ASEE'S VIRTUAL CONFERENCE
At Home with Engineering Education
JUNE 22 - 26, 2020 #ASEEVC
Paper ID #31429

Appropriate and Ethical Finite Element Analysis in Mechanical Engineering: Learning Best Practices through Simulation

Dr. Benjamin B Wheatley, Bucknell University

Benjamin Wheatley was awarded a B.Sc. degree in Engineering from Trinity College (Hartford, CT, USA) in 2011 and a Ph.D. in Mechanical Engineering from Colorado State University (Fort Collins, CO, USA) in 2017. He is currently an Assistant Professor in the Department of Mechanical Engineering at Bucknell University (Lewisburg, PA, USA). His pedagogical areas of interest include active learning approaches, ethics, and best practices as they relate to computational modeling. He runs the Mechanics and Modeling of Orthopaedic Tissues Laboratory at Bucknell, where they use computational and experimental techniques to better understand the mechanics of musculoskeletal soft tissues and human movement.

Appropriate Finite Element Analysis in Mechanical Engineering: Teaching Best Practices through Simulation

Finite element analysis (FEA) is a powerful computational tool employed in engineering industry, research, and in the classroom. While the finite element method was developed during the mid-twentieth century for civil and aeronautical applications, it has been adopted in mechanical engineering for the analysis of solids, fluids, and heat transfer, among others. Due to the incredible efficiency of the finite element method, improvement of computer aided graphical user interfaces, and the explosion of optimization and artificial intelligence tools, FEA has continued to grow in popularity.

However, the relative ease with which one can reasonably implement a finite element package without understanding the method itself presents a somewhat precarious circumstance. In some cases, users may not fully grasp the manner in which the solution presents an approximation of a real-world phenomenon. Furthermore, the robustness of FEA presents a circumstance in which users can design or modify their model to generate a preferred solution. The result of this disconnect between simulation results and user interpretation can lead to improper and even unethical modeling techniques.

This paper will discuss tools and best practices in teaching and learning FEA and interpreting results in the context of a junior-level mechanical engineering course. This course covers both the finite element method as well as use of a commercial finite element package. In addition to providing examples and activities from class coding the finite element method in MATLAB and using the commercial package Abaqus, this paper will highlight in-class FEA activity on deriving conclusions from finite element simulations. Topics include the assumption and accuracy of boundary conditions, mesh density and mesh convergence studies, material property selection, and interpretation of model outputs as they relate to model selection and failure criteria.

The primary objectives of this work are to 1) discuss the challenges of learning the numerical method versus application of FEA with commercial tools in a single semester and 2) highlight the importance of covering both topics by providing in-class and laboratory examples of developing and employing finite element analysis. Future work will be completed to assess the effectiveness of these activities in enabling proper modeling techniques by students. The long-term goals of these efforts are to improve practical and ethical simulation for engineering students and to further integrate these themes throughout the course.

Introduction

The finite element method is a numerical approach that uses discretization and numerical simplification to generate an approximate solution. Finite element analysis (FEA) is generally observed as the use of the finite element method to solve complex problems involving modeling of physical systems. While FEA is a robust tool that has been used for a range of applications across engineering and the physical sciences, it is commonly taught in mechanical engineering due to its relevance for the physical systems that mechanical engineers design, analyze, and maintain on a daily basis. While finite element analysis is often taught in mechanical engineering in the context of solid mechanics (Lissenden et al, 2002), heat transfer, fluid, and electrical domains are additional applications of FEA. Finite element analysis can be also implemented in static, transient, or highly dynamic ways. In short, FEA is becoming limited more so by what application one can construct versus the stability and robustness of the tool itself.

Teaching finite element analysis encompasses two main efforts within the classroom: 1) the mathematical foundation of the finite element method itself, and 2) the proper application of this method to engineering problems. Each effort may independently be worthy of a semester-long course, yet this is an unreasonable expectation in an undergraduate curriculum. It can either be introduced with coding such as MATLAB (Mueller, 2003), commercial tools (Kurowski, 2014), or both. However, for undergraduate mechanical engineering students, covering the content of both the finite element method and employing finite element analysis is a challenge (Smith and Davis, 2017). Some students struggle with the mathematical foundations necessary to succeed in employing the finite element method (Echempati et al, 2010) or with previous coding experience (Smith and Davis, 2017).

