Engineering Educators Bringing the World Together
2025 ASEE Annual Conference & Exposition
Palais des congrès de Montréal, Montréal, QC • June 22–25, 2025
Paper ID #48967

Improved Student Comprehension Through Student-Designed Tensile Testing Laboratory

Mr. Wesley David Klehm, Oral Roberts University

Wesley is from Tulsa, Oklahoma, and is 22 years old. Since he was a kid, he was always interested in how things worked and how they were made, with many disassembled toys to prove it. This curiosity inspired Wesley to pursue a degree in engineering to further satiate this desire. In 2021, Wesley Klehm and Jordan Swan founded Esque Box while students at Oral Roberts University to teach a new generation of kids what they wished they knew at the same age.

Emmelia Lichty, Oral Roberts University

Emmelia Lichty is an engineering major with a mechanical concentration in the School of Engineering at Oral Roberts University. She will earn a Bachelor of Science in Engineering Degree and a minor in Chemistry in May 2025.

Moriah Love Metellus, Oral Roberts University

Hello my name is Moriah Metellus, and I am an engineering major with a mechanical concentration in the School of Engineering at Oral Roberts University. I will earn a Bachelor of Science in Engineering in December of 2025.

Mr. Quadre John Antoine Moore, Oral Roberts University

My name is Quadre Moore, I am a senior Mechanical Engineering and Mathematics double major student. I have been at ORU for the past 5 years and have never participated in ASEE conference or published a paper, but this year I will submit two. I will be working for Engineering company in Tulsa called Tek Fins full time in May, I will also be getting married in May to my beautiful bride Hannah

Jonathan Ophus, Oral Roberts University

Jonathan Ophus has worked in several facets of the fabrication industry over the last 30 years. He developed several different pieces of equipment and processes for higher yields in the precious metals industry. In the construction industry, he designed equipment and new processes for delivery of 50% stronger concrete to be used in structural applications. He enjoys teaching and working with young people. He currently works with students and faculty at Oral Roberts University's School of Engineering as a machine shop technician.

Dr. John E Matsson, Oral Roberts University

John Matsson is a Professor of Mechanical Engineering at Oral Roberts University in Tulsa, OK. He earned M.S. and Ph.D. degrees from the Royal Institute of Technology in Stockholm, Sweden in 1988 and 1994 respectively.

Improved Student Comprehension Through Student-Designed Tensile Testing

Introduction

Abstract

Labs greatly enhance students' understanding by bringing to life often abstract concepts and equations. However, executing effective laboratory coursework for engineering courses is difficult because of a lack of preparedness students generally have for lab coursework. Another issue students experience is developing a thorough understanding of what the lab is teaching and retaining that knowledge. Nowhere is this more apparent than in the discipline of manufacturing processes and the study of material properties in material science.

Material properties lie at the heart of many engineering disciplines, as materials are what engineers incorporate in their disciplines. The study of material properties can be greatly accelerated in engineering courses by integrating a hands-on approach to the creation and testing of laboratory experiments through inquiry-based experiential learning. This approach will be achieved by allowing students to create, design, test, and write their own laboratory manual as well as experimental procedures, which will result in a holistic, hands-on approach to an integrated laboratory course. Students took advantage of knowledge and experience gained from previous pre-planned labs in the manufacturing processes course. Through this approach, students will be required to understand on a deeper level what they are achieving in laboratory coursework, and in doing so, will foster learning about the laboratory procedure and theory.

Throughout an entire semester, students designed a lab to test four different ¾" diameter test specimen composed of stainless and low-carbon steel, aluminum, and brass rods to determine the corresponding material properties of Young's modulus, yield strength, and ultimate strength. The ESL Test II Intelligent Materials Test System with software was measuring continuously the applied force using a load cell and resulting elongation of the test specimen was measured with strain gages. Students identified the material properties from stress-strain curves using data acquired from a Satec hydraulic tensile testing machine and strain gages. Ansys Explicit Dynamics simulations were included in order to offer a comparison using computational tools to compare with experimental testing. Ansys simulations offered the students a deeper understanding of the computational tools used in industry. Finally, students formally developed a laboratory assessment procedure to document and measure student learning outcomes. Implementing this strategy in engineering laboratory coursework will allow students to better understand and call upon this skill set in their future engineering careers.

Introduction

The engineering tensile stress-strain curve is obtained by the static loading of a test specimen and exhibits various regions of material behavior and properties such as Young's modulus, ultimate tensile strength, and the fracture stress, according to Hibbeler [1] and Vidosic [2]. Understanding and predicting material failure is crucial in engineering applications, for example in the systems of fuel cell and nuclear safety components, see Pham and Trinh [3].

By asking students to perform, write, and simulate this laboratory, it requires students to have a more in-depth approach than simply following the instructions within a manual. In doing so, students will have to demonstrate a higher level of comprehension. Project based learning has been proven by Fiteriani [4] to increase metacognitive thinking in students and can also lead to better comprehension of studied material than a control group. Metacognition is defined as internal cognitive monitoring by Flavell [5] or simply thinking about internal thoughts. This self-reflective cognition can be greater achieved in a project setting where students are given guidance and direction, but not necessarily the exact steps to completion. The metacognitive approach also applies to equipment, testing procedures, and technology used by the students in the laboratory project.

The purpose of this project is to aid students in the understanding of material properties using tensile tests. Students will learn to develop stress-strain curves for tested materials experimentally, as well as familiarize themselves with the associated tensile testing machinery and software. The tensile experiment will be simulated using Ansys Explicit Dynamics and the results for stress-strain curves and determined Young's Modulus compared to values found in literature and experimentally [6]. Students also will receive exposure to engineering standards, as the dimensions of the test rods are determined by the ASTM E8 Round Tension Test Specimen with Gage length 5 times the diameter standards [7]. Table 1 below lists the yield strength and ultimate strength for various materials that will be tested in this experiment.

Table 1 Tensile Testing Material Properties

Material	Specifications Met	Young's Modulus (ksi)	Yield Strength (ksi)	Tensile Strength (ksi)	70% Yield Load Limit (lbs)
AISI 304L Stainless Steel	ASTM A276	28,500	30.5	81.8	4100
Low-Carbon Steel AISI 1018	ASTM A108	29,000	53.7	63.8	7100
6061-T6 Alum	ASTM B221	10,000	40	45	5500
360 Brass Rod	ASTM B16	14,500	15	19	2000

Theory

Stress is defined as:

$$\sigma = \frac{F}{A_o} \quad (1)$$

where A_o is the original cross-sectional area of the test specimen and F is the applied force.

Strain is defined as:

$$\epsilon = \frac{\Delta L}{L} \quad (2)$$

where ΔL is the change in length and L is the original length of the test section of the test rod.

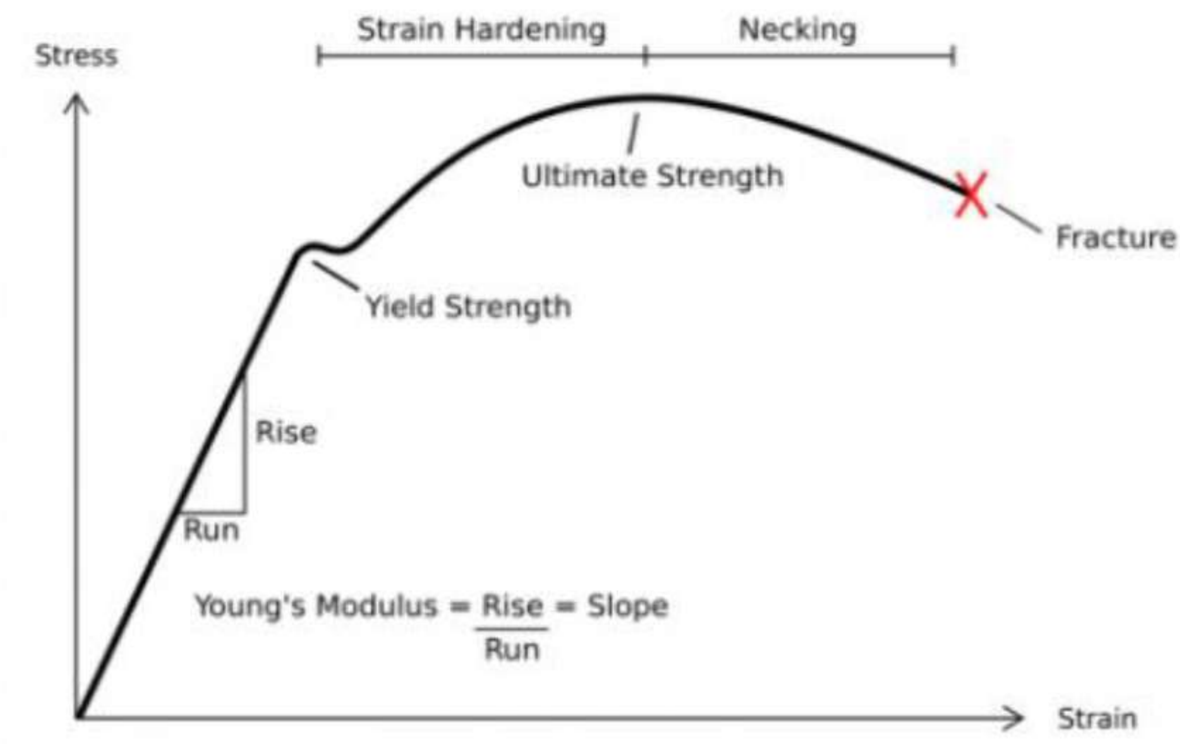


Figure. 1. Typical Stress-Strain Diagram

Above is a typical stress strain diagram. There are four regions on the stress-strain curve. The first is the elastic region, which is the linear region from the origin to the yield strength point on the curve [2]. This region defines the Young's Modulus for the test material. The outlined testing procedure above will not go past the yield limit for the specimens but still remains valid as a method to determine the Young's Modulus for materials.

The second region on the stress-strain curve is the yielding region. In this region the specimen will elongate without increasing the load placed on the specimen thus remaining flat [2]. (Note: this region is not shown on Figure 1 but occurs in between the yield strength point and the strain hardening region.) Material in this state exhibits plastic behavior as the material deforms plastically [2].

The third region occurs when the load begins to increase and is referred to as strain hardening. In this region the specimen can take more load but only up to the point of ultimate strength where the necking region begins [2]. The last region on the stress strain curve is the necking region. The specimen will undergo an elongation and a subsequent, localized reduction in cross sectional area will occur. This is referred to as necking [2]. At some point in this region, the specimen will fracture indicating the end of the test. The fractured test specimen will display a cup and cone pattern in the region where necking and subsequent fracture occurred. See Figure 2 for reference.

Figure 2. Necking Pattern Characteristics

Experimental Methods/Materials/Project Approach

Experimental Set-Up, Implementation and Procedure

Students were required to submit drawings to the engineering laboratory technician for the standard rods to be manufactured. Students were also required to add tolerances to the drawing as well as ensure the tolerances on the manufactured pieces were met. Using the engineering lab's Tensile Testing Machine (TTM), accompanying ESL View software, and a surface mounted strain gage with accompanying Model P3 Software from Micro-Measurements, students measured the load and strain that the test rod was subjected to.

Students loaded the test rod into the TTM and mounted the extensometer, see Figures 3 - 5. Students then used the ESL View Software to measure stress and used the Model P3 software to measure and record the strain imparted to the specimen until 70% of the yield stress had been reached. Students used Microsoft Excel to analyze their results.

Figure 3. Tensile Testing Machine

Figure 4. Loaded Test Rod

Figure 5. Connected Test Rod

Results

Experimental Results

Students encountered difficulty in using an extensometer to record strain for their experiment so the experiment shifted to using a surface mounted strain gage. Students mounted the strain gage and were able to test three different specimens: aluminum, low-carbon steel, and brass. The lab results are shown below in figures 6, 7, and 8 respectively.

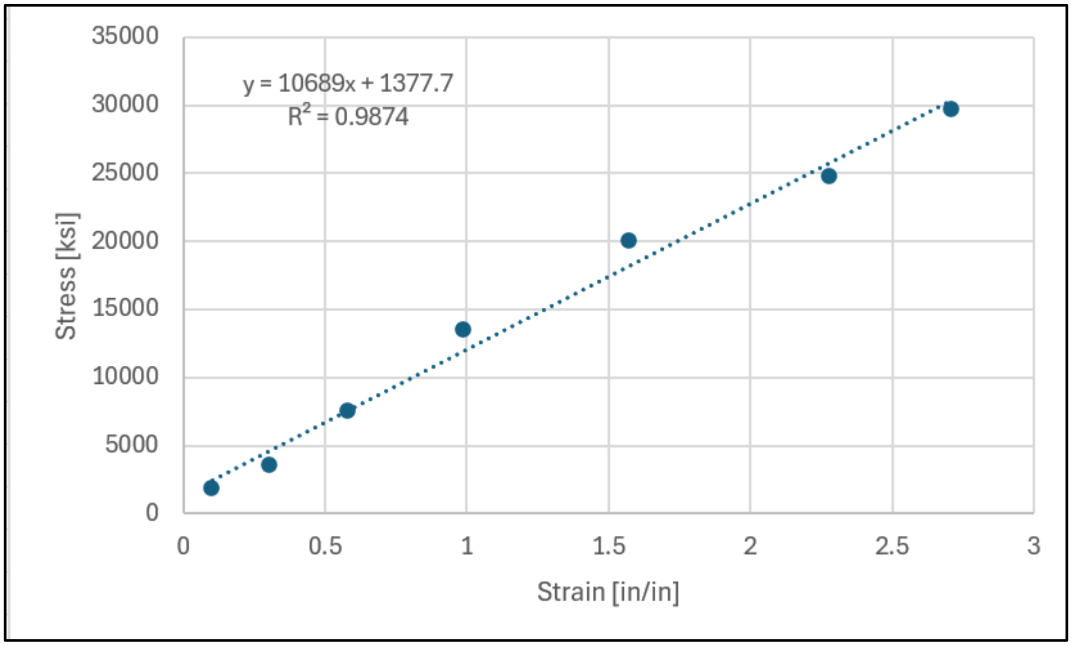


Figure 6. Aluminum Stress-Strain Curve

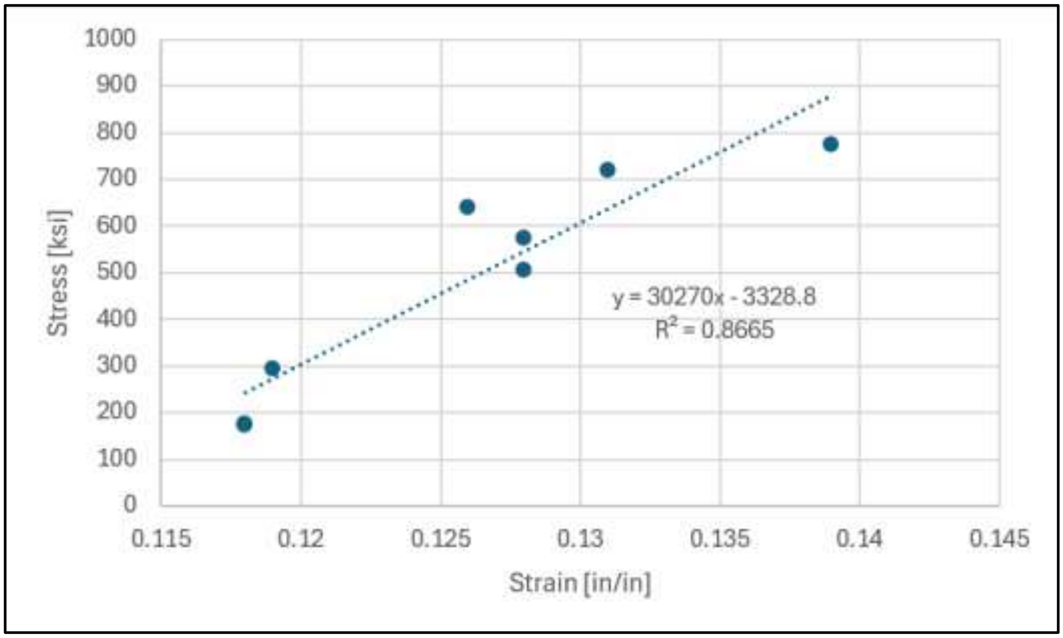


Figure 7. Low-Carbon Steel Stress-Strain Curve

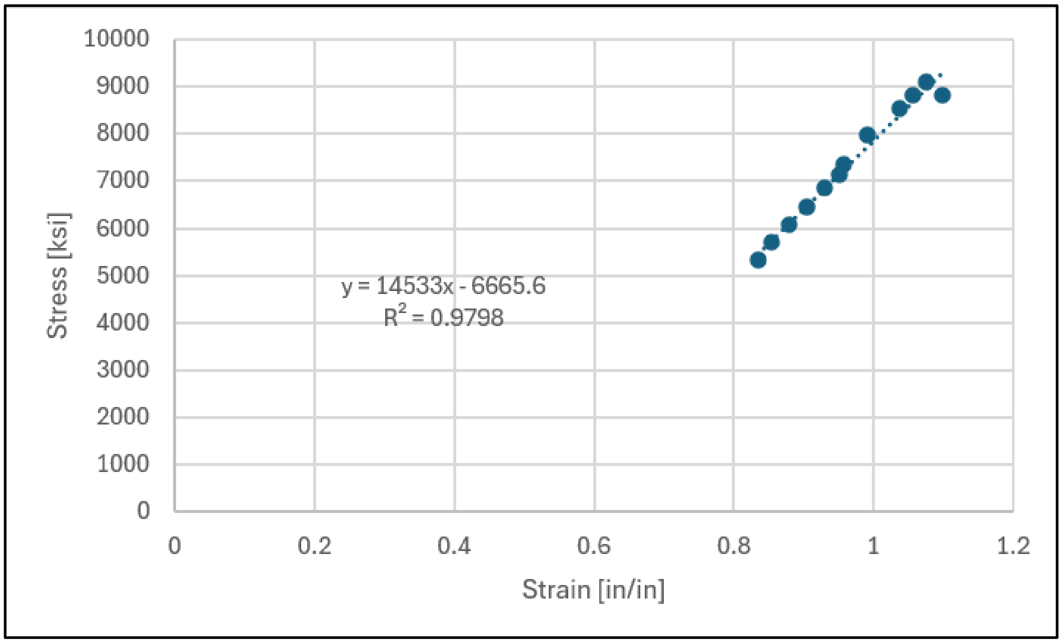


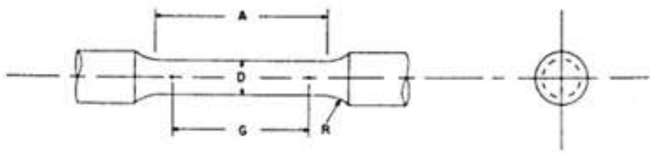
Figure 8. Brass Stress-Strain Curve

Students created a laboratory manual, appended in Appendix 1, which explained the process of performing the laboratory. This experiment project will be assigned in upcoming semesters in which this class is taken. The lab manual provided by the students will be used to help guide future students in writing and creating their own lab in order to stimulate hands-on learning about the subject of material properties.

