Industry-Academia Collaboration on Development of Learning Materials for Engineering Simulation

Dr. Rajesh Bhaskaran, Cornell University

Rajesh Bhaskaran is the Swanson Director of Engineering Simulation and senior lecturer in the Sibley School of Mechanical and Aerospace Engineering at Cornell University. His work seeks to promote the democratization of simulation through the effective integration of industry-standard simulation software into engineering education. He has helped introduce Ansys-based simulations into 17 Cornell engineering courses. Nearly 320,000 people from 173 countries have enrolled in his massive open online course on simulations at edx.org; this course was a finalist for the edX prize. He has developed two online Cornell certificates in finite-element analysis and computational fluid dynamics for working professionals to learn practical simulations.

Dr. Dipankar Choudhury, Lead Chief Technologist and Ansys Fellow, plays a central role in advanced technology and exploratory R&D in several key areas at Ansys Inc. (part of Synopsys). In addition, he leads Ansys's Academic Program which includes coordinating collaborative research and education partnership programs with Academia centered around engineering simulation. Prior to taking this role, Dipankar was responsible for Ansys Inc.'s technical directions in the Fluids Business Unit. Dipankar obtained his BE in Mechanical Engineering at BITS Pilani and his Ph.D. in Computational Fluid Dynamics and Heat Transfer from the University of Minnesota.

Dipankar Choudhury, ANSYS, Inc.

Dr. Dipankar Choudhury, Ansys Fellow, leads advanced technology and exploratory R&D in several areas at Ansys Inc. In addition, he leads Ansys's Academic Program which includes coordinating collaborative research and education partnership programs with Academia centered around engineering simulation. Prior to taking this role, he helped set Ansys Inc.'s technical directions as the Chief Technologist in the Fluids Business Unit. Dipankar obtained his BE in Mechanical Engineering at BITS Pilani and his Ph.D. in Computational Fluid Dynamics and Heat Transfer from the University of Minnesota. Subsequently, he held engineering and R&D positions at Creare Inc., transitioning to the Chief Technology Officer role at Fluent Inc. which joined the Ansys family in 2016. Dr. Choudhury is an ASME, AIAA, and ASEE member, an adjunct faculty at the University of Notre Dame, and is a frequently invited speaker on campuses.

Industry-Academia Collaboration on Development of Learning Materials for Engineering Simulation

Abstract

Simulation technologies such as finite-element analysis for structural mechanics and computational fluid dynamics straddle the gap between theory, which is part of the core engineering curriculum, and industry applications. For instance, simulation tools enable students to solve relevant partial differential equations over complex domains that appear in practical applications. Industry-standard simulation tools provide a convenient platform for bringing industry knowledge and applications into the classroom in a way that connects with conventional theoretical material. As an example, Cornell university has collaborated with Ansys Inc. to develop industry-relevant learning modules that are used in the undergraduate curriculum at the university. The software tools are used to illustrate, augment and apply theoretical knowledge gained in the classroom. This paper describes some of the industry-relevant examples developed and the approach used to embed them in coursework. Some of the classroom-tested learning modules developed are available for free on the internet for other institutions to use in their teaching.

Introduction

Simulation technologies such as finite-element analysis (FEA) and computational fluid dynamics (CFD) have revolutionized engineering practice. The conventional curriculum doesn't often connect theory to practical examples. Simulation software can serve as a mechanism to overcome this lack of connection since it connects to CAD and as well as fundamental theory such as governing equations and boundary conditions. The connection to CAD means one can consider analyzing realistic geometries in the classroom. The boundary conditions would be applied to a region defined via the CAD geometry. The extension to design is straightforward since the CAD geometry can be readily modified and re-analyzed in the simulation software until design goals are met. Industry-academia collaboration is essential for bringing in industry-relevant content on simulations and design into the engineering curriculum. Industry partners can help educators define example problems that have realistic geometries and physics.