This can be addressed by introducing finite element analysis either through activities in multiple classes (Howard et al 2001), emphasis in a single class (Pike, 2001), or by a standalone course that focuses on the implementation of commercial software (Le et al 2019). However, recent work suggests that teaching the fundamentals of the finite element method helps students employ FEA through commercial software (Le et al, 2019, Smith and Davis, 2017). Furthermore, we believe that the increase in applications of FEA, expanding capabilities of commercial software, and decreased cost associated with computational simulations emphasize to a greater extent the need for a foundational understanding of the inherent approximate nature of finite element analysis, which is best taught through the method itself.

The combination of increased stability and capability of commercial packages and difficulty learning the finite element method can lead to erroneous use of FEA as an engineering tool (Dues, 2006). Early use of finite element analysis required expertise in the finite element method because commercial tools were not available, thus any user must also be the developer. This is no longer the case, and has not been for quite some time. The availability of finite element solvers in computer aided design platforms such as SolidWorks (Dassault Systèmes SolidWorks Corporation) further make FEA accessible for a wide range of engineers and students (Kurowski, 2014). This increased availability of finite element solvers is certainly a benefit to the engineering and educational community, but comes at a cost of providing powerful simulation methods to those who may not understand best practices with the tool.

Course Overview

At Bucknell University, finite element analysis is introduced as a junior-level course in the mechanical engineering department. The course meets four times a week, with three in-class 52-minute sessions per week and one laboratory session of 112 minutes. Due to the computational nature of the course, both class and laboratory sessions were held in the same room that includes a desktop computer for each student equipped with commercial finite element software and MATLAB (The Mathworks, Inc.) among other relevant packages. While various commercial finite element packages are available and each presents advantages and disadvantages, the commercial finite element software of choice for this work was Abaqus (Dassault Systèmes). This was chosen due to the combination of a highly functional user interface, robustness in use for mechanical engineering content, and use in industry. A textbook "Introduction to Finite Element Analysis Using MATLAB and Abaqus" (Khennane, 2013) was assigned, primarily as a reference for Abaqus tutorials and MATLAB code.

The course learning goals, as provided in the syllabus to all students, are provided below, and map to ABET student outcomes 1, 3, 5, and 6:

1. Formulate element stiffness matrices analytically for solid mechanics and heat transfer elements.
2. Assemble global stiffness matrices, apply boundary conditions, and solve solid mechanics or heat transfer finite element analysis problems.
3. Use FE software to analyze, design, and redesign solid mechanics or heat transfer components.
4. Evaluate and interpret FE results in terms of failure modes, mesh convergence, verification and validation, and other modeling procedures.

These four course learning goals can be generally observed as covering the following four components of the finite element method and finite element analysis: 1) recognizing that elements are the foundation of FEA and proficiency in generating their equations, 2) all other major components of the (linear) finite element method, 3) proficiency in using a commercial tool to perform FEA, 4) proficiency in understanding the nature of FEA as an approximation and not an exact solution. These learning goals were specifically identified to minimize the overall number of outcomes (and emphasize the importance of each) and to cover both theory and application regarding FEA.

The general progress of the course was as follows: 1) 1D linear problems (mostly by hand), 2) 2D trusses and beams (emphasis on solids), 3) weighted residual methods (emphasis on heat transfer), 4) planar problems, 5) 3D problems, and 6) remaining specific topics such as verification and validation, sensitivity/parametric analyses, and singularities. Topics such as mesh convergence, approximate versus exact solutions, and failure criteria were dispersed throughout the course.

Specific Considerations

While a range of technical topics such as formulating element stiffness matrices, applying boundary conditions to matrix equations, and numerical integration were covered in the course, this paper will not detail these efforts. We will instead focus on teaching and learning appropriate finite element procedures by linking theory and application. Nearly all of these assignments involved students performing finite element analysis or writing finite element code, as active learning is an effective approach for FEA coursework (Watson et al, 2017). While specific course undertakings are provided in the following section, they include in-class and laboratory assignments, graded and non-graded in-class activities, and a final project. This range of content (in both evaluated and non-evaluated form) was designed to bridge this gap between theory and practice through various efforts, but more importantly provide specific approaches for students to use to employ appropriate finite element analysis in the future.