Ansys Simulation Results

Ansys simulations were conducted alongside physical experiments to verify the results of the physical experimentation as well as to give students experience using finite element analysis software to supplement laboratory work. Students used Ansys Explicit Dynamics which is a time integration method used to perform quick dynamic simulations according to Ansys [6]. Explicit Dynamics is helpful for simulating engineering challenges with short timescales and complex body changing interactions, such as a breaking test specimen as described by Ansys [6].

To perform Ansys tests, standard test rods were modeled using Dassault Systems' SOLIDWORKS according to ASTM E8 standards for tension testing rods with gage length 5 times diameter size (standard section shown in Figure 9 and rod drawing shown in Figure 10).



Dimensions, mm [in.]

For Test Specimens with Gauge Length Five times the Diameter [E8M]

	Standard Specimen	Small-Size Specimens Proportional to Standard			
	Specimen 1	Specimen 2	Specimen 3	Specimen 4	Specimen 5
G—Gauge length	62.5 ± 0.1 [2.500 ± 0.005]	45.0 ± 0.1 [1.750 ± 0.005]	30.0 ± 0.1 [1.250 ± 0.005]	20.0 ± 0.1 [0.800 ± 0.005]	12.5 ± 0.1 [0.565 ± 0.005]
D—Diameter (Note 1)	12.5 ± 0.2 [0.500 ± 0.010]	9.0 ± 0.1 [0.350 ± 0.007]	6.0 ± 0.1 [0.250 ± 0.005]	4.0 ± 0.1 [0.160 ± 0.003]	2.5 ± 0.1 [0.113 ± 0.002]
R—Radius of fillet, min	10 [0.375]	8 [0.25]	6 [0.188]	4 [0.156]	2 [0.094]
A—Length of reduced section, min (Note 2)	75 [3.0]	54 [2.0]	36 [1.4]	24 [1.0]	20 [0.75]

Figure 9. ASTM E8 Tensile Test Rod Standards [7].

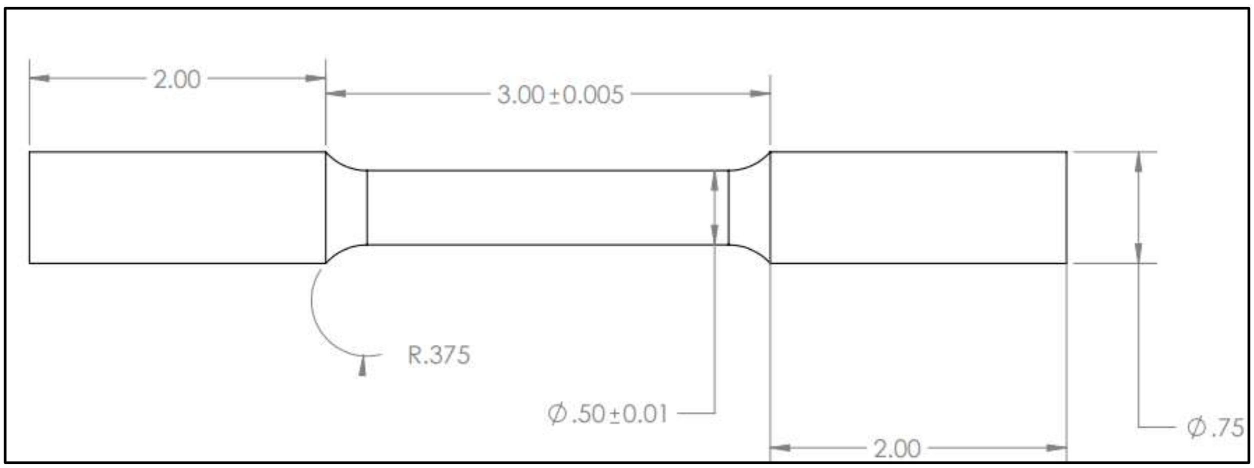


Figure 10. Modeled Tensile Test Rod Drawing.

To perform the Explicit Dynamics simulations, the modeled SOLIDWORKS file was uploaded into Ansys by the students. All four materials, stainless steel, brass, aluminum, and low-carbon steel, included in the Ansys material Engineering Data library were assigned to the uploaded model and a simulated tensile test was performed for each. Post-processing was also performed in Explicit Dynamics to find stress over time and the strain over time for the tests. The students then exported this data to Excel and created the stress strain curve to determine the Young's Modulus. These curves can be seen in Figures 11 - 14.

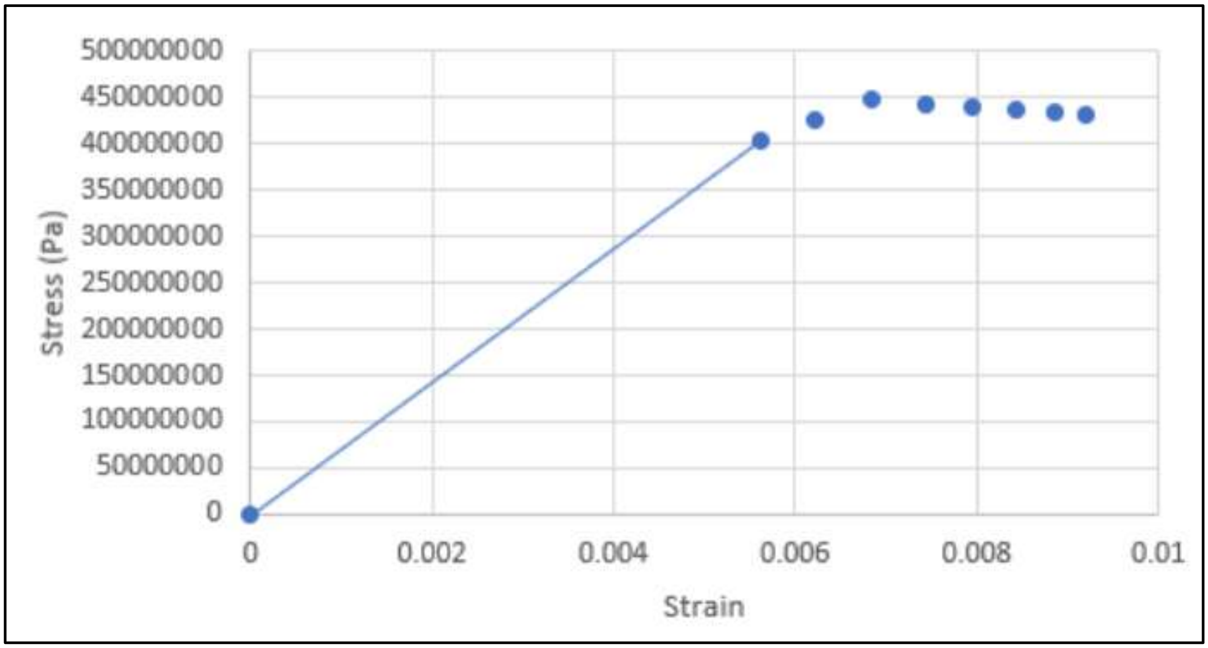


Figure 11. Ansys Aluminum 6061 Stress-Strain Curve

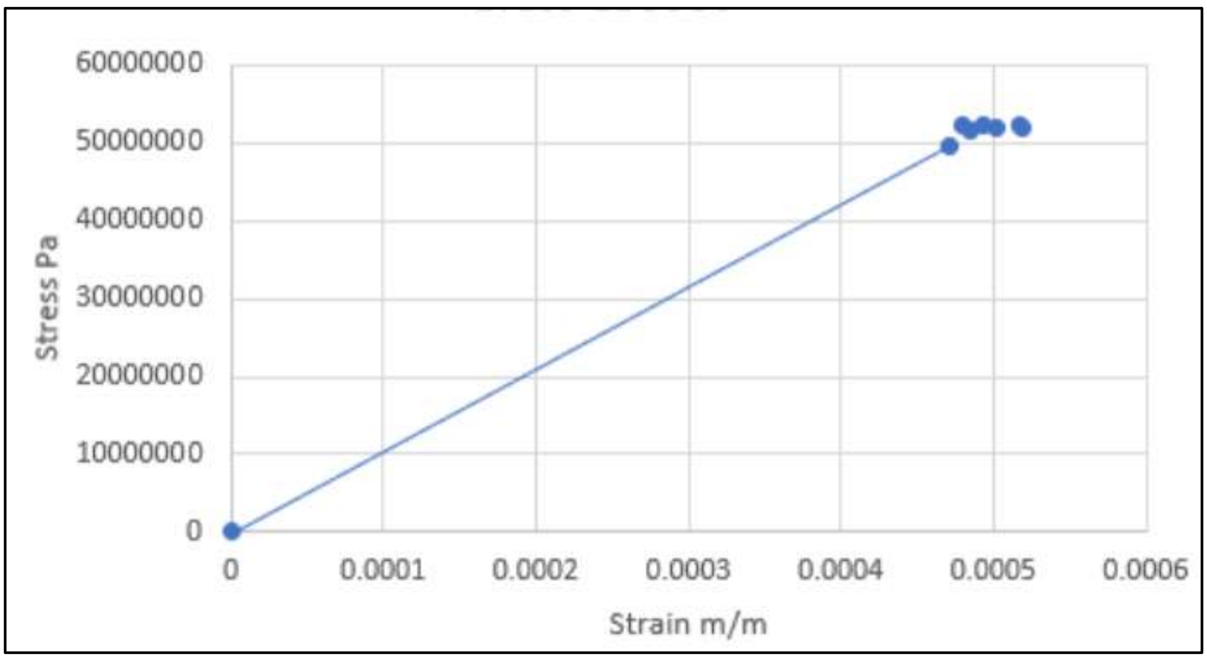


Figure 12. Ansys C36000 Stress-Strain Curve

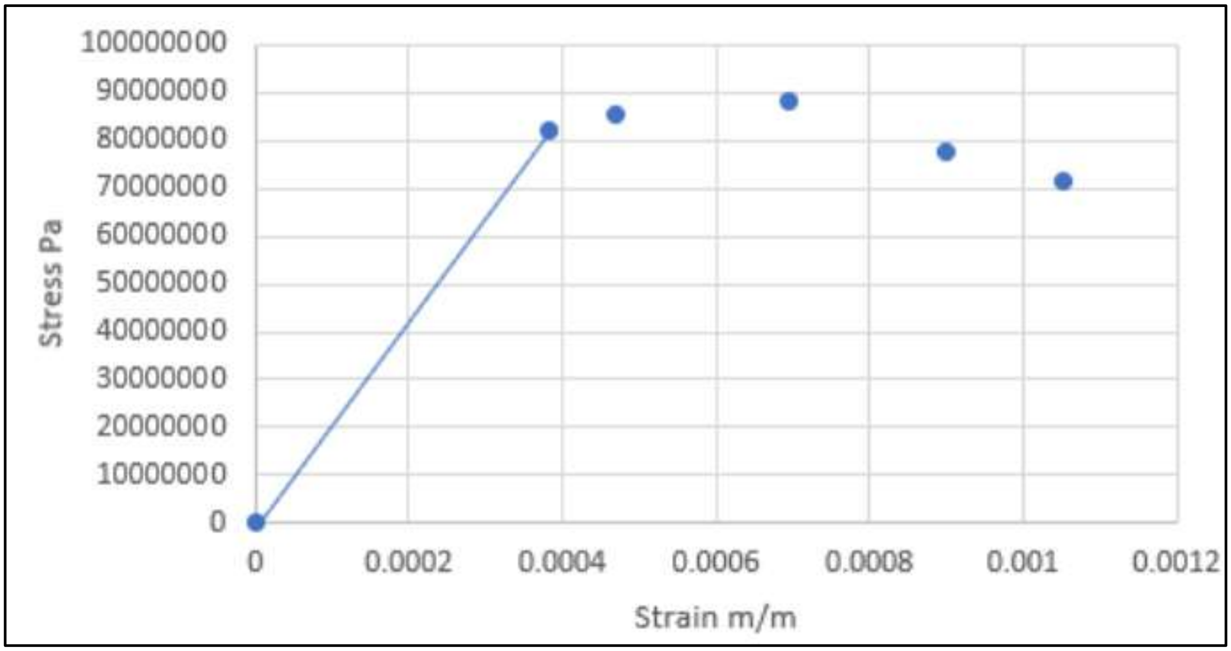


Figure 13. Ansys SS 304 Stress-Strain Curve

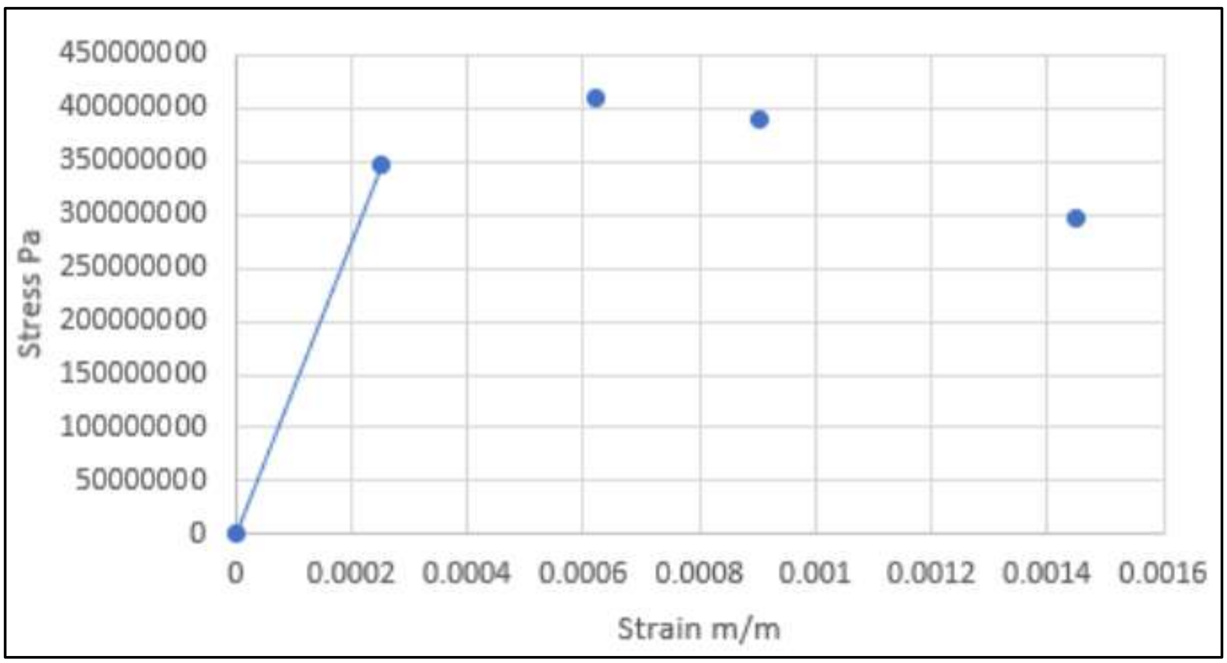


Figure 14. Ansys Low-Carbon Steel Stress-Strain Curve

Table 2 shows the material properties determined from performed Ansys Simulations as well as the percent difference from data provided in Table 1. Table 2 also shows the experimentally determined Young's Moduli and the calculated percent difference.

