Engineering Educators Bringing the World Together
2025 ASEE Annual Conference & Exposition
Palais des congrès de Montréal, Montréal, QC • June 22–25, 2025 ASEE
Paper ID #46001

BOARD # 182: Integrating Impact Experiments and Simulations in a Finite Element Method Course

Dr. Yunfeng Wang, The College of New Jersey

Professor of Mechanical Engineering at The College of New Jersey

Dr. Karen Chang Yan, The College of New Jersey

Karen C. Yan is a Professor in the Mechanical Engineering Department at the College of New Jersey. Her teaching and research interests include biomaterials with tissue engineering applications, composite materials, and materials science.

Dr. Manish Paliwal, The College of New Jersey

Manish Paliwal is a Professor of mechanical engineering at The College of New Jersey. His teaching and research interest areas include solid mechanics, biomechanics, vibration analysis, and design/analysis of mechanical systems.

Integrating Impact Experiments and Simulations in a Finite Element Method Course

Abstract

This paper investigates the integration of impact analysis into an undergraduate Finite Element Method (FEM) course. While dynamic simulations are widely used in industrial product design, impact analysis is often overlooked in traditional FEM curricula. This study outlines the methodology and content of a dedicated impact analysis module. Pre- and post-activity surveys are developed to assess the effectiveness of this teaching approach.

1. Introduction

Finite Element Method (FEM) is a core course for Mechanical Engineering. This course is offered in the first semester of senior year at the authors' department. The FEM course taught by the authors is developed using the textbooks written by Logan [1] and Lee [2]. In addition to the fundamental theories and manual calculations, ANSYS Workbench [3], a leading Computer Aided Engineering (CAE) software, is integrated into the course. This approach allows students to gain practical experience with Finite Element Analysis (FEA) software and develop a highly sought-after valuable skill by employers. The students utilize ANSYS Workbench (Ansys Inc, Canonsburg, PA) for a wide range of analyses, including structural, thermal, fluid, failure, and optimization. Since ANSYS Workbench is incorporated in the class, the seniors have utilized ANSYS Workbench to perform various engineering analyses and simulations in all of their senior design projects. Recognizing the growing importance of impact analysis in industry and the specific needs of the students involved in projects like SAE Baja and Autonomous Vehicles, the authors have developed and integrated an impact analysis module into the FEM course. Educational studies have been conducted on undergraduate FEA courses [4, 5, 6]. However, research specifically addressing the teaching and learning of impact analysis within these courses is lacking.

FEA-based impact analysis has a wide range of applications, including automobile crashes [7, 8], sports equipment protection[9, 10], ballistic impacts [11, 12], armour development [13], and electronics dropped on the ground. LS-DYNA within the ANSYS Workbench platform is a widely used and powerful tool for these types of simulations, making it the chosen software for this course.

In addition, the authors have developed an affordable and portable drop impact testing apparatus to complement the LS-DYNA simulations. This hands-on component allows students to conduct experiments and directly compare the observed deformations with the simulation results, fostering a deeper understanding of both theoretical and computational methodologies.

This paper presents the development of a teaching module integrating impact experiments and simulation into an undergraduate finite element course. The learning objectives of this module are to:

- Introduce students to the concepts of explicit dynamics.
- Provide hands-on experience using ANSYS LS-DYNA to solve impact problems.
- Develop an understanding of solution techniques through theoretical knowledge, simulation results, and experimental data.

2. Proposed Teaching Module

The proposed teaching module uses two 80-minute class periods and includes three elements: lectures, hands-on experiments with data collection, and in-class exercises. It is scheduled for the latter part of the semester, subsequent to the learning activities in fundamental FEM concepts and static structural analysis.

- **First 80-minute class**

  This class period aims to cover the relevant theoretical knowledge and train students in the use of LS-DYNA software.

  The class commences with a 20-minute lecture providing background information on impact analysis and relevant finite-element modeling techniques. Building upon prior coursework in Dynamics [14] and Mechanical Design Analysis I [15], where students acquired foundational knowledge of impact dynamics and loading, this lecture serves as a refresher and introduction to the specific concepts relevant to the impact analysis module.

Following the lecture, a 60-minute interactive session guides students through a step-by-step simulation of a smartphone drop test using LS-DYNA, allowing them to gain hands-on experience with the software.

- **Second 80-minute class**

  This session focuses on applying the knowledge and skills acquired during the first session to solve and analyze a new drop weight problem, both experimentally using a drop tester and computationally using LS-DYNA.

