi; $E = 29.7(10^3)$ ksi].
3. The boom shape is a (3 + d)-ft straight section followed by a 5-ft-radius quarter-circle (radius along the beam centerline).
4. As an initial load approximation, ignore the hook eye at the tip of the boom, and apply force F uniformly to the beam face at C.

5. As an initial restraint approximation, assume that the bolted connection represents a fixed boundary condition at the base.
6. Perform a mesh refinement study (see Chapter 2 in Reference 1) to determine the impact of mesh density on the solution (stress and displacement).
7. Refine the load application by explicitly modeling the hook eye.
8. Refine the restraint condition by modeling the bolted connection.

Each student received values for *d*, *e*, and θ that were unique to his or her version of the problem.

Two electronic files were to be submitted:

- A SolidWorks® part file containing the 3D solid model created for the crane boom including the hook eye and bolted connection.
- A pdf file containing color stress plots that clearly show the von Mises stress field generated using FEA, including the specific values at points *A* and *B*.

In addition, the following documentation was required in hardcopy:

- A set of calculations showing how the von Mises stresses at points *A* and *B* were determined using the analytical approach covered in the prerequisite solid mechanics course.
- A description of the mesh refinement study, with commentary on what it implies about the final FEA solution.
- A discussion of the quantitative impact of refinements in the modeling of loads and restraints on the results at *A* and *B* as well as at locations nearer to the load and restraint application points.
- A clear and meaningful quantitative comparison of the analytical and FEA-generated von Mises stresses at points *A* and *B.* Are they close? If not, why not?

This project supported the attainment of course learning outcomes II, III, and IV listed previously.

<u>Results and Student Performance</u>

Only one (1) student out of 17 was able to correctly calculate the von Mises stresses for this project analytically. In contrast, nine (9) out of 17 students were able to generate accurate finite-element stress results and four (4) others had results that were within 5 percent of the correct values. FEA errors were primarily due to relatively small mistakes in creating the component geometry.

The mesh refinement studies conducted by the students were generally thorough, and they were able to observe convergence of displacement and stress results as the mesh density was increased. They also noted that the effects of refinements in the ways that the load and restraints were implemented diminished rapidly as one moved away from the locations of application. The results at points *A* and *B* were not substantially affected by the refinements.

Student comparisons between the analytical and FEA-generated results at *A* and *B* were of lower quality, primarily because the errors in their analytical calculations often made meaningful comparisons impossible. Their proposed reasons for the discrepancies they observed often missed the mark.

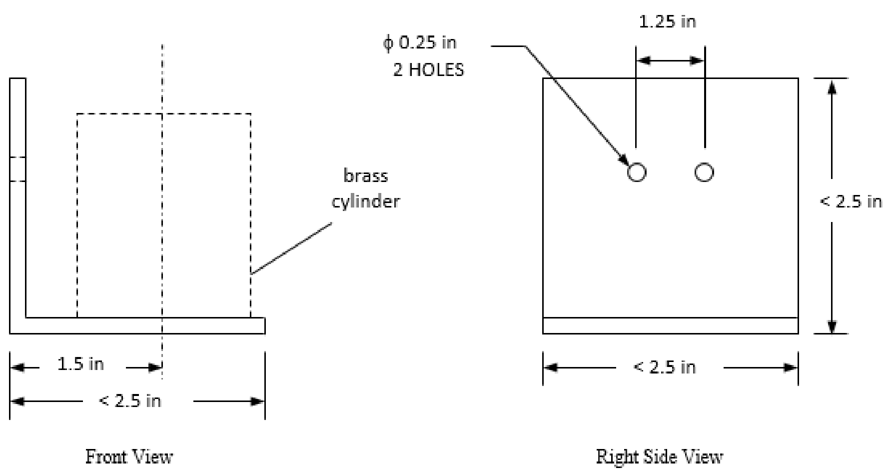
This project represented a tractable introduction to FEA that included comparisons with stress predictions calculated using analytical methods that the students learned earlier in the curriculum. By varying the approximations for the loads and restraints and the mesh density, the students were able to examine the impact of some of their modeling choices on the results. What this project did not do was give the students the opportunity to design a part or to use FEA to improve a design, nor did it provide any ties to testing of a physical prototype. This brings us to Project II.