Table 2. Material Properties

Material	Ansys Young's Mod. (ksi)	Exp. Young's Mod. (ksi)	Given Young's Mod. (ksi)	Ansys Percent Diff. (%)	Exp. Percent Diff. (%)
Aluminum	10,421	10,689	10,000	4.21	6.89
Brass	15,464	14,533	14,500	6.65	0.23
Stainless Steel	31,331	Not Tested	28,500	9.93	Not Tested
Low-Carbon Steel	29,670	30,270	29,000	2.31	4.38

Discussion

Student Assessment

In the Tensile Testing Laboratory project, students can assess their performance by actively comparing experimental data with theoretical predictions and industry standards. For example, students calculated Young's Moduli from experimentally developed stress-strain curves and compared these results with standard values provided in the given tables. Using tools like Ansys simulations alongside experimental setups allows them to verify their computational and hands-on outcomes, ensuring accuracy and consistency. Reflecting on their results through post-lab reports and discussions with peers further enables students to identify gaps in understanding or procedure. By analyzing deviations and justifying them with logical reasoning, students gage their grasp of both the experimental process and the underlying material properties as well as the corresponding theory.

Figure 15 shows the student outcomes required for students to achieve to pass the class, as well as meet ABET standards for the course. In the course of this project, students were required through the experimentation, lab manual, Ansys manual, and Ansys simulations to demonstrate proficiency in some area of study for the course and the ABET requirements. Students were required in this project to understand machining, material properties, soldering, and were required to apply engineering, scientific, and mathematical principles. In this manner students will be assessed on their application of engineering, scientific, and mathematical principles as well as their understanding of material properties, machining, and soldering.

OUTCOMES	Significant	Moderate	Minimal
Intellectual Pursuit			
ABET: An ability to identify, formulate, and solve complex engineering problems by applying principles of engineering, science, and mathematics.		**X**	
Discuss and explain casting processes.	**X**		
Discuss and explain hot and cold working of metals.		**X**	
Discuss and explain joining: soldering, brazing and welding bonding.	**X**		
Discuss and explain machining.	**X**		
Discuss and explain plastic processing: molding and extruding.	**X**		

Figure 15. Student Outcome Table for Manufacturing Processes.

In their first attempt to ascertain stress-strain curves, students ran into many issues such as faulty extensometers, test specimens being stretched beyond their elastic region, difficulty in installing strain gages, etc. Students demonstrated principles of science, mathematics, and engineering in solving these problems in order to achieve results which are outlined in Table II. One specific example is the development of the 70% yield strength cutoff in order to prevent the stretching of the specimen beyond the elastic region and breaking the strain gage, demonstrating scientific knowledge. Engineering problem solving principles were demonstrated in the pivot to using surface mounted strain gages in the place of an extensometer. Students demonstrated application of mathematical principles in their development of the Young's Modulus, applying their scientific understanding of the linearity of the elastic region of the stress-strain curve. Knowledge of machining was adequately demonstrated in the manufacturing of new unstretched test rods. Students were required to electrically connect strain gages and developed an innovative and easily reusable connection method and in doing so, demonstrated significant knowledge in soldering and soldering techniques.

Student Involvement

Students engaged deeply in the Tensile Testing Laboratory by directly managing tasks like specimen preparation, experimental setup, and data analysis. Collaboration was fostered as teams worked together to configure the tensile testing machine and strain gages, ensuring proper alignment and calibration. They also actively incorporated computational tools, such as Ansys, to simulate stress-strain behavior, which complements hands-on experiments. By presenting their findings in structured lab reports and participating in peer evaluations, students reinforced critical thinking and problem-solving skills. This multifaceted involvement ensured they gain a holistic understanding of tensile testing principles while connecting theoretical knowledge with practical applications.

Student Recommendations

There are many different challenges that students can face and will encounter during their time working with this project. However, there are a few ways to help the learning process and boost success with the project as students work. It would be helpful for the instructor to clearly lay out what objectives are required to be met and assign a grading scale to them, so students can track their progress and know the amount each outcome weighs on their grade. It also would be helpful to ensure students have updated equipment for anything the students might have to use, to help with the repeatability and accuracy of this project. Lastly, it would be beneficial for the instructor to have a working understanding of the procedure or have completed the lab themselves, to ensure they can help guide and assist students with the questions or problems they will inevitably face.

Benefits of Reproducing this Project in a Classroom Setting

In the course of this project, students are not given a comprehensive manual that outlines fully how to complete their project. Students are required to research, collaborate, and critically think about how to approach the lab and how to ascertain results. Through the exclusion of an all-inclusive lab manual, the students must become resourceful in applying their coursework to the project and reaching out to professors, lab technicians, and fellow students in order to successfully complete the laboratory course. Students also are required to demonstrate a more holistic understanding of how the laboratory is completed, due to the lab manual deliverable required by the project. Materials science is at the heart of the engineering discipline so introducing this method of laboratory pedagogy with varying complexity of tensile tests can only serve to benefit future engineers. Tensile testing in particular is a good candidate for this style of laboratory pedagogy as undergraduate engineers must have a grasp on this idea to meet ABET requirements for engineering programs.

Many students appear to achieve proficiency in a laboratory setting when filling out a lab data sheet and an ensuing lab report, but with the rise of artificial intelligence, student work has, by anecdotal evidence, been outsourced to large language models. The goal of this laboratory approach, and its future works, is to require students to demonstrate proficiency in their laboratory coursework and develop an assignment that intrinsically requires students to demonstrate understanding, while preventing students from generating an entire lab report that meets assessment standards but does not achieve student understanding. In requiring students to produce both a laboratory and Ansys manual, the aim of this continuing work is to require students to demonstrate holistic understanding of their coursework and project.

Inclusiveness and Adaptability

The student-designed tensile testing laboratory presented in this paper can be made more inclusive and adaptable by including flexibility, support structures, and differentiated learning opportunities.

- Students can choose from a range of materials and testing parameters.

- The lab can be structured in different phases including proposal, design, execution, and analysis.
- Students can select or rotate through roles that match their strengths.
- Teams with advanced students can mentor peers who may need more guidance.
- Availability of comprehensive instructional materials in various formats.
- Students can present their findings through different formats.
- Assessment based on effort and understanding.

Challenges and Scalability Across Institutions

Some of the challenges of a student designed tensile testing laboratory include

- Students enter the course and project with differing levels of technical knowledge.
- Tensile testing machines can be expensive, making access difficult.
- Supervising and evaluating student-designed labs require significant instructor time for project reviews and feedback.

Scalability across institutions can be achieved through

- Designing a modular lab framework, institutions with different resources and expertise can adopt the tensile testing lab.
- Developing a shared repository of lab designs, datasets, and experimental protocols can help institutions save time and ensure quality control.
- Scaling the lab enables collaborative projects where institutions can compare results from student-led designs.

Changes to Specific Guidelines for Project

When updating student specific guidelines for the project, the instructor will ensure clarity, alignment with the learning objectives and adaptability with student preparedness level.

- Allowing a broader range of materials including 3D printed polymers.
- Allow flexibility in test specimen geometry and strain rate.
- Emphasize equipment calibration in more detail.
- Define clear team roles for student team members.
- Include a student sign-in sheet when students work on project.
- Increasing the open hours of the machine shop so that students can work more on project.
- Emphasize safety protocols and machine training.
- Provide clear project grading rubrics for project progress reports and final project report.
- Hand out check lists for design, testing and reporting.

Conclusion

In the laboratory classroom approach outlined in this paper, students will have robust and foundational laboratory projects that will give undergraduate students experience with writing their own lab manuals and understanding the lab tools and theory. This laboratory approach will expose engineering students to different types of manufacturing processes needed to complete the lab. Students will learn how to complete assignments with limited direction and develop the best way for students to accomplish the very assignment given to them. The tensile testing laboratory coupled with the laboratory approach hopes to yield a significant improvement in student understanding. This more holistic approach to the execution of laboratory coursework encourages students to truly understand their course material, laboratory processes, and simulation methodology, the last of which gives students an added skill for which they can use to make themselves more attractive to potential employers.

Future Work

Each ensuing class of mechanical engineering students in manufacturing processes will design the same experiment with the same basic guidelines provided. There are various other manufacturing process concepts this idea could be translated to such as experiments involving blow molding or V-bending. Even beyond manufacturing processes courses, this versatile procedure can be applied to nearly any theory such as determining the gravitational constant or spring constant in physics, or a falling ball viscometer in fluid mechanics. The instructor will be able to compare the findings and quality of lab manuals among their students from year to year. The results of this education research experiment will be evident through comparing the results of past students that simply conducted the experiment according to a written lab manual to current students creating the experiment and writing the lab manual.

References

[1] R. C. Hibbeler, "The Stress-Strain Diagram," in *Mechanics of Materials*, 8th ed, vol. 1, Upper Saddle River, New Jersey: Pearson Prentice Hall, 2011, pp. 84–85.

[2] J. P. Vidosic, "Mechanics of Materials," in *"Mark's Standard Handbook for Mechanical Engineers"*, 8th ed, New York, New York: McGraw-Hill, 1978, pp. 5-16-5–18

[3] Pham, Q and Trinh, H.T., Simulation and Experimental Test in Tensile Behavior of Austenitic Stainless Steels, Advances in Materials, 8(3): 108-111, 2019.

[4] I. Fiteriani, R. Diani, A. Hamidah, and C. Anwar, "Project-based learning through STEM approach: Is it effective to improve students' creative problem-solving ability and metacognitive skills in physics learning? ," Journal of Physics: Conference Series, vol. 1796, no. 1, p. 012058, Jan. 2021.

[5] J. H. Flavell, "Metacognition and cognitive monitoring: A new area of cognitive–developmental inquiry.," American Psychologist, vol. 34, no. 10, pp. 906–911, Oct. 1979.

[6] "Ansys Explicit Dynamics," Ansys.com, https://www.ansys.com/content/dam/product/structures/ansys-explicit-dynamics-brochure-140.pdf (accessed Dec. 1, 2024).

[7] *Standard Test methods for Tension Testing of Metallic Materials*, E8/E8M-13a, Jul. 2013.

Appendix 1: Lab Manual for Tensile Testing Project

<u>Manufacturing Processes</u>
<u>Tensile Testing Lab</u>

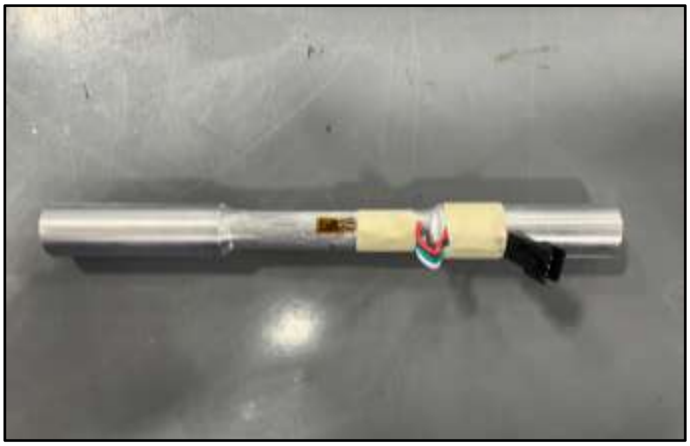


Figure 1. Tensile Test Rod with Applied Strain Gage

Introduction

The following laboratory experiment is meant to aid students in the understanding of material properties using tensile tests. Students will learn to develop stress-strain curves for tested materials, as well as familiarize themselves with tensile testing machinery and software.

The tensile test specimen will be pulled up to 70% of their yield strength as the load and strain are measured until the 70% yield has been reached, see Figure 1. The ESL Test II Intelligent Materials Test System and software will continuously measure the load, while a surface-mounted strain gage will measure the strain which will be read by the P3 Strain Indicator and Recorder by Micro-Measurements. The SATEC hydraulic tensile testing machine (TTM), with a capacity of 200,000 lb, will load the specimen up to its 70% yield limit. The SATEC TTM is equipped with specially designed chucks to grip the round test specimen, and a mounted strain gage will measure the strain of the test specimen.

The dimensions of the test rod are determined by the ASTM E8 Round Tension Test Specimen with Gage length 5 times the diameter standards. For the test specimen used in this manual, the diameter of the test area is 0.5 in., and the gage length is 2.5 in.

The table on the following page lists the yield strength and ultimate strength for various materials that will be tested in this experiment. The applied load in pounds will be measured as the specimen is stretched until the 70% yield limit. Stress-strain curves will be developed for each material and analyzed to determine Young’s Modulus. The experimental value for Young’s Modulus will be compared with the published values in Table 1 below.

Table 1. Material Properties of Tensile Testing Material.

Material	Specifications Met	Young's Modulus (ksi)	Yield Strength (ksi)	Tensile Strength (ksi)	70% yield load limit (lbs)
AISI Type 304L Stainless Steel	ASTM A276	28,500	30.5	81.8	4100
Low-Carbon Steel AISI 1018	ASTM A108	29,000	53.7	63.8	7100
6061-T6 Aluminum	ASTM B221	10,000	40	45	5500
360 Brass Rod	ASTM B16	14,500	15	19	2000

SATEC Testing Procedure

1. Measure the test specimen's cross-sectional area before loading to verify the upper 70% load limit. Also, measure the gage length of the test specimen.

$$Load\ Limit\ = 0.7\sigma_{Yield} \cdot A_{Test\ Section} \quad (1)$$

2. Start the ESL View software and proceed to the main ESL View interface, see Figure 2.

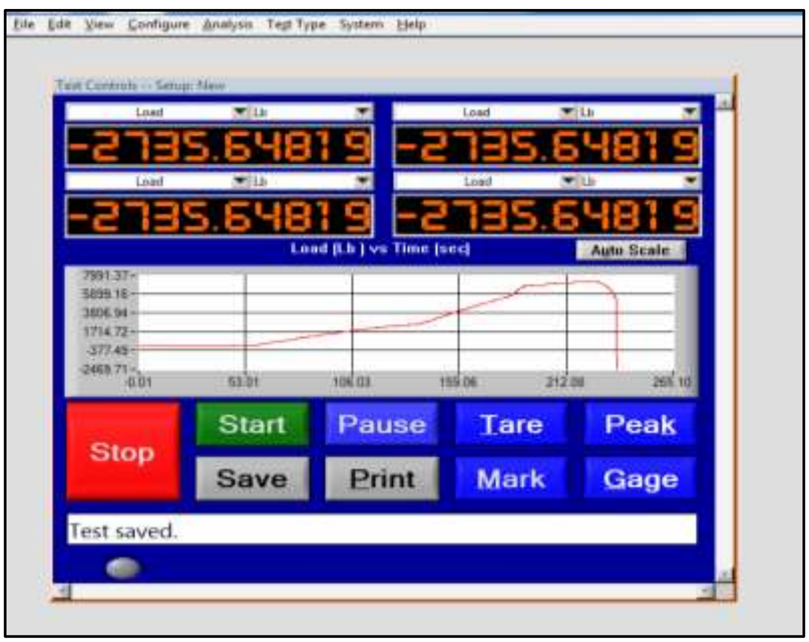


Figure 2. ESL View main interface

3. Start the Model P3 software and ensure the laptop is plugged into the P3 strain recorder. Connect the laptop to the box by hitting retry in the popup menu until the box and laptop are connected, see Figure 3. If the box still will not connect to the laptop, unplug the cord from the box, wait ten seconds then try again.

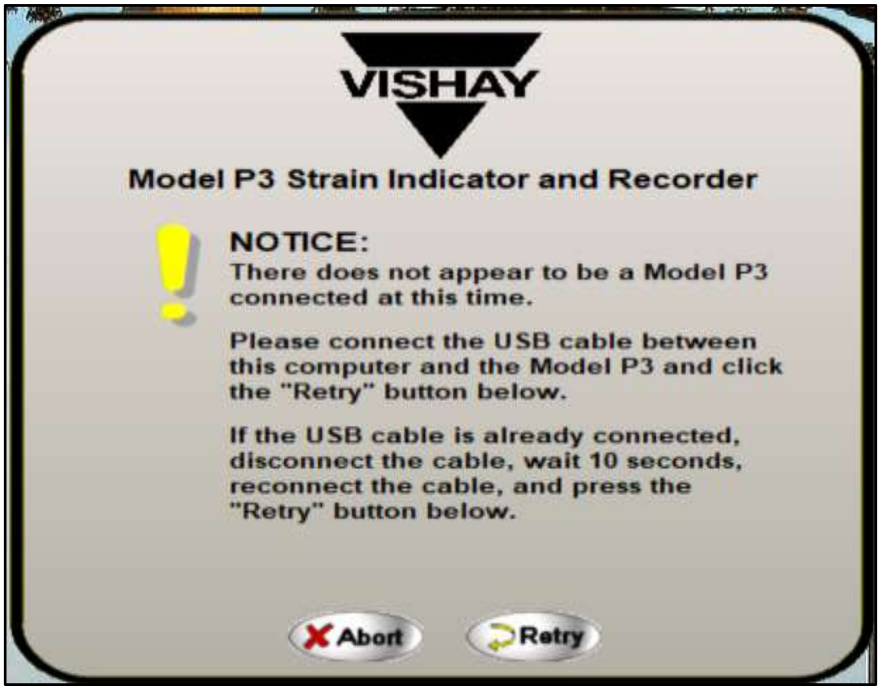


Figure 3. P3 Strain Indicator Connection Portal

Select the blue Tare button and ensure the numerical reading for the load is continuously varying over time.

4. Turn on the TTM. The ON switch is located on the right side of the pump control box located next to the TTM, see Figure 5a.