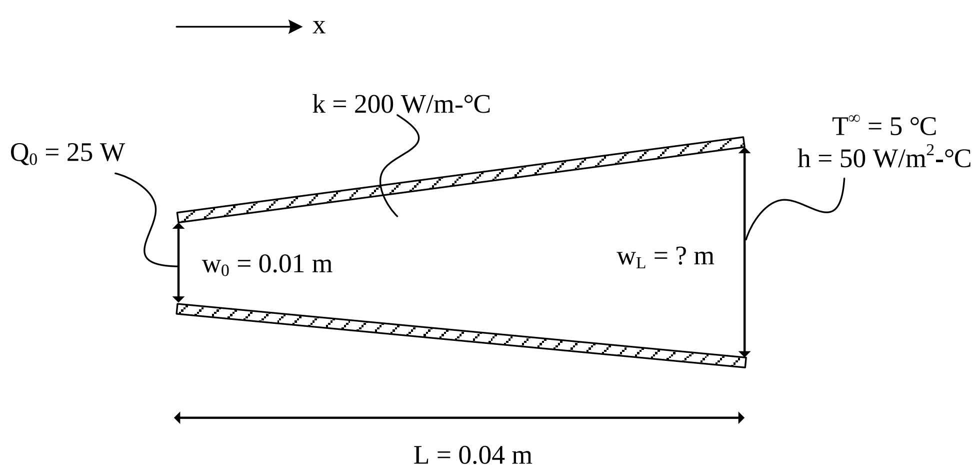
Course Undertakings

This section highlights specific in-class activities, homework assignments, and laboratory sessions that were either designed specifically with the above context in mind or are relevant to this context.

Discretization and Model Resolution

One of the most commonly emphasized components of finite element modeling is mesh convergence. Most generally, performing a mesh convergence study is ensuring that a model has sufficient geometrical resolution to answer a specified problem. Or functionally, mesh convergence is confirmation that including more elements in a model does not significantly affect the outcome of one's simulation. The topic of mesh convergence was addressed at various intervals throughout the course, with the first major emphasis in a laboratory assignment-quiz pairing in which students developed "complete" finite element code of a linear 1D heat transfer system (Figure 1).

Here students were tasked with developing MATLAB code that uses the direct stiffness method and is capable of simulating various boundary conditions, 1D geometries, material properties, and cases of internal heat generation. The code must output nodal solutions (temperatures), boundary heat flux, and solution derivatives and must ask the user to select a number of elements. Students were first tasked with developing the code as a laboratory assignment, and one week later were given an in-lab quiz where they used their code to answer the problem provided in Figure 1.



Determine the width of the heat sink at the right boundary w_L (in meters) that is necessary to keep the left boundary below 90 °C (T(0) < 90 °C). One of your design criteria is to keep the size of the heat sink to a minimum. **You**

must select the number of elements that you believe is sufficient.
After you run your MATLAB code, either publish the code and print or print the command window.

Figure 1. Example MATLAB quiz question. Students were tasked with first developing 1D linear heat transfer FEA code (using the direct stiffness method) as a laboratory assignment, then using such code in an in-class quiz to answer the above problem.

This exercise was designed to highlight two specific topics in FEA: 1) the general workflow of commercial finite element packages, as applied to a linear 1D system, and 2) the vital role of mesh density in model outputs. This quiz problem (Figure 1) required students to determine a specific number of elements, and as discussed in class this can be done by varying element number and observing nodal temperatures. While not shown here, further examples of mesh convergence studies that include both local values (such as nodal temperatures and integration point stresses) and global values (strain energy) were performed both in class and in laboratory and homework assignments.

While commercial finite element packages provide output data such as color contour plots, maximum values (such as stress or strain), and total energy, these data can be misleading. Color contour plots, for example, are a highly useful visualization tool but suggest output variables have a continuous nature within the models. To address the point of actual resolution versus continuous interpolation, a homework assignment requiring students to determine von Mises stress in a 2D first-order plane strain element was assigned (Figure 2). Here, a first-order quadrilateral element contains four integration points where the solution is "known", and all other locations in the element are determined via an interpolation scheme. This problem also required students to answer a conceptual question regarding outputs provided in commercial software such as Abaqus.