Project II – Computer-Aided Design of Bracket

Project Description

The second project was assigned following one additional 75-minute class period dedicated to exploring a few more-advanced features of FEA as implemented in SolidWorks® Simulation. Students worked through one more example problem from Reference 1, which dealt with adaptive mesh generation. After this, the students were assigned the project summarized in Figure 2.

A bracket is required to support a small sculpture for display. The sculpture can be approximated as a 0.5-kg brass cylinder 2.00 inches tall and 1.625 inches in diameter. The bracket will be bolted to a vertical metal plate using two machine screws and associated hex nuts, and the cylinder will be oriented vertically. An example showing a crude hypothetical bracket is presented in the figure below. Your task is to design the actual bracket.



Example Bracket Design with Dimensions

Figure 2. Core Elements of Project II

The students were provided with the following additional specifications and instructions.

1. The bracket height, width, and depth are each limited to 2.5 inches. The cylinder is to be placed so that its vertical centerline is 1.5 inches from the back face of the bracket. The mounting holes for the bracket have a center-to-center horizontal distance of 1.25 inches and are 0.25 inches in diameter. (See Figure 2.)
2. The back of the bracket will mount flush against a vertical metal plate (1/4-inch thick) and will support the brass cylinder from below. The bracket will keep the cylinder oriented as shown in Figure 2.
3. When loaded with the cylinder, no point on the bracket shall be vertically displaced more than 0.05 in. from its unloaded position.
4. Based on the conditions associated with this installation and application, company guidelines call for a factor of safety (FOS) of 2 in your bracket design with respect to yielding.
5. The bracket will be manufactured using the Stratasys Object Model 30 3D printer in the Mechanical Engineering Lab. The material of manufacture will be VeroWhitePlus. Assume that the yield strength of this material is 40% of the listed tensile strength and that Poisson's ratio is 0.4.
6. The bracket should include geometric features that make it easy to properly align the cylinder. These features do <u>not</u> have to secure the cylinder against impacts, earthquakes, or other significant forces that could dislodge it.
7. The bracket will be anchored in place using two round-head steel machine screws (1/4 - 20 UNC × 1.25) and associated hex nuts. The bolt holes in the bracket shall not be threaded. The hole pattern is shown in Figure 2.
8. The bracket should be lightweight while still meeting the design requirements above.
9. Use finite-element analysis (FEA) to ensure that the requirements in items 3 and 4 are met.
10. Because this is an artistic display, you are encouraged to use your creativity to design an aesthetically pleasing bracket. The 3D printer can create intricate shapes cost-effectively. Balance this guidance with the requirement in item 8.

This project supported the attainment of all five course learning outcomes listed previously.