   The tensile testing machine is loaded using two handles that control the upper and lower dies, and a joystick that will load the machine with a small amount of force to create tension to hold the test specimen, as shown in Figure 4.

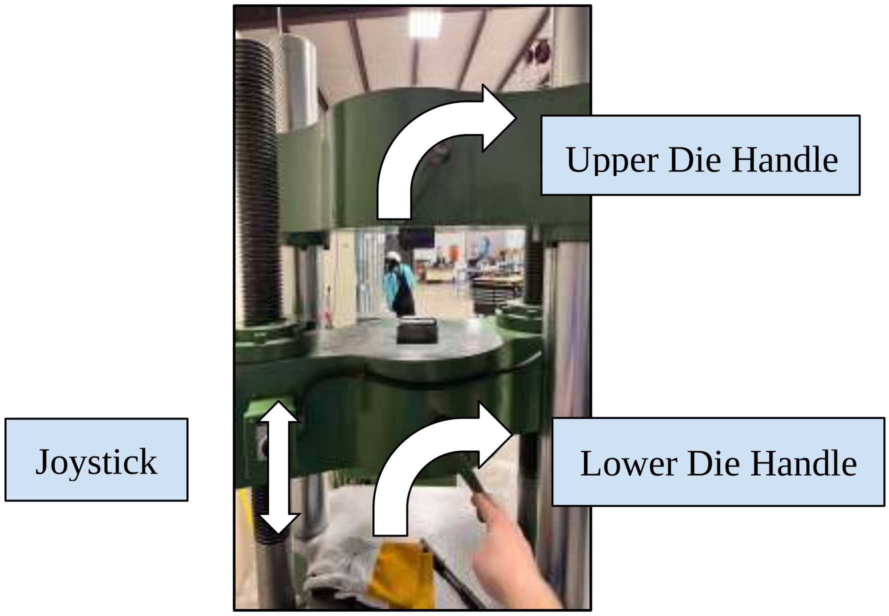


Figure 4. Tensile Testing Machine Loading Controls

Ensure the lower half of the TTM is low enough to fit the rod into the upper jaws. If not, use the joystick to lower the bottom half of the machine until the rod can be placed.

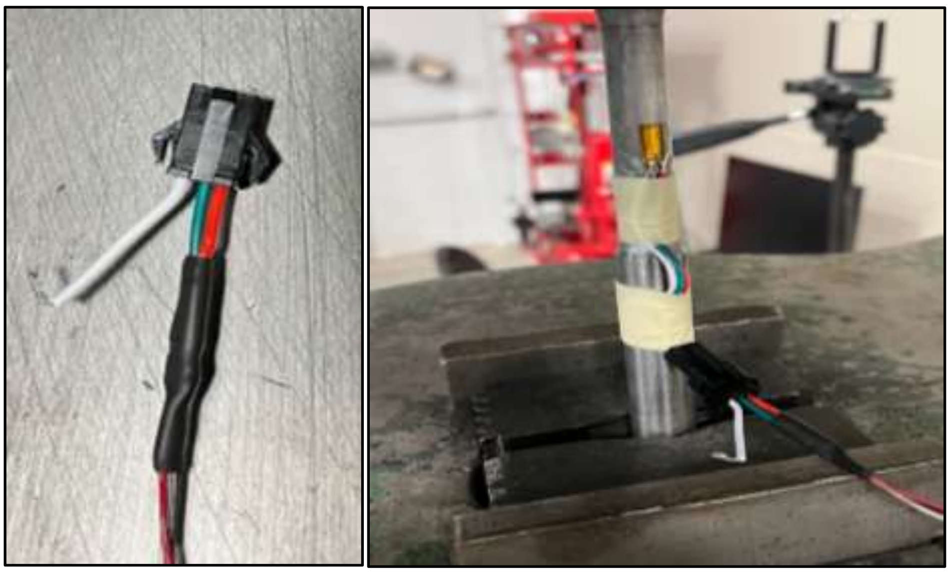
5. Use the upper die handle to load the test specimen in the upper jaws of the machine. The machine must grab at least 1.5 inches of the test rod to ensure a good grip. Use the joystick to raise the machine high enough to grab the bottom part of the rod. Raise the lower half of the TTM so that the die chucks are half an inch below the start of the first fillet on the rod. Use the lower die handle to grab the test rod and keep your hand on the handle. Use the joystick to pre-load the test rod just enough to hold it in place but not enough to stretch the specimen, see Figure 5.

TTM On-Off Switch | Upper Load | TTM Lower half raised | Fully Loaded Test Rod

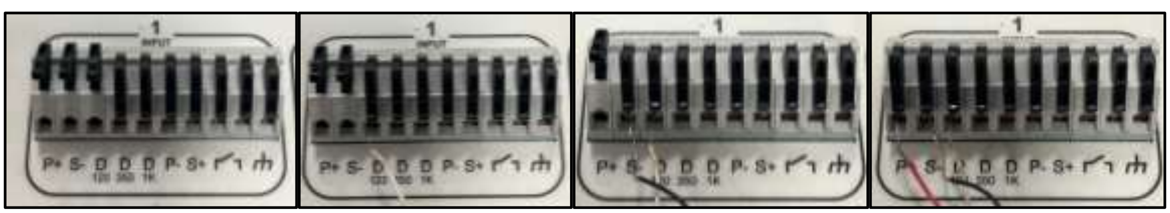
Figures 5. On-Off Switch and Test Rod Loading Steps

IMPORTANT: For the brass test specimen, the process of loading the brass specimen can **AND WILL STRETCH THE SPECIMEN.** To safely load the brass specimen, follow the process described in step 6 but for the lower jaws, grab the test rod and lightly tap the joystick. Have someone monitor the load on the laptop and ensure the load does not exceed 100 lbs. This can happen very quickly.

6. Once the test rod has been mounted in the jaws, connect the strain gage to the cord, see Figure 6a, by plugging in the data cord to the test rod, see Figure 6b. To connect the cord to the box, with the black locking switches in the up position, see Figure 7a, insert the white wire into the D120 port on Channel 1 Input. Then insert the black wire into the S- port and the red wire into the P+ port on Channel 1 Input.



Figures 6. Strain Gage Data Cord and Cord Connected to Test Rod



Lever Locks Up | White Wire Installed | Black Wire Installed | Red Wire Installed

Figures 7. Strain Gage Data Cord Connection to the P3 Strain Recorder

7. The P3 box should already be connected to the laptop and to be sure that the box is sending data to the laptop, the box and laptop display should be showing the same values for Channel 1 on the box.

8. Now that all mechanical equipment has been loaded and turned on, navigate to the control box for the TTM and ensure the laptop with the tensile testing software is open and operating. Data is collected by the ESL Test Box shown in Figure 8 and displayed using software.

Figure 8. ESL Test II Machine

Ensure that the tensile testing software is measuring load, see Figure 2. Stress can be measured as well but the software assumes a test area of 1in^2 which will output the same values for load and stress.

9. Figure 9 shows the controls for the TTM. The bottom knob will control how fast the load is applied to the specimen, and the handle, when pushed toward the load section on the bottom, will apply more load to the machine. One method to apply load is to set the wheel to a low setting and crank the handle all the way, and another method is to set handle and adjust knob upward. For steel rods, use the first method. For the brass and aluminum, use the second method described. Before beginning testing, turn bottom knob clockwise to the right until the knob won't turn anymore. This will zero-out this control.

Figure 9. Test Controls on the TTM Control Box

10. Navigate to the control screen for the P3 software home and click record, see Figure 10.

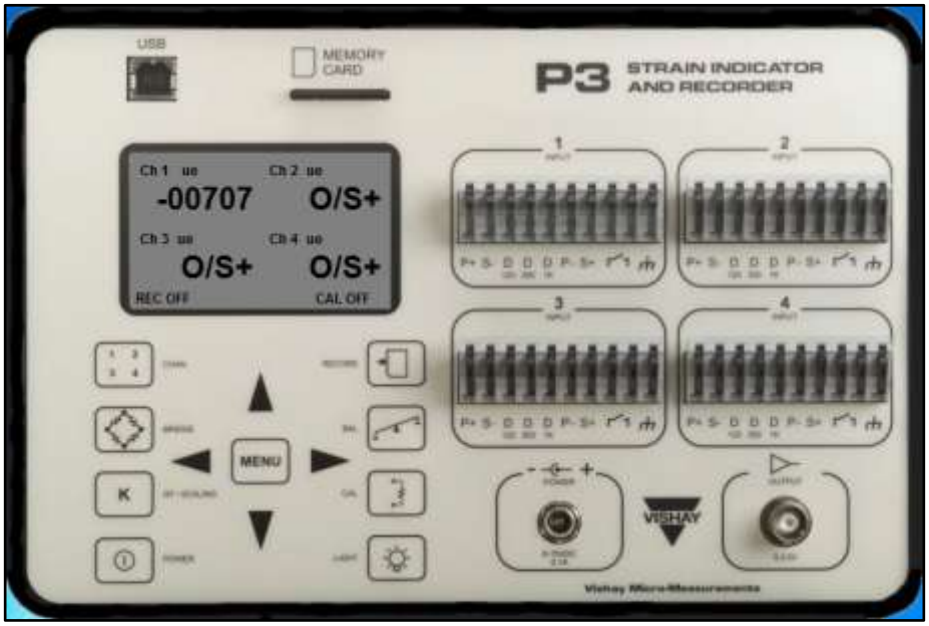


Figure 10. P3 Software Home Screen.

11. A recording menu will pop up, select Time-Based Recording, Channel 1 (or whatever input channel the wires are installed in), and On This Computer. See Figure 11a for recording menu settings. An example of output data is shown in Figure 11b.

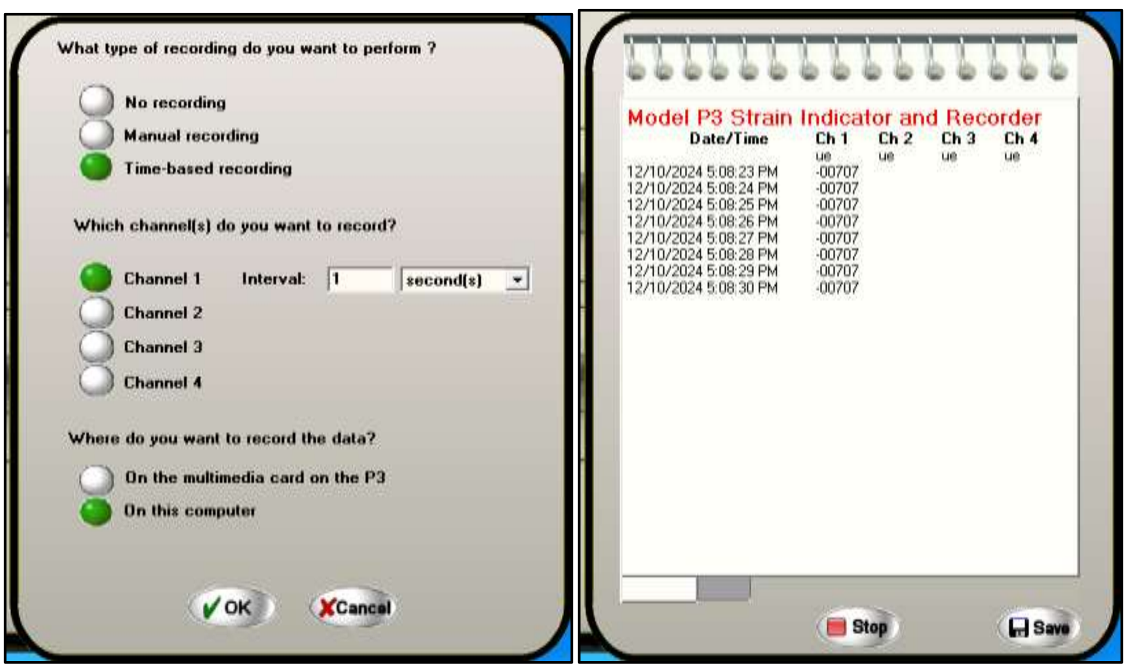


Figure 11. Recording Settings and Sample Recording Output

IMPORTANT: Pressing start on the ESL software will calibrate the software but the P3 software starts recording data immediately. To start the tests at the same time, press start on the ESL software, then once the software stops calibrating, press start on the P3 software. It is recommended to test this setup before recording data

Start the test by pressing start on the test control software and begin loading the test specimen. The TTM has a high capacity and the load can quickly jump up to the yield limit so be sure to keep a steady eye on the load limit listed in Table 1. Once the load limit is reached, unload the specimen.

Select the red Stop button on the ESL View interface and the P3 software when the specimen load limit is reached. Select File>>Export from the menu in the ESL software interface and save the data file. Save the data from the P3 software as well.

Unload the specimen from the jaws by rotating the corresponding knobs. Push the button for Pump Stop shown in Figure 9. Turn off the SATEC testing machine using the switch shown in Figure 5a.

Repeat this procedure for the following rod specimens of the materials listed in Table 1. and collect the test data.

Analyzing Data in Excel

Microsoft Excel will now be used to analyze the collected data. The data must be cleaned up and corrected before a usable Young's Modulus can be determined for the tensile test.

12. Open Excel. Click File>> Open>> This PC>> then select your data file for the load data from the ESL software. If not visible, select format All Files ("*.*") in Excel to open the saved file. In Text Import Wizard - Step 1 of 3, set Start Import at Row: to 1. Check the box for My Data has Headers. Select Next > in the Text Import Wizard as shown in Figure 12.

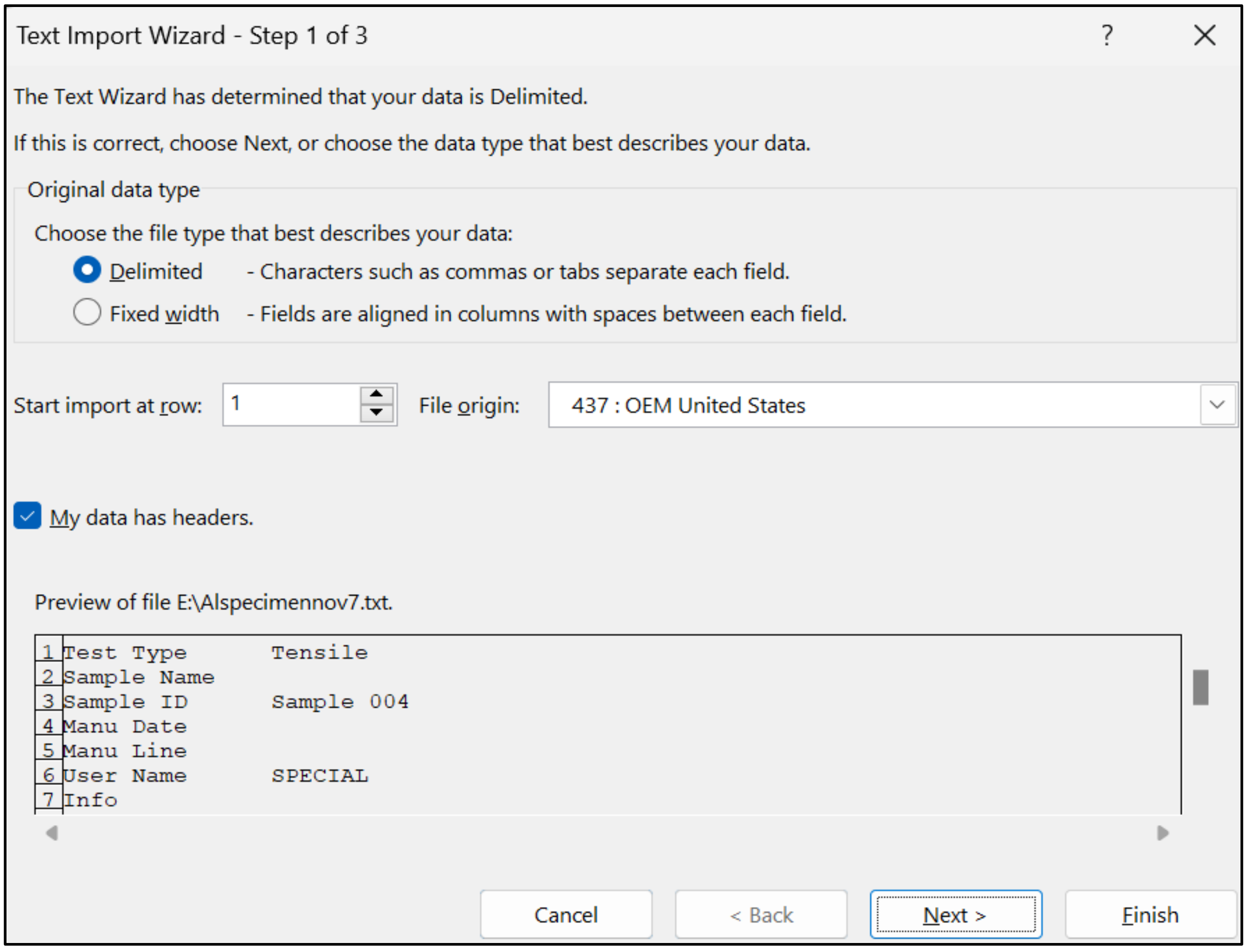


Figure 12 Text Import Wizard Step 1 of 3

13. Select Tab as Delimiter in Step 2 and select Next> once again. Select General as Column data format in Step 3 and select Finish as shown in Figures 13a and 13b.