  The class starts with a 30-minute experimental activity. Students conduct a drop weight test and capture the impact event using the integrated video recording feature of the tester or the camera function of their smartphones. Subsequently, they solve the same drop test problem using LS-DYNA. The culmination of this activity involves students producing a discussion and analysis report, critically examining the experimental and simulation results, and comparing their findings.

3. Learning LS-DYNA through A Smartphone Drop Test Example

LS-DYNA is a nonlinear explicit finite element software widely employed in industries for analyzing complex problems involving contact, large deformation, nonlinear materials, transient response, and those requiring explicit solutions. Common applications include drop tests, impact and penetration, smashes and crashes, safety, and stability [16]. The demonstration example of using LS-DYNA is a smartphone drop test developed based on the training materials provided by ANSYS [17, 18].

The following subsections describe the steps involved in the demonstration example.

3.1 Create a Project in LS-DYNA

Launch the ANSYS Workbench Software. Drag and drop the **LS-DYNA** from the **Analysis Systems** Toolbox into the **Project Schematic** to create a standalone impact analysis project shown in Figure 1.

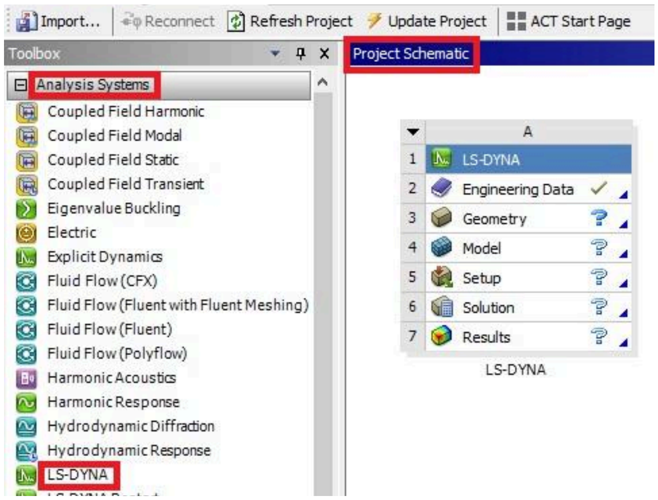

Figure 1: Launch LS-DYNA Project

3.2 LS-DYNA Workflow

A typical workflow has three major stages: define material and geometry of the objects, define boundary and loading conditions during Pre-Process, and solve and review the results during Post-Process. Figure 2 illustrates the drop test workflow.