This problem incorporated many of the complexities of 2D and 3D finite element analysis that are difficult to represent in 1D problems or tri/tet elements such as Gauss quadrature and calculation of the element stiffness matrix at integration points. In conjunction with the problem provided in Figure 1, this broadly encompasses much of the finite element method by coding in MATLAB, but does not task undergraduate students with developing full 2D or 3D FEA code. These two examples highlighted how mesh density and element order can drastically affect model resolution and thus accuracy. The purpose of these activities was to enforce the notion that model data – particularly interpolated contour data –should be interpreted with skepticism if a convergence study is not performed.

Given the following geometry (dimensions in mm), thickness 2.5 mm, and a material E = 135 MPa and ν=0.35. You may assume plane strain. Note that because there are four nodes, the element is a first-order quadrilateral.

a. Determine the element stiffness matrix.
b. If v_1 = 2 and u_2 = 1.5, determine the value and location (in actual coordinates) of the maximum von Mises stress in the element.
c. How many locations within the element do you calculate stress? How does this affect the accuracy of your solution? What does

this mean regarding stress outputs in FEA software such as Abaqus?

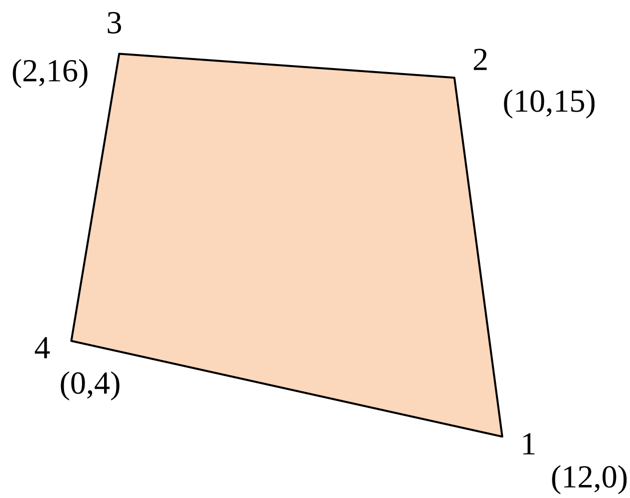


Figure 2. Example MATLAB homework problem addressing solution resolution through a 2D linear quadrilateral plane strain element.

Verification and Validation, and Sensitivity Studies

Following an emphasis that finite element models are inherently approximations, one may wonder what specific tools and/or best practices can be used to interpret model results. Confidence in finite element results can be achieved by either robust verification and validation methods, performing relevant sensitivity study, or by a combination of these approaches. These specific implementations of FEA can help reduce model errors when compared to experimentally observed phenomena and make informed conclusions about these phenomena, design solutions, and the safety of existing systems.

Verification and validation are critical steps in ensuring that finite element simulations are correctly formulated mathematically, employed effectively, and interpreted properly (The Materials Society, 2019). While verification and validation are typically discussed in conjunction, they are distinctly different procedures. Verification is defined by ASME as (American Society of Mechanical Engineers, 2006):

> "The process of determining that a computational model accurately represents the underlying mathematical model and its solution."

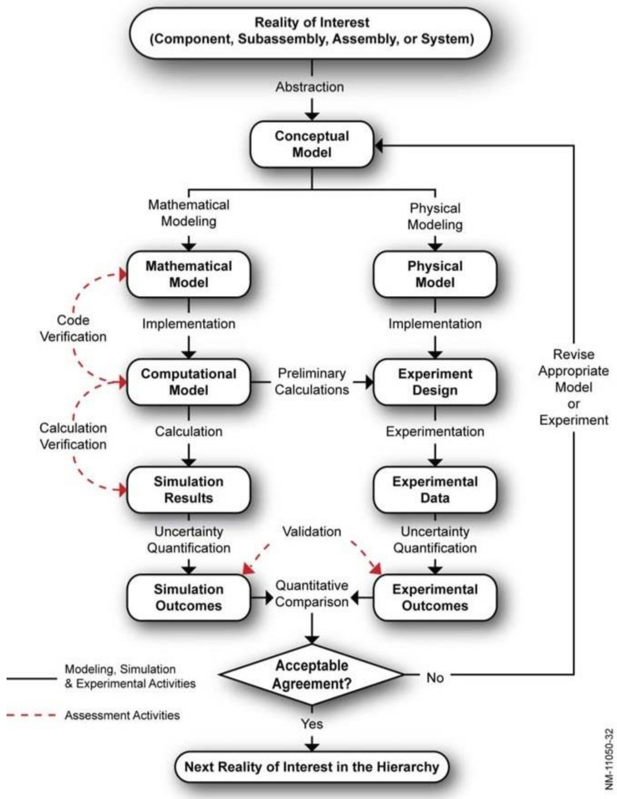
Similarly, validation is defined by ASME as (American Society of Mechanical Engineers, 2006):

> "The process of determining the degree to which a model is an accurate representation of the real world from the perspective of the intended uses of the model."