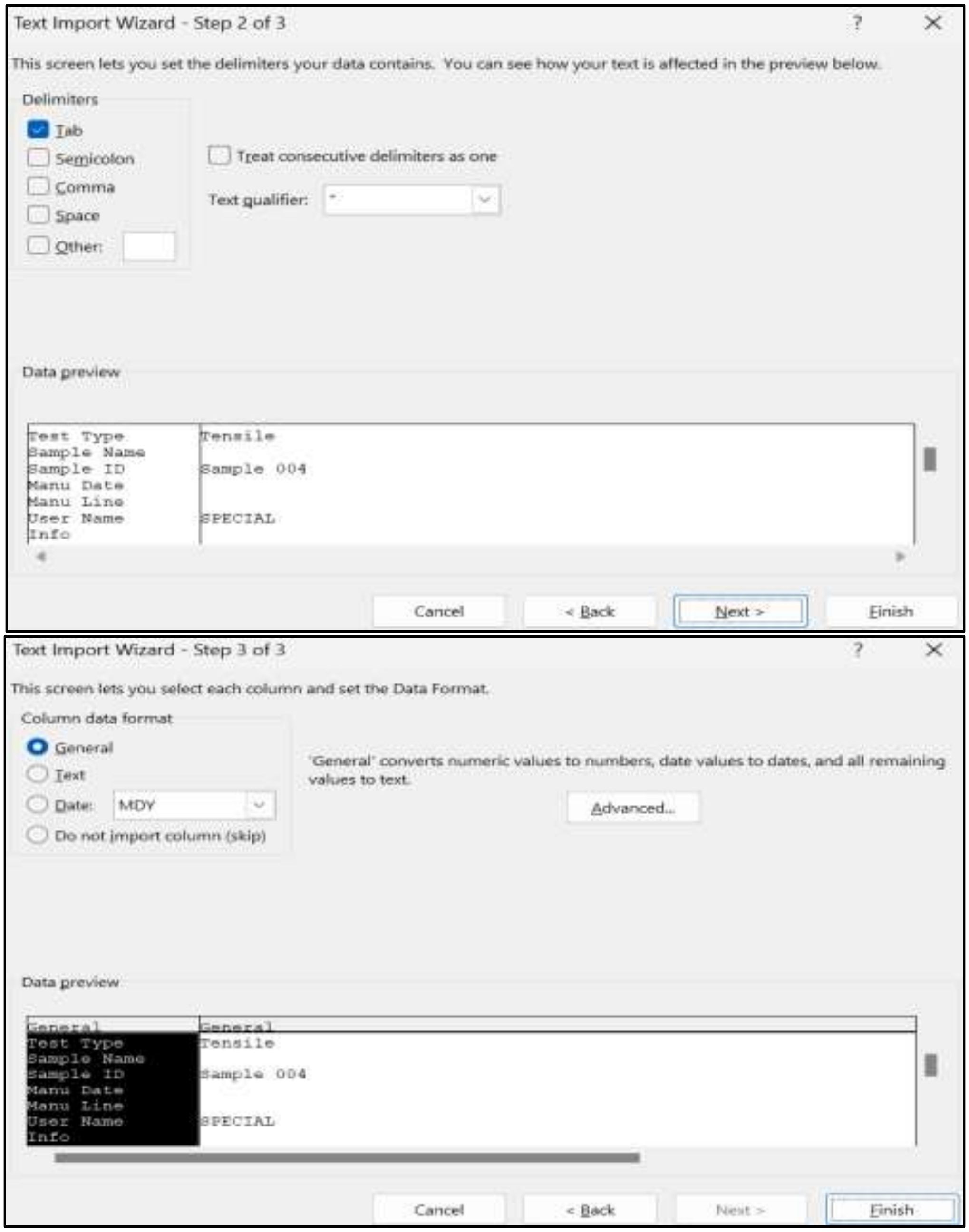


Figure 13. Text Import Wizard Steps 2 and 3 of 3

14. The data must be filtered and analyzed to create the stress-strain curve. Insert a new column to the right of Column E labeled Stress. Label this new column F Corrected Stress. Insert a formula in the first cell under the label of the new column that divides the stress value by the original cross-sectional area of the test specimen. Explicitly, in cell F31, enter the formula =E31/ original measured cross-sectional area.

    The ESL software records data every 0.1 second but the P3 data records data every second. Before combining the data, delete all non-integer data points (i.e. 2.1, 5.6, 7.9) and collect the data into one continuous column.

Open Excel. Click File>> Open>> This PC>> then select your data file for the strain data from the P3 software. Divide the strain data in column A by 1000 in column B and title this column Corrected Strain [in/in]. Copy and paste the stress data into the strain Excel file. If the tests were started at the same time, then line up the first two data points. If not, line up the data according to the starting times listed on the respective data files Select both corrected Stress and Strain columns and insert a scatter plot, see Figure 14.

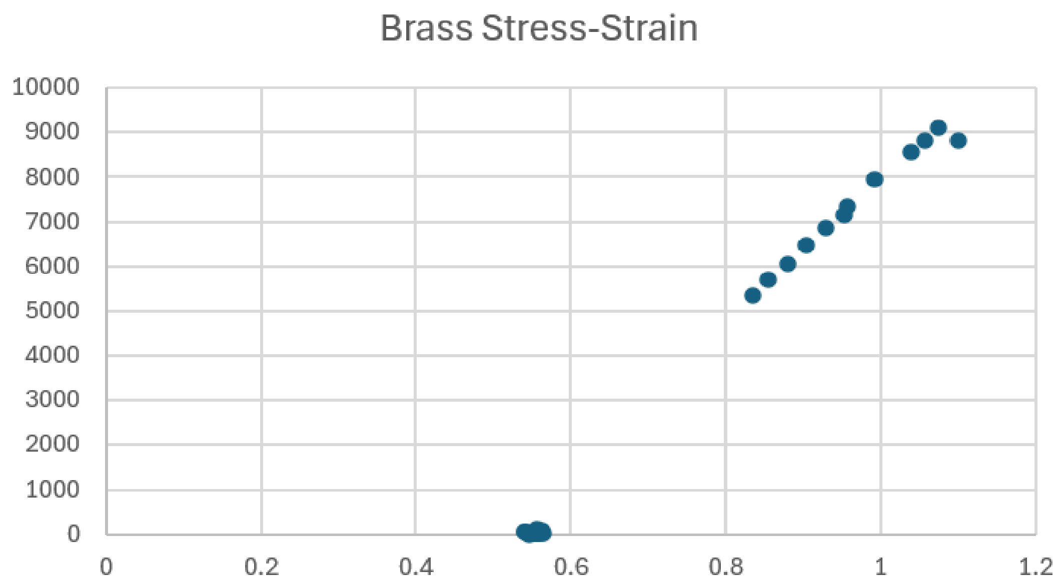


Figure 14. Initial Data Stress-Strain Curve

15. This data should look relatively linear. Add a trendline and axes titles. Label the axes. Click on the graph, then click on the green plus sign, then click the arrow next to trendline >>More Options and check the boxes for “Display Equation on chart” and “Display R-squared value on chart. Data will need to be refined. The easiest way to accomplish this is to click on the graph and refine outlying data points by dragging down the highlighted purple and blue boxes indicating the displayed data. Check the graph periodically until the linear region only is displayed and check the displayed equation. The coefficient in front of the x is the measured Young’s Modulus for the specimen. The refined data will look like Figure 15.

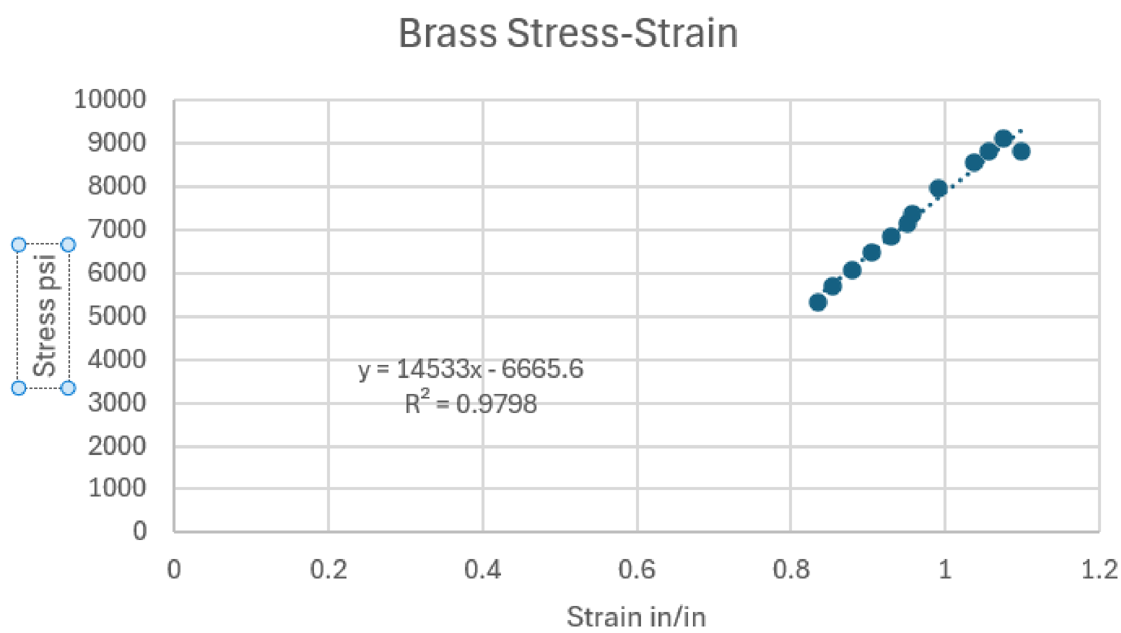


Figure 15. Filtered Stress-Strain Curve

Record the Young's Modulus and compare your result with the given values in Table 1. Save the combined data file as the specimenmaterial_month_day.

Theory

16. Stress is defined as:

$$\sigma = \frac{F}{A_o} \tag{2}$$

where A_o is the original cross-sectional area of the test specimen, and F is the applied force[1].

Strain is defined as:

$$\epsilon = \frac{\Delta L}{L} \tag{3}$$

where ΔL is the change in length, and L is the original length of the test section of test rod[1].

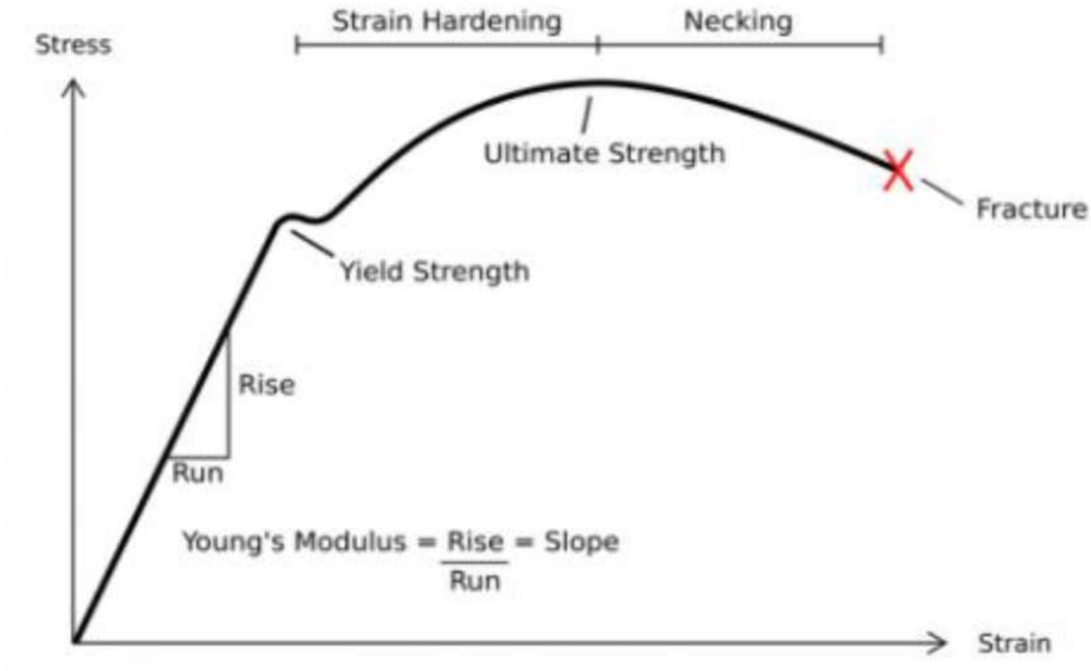


Figure 16. Typical Stress-Strain Diagram

Above is a typical stress-strain diagram. There are four regions on the stress-strain curve. The first is the elastic region, the linear region from the origin to the yield strength point on the curve[2]. This region defines the Young's Modulus for the test material. The outlined testing procedure above will not go past the yield limit for the specimens but still remains valid as a method to determine the Young's Modulus for materials.

The second region on the stress-strain curve is the yielding region. In this region, the specimen will elongate without increasing the load placed on the specimen, thus remaining flat[2]. (Note: this region is not shown in Figure 16 but occurs in between the yield strength point and the strain hardening region.) The material in this state exhibits plastic behavior as the material deforms plastically[2].

The third region occurs when the load increases and is referred to as strain hardening. In this region, the specimen can handle more load but only up to the point of ultimate strength where the necking region begins[2]. The last region on the stress-strain curve is the necking region. The specimen will undergo an elongation, and a subsequent, localized reduction in cross-sectional area will occur. This is referred to as necking[2]. At some point in this region, the specimen will fracture, indicating the end of the test. The fractured test specimen will display a cup and cone pattern in the region where the necking and subsequent fracture occurred, see Figure 17 for reference.

Figure 17. Necking Pattern Characteristics

References

1. Vidosic, J. P. (1978). Mechanics of Materials. In *Mark's Standard Handbook for Mechanical Engineers* (8th ed., pp. 5-16-5–18). section, McGraw-Hill.
2. Hibbler, R. C. (2011). The Stress-Strain Diagram. In *Mechanics of Materials* (8th ed., Vol. 1, pp. 84–85). essay, Pearson Prentice Hall.

Appendix 2: Ansys Tutorial for Tensile Testing Project

TENSILE TESTING ROD

A. Objectives

- Using Ansys Explicit Dynamics to simulate tensile testing of standard test rod
- Creating mesh and displacements
- Running the calculations
- Using deformations for visualizations
- Comparing Ansys solution with theory

B. Problem Description

We will study tensile testing of a test rod made of aluminum, brass, stainless, and low-carbon steel.

C. Launching Ansys Workbench and Selecting Explicit Dynamics

1. Start by launching the Ansys Workbench. Select Tools>>Options… from the menu. Select Appearance on the left-hand side. Select Uniform as Background Style under Graphics Style. Set Background Color and Background Color 2 to white. Set Text Color to black and select OK. Double click on the Explicit Dynamics Analysis System.

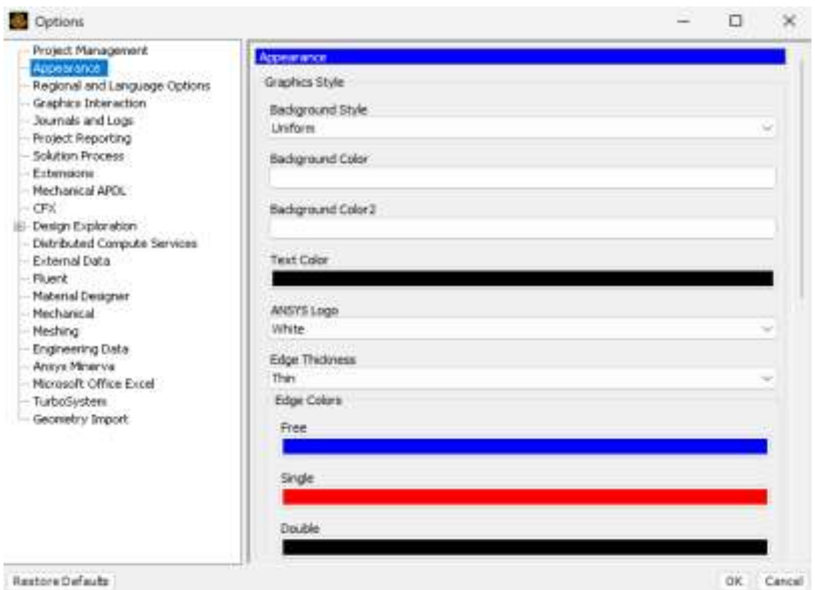


Figure 1a) Appearance settings

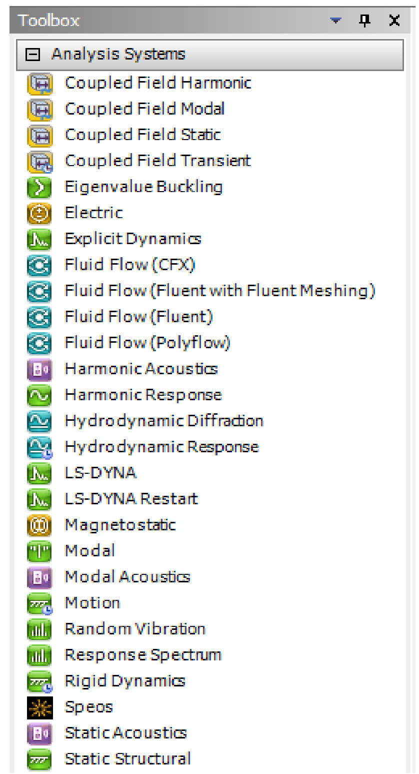


Figure 1b) Explicit Dynamics

2. Double click on Engineering Data in the Project Schematic window. Select the Engineering Data Sources tab and select Explicit Materials. Highlight AL 6061-T6 under Material in the Outline of Schematic. Select the plus sign next to AL 6061-T6. Click on the Project tab under Tools in the menu.