There has been much prior work on integrating industry-standard FEA and CFD software into the engineering curriculum. Szatmary (2025) [1] provides a detailed account of previous efforts to incorporate simulations into the mechanical engineering curriculum. Most of the prior work deals with introducing simulations into courses at a relatively basic level and the pedagogical approach used to do this (see, for example, [2]). In contrast, this paper focuses on incorporating industry-relevant simulation exercises into the curriculum through industry-academia collaboration. This is done after sufficient scaffolding has been built so that students are prepared to tackle realistic problems involving complex geometries and physics. Bhaskaran (2025) [3] details the pedagogical approach we use to build this scaffolding [3]. Here, we discuss how this scaffolding extends to industry problems defined in collaboration with a leading simulation software vendor, Ansys Inc. (now a part of Synopsys).

To situate our work in context, we list some of other efforts to incorporate industry-level simulations into mechanical engineering courses. Davis et al (2017) [4] discuss a semester finite element project developed by a company, in which students are asked to design and analyze an aluminum commercial truck wheel. Sracic [5] used best practices from industry to generate a checklist to guide students through the steps of analyzing parts based on real-life objects. In our case, we worked with Ansys Inc. to define practical FEA and CFD examples to teach application of simulations to industry problems. Ansys engineers were able to draw on their broad experience working with different industries to craft examples that highlight key simulation capabilities and approaches relevant to multiple companies and industries. The collaboration with Ansys also helped us bring in best practices from industry in a way that integrates with the pedagogical approach described by Bhaskaran [3] to build the relevant theoretical background.

We discuss below some industry-relevant examples from structural mechanics and fluids that have been defined in collaboration with Ansys Inc. and developed into classroom modules that also connect to the relevant theory. In the process, students are also honing practical computer-based modeling skills that are valued in the engineering profession.

Pedagogical Approach

We summarize below some of the key elements of our pedagogical approach to embed simulations into the engineering curriculum which are described in detail by Bhaskaran [3].

- Problem-based learning
- Just-in-time teaching of theory focusing on helping students develop a deep conceptual understanding
- A uniform process for implementing simulations in different subject areas which foregrounds identifying the underlying mathematical model and numerical solution strategy, inputting the mathematical model into the tool and getting it to provide an approximate numerical solution
- Teaching of the theory as well as software implementation in a holistic manner using a series of short lecture videos
- Using the lecture videos to flip the classroom and make the in-class learning experience active

In this paper, we describe how this pedagogical approach can be extended to integrate industry-relevant simulation examples in coursework as well as assessment of the impact on student learning.

Industry-Relevant Examples

The involvement of Ansys as an industry partner with broad and deep expertise in the use of simulations to design automobiles, wind turbines, airplanes, rockets, engines, etc., has enabled us to develop a rich set of examples that are practically relevant and also connect to foundational theory such as governing equations, boundary conditions, numerical solution strategies and hand calculations for verifying results. In our experience, students find these examples exciting since they readily see their practical importance and are able to visualize the results on realistic CAD geometries to build physical intuition.

Listed below are some industry-relevant examples in structural mechanics/dynamics and fluid dynamics developed that are used in the classroom at Cornell university.

- Pressure vessel static analysis
- Wind turbine blade buckling
- Vibration of a turbine disk
- High-speed flow past an airplane

These examples involve realistic geometries and physics as shown in Figures 1 and 2. The pressure vessel and airplane examples are discussed in more detail below. The pressure vessel example is available as part of a free self-paced online course entitled "Finite Element Analysis: A Hands-on Introduction" [6]. Select CFD modules are available in a similar free online course [7].

Pressure vessel static analysis: This example involves predicting the displacements, stresses and safety factor for a pressure vessel with realistic geometry and loads. The geometry is shown in Figure 1 and is provided to students as a CAD file in STEP format. The underlying governing equations are the 3D elasticity equations with a linear constitutive model. We first derive the governing equations in 1D and develop a finite-element solution to these equations by hand. This introduces major concepts such as derivation of the relevant boundary value problem as well as the numerical solution through discretization, interpolation, derivation of algebraic equations for the discrete displacements, solving the algebraic equations to determine the discrete displacements and post-processing to calculate stresses. These key theoretical concepts are then extended to 2D and 3D. The 3D boundary value problem for the pressure vessel is solved using industry-standard finite-element software since it is too complicated to be solved by hand. By grounding the 3D problem in accessible 1D concepts, students can gain a deep conceptual understanding of 3D elasticity models. In the process, they are able to connect elasticity theory to a practical example.