<u>Fabrication and Testing of Physical Prototypes</u>

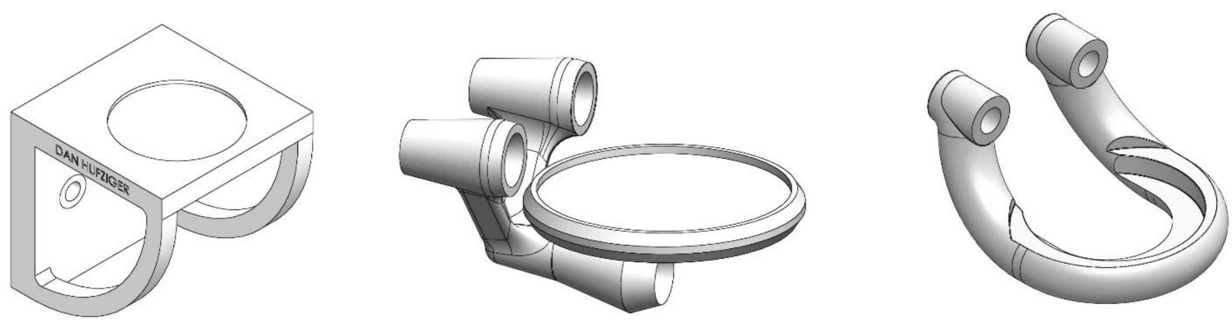
Each student created a solid computer model of his or her own design using SolidWorks® and then conducted FEA to identify areas of unacceptable stress or deflection. If the design was found to be too robust, the geometry was modified to eliminate unnecessary material. If simulations indicated that the design failed to meet one or more requirements, it was adjusted to do so. This process generally required several iterations. Once all of the student designs were finalized, the 17 brackets were fabricated using the Stratasys 3D printer. Support material was removed using a water jet apparatus, and the resultant prototypes were then tested to determine whether they met the mounting and vertical deflection requirements specified in the project statement. These tests consisted of bolting each bracket to a test stand using two round-head steel machine screws and associated hex nuts, placing a 0.5-kg brass cylinder on the bracket, and measuring the maximum vertical deflection of the bracket using a laser micrometer.

Unfortunately, geometric limitations associated with the micrometer made it impossible to fit some of the brackets into the device's measurement field. This was recognized as a possibility by the instructor, and a small, lightweight extension was temporarily attached to each bracket at the point farthest from the mounting plate. The vertical position of this extension piece was measured using the laser micrometer under unloaded and loaded conditions. The vertical deflection of the extension was calculated as the difference between the two positions, and geometric scaling was used to infer the approximate maximum vertical displacement of the bracket.

No experimentally-based evaluations of strain or associated stress were performed.

Results and Student Performance

The student bracket designs were both varied and creative as evidenced by the three examples shown in Figure 3. Design (a) represented a fairly traditional inverted L-bracket supported from below by two curved members. Design (b) was modeled loosely after the starship U.S.S. *Enterprise* from the Star Trek television series, with the "saucer" section serving as the platform for the sculpture. Design (c) was perhaps the most aesthetically interesting, utilizing a curved tube configuration with a portion of the tube cut away to hold the sculpture.



(a) (b) (c)

Figure 3. Examples of Student Bracket Designs for Project II (Not to Scale)

According to their FEA results, all seventeen (17) of the students met the minimum FOS > 2 requirement for von Mises stress with respect to yielding. Twelve (12) of the designs, however, were significantly over-engineered, with FOS values in excess of 5.0. Physical testing verified that all seventeen (17) brackets were able to be successfully mounted to the test stand and that the brass cylinder fit properly in its support guides. All brackets also met the maximum vertical displacement criterion, but as might be expected from the FOS calculations, most were stiffer than required, exhibiting maximum deflections that were significantly less than the allowed value (0.05 in.).

An example of one of the FEA simulations is given in Figure 4, which shows the mesh and the von Mises stress results for the *Enterprise* bracket. The corresponding fabricated prototype is shown in Figure 5. Quantitative FEA and experimental results for the three brackets from Figure 3 are presented in Table 1.