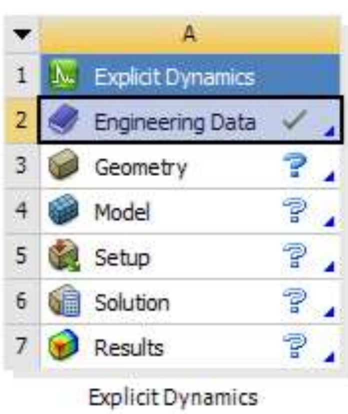


Figure 2a) Selecting Engineering Data

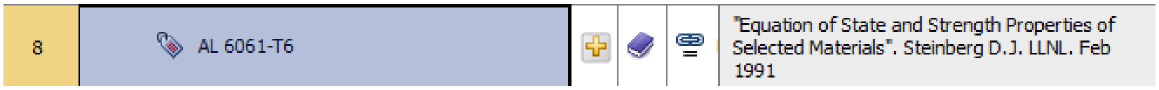


Figure 2b) Highlighting Aluminum AL 6061-T6

Properties of Outline Row 8: AL 6061-T6

	A	B	C
1	Property	Value	Unit
2	Density	2703	kg m^-3
3	Specific Heat Constant Pressure, C_p	885	J kg^-1 C^-1
4	Steinberg Guinan Strength		
5	Initial Yield Stress Y	2.9E+08	Pa
6	Maximum Yield Stress Ymax	6.8E+08	Pa
7	Hardening Constant B	125	
8	Hardening Exponent n	0.1	
9	Derivative dG/dP G'P	1.8	
10	Derivative dG/dT G'T	-1.7E+07	Pa C^-1
11	Derivative dY/dP Y'P	0.018908	
12	Melting Temperature Tmelt	946.85	C
13	Shear Modulus	2.76E+10	Pa
14	Shock EOS Linear		

Figure 2c) Properties of Aluminum AL 6061-T6

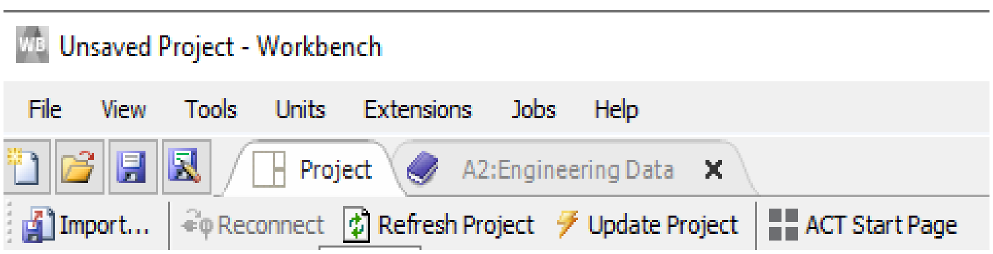


Figure 2d) Returning to the Project

D. Launching Ansys Mechanical

3. Right click on Geometry in the Project Schematic window and select Import Geometry>>Browse. Select the file *Correct Rod.step* Double click on Model in the Ansys Workbench Project Schematic to open the Ansys Mechanical window. **Select the Home tab from the menu and select Tools>>Units>>Metric (m, kg, …)**. Open Geometry in the Outline, select Correct Rod-FreeParts|Fillet1 and assign AL 6061-T6 as the Material in Details View for the lap joint.

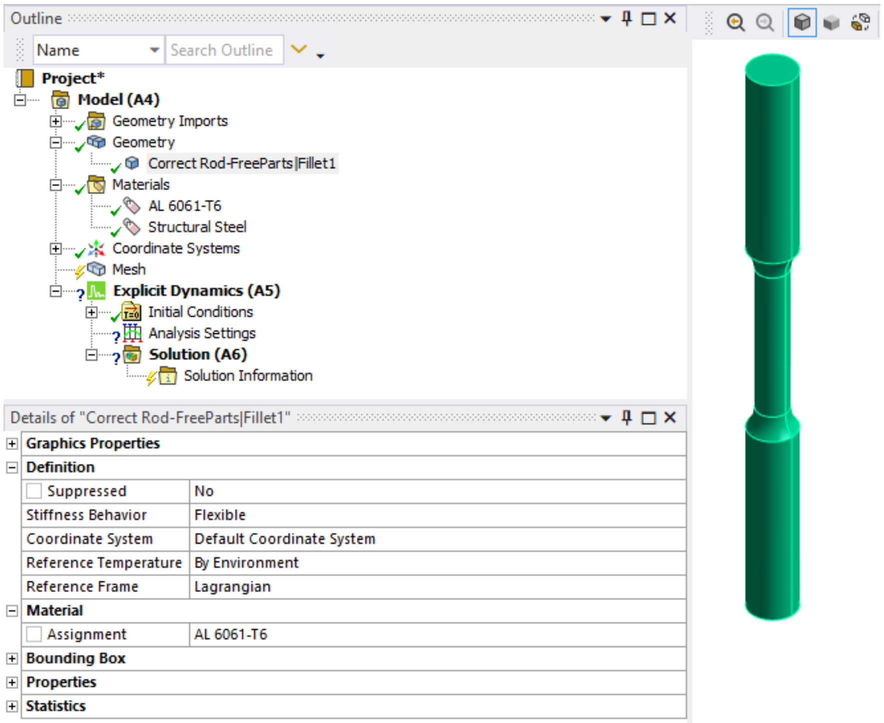


Figure 3 Assigning AL 6061-T6 as material for test rod

4. Right click on Mesh in the Outline and select Generate Mesh. The mesh has 984 Nodes and 720 Elements.

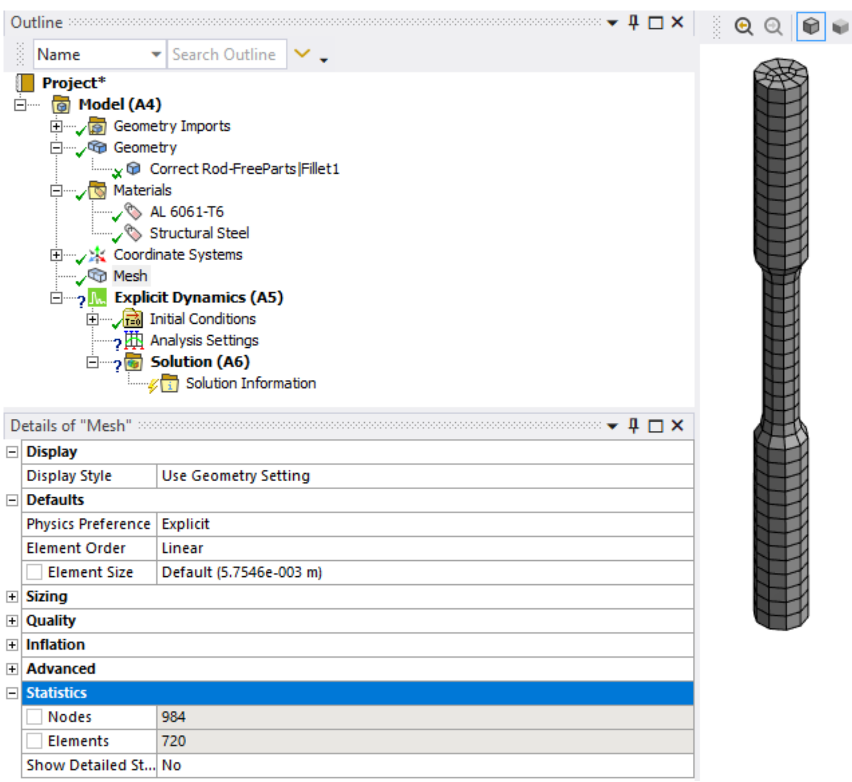
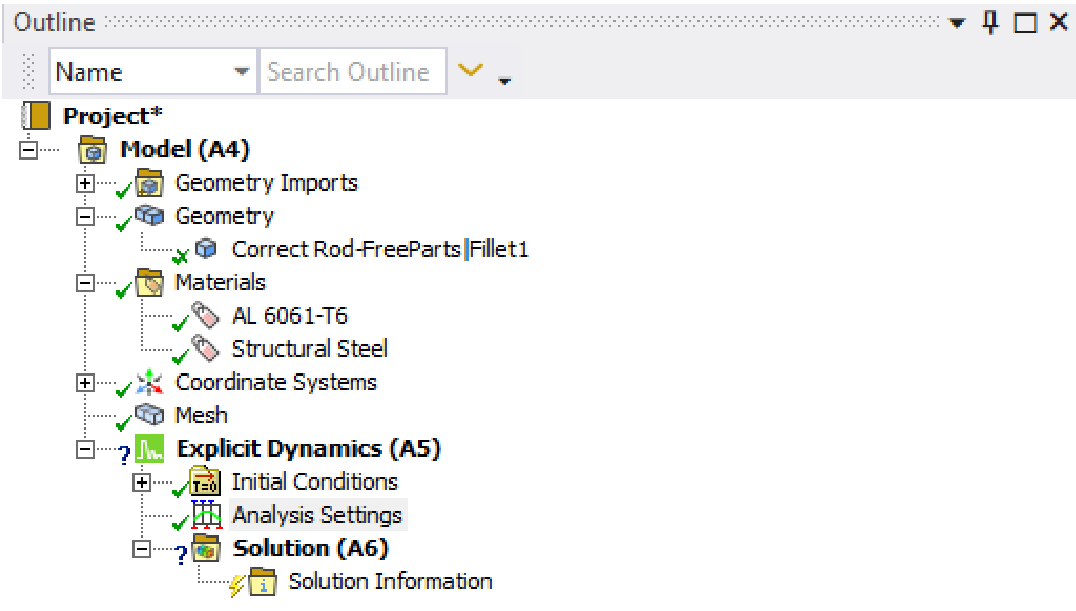


Figure 4 Details of mesh for test-rod

5. Select Analysis Settings under Explicit Dynamics in the Outline. Set the End Time to 0.001 s.



Details of "Analysis Settings"

Analysis Settings Preference	
Type	Program Controlled
Step Controls	
Number Of Steps	1
Current Step Number	1
Load Step Type	Explicit Time Integration
End Time	1.e-003 s
Resume From Cycle	0
Maximum Number of Cycles	1e+07
Maximum Energy Error	0.1
Reference Energy Cycle	0
Initial Time Step	Program Controlled
Minimum Time Step	Program Controlled
Maximum Time Step	Program Controlled
Time Step Safety Factor	0.9
Characteristic Dimension	Diagonals
Automatic Mass Scaling	No

Figure 5 Details of analysis settings

6. Right click on Explicit Dynamics in the Outline and select Insert>>Displacement. Select the top two vertical cylindrical faces of the upper part of the rod and Apply these faces as Geometry. Select Y Component under Definition in Details of “Displacement”. Enter the value 0.1 m shown in figure 9.6. Right click on Explicit Dynamics in the Outline and select Insert>>Fixed Support. Select the bottom two vertical square faces of the lower plate and Apply these faces as Geometry.

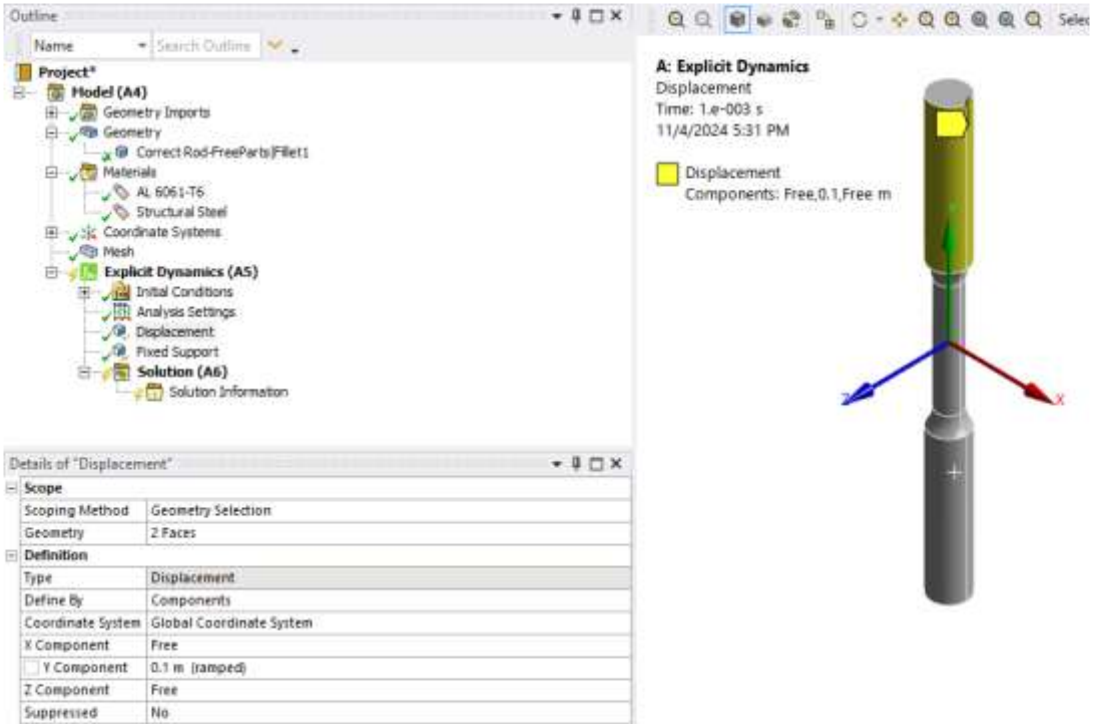


Figure 6a) Inserting Y-component of displacement for test-rod

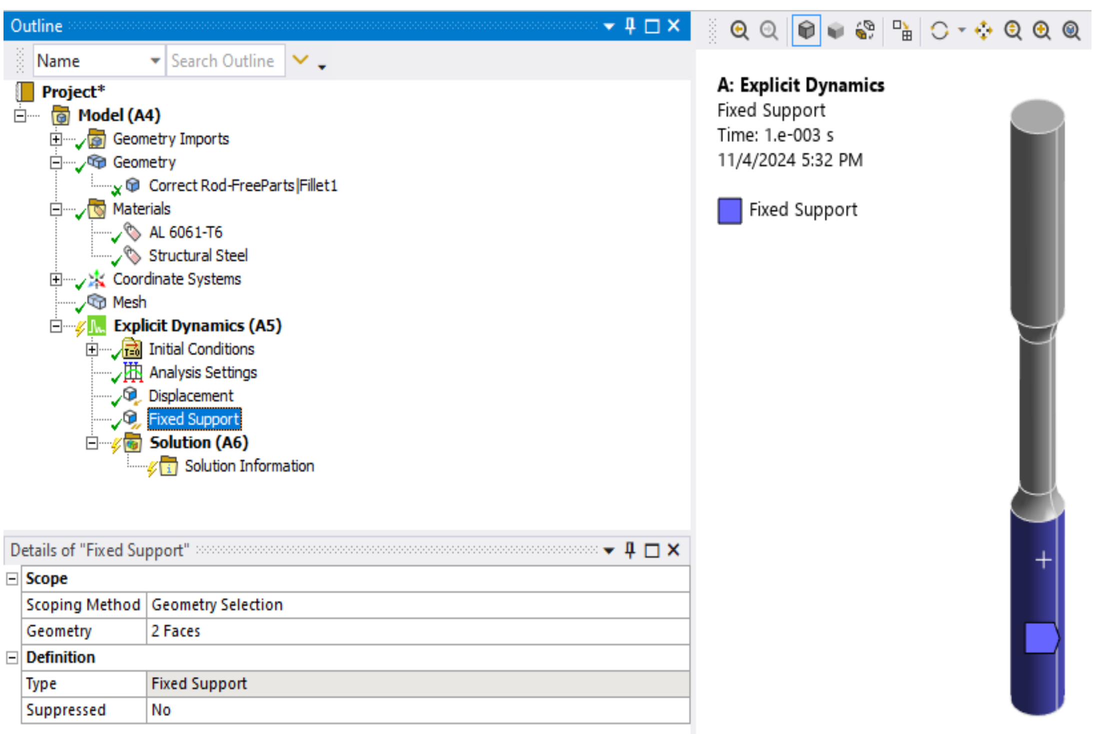


Figure 6b) Inserting fixed support for test-rod

Right click on Explicit Dynamics in the Outline and select Insert>>Standard Earth Gravity. Select -Y Direction under Definition in Details.