Lecture videos have been developed which take students through the relevant theory and application to the pressure vessel using finite-element software. The theory and application are part of a holistic problem-solving process with students accessing the software in class through the cloud. This is in contrast to a more conventional disjointed approach seen in other classes where the theory is taught in class and the software application is taught in a separate computer lab section.

Practical topics covered in the holistic problem-solving process for the pressure vessel include the following:

- How to define boundary conditions that incorporate effects of a part of the structure that is not modeled in the CAD geometry to simplify the analysis.
- How to perform hand calculations to predict the expected reaction forces for a complex geometry for comparison with the finite-element result as part of verification

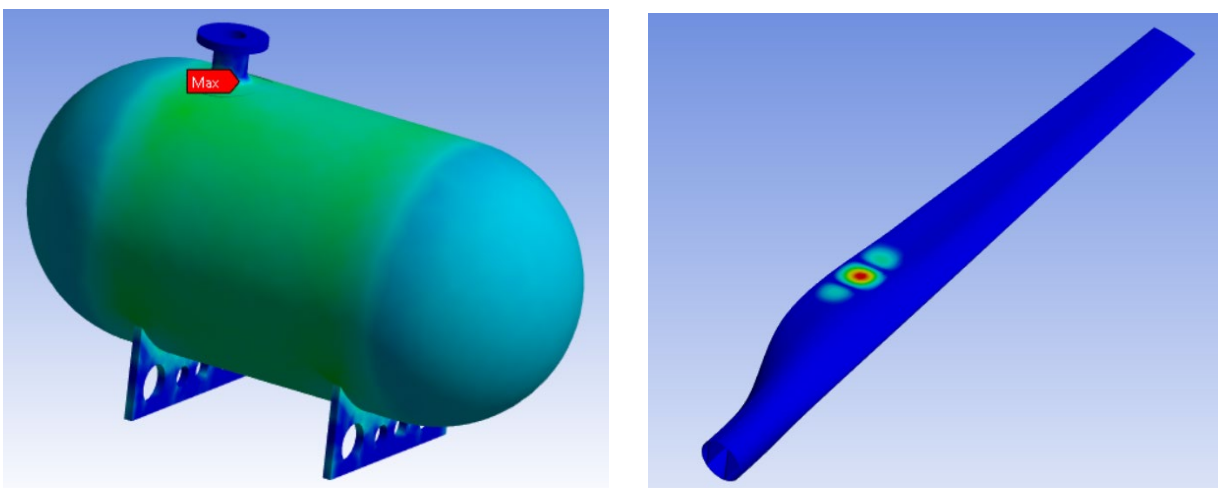
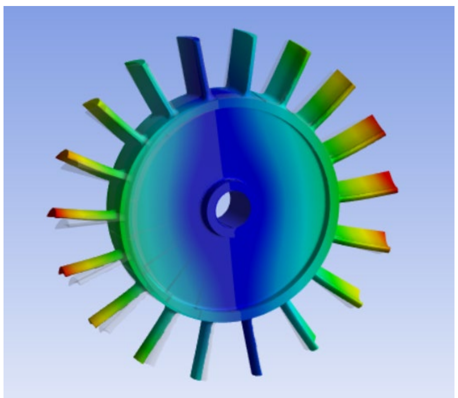


Figure 1: At left is the Von Mises stress distribution obtained from pressure vessel static analysis. At right is a mode shape obtained from the wind turbine blade buckling analysis showing the region where buckling is likely to occur.