More generally, verification may be simplified to "using the mathematics correctly", while validation can be simplified to "using the correct mathematics". These two processes are necessary for any model that aims to predict the behavior of a physical system. Verification remains a difficult and often under-discussed component of undergraduate finite element courses. This may be due to the fact that commercial software such as Abaqus does not require verification unless the user were to develop a subroutine. Nonetheless, the problem outlined in Figure 2 presents an excellent opportunity for students to perform validation on the MATLAB code that was developed for this problem. By creating an identical element in Abaqus with the same boundary conditions and material properties, one can compare the integration point von Mises stress in the newly developed code against a benchmark (Abaqus) as a means of verification. Previous work has also highlighted that a lack of 1D capabilities in commercial finite element packages limits the educational effectiveness of these tools (Smith and Davis, 2017), where further verification exercises would be appropriate.

While validation is of utmost importance for properly tuned finite element models, a complete validation exercise was outside of the scope of this course. Certainly, future iterations of the could include an experimental laboratory or class session in which data are generated for model validation. Nevertheless, students were tasked with designing a parametric study for their final project in which they performed a redesign on a physical object (Figure 3 bottom). This assignment required students to identify specific modeling outputs to compare to experimental measures, and to estimate the allowable error between their experimental and computational data. The purpose of this assignment was for students to conceptualize the extensive processes that may be necessary to validate a single model.

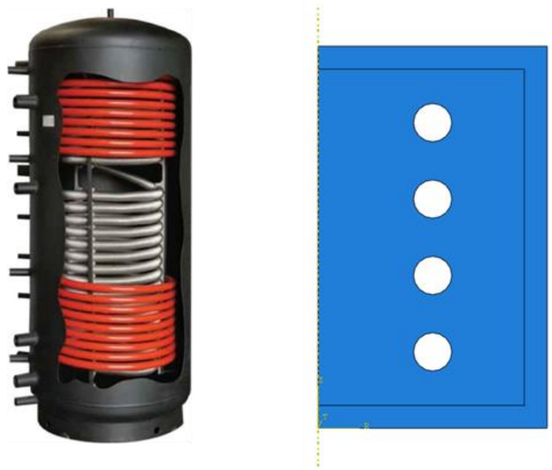
Finally, a workflow of verification and validation was introduced in class (Figure 3 top) that highlights the iterative process of verification and validation. Intuitively, one may think that these follow a structured path of verification first, then once that is completed validation can occur. While many repeated attempts at verification can lead to improper "tuning" of finite element models, using an iterative approach that includes preliminary experiments and simulations generally strengthens these models. Thus, future course iterations will consider introducing verification and validation throughout the curriculum or course and perhaps not as a stand-alone unit to further emphasize this point.



1. Design a validation study for your final project. You do not actually have to do it for the project. This must include the following (approximately 1-2 pages total):
   a. Specific experimental analyses that you would perform. Be as specific and concise as possible. A figure is worth 1,000 words here.
   b. Specific modeling outputs that you will compare. You must have a **minimum** of two outputs to compare to specific experimental measures.
   c. Allowable error between your model predictions and experiment. Again, be as specific and concise as possible.
   d. Citations as necessary.

Figure 3. Verification and validation workflow provided by ASME (top) (American Society of Mechanical Engineers, 2006) and example assignment from class that requires students to identify specific model outputs for a validation study (bottom).

A sensitivity (or parametric) study is a method to investigate the sensitivity of model outputs to model inputs. In the case of finite element analysis, these inputs are often material properties, geometry, and boundary conditions, and outputs may be stress, displacement, temperature, etc. Students were then tasked with performing a sensitivity study on an axisymmetric thermal storage tank (Figure 5) to investigate the effect of insulation thickness and boundary convection coefficient. This activity requires students to perform repeated analyses, and more importantly draw conclusions from model outputs. Specifically, students had to weigh factors such as cost and ease of manufacturing when selecting a final design. This addressed a common misconception that the difficulty in FEA is in developing and employing these models, while scoping problems and interpreting results can be a more daunting task in many cases.