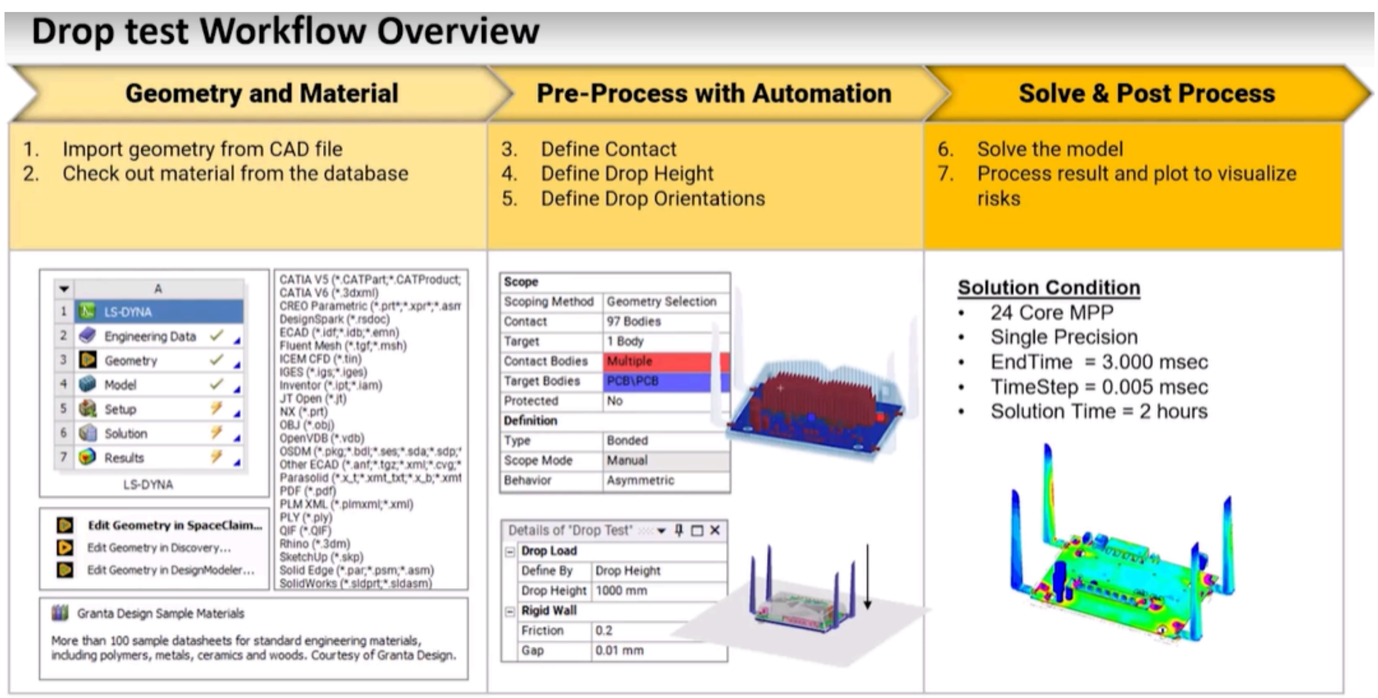

Figure 2: Drop Test Workflow [18]

Stage 1: Geometry and Materials

ANSYS provides a comprehensive engineering material database that includes polymers, metals, ceramics, and woods. Users can either assign the properties of the materials or select an existing one from the material database. The default material is structural steel. To select or define the material's properties, double-click on the **Engineering Data** icon of the LS-DYNA analysis box to launch the Engineering Data module.

The LS-DYNA analysis box's **Geometry** icon allows users to create or import CAD models of objects. Right-clicking the **Geometry** icon launches a built-in geometry modeling tool or import a CAD file. LS-DYNA provides a variety of CAD models, such as barriers, the human body, tires, and vehicles, for users to apply directly.

Stage 2: Pre-Process

Double-clicking the **Model** icon of the LS-DYNA analysis box launches the pre-process module (Figure 3). Major work for this stage is to mesh the objects, define boundary, loading, contact conditions, assign drop test parameters, and set up drop orientations. The meshes of the smartphone are shown in Figure 4. The bonded contacts are used and are defined under **Connections** of the project model tree.

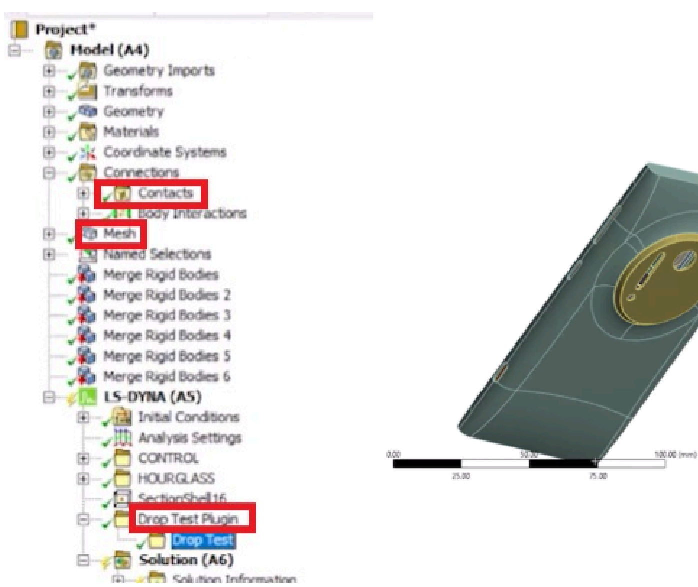

Figure 3: Pre-process Module

Figure 4: The Mesh of the Smartphone

The drop test parameters, such as drop height, initial velocity, friction coefficient, and the gap between the rigid wall and the model, are assigned using the **Drop Test** under the **Drop Test Plugin** of the project model tree, as shown in Figure 5.

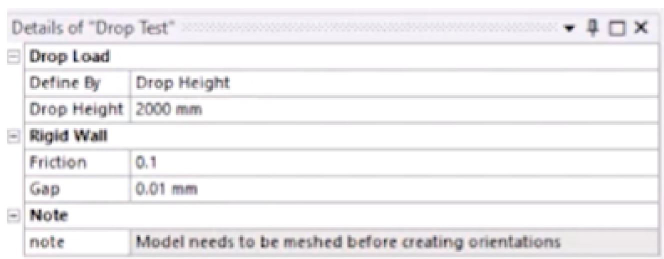

Figure 5: The Window for Assigning Drop Test Parameters

The last step of the pre-process is to set up the drop orientations. Insert the **Drop Case** to the **Drop Test Plugin.** There are four different ways to define the drop orientations: Vector with 2 Points, Global Plane, Plane with 3 points, and Axis with 2 points, as shown in Figure 6. The example analyzes two different scenarios: a corner drop and a bottom drop, as shown in Figure 7. **Vector with 2 Points** is used to set up the corner drop and **Global Plane** for the bottom drop.