High-speed flow past an airplane: This example involves simulating the compressible turbulent flow past a simplified airplane geometry shown in Figure 2. The airplane geometry used in this example is the DLR F4 model used as a benchmark in the 1st AIAA CFD Drag Prediction Workshop [8]. The DLR-F4 is an idealized wing-fuselage model of a passenger jetliner that has been tested in different wind tunnels for comparison with computational predictions. The flow is turbulent and transonic with the free stream Mach number being 0.7.

The simulation involves solving the governing equations which are the compressible version of the Navier-Stokes equations. The boundary conditions on the relevant flow variables like the velocity and pressure are applied to a flow domain surrounding the airplane geometry. We first derive the Navier-Stokes equations for incompressible flow, focusing on the chain of reasoning used to derive the equations starting with Newton's second law. We then show how turbulence can be incorporated through Reynolds-averaging and introduction of a turbulence model to calculate unknown Reynolds stresses that result from the averaging process. Students also see how the governing equations can be solved numerically using the finite-volume method by drawing on control volume analysis that is a part of introductory fluids courses.

Subsequently, the incompressible governing equations are extended to account for the effect of compressibility. Favre-averaging, similar to Reynolds-averaging for the incompressible case, is used to account for turbulence and the averaged equations are closed using the k-omega turbulent model. The above teaching strategy in effect gradually builds up the fundamental ideas surrounding the derivation of the governing equations so that students get deep insights into the physical significance of each term in the governing equations. Similarly, key ideas surrounding the numerical solution of the governing equations using the finite-volume method are also introduced concomitantly. The focus throughout is on concepts since the nitty-gritty mathematical details are taken care of by the software.



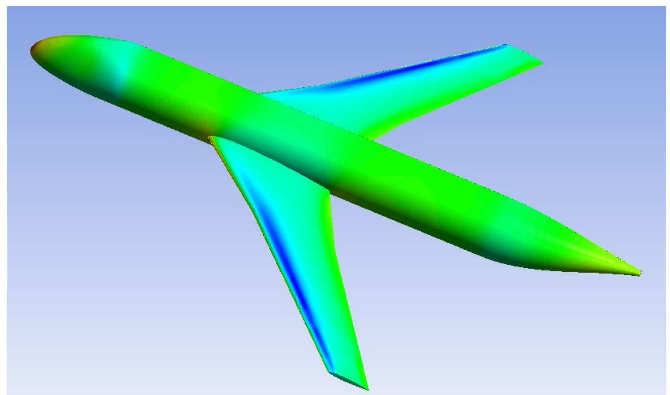


Figure 2: At left is shown a mode shape obtained from modal analysis of a turbine disk. At right is the pressure distribution obtained from solving the compressible form of the Navier-Stokes equations.

Once students have a good grasp of the governing equations and numerical solution strategy, the numerical solution is implemented for the DLR F4 model using industry-standard CFD software. The software implementation is connected closely to the underlying theory discussed earlier in order to help students make sense of the software inputs and outputs. In the process, students are connecting the fundamental theory to the solution of a practical problem, similar to the pressure vessel case in structural mechanics discussed earlier.

Assessment of Impact on Learning

The pressure vessel and airplane examples have been incorporated into senior-level elective courses using instructional videos. The content is chunked into short videos followed by "Check your understanding" quiz questions. The pressure vessel instructional module consists of 24 short videos totaling about 3.5 hours of instruction. Student performance on this module from Spring 2025 is shown in Table 1.

Assessment type	Total no. of questions	Average	Standard deviation	Findings
Multiple-choice quiz questions following videos	15	94%	9%	Students were successful in immediately applying the knowledge in the videos
Simulation submission	11	95%	5%	Students were able to implement the pressure vessel simulation

Table 1: Student performance on the pressure vessel exercise during the Spring 2025 semester. The sample size was 43 students.

In the course survey that students filled out at the end of the semester, they were asked the following question:

- Were the number and variety of examples and practical applications presented appropriate to the course content and beneficial to your learning?