You work for Environmental Engineering and Elements, an engineering firm that specializes in solving problems related to environmental health and green energy with finite element analysis. You have been tasked with the design of a thermal storage tank that uses a ceramic to store heat. The tank is comprised of clay as a storage material and wool as an insulator, making it low-cost and reducing the environmental impact relative to other systems. Use a sensitivity study to investigate **the effect of the thermal insulation thickness and boundary convection coefficient on internal temperature over a 24-hour period**.

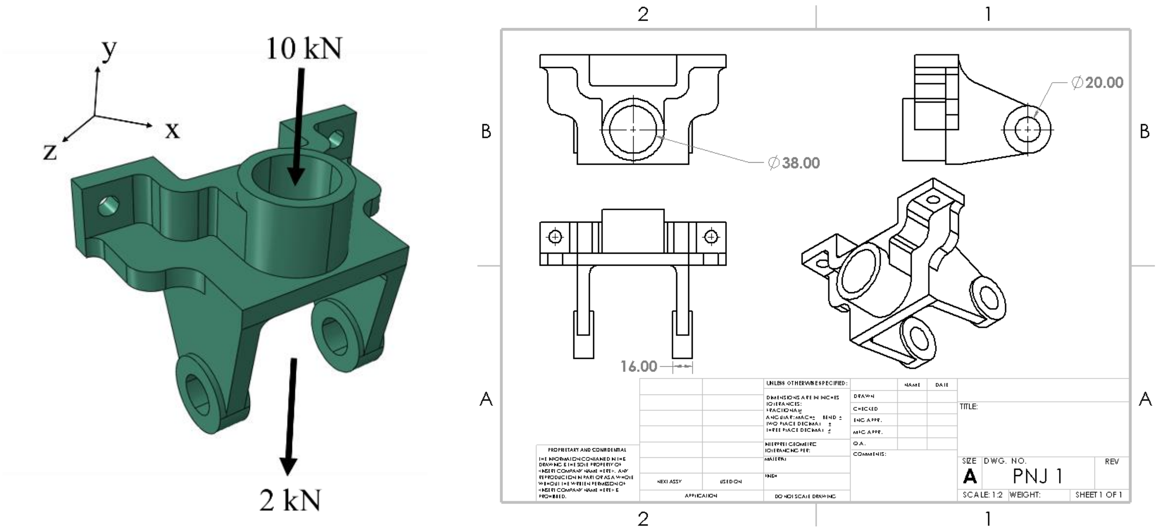
Figure 5. Parametric study in-class example, with a thermal storage tank schematic (left), and idealized axisymmetric geometry of a similar system (right).

While the topics of verification and validation and sensitivity studies required some of the simplest finite element models that were employed in the course, the difficulty and meaning of these topics was in properly scoping the work and interpreting results. Anecdotally, students were surprised at the level of difficulty in making these conclusions based on the simplicity of the models. However, scoping out and completing such assignments also provided students with specific tools to use in future FEA efforts.

Finite Element Analysis and Decision Making

The final topic highlighted in this paper is the difficulty of concrete decision making based on finite element analysis. An example in-class activity was administered that involved material selection of a bracket (Figure 6). One material (aluminum 6061) was more cost effective, while another material (aluminum 2024) was safer (Figure 6 top). Students were tasked with determining their own boundary conditions, including modeling of the bracket mount and application of loads (the values were fixed but they could by applied however students felt was appropriate).

> "The loads we determined are approximately worst-case scenario, and in all honesty are very likely on the high end. Money has been tight, and the 6061 aluminum would save us significant cost. However, safety is our first priority, so please carefully consider failure for the 6061 as an alternative to the 2024 we currently use."



Aluminum Alloy Type	Young's Modulus [GPa]	Poisson's Ratio	Yield Stress [MPa]
6061	70	0.3	276
2024	70	0.3	325

Figure 6. Example in-class activity addressing uncertainty and decision making in finite element analysis, including representative quote from a potential manager (top), bracket design and loads (middle), and different material options (bottom).