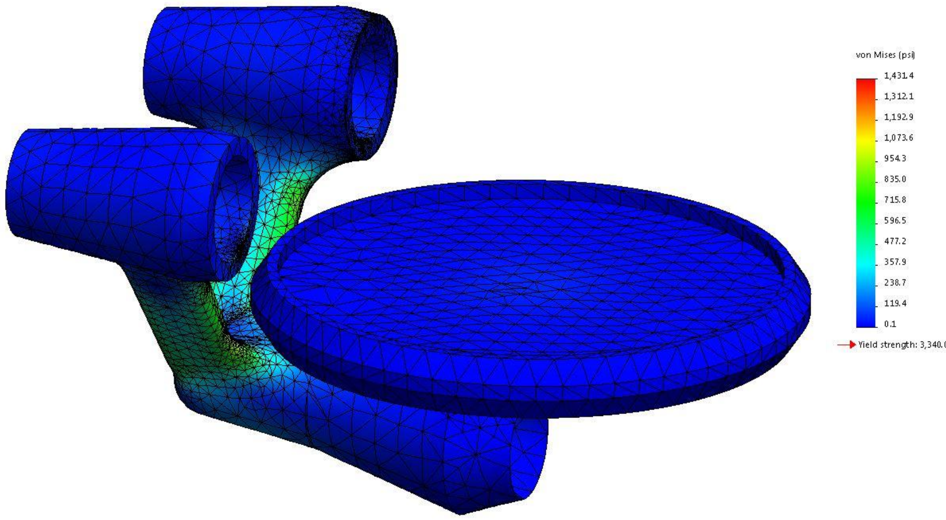


Figure 4. von Mises Stresses Calculated using FEA for the Bracket in Figure 3(b).

Figure 5. The Fabricated Prototype of the Bracket in Figure 3(b).

Table 1. Minimum Factor of Safety (FOS) and Maximum Vertical Displacement Results for the Three Brackets Shown in Figure 3

	FOS (von Mises stress) (calculated)	Maximum Vertical Displacement (in.) FEA (calculated)	Maximum Vertical Displacement (in.) Experimental
Bracket (a)	147	0.003	0.002
Bracket (b)	2.4	0.028	0.03
Bracket (c)	2.5	0.006	0.005

The students were also required to generate traditional design drawings for their brackets. In many cases, they created brackets whose geometric features were so complex that it was difficult for them to create drawings that provided all of the information needed for a machinist to accurately fabricate the part.

Student Feedback

At the end of the course, the students were asked to complete a questionnaire in which they rated how confident they were in their ability to demonstrate mastery of each course learning outcome using a five-point scale (1 – low confidence; 5 – high confidence). The mean values for each outcome are listed in Table 2 for the 2012/2013 classes combined (no 3D printing) and the 2014 class (with 3D printing). Each class included two projects that were comparable across the class years, with the only substantial difference being that, in 2014, Project II included fabrication of prototypes using 3D printing as well as subsequent performance testing. A two-sample t-test of the means was conducted for each outcome, and the resultant P-values are also shown in Table 2. It can be seen that the mean student assessment scores for all five learning outcomes increased in 2014. However, statistical analysis revealed that only two outcomes had P-values below a significance level of 0.05, which we consider to be an appropriate threshold. This indicates that the increases observed for Outcomes III and IV, which specifically dealt with FEA, were statistically significant. While these indirect evaluation results are not considered conclusive by themselves, they are promising, indicating that the students believed that their own performance relative to the course learning outcomes was enhanced by the inclusion of 3D printing.

Table 2. Student Self-Assessment Results for the Five Course Learning Outcomes*

Course Learning Outcome	Student Self-Evaluation Mean Score 2012-13	Student Self-Evaluation Mean Score 2014	Change	P-Value
I. Engineering drawings	4.6	4.8	+ 4.6%	0.288
II. Solid models	4.7	4.8	+ 1.3%	0.662
III. Applying FEA	4.0	4.9	+ 20.5%	0.00006
IV. Evaluating FEA results	4.0	4.6	+ 14.3%	0.013
V. Design and document a component	4.5	4.7	+ 4.2%	0.266

* Scale runs from 1 (low proficiency) to 5 (high proficiency)

The students were also asked in the questionnaire to identify the aspects of the course that were the most interesting and the most challenging, to comment qualitatively on how the course was conducted, and to make suggestions as to how it could be improved. With regard to Project I, the most frequent and relevant comment was that the solid mechanics hand calculations were quite challenging, and that more review would be helpful Seven (7) of the 17 students indicated that designing, analyzing, and actually fabricating a real part in the final project was the most interesting part of the course. Three (3) indicated that learning about and applying FEA was, in and of itself, their favorite part the of course. A suggestion was made to show a physical "sample" bracket up front for Project II to aid in visualizing the function and scale of the intended component. Other suggestions included providing additional instruction on how to effectively evaluate the results of FEA simulations and adding more assignments that used FEA, in general.