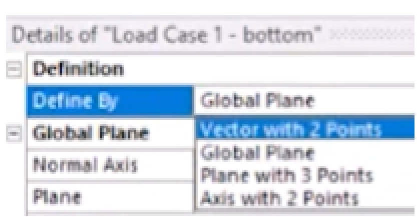

Figure 6: Windows for Defining Drop Orientation

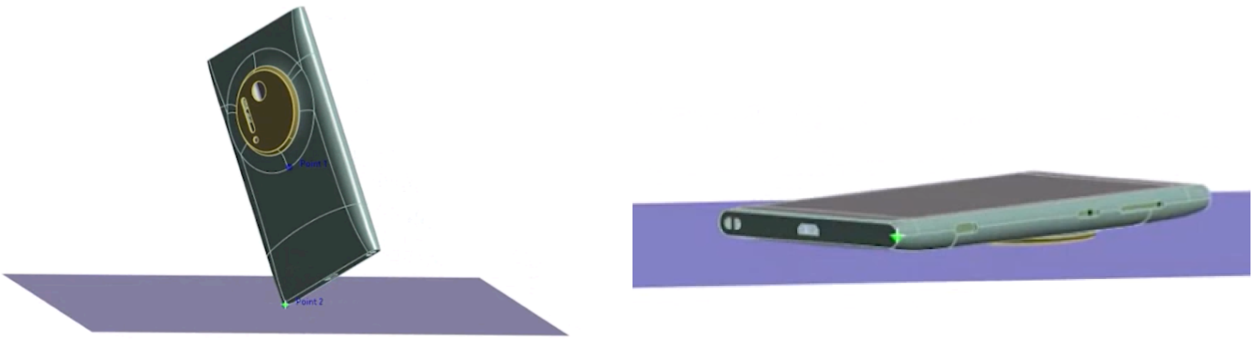

Figure 7: Corner Drop (left) and Bottom Drop (right)

Stage 3: Post-Process

Post-processing involves solving the model and subsequently visualizing and analyzing the results. Access the post-processing module by importing the **Export K Files** under the **Drop Test Plugin** (Figure 8). The results such as the deformation, stress, effective plastic strain, acceleration and velocity can be obtained. Figure 9 presents a snapshot of the von Mises stress for a corner drop scenario.