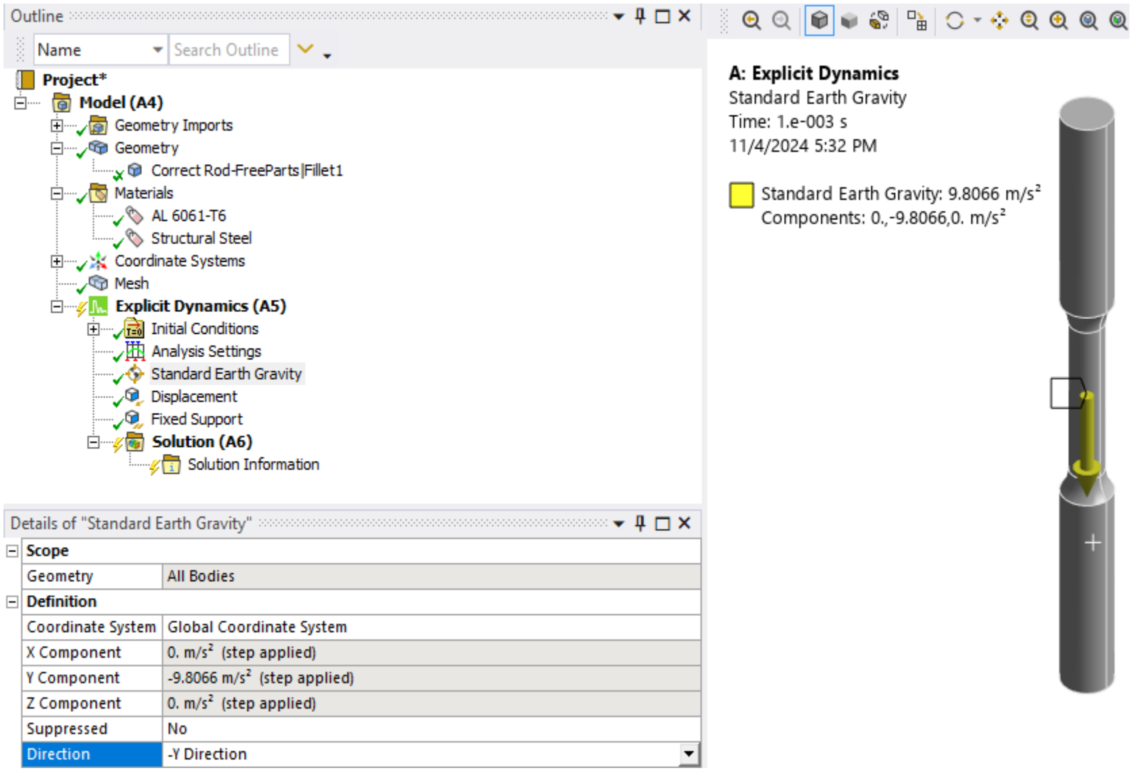


Figure 6c) Inserting gravity

Right click on Solution in the Outline and select Insert>> Deformation>>Total. Right click on Solution in the Outline and select Insert>> Deformation>>Directional. Select Y Axis as Orientation under Definition in Details.

Right click on Solution in the Outline and select Insert>>Probe>>Force Reaction. Select Boundary Condition as Location Method under Definition in Details. Select Displacement as Boundary Condition in Details.

Right click on Solution in the Outline and select Insert>>Stress>>Normal. Select Y Axis as Orientation under Definition in Details. Right click on Solution in the Outline and select Insert>>Strain>>Normal. Select Y Axis as Orientation under Definition in Details.

Set the number of Cores to 4 under the Home tab in the menu. **Uncheck the Distributed check box located above Cores in the Home>>Solve tab.** Select Save As from the menu and save the project with the name *TensileTestRodAL6061*. Select Solve from the menu.

E. Post-Processing

7. Double click on Force Reaction under Solution in the Outline. Select 100 Frames as Total number of frames in animation. Select 10 Sec as Play duration for animation. Export Video File and save the file with the name *Tensile Testing Force Reaction (Y) [N] Video*.mp4. Uncheck the boxes for Force Reaction (X) [N], Force Reaction (Z) [N] and Force Reaction (Total) [N] under Tabular Data.

Figure 7a) Exporting a video file

Drag the Force Reaction (Y) [N] column of data towards the Time [s] column. Select the row number in Tabular Data corresponding to the maximum Force Reaction (Y) [N]. In this case, that is row number 4 at Time [s] = 1.5007e-004 s corresponding to maximum Force Reaction (Y) [N] 2.1979e+005 N. Right click on row 4 and select Retrieve This Result.

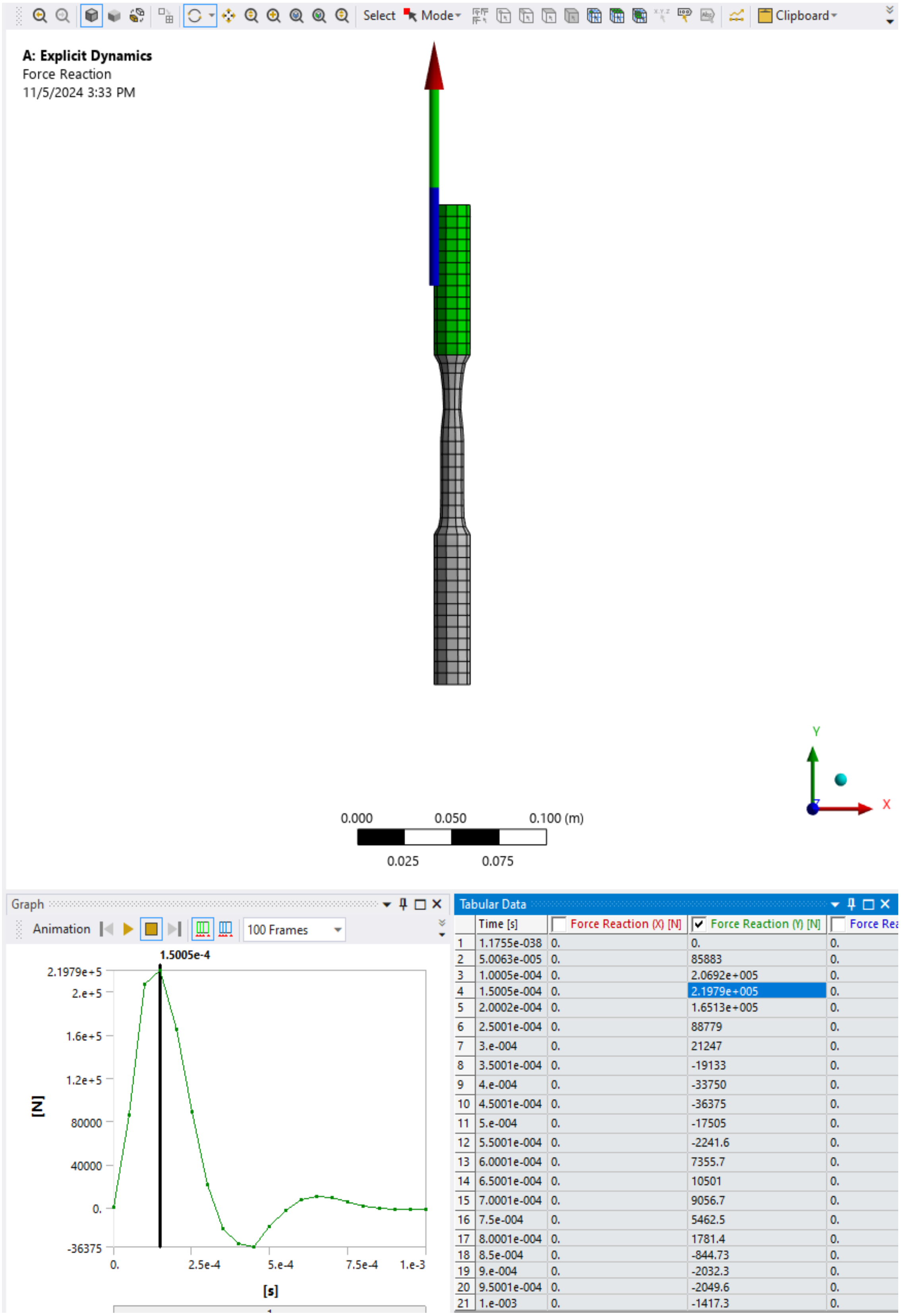


Figure 7b) Maximum Force Reaction (Y) [N] at *t* = 1.5007e-4 s

Double click on Directional Deformation (Y Axis) under Solution in the Outline. Select 100 Frames as Total number of frames in animation. Select 10 Sec as Play duration for animation. Export Video File and save the file with the name *Tensile Testing Directional Deformation Y Axis Video*.mp4. Uncheck the boxes for Minimum [m] and Average [m] under Tabular Data.

Select the row number in Tabular Data corresponding to the maximum Force Reaction (Y) [N]. In this case, that is row number 4 at Time [s] = 1.5007e-004 s corresponding to Maximum [m] = 1.5007e-002. Right click on row 4 and select Retrieve This Result. Drag the Maximum [m] data column of data towards the Time [s] column.

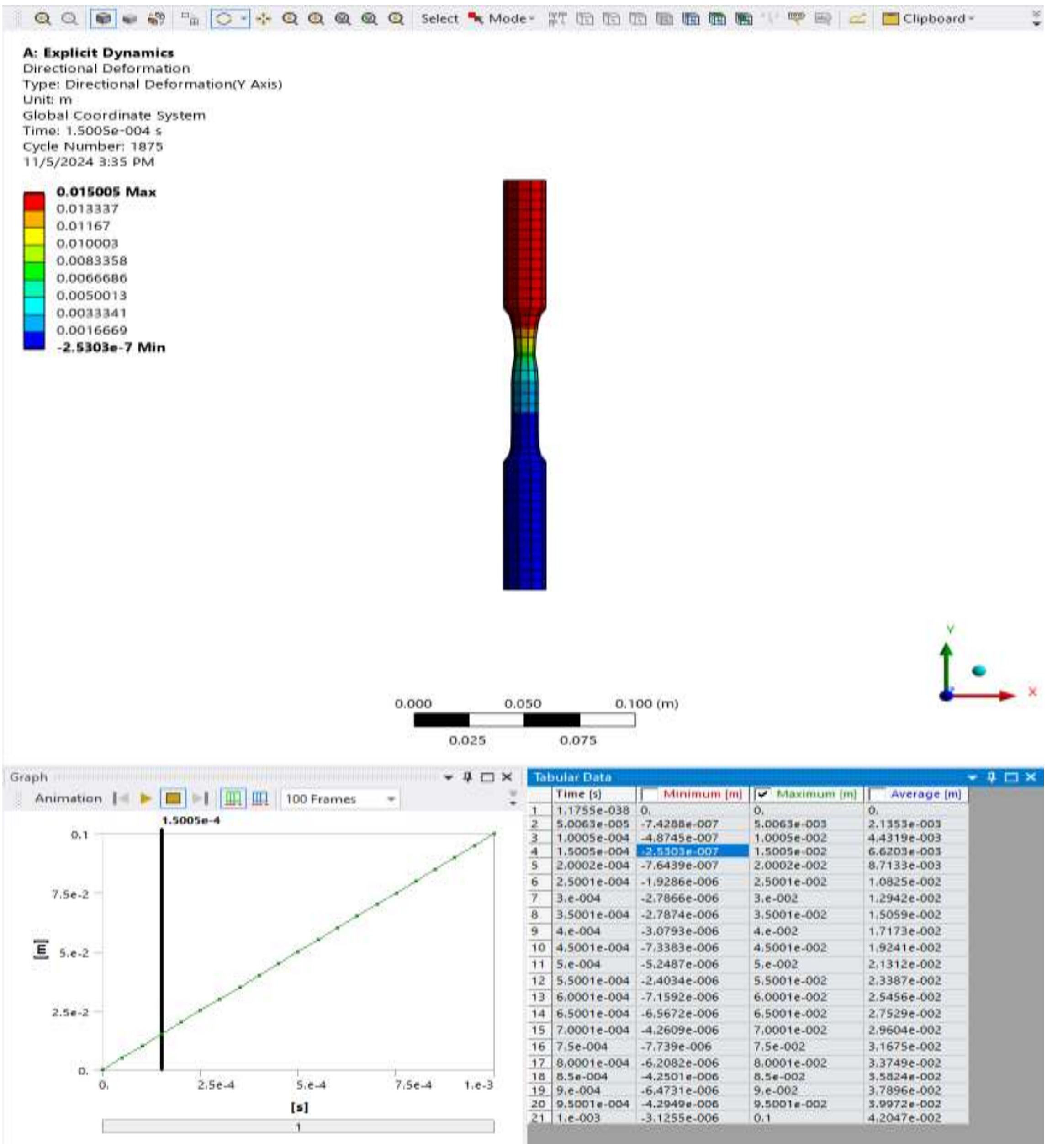
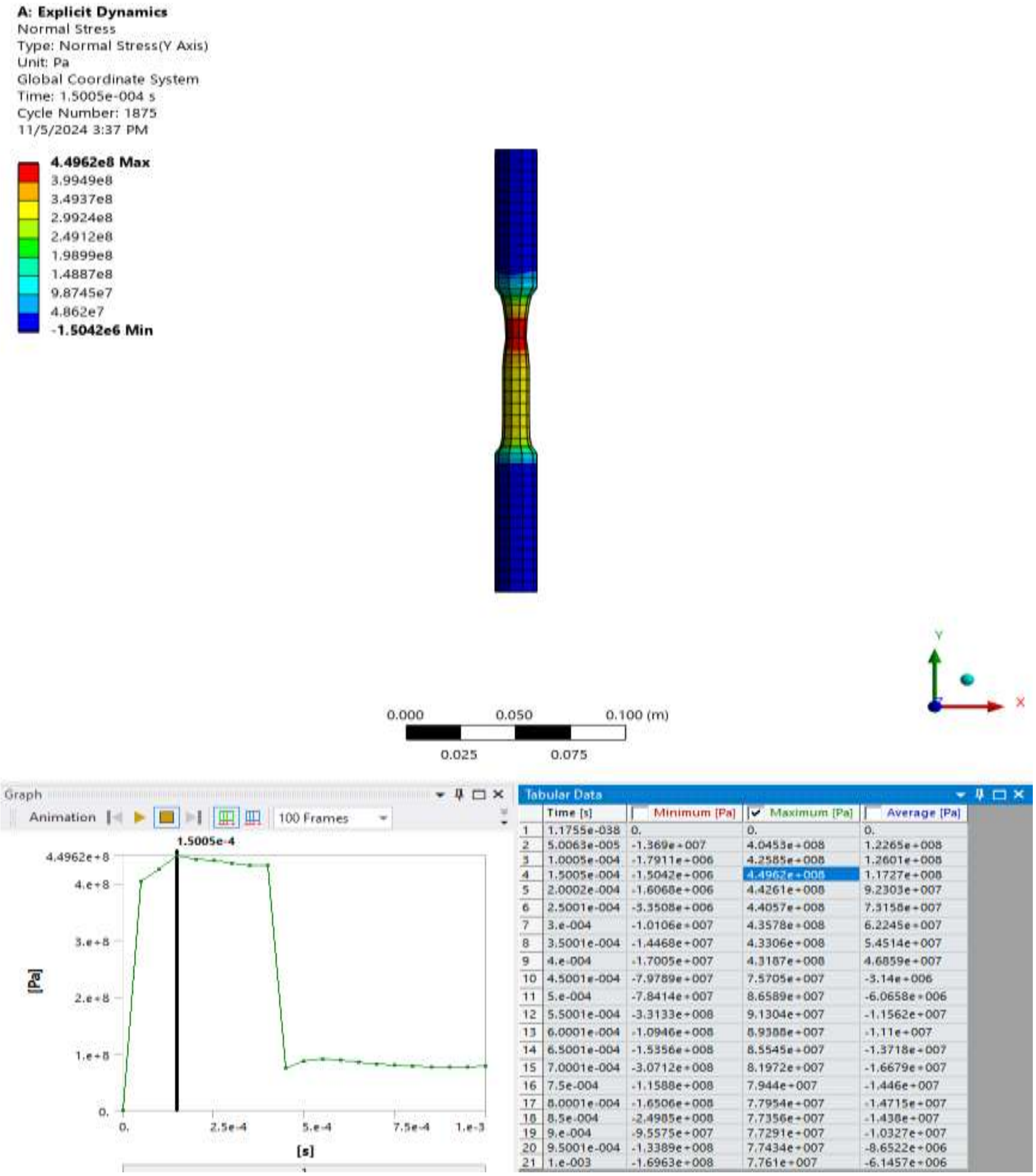


Figure 7c) Maximum Directional deformation Y-axis at t = 1.5005e-4 s.

Double click on Normal Stress (Y Axis) under Solution in the Outline. Select 100 Frames as Total number of frames in animation. Select 10 Sec as Play duration for animation. Export Video File and save the file with the name *Tensile Testing Normal Stress Y Axis Video*.mp4. Uncheck the boxes for Minimum [Pa] and Average [Pa] under Tabular Data.

Select the row number in Tabular Data corresponding to the maximum Normal Stress Y Axis . In this case, that is row number 4 at Time [s] = 1.5005e-004 s corresponding to Maximum [Pa] = 4.4962e+008. Right click on row 4 and select Retrieve This Result. Drag the Maximum [Pa] data column of data towards the Time [s] column.