Limitations in 3D Printing

Before moving on to close this paper by stating our conclusions and path forward, we take a moment to mention some of the important restrictions associated with 3D printing. This powerful additive manufacturing technology has brought rapid-prototyping capabilities to users, including university engineering programs, at an affordable price point[8,9]. It also includes limitations, most notably in the materials that can be used and the size of the parts that can be produced. In the case of the Stratasys Objet Model 30, the maximum build dimensions for a single part are 11.57 in.× 7.55 in. × 5.85 in.[16]. Available rigid opaque photopolymers include VeroWhitePlus, VeroGray, VeroBlue, and VeroBlackPlus as well as a polypropylene-like material, DurusWhite[17]. Other 3D printers use a variety of other polymers, most commonly ABS and PLA. Additive manufacturing techniques involving metals (*e.g*., laser sintering of metallic powders) do exist, but they are used primarily in specialized industrial applications, and the cost of the associated equipment is prohibitively high for most educational institutions.

Because engineering applications often call for stronger materials such as metal alloys, educational exercises that rely on 3D printing cannot include performance requirements that are outside the scope of the available plastic materials. In addition, the stress-strain, yield, and creep behaviors of 3D printing polymers, in addition to their resistance to chemical attack and extremes in temperature, are, of course, often significantly different from non-polymeric materials. These limitations notwithstanding, 3D printing allows students to experience the modern design cycle and to examine the accuracy of their FEA predictions. The lessons learned during these experiences have direct relevance to successful product design activities even when other materials are involved.

Conclusions and Path Forward

The educational projects described in this paper are works in progress, but overall, we consider the combination of FEA and 3D printing in the final course project to have been a success. The students self-reported an increased level of proficiency in all five course learning outcomes, though the increases were only statistically significant for two of the outcomes (III and IV).

Student written comments indicated that the they found the design experience embodied in the second project (including conceptual design, computer model development, FEA, fabrication via 3D printing, and testing relative to performance requirements) to be interesting and enjoyable. The author's own informal observations of student performance and engagement suggest that the anticipation of actually fabricating and testing their designs seemed to create stronger student motivation than in prior years. Their efforts "felt" different. They were more inquisitive and less reluctant to try out design modifications. They were more concerned about understanding their FEA results and in getting their simulations right, likely because they realized their real product would have to physically perform. Their designs also reflected more variety and creativity. These are, of course, somewhat subjective observations, but the author has been teaching engineering for 15 years and this particular course for 6 years, so there is experience behind them.

Based on student performance and feedback, several changes are being considered for the next time the course is taught. A short refresher session on combined stress analysis and Mohr's circle was conducted in preparation for Project I, but students generally find combined stress analysis to be among the more challenging topics in their solid mechanics course, so it is perhaps not surprising that they struggled with this part of the project. This refresher will be expanded to include an in-class, hands-on group exercise that requires the students to evaluate longitudinal, bending, and shear stresses and to examine the combination of these stresses at particular locations on an object.

For Project II, the method for physically measuring maximum vertical displacement of the manufactured part will be changed to eliminate the need to attach an extension piece. We will also consider mounting a strain gage onto one or more of the parts to facilitate direct measurement of strain and subsequent calculation of the state of stress. This would directly support outcome IV.

In terms of evaluation, the weaknesses associated with only using a subjective survey to assess the impact of 3D printing on student learning will also be addressed the next time the course is taught. We will likely administer a pre- and post-quiz dealing with FEA to quantify the degree of learning. Unfortunately, we cannot go back and administer such a test for years already past, and we are now quite reluctant to remove 3D printing from the course to generate a control group. We are, however, taking a more detailed look at the archived student work to see whether performance indicators from the graded projects can be identified that could form the basis for an objective measure of the impact of 3D printing on outcomes III and IV. This exercise could not be completed in time for inclusion in this paper.

As previously mentioned, an unintended consequence of the flexibility afforded by 3D printing was that some students created very intricate designs that were then difficult to document effectively in traditional design drawings. We plan to point this out to the students up front and ask them to balance their creativity with a measure of simplicity, always giving thought to how they will communicate design intent in their drawings.