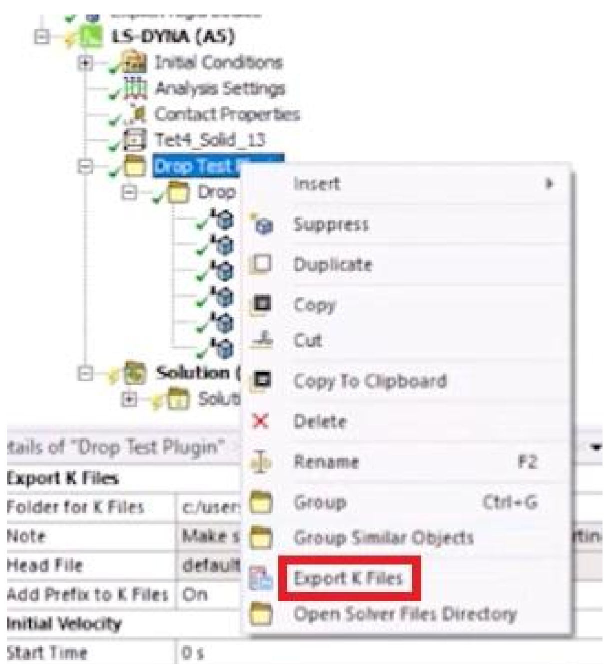

Figure 8: Post-Process

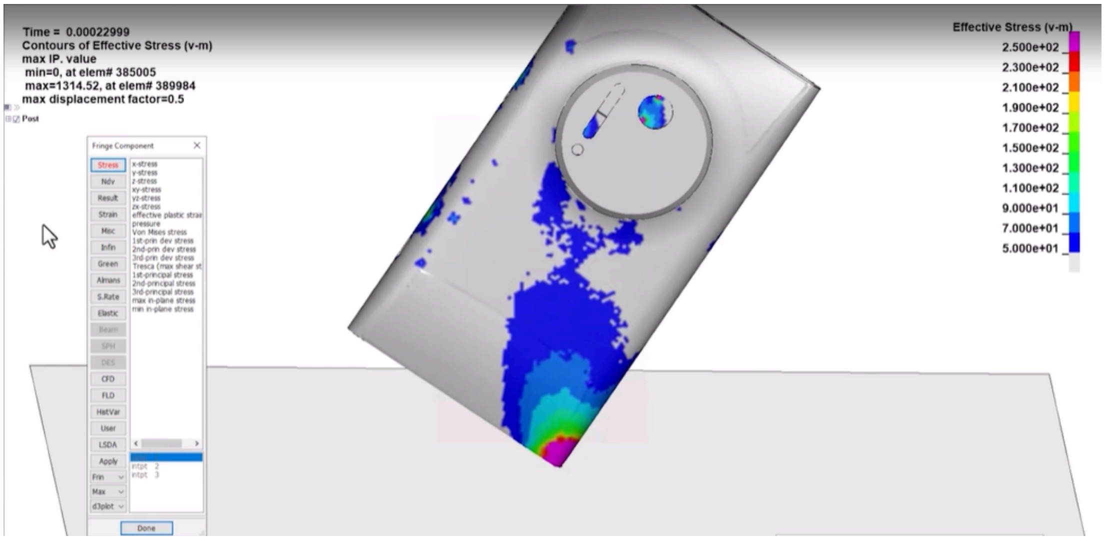

Figure 9: A Snapshot of von Mises Stress for a Corner Drop Scenario

4. In-Class Exercises: Drop Weight Impact Experiment and Simulation

A new drop weight problem that can be experimented with the impact tester is developed for students to work by themselves. The weight is a 2-lb steel striker with a semisphere tip, measuring 5.5 inches in length and 0.625 inches in diameter. The striker drops from a height of 8.5 inches onto a polycarbonate specimen. The specimen is 1/16 of an inch thick, 4.5 inches long, and 1.75 inches wide. Polycarbonate was chosen for the specimen due to its inherent flexibility and elasticity, which prevent the material from breaking upon impact.

Student Tasks:

1. **Experimental Measurement:**
    - Conduct the impact test.
    - Measure the maximum deformation of the polycarbonate specimen after impact using the cameras for capturing the deformation process.
2. **Computational Simulation (LS-DYNA):**
    - Create a model of the impact scenario in LS-DYNA, including the striker and specimen geometries, material properties, and impact conditions (velocity derived from drop height).
    - Run the simulation to predict the maximum deformation of the specimen.
3. **Validation and Comparison:**
    - Compare the simulation results with the experimental data.
    - Analyze any discrepancies between the two and discuss potential sources of error (e.g., measurement inaccuracies, material property assumptions in the simulation, limitations of the simulation software).

4.1 Impact Tester

A portable drop weight-based impact tester has been designed and built in-house [19], as shown in Figure 10. This apparatus has three major components: a clamping system, a height-adjustable guide tube, and drop strikers. The right picture of Figure 10 shows the three different strikers: semisphere, flat, and conical tip. A separable video recording feature is added to the tester to capture the impact process (Figure 11). The recorded videos can be imported into a dedicated image processing program developed using MATLAB to obtain the results on deformation, strain, moment, and shear force from the recorded videos [19]. The video recording feature is implemented with two cameras set up to capture the side and bottom surfaces of the specimen.

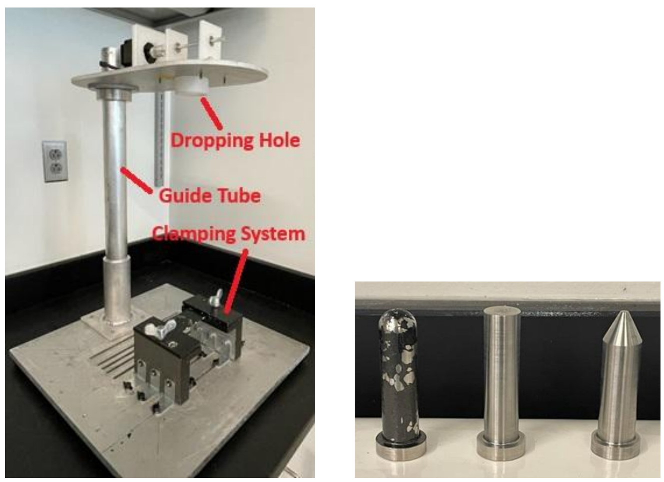

Figure 10: Impact Tester (left) and Strikers (right)