Students could choose from the following five options:

1. No, almost no examples
2. A few, but insufficient number and/or mostly trivial
3. Some, but more or higher quality would have been helpful
4. Yes, including some very good ones
5. Excellent use of examples and applications that significantly increased my understanding of the material

The mean on this question from 33 student responses was 4.7/5, indicating a high level of satisfaction with the examples and practical applications.

Mechanics of Industry-Academia Collaboration

The relationship between Cornell University and Ansys Inc. in the teaching domain is long standing and maintained through a strategic partnership. The industry-relevant examples described above were first drafted by Ansys engineers based on their work with customers to highlight common types of simulation problems encountered in industry. Cornell faculty developed these into online learning modules with continual input from Ansys engineers and then implemented them in the classroom.

Complementary to the classroom-tested educational resources provided by Cornell University is a diverse set of free educator resources in materials, fluids, structures and electronics hosted on Ansys Inc.'s website [9]. The resources include case studies, lecture presentations, tutorials and teaching packages. The usage statistics for these resources for 2025 are shown in the following table. The countries with the most visitors were, in order, Germany, USA and India.

Unique visitors	113k

Pageviews	360 k
Resources	290
Downloads	57k
Countries	25

Conclusion

The objective of this work is to use industry-standard simulation software as a platform for connecting foundational theory to industry-relevant content in analysis and design. In the process, students also develop modeling skills that are important in the workplace.

Our pedagogical approach uses problem-based learning and just-in-time teaching of necessary theoretical concepts, providing practical context for foundational theoretical concepts. The framework used to connect theory to practical examples and software steps is broadly applicable and has been implemented in diverse courses on structural mechanics, fluid dynamics and heat transfer. In the process, students follow the same problem-solving process across disciplines.

We have described above examples in structural mechanics and fluid dynamics developed through collaboration between Ansys Inc. and Cornell University to incorporate industry-relevant simulation exercises into courses. Select learning modules related to these examples are available for free online for other institutions to use in their teaching. Assessment of student performance indicates that they were largely able to answer quiz questions on the underlying theory and implement the simulation in the software. Student responses on a final course survey indicated a high level of satisfaction with the practical examples. Usage statistics for related online educational resources shows international interest. Future work would involve assisting interested faculty adopt these resources in their teaching.

References

[1] A. C. Szatmary, "Simulation across the Mechanical Engineering Curriculum," in *2025 ASEE Annual Conference & Exposition* , Montreal, Quebec, Canada, 2025.

[2] A. Raich, "Self-paced, Problem-solving Approach to Teaching Finite Element Analysis," in *ASEE Annual Conference & Exposition*, 2016.

[3] R. Bhaskaran, "Connecting Theory to Applications Through Simulations Using Industry-Standard," in *2025 ASEE Annual Conference and Exposition*, Montreal, Canada, 2025.

[4] J. L. Davis, N. Smith and M. McLeod, "Collaborating with Industry Partner within an Undergraduate Finite Element Course," in *2017 ASEE Annual Conference & Exposition*, Columbus, Ohio, 2017.

[5] M. W. Sracic, "An Approach to Teaching the Finite Element Method that Uses Best," in *ASME International Mechanical Engineering Congress and Exposition*, 2016.

[6] "Finite Element Analysis: A Hands-on Introduction," eCornell, [Online]. Available: https://ecornell.cornell.edu/portal/fea/. [Accessed 29 April 2026].

[7] " Fluid Dynamics Simulations Using Ansys: A Hands-on Introduction," eCornell, [Online]. Available: https://ecornell.cornell.edu/portal/fds-ansys/. [Accessed 29 April 2026].

[8] D. Levy, T. Zickuhr, J. Vassberg, A. Shreekant, R. Wahls, S. Pirzadeh and M. Hemsch, "Summary of Data from the First AIAA CFD Drag Prediction Workshop," in *40th AIAA Aerospace Sciences Meeting*, 2002.

[9] "Ansys Education Resources," [Online]. Available: https://www.ansys.com/academic/educators/education-resources#tab=EducationResourcesTab.