Of the 33 students in two sections, eighteen selected the 2024, while fifteen selected the 6061. For this assignment, there was no "correct" answer, as a case could be made for either material. Upon discussion of the nearly 50/50 split between student selections, emphasis was made on the following points:

1) Making specific design decisions based on finite element models is difficult and requires appropriate context
2) Models can be easily manipulated to "tell the story" that is desired by a user (6061 versus 2024 by altering boundary conditions, for example)
3) Ensuring ethical engineering practice is one way to help with the difficulty of interpreting FEA results

These points further highlight the nature of finite element analysis as an approximate solution, the need for proper procedures such as mesh convergence and verification and validation, and the benefit of techniques such as a sensitivity study. As an instructor, one can only hope that in addition to students gaining technical skills in a finite element course, that they also gain a contextual understanding of how to properly employ this technique. While ethical use of finite element model is certainly a topic of great relevance, it is outside of the scope of this work.

However, the point should be made that ethical decision making can assist with difficult problems in cases where some outcomes may seem cheaper or easier.

Assessment
Students were provided an opportunity to reflect on the course learning goals identified in the course at the end of the semester, prior to the final examination. Of the 33 students enrolled, 32 completed the survey as component of the course evaluation. Specifically, they were asked to answer the question "How well did you progress on the following learning goals:" with a Likert scale set of options that progressed as follows: "extremely well", "very well", "moderately well", "slightly well", "not well at all". Below are the learning goals (also provided in the Course Overview section).

1. Formulate element stiffness matrices analytically for solid mechanics and heat transfer elements.
2. Assemble global stiffness matrices, apply boundary conditions, and solve solid mechanics or heat transfer finite element analysis problems.
3. Use FE software to analyze, design, and redesign solid mechanics or heat transfer components.
4. Evaluate and interpret FE results in terms of failure modes, mesh convergence, verification and validation, and other modeling procedures.

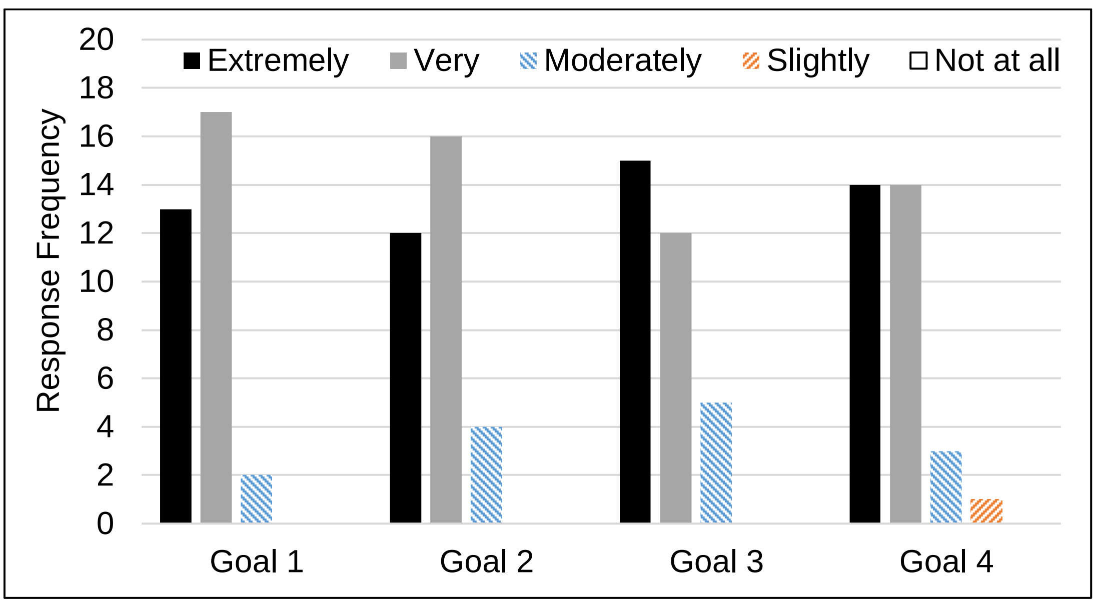


Figure 7. Student responses to Likert scale self-reporting of how well they progressed throughout the course regarding learning goals.

In general, students self-reported overall strong progression of the learning goals. More importantly, there seems to be a consistent response across all four learning goals, suggesting that the methods employed in the course to engage students in both technical theory content and practice of applying the finite element method to real world problems. While these specific numerical results do not directly evaluate student proficiency, that was beyond the scope of the work. Such an effort may benefit from a control group as well. Finally, one noteworthy student comment highlighted the need for both hand-written (or MATLAB coded), theory-based activities as well as use of commercial software, and how it was captured in the course:

> "The combination of on-paper problems and software really tied together how incredible these [software platforms] are and how to interpret the results they produce."