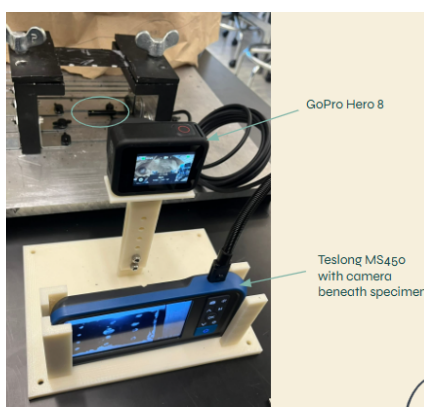

Figure 11: Video Recording Systems with Two Cameras.

4.2 Drop Weight Impact Experiment

The operation of the tester involves five steps: prepare the specimen, secure the specimen in the clamping system, adjust the height of the dropping platform, drop the weight striker while video recording the process, and import the recorded video to the dedicated imaging processing program to obtain the results.

In order to use the imaging processing program, the specimen needs to be painted in black and mark up a side surface of the specimen and its bottom surface with white dots using white paint or white-out, as illustrated in Figure 12. The specimen is loaded into the clamping system with the painted side surface facing the camera stand and the painted bottom surface facing the floor or the base plate of the device.

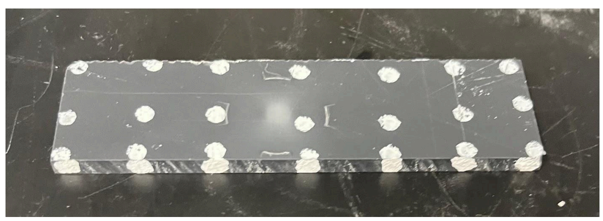

Figure 12: Painted Test Specimen

Four impact tests were conducted using the semi-spherical striker, and the results are presented here. The average maximum deformation (in the z-direction) was measured to be 0.46 inches, with a standard deviation of 0.035 inches. Figure 13 shows two snapshots from the experiments: one capturing the moment the striker just touches the specimen surface, and the other showing the specimen at its maximum deformation.

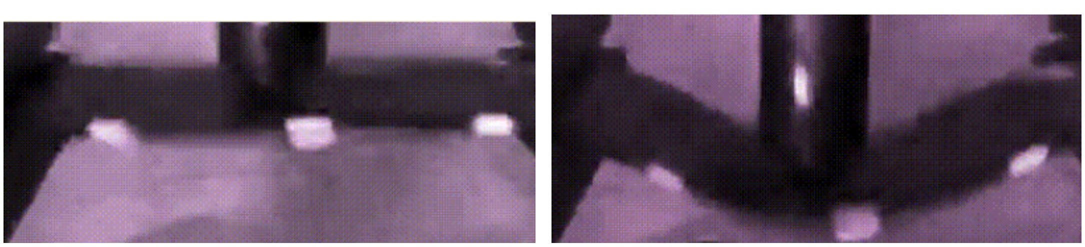

Figure 13: Two snapshots of the experiment: touch surface (left), maximum deformation (right)

4.3 Drop Weight Impact Simulation using LS-DYNA

The drop weight problem is also solved computationally with LS-DYNA. The maximum z-directional deformation is found to be 0.386 inches, visualized as the dark blue area in Figure 14. Comparing the simulation result with experimental data revealed a 19.3% difference. This relatively small discrepancy suggests that the acquired simulation results for z-directional deformation are accurate. The z-directional deformation as a function of time is also obtained and plotted in Figure 15.

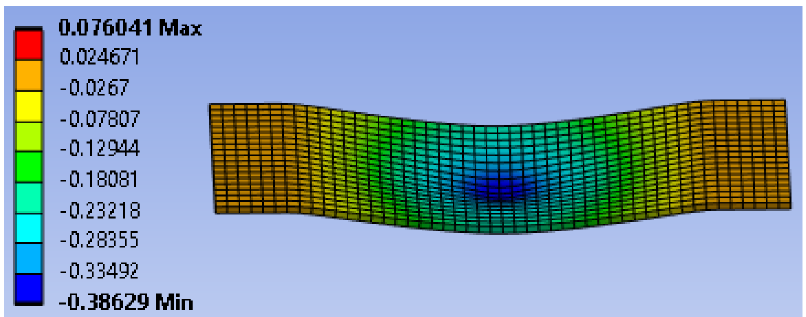

Figure 14: Maximum Z-directional Deformation (Unit: inch)