Finally, we would like to extend Project II to include a redesign of each part to address shortcomings identified by physical fabrication and testing. This would allow the students to experience full richness of the engineering design and product development process first hand.

We are anxious to see what our students will create next.

Acknowledgments

The author would like to thank Dr. Yanko Kranov of Loyola University Maryland for his assistance with the prototype testing portion of Project II.

References

1. Kurowski, P., *Engineering Analysis with SolidWorks® Simulation 2014*, SDC Publications, Mission, KS, 2014.
2. ESI-Group, *CFD-ACE+ V2013.4 User Manual*, Part I, ESI CFD, Inc., Huntsville, AL, 2014.
3. Ural, A. and Yost, J., "Integration of Finite Element Modeling and Experimental Evaluation in a Freshman Project," in *Proceedings of the Fall 2010 ASEE Mid-Atlantic Section Conference*, Philadelphia, PA, October 15-16, 2010.
4. Abdel-Mohti, A. and Khasawneh, M., "Teaching Finite Element Analysis in Undergraduate Courses, in *Proceedings of the 2012 ASEE North-Central Section Conference*, Ada, OH, March 23-24, 2012.
5. Brown, A., Crawford, R., Jensen, D., Rencis, J., Liu, J., Watson, K., Jackson, K., Hackett, R., Schimpf, P., "Assessment of Active Learning Modules: An Update of Research Findings," in *Proceedings of the 2013 ASEE Annual Conference and Exposition*, Atlanta, GA, June 23-26, 2013.
6. Shih, R., *SolidWorks® 2014 and Engineering Graphics: An Integrated Approach*, SDC Publications, Mission, KS, 2014.
7. Engineering Accreditation Commission (EAC), "2014 – 2015 Criteria for Accrediting Engineering Programs," ABET, Baltimore, MD, 2013.
8. Jaksic, N., "3D Printers Open Doors to Novel Experiential Learning Practices in Engineering Education." in *Proceedings of the 2014 ASEE Annual Conference and Exposition*, Indianapolis, IN, June 15-18, 2014.
9. Ault, H., "The Use of Rapid Prototype Models in Mechanical Design Courses," in *Proceedings of the 2009 ASEE Annual Conference and Exposition*, Austin, TX, June 15-18, 2009.
10. Stratasys, "PolyJet Technology," http://www.stratasys.com/3d-printers/technologies/polyjet-technology, accessed January 5, 2015.
11. Fischer, F., "FDM and PolyJet 3D Printing: Determining which Technology is Right for Your Application," Stratasys, Eden Prairie, MN, 2013.
12. Bishop, J. and Verleger, M., "The Flipped Classroom: A Survey of Research," in *Proceedings of the 2013 ASEE Annual Conference and Exposition*, Atlanta, GA, June 23-26, 2013.
13. Allam, Y., Whitfield, C., Phanthanousy, J., "Scaffolding Provided to Engineering Students in Cornerstone Design Project Scenarios Related to Practices of Expert Designers," in *Proceedings of the 2012 ASEE Annual Conference and Exposition*, San Antonio, TX, June 10-13, 2013.
14. McHenry, A., Depew, D., Cryenfurth, M., Dumlap, D., Keating, D., Stanford, T., Lee, P., and Deloatch, G.. "Constructivism: The Learning Theory that Supports Competency Development of Engineers for Engineering Practice and Technology Leadership through Graduate Education," in *Proceedings of the 2005 ASEE Annual Conference and Exposition*, Portland, OR, June 12-15, 2005.

15. Hibbeler, R., *Mechanics of Materials*, 7th Ed., Pearson-Prentice Hall, Upper Saddle River, NJ, 2008.

16. Stratasys, "Objet30: A Powerful Desktop 3D Printer," http://www.stratasys.com/3d-printers/design-series/objet30, accessed January 5, 2015.

17. Stratasys, "PolyJet Materials," http://www.stratasys.com/materials/polyjet, accessed January 5, 2015.