Conclusions

The infamous quote from George Box, the late British statistician, provides a simple take-home message broadly regarding the use of modeling: "all models are wrong, but some are useful" (Box and Draper, 1987). It is certainly applicable to the use of finite element software and the finite element method, as FEA is by its very nature an approximation. One may see the parallel to the use of statistical methods just as George Box intended with his quote, as statistical analyses can and often do lead to distinctly different conclusions based on different assumptions and model selections. The standardization of p-value statistical significance set at 0.05, for example, can lead to one interpreting a p-value of 0.049 to mean two groups are "different" while a value of 0.051 suggests groups are "not different". These values are nearly the same, however, and one may see how this improper use of statistical techniques can lead to erroneous conclusions.

Is a finite element design that predicts a maximum von Mises stress "safe" because it is below the yield stress of 250 MPa? Is it "significantly" different than another model if there is a difference of 25 MPa? The answer to these questions is "we don't know" because there is a lack of context. Providing this context and encouraging students to view problems comprehensively assists in developing the appropriate toolkit for engineers to make decisions (at times difficult decisions at that) based on models that are "wrong". Emphasizing the inherent power of finite element analysis using commercial software and along with it the responsibility to perform appropriate – and ethical – FEA is vital to any finite element course. While we have not explicitly discussed ethical use of FEA in this paper, it remains a relevant topic for future efforts.

It may be difficult for novice users to grasp the concept that FEA is approximate due to the incredible power of available commercial packages. However, it is imperative that undergraduate students exposed to FEA understand this well. We suggest that programs that do not provide a full FEA course or a FEA thread throughout multiple courses ensure that this point is emphasized. Programs that do include an FEA course or a thread should repeatedly emphasize this point with examples where appropriate. We have highlighted specific activities for undergraduate coursework to exhibit the nature of finite element analysis as an approximation and how to use this powerful tool in spite of this fact.

References

The American Society of Mechanical Engineers: *"Guide for Verification and Validation in Computational Solid Mechanics*, ASME V&V 10 ASME V&V 10, 2006.

G.E.P. Box and N.R. Draper, *Empirical Model-Building and Response Surfaces*, John E. Wiley & Sons, 1987.

J. Dues, "Avoiding Finite Element Analysis Errors," *ASEE Annual Conference and Exposition*, 2006.

R. Echempati, E. Mahajerin, and A. Sala, "Assessment of a Common Finite Element Analysis Course," *ASEE Annual Conference and Exposition*, 2010.

W.E. Howard, J.C. Musto, and V. Prantil, "Finite Element Analysis in a Mechanics Course Sequence," *ASEE Annual Conference and Exposition*, 2001.

A. Khennane, *Introduction to Finite Element Analysis Using MATLAB and Abaqus*, CRC Press, 2013.

P.M. Kurowski, *Engineering Analysis with SolidWorks Simulation 2014*. SDC Publications, 2014.

X. Le, R.L. Roberts, and A.W. Duva, "Teaching Finite Element Analysis for mechanical undergraduate students," *ASEE Annual Conference and Exposition*, 2019.

C.J. Lissenden, G.S. Wagle, and N.J. Salamon, "Applications of Finite Element Analysis for Undergraduates," *ASEE Annual Conference and Exposition*, 2002.

The Materials Society, *Verification & Validation of Computational Models Associated with the Mechanics of Materials*, The Minerals, Metals & Materials Society, 2019.

D.W. Mueller, "Introducing the Finite Element Method to Mechanical Engineering Students Using MATLAB," *ASEE Annual Conference and Exposition*, 2003.

M. Pike, "Introducing Finite Element Analysis in Statics," *ASEE Annual Conference and Exposition*, 2001.

N. Smith and J.L. Davis, "Connecting Theory and Software: Experience with an Undergraduate Finite Element Course," *ASEE Annual Conference and Exposition*, 2017.

K.A. Watson, A.O. Brown, and J. Liu, "Finite Element Analysis Active Learning Modules Embedded Throughout A Curriculum: Implementation and Assessment of Results Based on Student GPA," *ASEE Annual Conference and Exposition*, 2017.