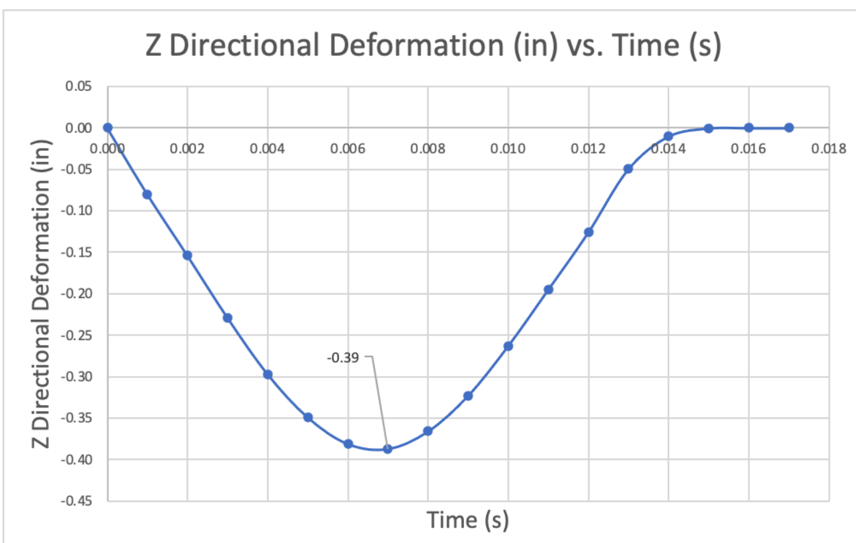

Figure 15: Z-direction Deformation vs Time from LS-DYNA

5. Assessment: Pre-post Activity Surveys

Pre- and post-activity surveys are designed to assess the effectiveness of the learning module in achieving its learning objectives. The pre-activity survey will gauge students' prior knowledge of explicit dynamics and impact analysis, as well as their perceptions of the field's importance and applications. The post-activity survey will prompt students to reflect on their learning and overall learning experience.

Both surveys will collect quantitative data and open-ended responses. For quantitative analysis, responses to questions 1 and 5 (likely related to knowledge or understanding) will be evaluated based on the percentage of correct or complete answers. Questions 4, 6, and 7 (likely related to perceptions or attitudes) will use a 5-point Likert scale, ranging from "Extremely" to "Not at all." By comparing pre- and post-survey results, the module's impact on student knowledge and skill development can be measured. Additionally, student reflections on their learning experiences will provide valuable feedback for refining and improving the teaching module in the future.

Pre-activity Survey

Question 1: What does explicit dynamics entail? Circle all the correct answers.

1. Involves a time integration method for problems when speed is important
2. Static analysis with low strain rates and non-linear problems.
3. Used to solve problems for quickly changing conditions such as free falls and high-speed impacts.
4. It deals with forces and motions that change slowly with time.

Question 2. Please provide three to five examples of real-world applications of impact analysis.

Question 3. What is an impact factor in impact analysis?

Question 4. How comfortable are you with finding deflection in a beam due to impact using theoretical mechanics (e.g., a weight dropping from a certain height)? Please cycle one answer.

Extremely	Very	Somewhat	Not very	Not at all

Question 5. Are you aware of the ANSYS LS-DYNA? Please cycle one answer.

Yes	No

Question 6. How comfortable are you with using the ANSYS LS-DYNA to solve explicit dynamics problems? Please cycle one answer.

Extremely	Very	Somewhat	Not very	Not at all

Question 7. Hands-on experiments enhance or facilitate your understanding of the topic. Please cycle one answer.

Extremely	Very	Somewhat	Not very	Not at all

Question 8. What are the limitations of a simulation study?

Post-activity Survey

Question 1: What does explicit dynamics entail? Circle all the correct answers.

1. Involves a time integration method for problems when speed is important
2. Static analysis with low strain rates and non-linear problems.
3. Used to solve problems for quickly changing conditions such as free falls and high-speed impacts.
4. It deals with forces and motions that change slowly with time.

Question 2. Please provide three to five examples of real-world applications of impact analysis.

Question 3. What is an impact factor in impact analysis?

Question 4. How comfortable are you with finding deflection in a beam due to impact using theoretical mechanics (e.g., a weight dropping from a certain height)? Please cycle one answer.

Extremely	Very	Somewhat	Not very	Not at all

Question 5. Are you aware of the ANSYS LS-Dyna? Please cycle one answer.

Yes	No

Question 6. How comfortable are you with using the ANSYS LS-Dyna to solve explicit dynamics problems? Please cycle one answer.

Extremely	Very	Somewhat	Not very	Not at all

Question 7. Hands-on experiments enhance or facilitate your understanding of the topic. Please cycle one answer.

Extremely	Very	Somewhat	Not very	Not at all

Question 8. What are the limitations of a simulation study?

Question 9. Please provide feedback on your experience learning the ANSYS LS-DYNA in conjunction with theory and experiment. Use the space below.

6. Implementation Progress and Plan

This project is progressing well. All teaching materials, including the LS-DYNA tutorial, impact tester instructions, and in-class exercises, are complete. Pre- and post-activity surveys have also been developed. All authors have completed the necessary IRB training. The IRB application is currently under review by the College committee. The teaching module is scheduled to be offered in Fall 2025, at which time the pre- and post-surveys will be administered.

References

[1] D. Logan, *A First Course in the Finite Element Method*, 6th ed., Cengage Learning, 2022.

[2] H. Lee, *Finite Element Simulations with ANSYS Workbench 2023: Theory, Applications, Case Studies*, SDC Publications, 2023.

[3] ANSYS Workbench, https://www.ansys.com/products/ansys-workbench

[4] N. Smith, J.L. Davis, "Connecting theory and software: Experience with an undergraduate finite element course," *The ASEE Annual Conference and Exhibition*, Seattle, June 2015.
[5] K.A. Watson, A.O. Brown, R.K. Hackett, A. Pham, " Finite Element Analysis Learning Modules for an Undergraduate Heat Transfer Course: Implementation and Assessment," *The ASEE Annual Conference & Exposition*, June 2012.
[6] S. Higbee, S. Miller, "Finite Element Analysis as an Iterative Design Tool for Students in an Introductory Biomechanics Course," *Journal of Biomechanical Engineering*, 143(12), 2021.
[7] A. Hickey, S. Xiao, "Finite element modeling and simulation of car crash," *International Journal of Modern Studies in Mechanical Engineering*, Vol. 3, pp. 1-5, 2017.
[8] U. Idrees, S. Ahmad, I. Shah, M. Talha, R. Shehzad, M. Amjad, S. Kolloor, "Finite element analysis of car frame frontal crash using lightweight materials," *Journal of Engineering Research*, Vol 11, March 2023.
[9] S. Imam, A. Hughes, M. Carre, H. Driscoll, K. Winwood, P. Venkatraman, T. Allen, " A finite element model for predicting impact-induced damage to a skin simulant," *Scientific Reports*, Vol. 14:12810, 2024.
[10] B. Campshure, P. Drane, J. Sherwood, " An investigation of wood baseball bat durability as a function of bat profile and slope of grain using finite element modeling and statistical analysis," *Applied Sciences*. Vol. 12, Issue 7, 2022.
[11] M. Silva, C. Cismaşiu, C. Chiorean, "Numerical simulation of ballistic impact on composite laminates," *International Journal of Impact Engineering*, Vol. 31 pp. 289–306, 2005.
[12] D. Bürger, A. De Faria, S. De Almeida, F. De Melo, M. Donadon, "Ballistic impact simulation of an armour-piercing projectile on hybrid ceramic/fiber reinforced composite armours," *International Journal of Impact Engineering*, vol 43, pp. 63–77, 2012.
[13] S. Choudhary, "Ballistic impact behaviour of newly developed armour grade steel: An experimental and numerical study," *International Journal of Impact Engineering*, 15:103557, 2020.
[14] R. Hibbeler, "*Engineering Mechanics: Dynamics*", 15th ed., Pearson, 2021.
[15] R. Juvinall, K. Marshek, "*Fundamentals of Machine Component Design*,", 7th ed., Wiley, 2019.
[16]ANSYS LS-DYNA, https://www.ansys.com/products/structures/ansys-ls-dyna

[17] Ansys Workbench LS-DYNA Course, https://www.ansys.com/training-center/course-catalog/structures/ansys-workbench-ls-dyna#tab1-4

[18] Drop Test Simulation Demo, https://www.ansys.com/resource-center/demo/drop-test-demo

[19] M. Grafer, P. Temporado, S. Schneiderman, K. C. Yan, "*Low Velocity Impact Tester (LVIT) with Imaging Capabilities*", Senior Design Project Report, Department of Mechanical Engineering, The College of New Jersey, NJ, 2024.