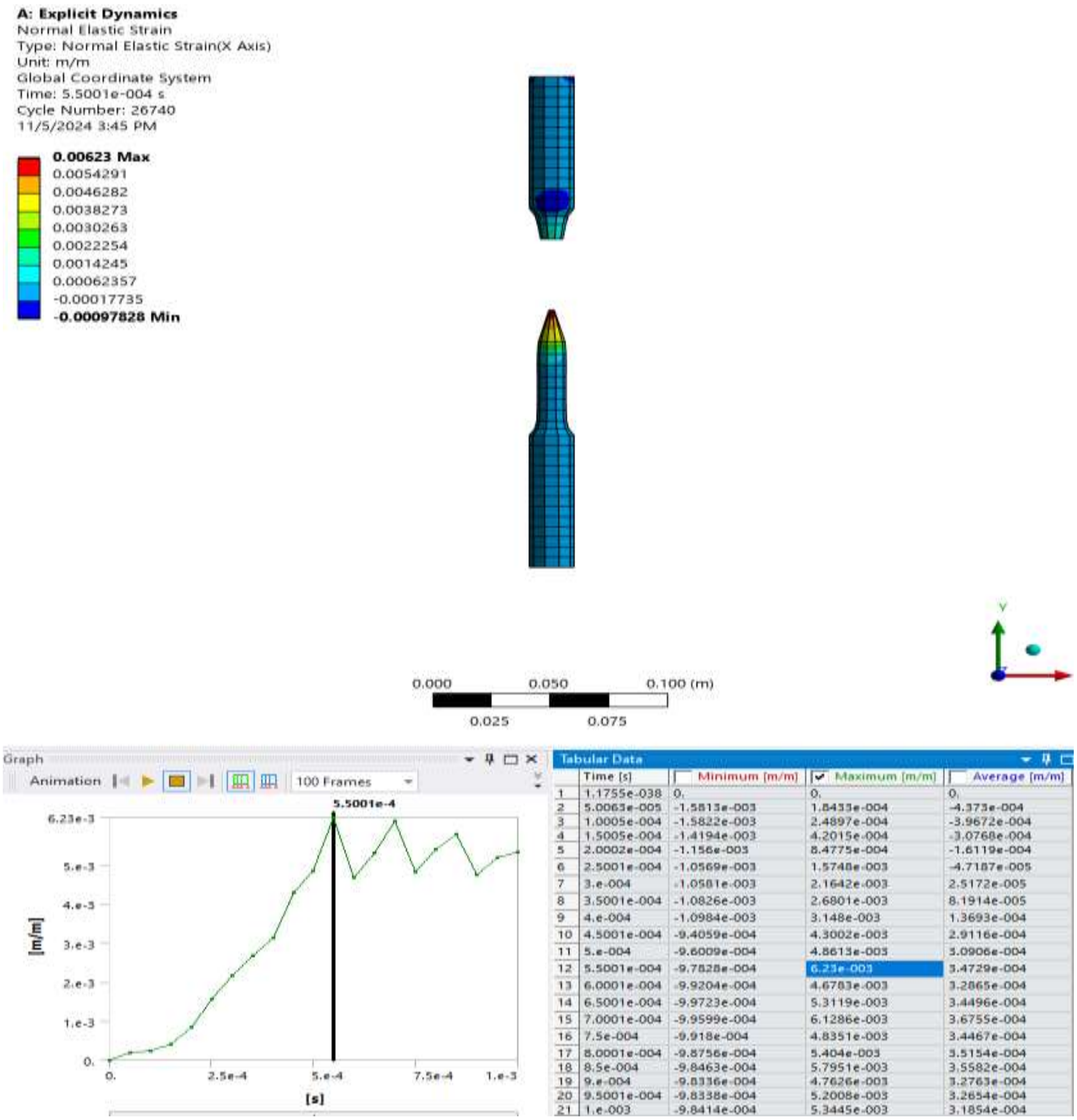


	Time [s]	Minimum [Pa]	Maximum [Pa]	Average [Pa]
1	1.1755e-038	0.	0.	0.
2	5.0063e-005	-1.369e+007	4.0453e+008	1.2265e+008
3	1.0005e-004	-1.7911e+006	4.2585e+008	1.2601e+008
4	1.5005e-004	-1.5042e+006	4.4962e+008	1.1727e+008
5	2.0002e-004	-1.6068e+006	4.4261e+008	9.2303e+007
6	2.5001e-004	-3.3508e+006	4.4057e+008	7.3158e+007
7	3.e-004	-1.0106e+007	4.3578e+008	6.2245e+007
8	3.5001e-004	-1.4468e+007	4.3306e+008	5.4514e+007
9	4.e-004	-1.7005e+007	4.3187e+008	4.6859e+007
10	4.5001e-004	-7.9789e+007	7.5705e+007	-3.14e+006
11	5.e-004	-7.8414e+007	8.6589e+007	-6.0658e+006
12	5.5001e-004	-3.3133e+008	9.1304e+007	-1.1562e+007
13	6.0001e-004	-1.0946e+008	8.9388e+007	-1.11e+007
14	6.5001e-004	-1.5356e+008	8.5545e+007	-1.3718e+007
15	7.0001e-004	-3.0712e+008	8.1972e+007	-1.6679e+007
16	7.5e-004	-1.1588e+008	7.944e+007	-1.446e+007
17	8.0001e-004	-1.6506e+008	7.7954e+007	-1.4715e+007
18	8.5e-004	-2.4985e+008	7.7356e+007	-1.438e+007
19	9.e-004	-9.5575e+007	7.7291e+007	-1.0327e+007
20	9.5001e-004	-1.3389e+008	7.7434e+007	-8.6522e+006
21	1.e-003	-1.6963e+008	7.761e+007	-6.1457e+006

Figure 7d) Maximum Normal Stress Y-axis at t = 1.5005e-004 s.

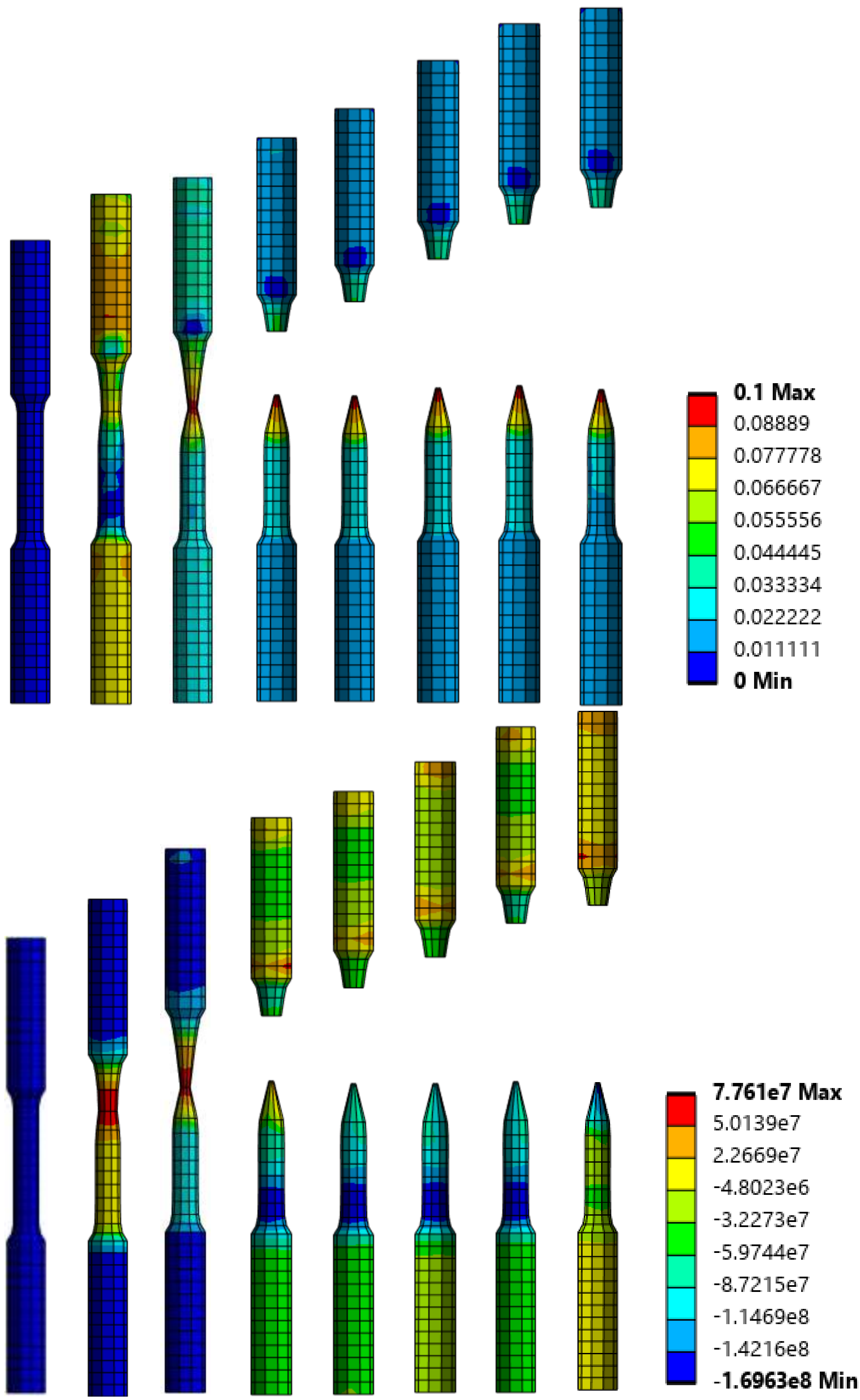
Double click on Normal Elastic Strain (Y Axis) under Solution in the Outline. Select 100 Frames as Total number of frames in animation. Select 10 Sec as Play duration for animation. Export Video File and save the file with the name *Tensile Testing Normal Elastic Strain Y Axis Video*.mp4. Uncheck the boxes for Minimum [m/m] and Average [m/m] under Tabular Data.

Select the row number in Tabular Data corresponding to the maximum Normal Elastic Strain Y Axis . In this case, that is row number 14 at Time [s] = 5.5001e-4 s corresponding to Maximum [m/m] = 6.23e-003. Right click on row 12 and select Retrieve This Result. Drag the Maximum [m/m] data column of data towards the Time [s] column.



	Time [s]	Minimum [m/m]	Maximum [m/m]	Average [m/m]
1	1.1755e-038	0.	0.	0.
2	5.0063e-005	-1.5813e-003	1.8433e-004	-4.373e-004
3	1.0005e-004	-1.5822e-003	2.4897e-004	-3.9672e-004
4	1.5005e-004	-1.4194e-003	4.2015e-004	-3.0768e-004
5	2.0002e-004	-1.156e-003	8.4775e-004	-1.6119e-004
6	2.5001e-004	-1.0569e-003	1.5748e-003	-4.7187e-005
7	3.e-004	-1.0581e-003	2.1642e-003	2.5172e-005
8	3.5001e-004	-1.0826e-003	2.6801e-003	8.1914e-005
9	4.e-004	-1.0984e-003	3.148e-003	1.3693e-004
10	4.5001e-004	-9.4059e-004	4.3002e-003	2.9116e-004
11	5.e-004	-9.6009e-004	4.8613e-003	3.0906e-004
12	5.5001e-004	-9.7828e-004	6.23e-003	3.4729e-004
13	6.0001e-004	-9.9204e-004	4.6783e-003	3.2865e-004
14	6.5001e-004	-9.9723e-004	5.3119e-003	3.4496e-004
15	7.0001e-004	-9.9599e-004	6.1286e-003	3.6755e-004
16	7.5e-004	-9.918e-004	4.8351e-003	3.4467e-004
17	8.0001e-004	-9.8756e-004	5.404e-003	3.5154e-004
18	8.5e-004	-9.8463e-004	5.7951e-003	3.5582e-004
19	9.e-004	-9.8336e-004	4.7626e-003	3.2763e-004
20	9.5001e-004	-9.8338e-004	5.2008e-003	3.2654e-004
21	1.e-003	-9.8414e-004	5.3445e-003	3.1854e-004

Figure 7e) Maximum Normal Elastic Strain Y-axis at t = 5.5001e-4 s.



t = 0 s, 1.5e-4s, 3e-4s, 4.5e-4s, 6e-4s, 7.5e-4s, 9e-4s, 1e-3s

Figure 7f) Directional deformation Y-Axis and normal stress at different times

8. Double click on Directional Deformation under Solution in the Outline. Select the Time [s] column, right click and select Copy Cell. Open Excel and paste the data in the first Excel column. Include a heading labeled Time [s].

   Double click on Directional Deformation under Solution in the Outline. Select the Maximum [m] column, right click and select Copy Cell. Paste the data in the second Excel column. Include a heading labeled Elongation [m].

   Double click on Force Reaction under Solution in the Outline. Select the Force Reaction (Y) [N] column, right click and select Copy Cell. Paste the data in the third Excel column. Include a heading labeled Force [N].

   Double click on Normal Elastic Strain under Solution in the Outline. Select the Maximum [m/m] column, right click and select Copy Cell. Paste the data in the fourth Excel column. Include a heading labeled Strain.

   Double click on Normal Stress under Solution in the Outline. Select the Maximum [Pa] column, right click and select Copy Cell. Paste the data in the fifth Excel column. Include a heading labeled Stress (Pa).

Time (s)	Elongation (m)	Force (N)	Strain	Stress (Pa)
1.18E-38	0	0	0	0
5.01E-05	5.01E-03	85883	1.84E-04	4.05E+08
1.00E-04	1.00E-02	2.07E+05	2.49E-04	4.26E+08
1.50E-04	1.50E-02	2.20E+05	4.20E-04	4.50E+08
2.00E-04	2.00E-02	1.65E+05	8.48E-04	4.43E+08
2.50E-04	2.50E-02	88779	1.57E-03	4.41E+08
3.00E-04	3.00E-02	21247	2.16E-03	4.36E+08
3.50E-04	3.50E-02	-19133	2.68E-03	4.33E+08
4.00E-04	4.00E-02	-33750	3.15E-03	4.32E+08
4.50E-04	4.50E-02	-36375	4.30E-03	7.57E+07
5.00E-04	5.00E-02	-17505	4.86E-03	8.66E+07
5.50E-04	5.50E-02	-2241.6	6.23E-03	9.13E+07
6.00E-04	6.00E-02	7355.7	4.68E-03	8.94E+07
6.50E-04	6.50E-02	10501	5.31E-03	8.55E+07
7.00E-04	7.00E-02	9056.7	6.13E-03	8.20E+07
7.50E-04	7.50E-02	5462.5	4.84E-03	7.94E+07
8.00E-04	8.00E-02	1781.4	5.40E-03	7.80E+07
8.50E-04	8.50E-02	-844.73	5.80E-03	7.74E+07
9.00E-04	9.00E-02	-2032.3	4.76E-03	7.73E+07
9.50E-04	9.50E-02	-2049.6	5.20E-03	7.74E+07
1.00E-03	0.1	-1417.3	5.34E-03	7.76E+07

Figure 8a) Data from Ansys simulation

Select the Strain and Stress (Pa) columns in Excel. Select Insert>>Charts>>Scatter>Scatter from the menu. Label the horizontal axis as Strain and the vertical axis as Stress (Pa). From the raw data it is clear that we can delete the last twelve rows in the Excel data file. Plot the same graph once again after data reduction. Insert a line from the origin to the next point as shown in Figure 8c). Use Insert>>Illustrations>>Shapes>>Line from the Excel menu to insert the line. Drag the line between the two points. Determine the slope for this line which is Young's modulus.

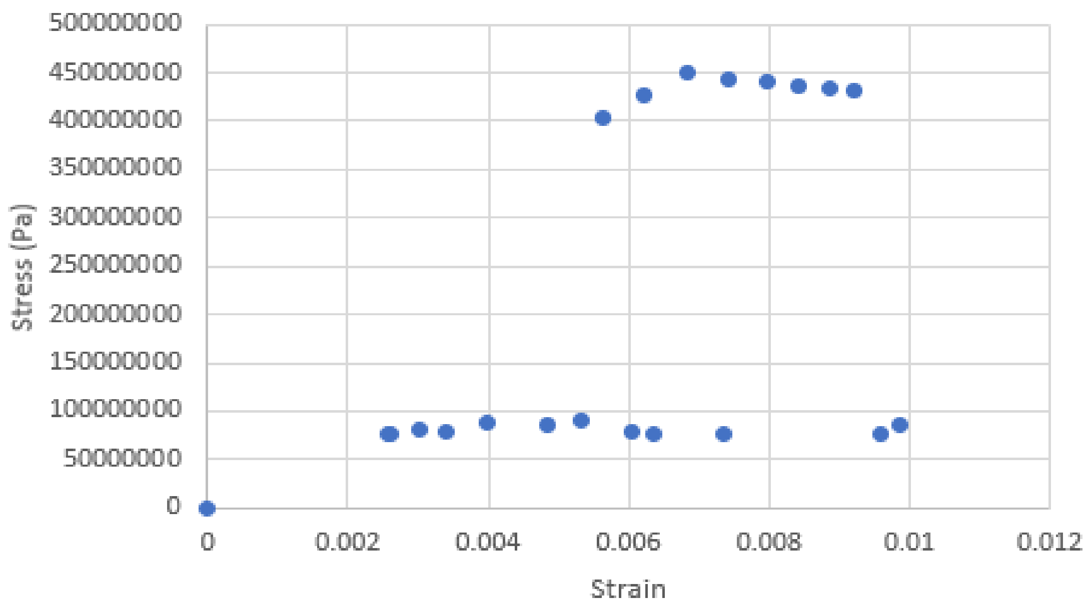


Figure 8b) Raw data of stress (Pa) versus strain from Ansys simulation.

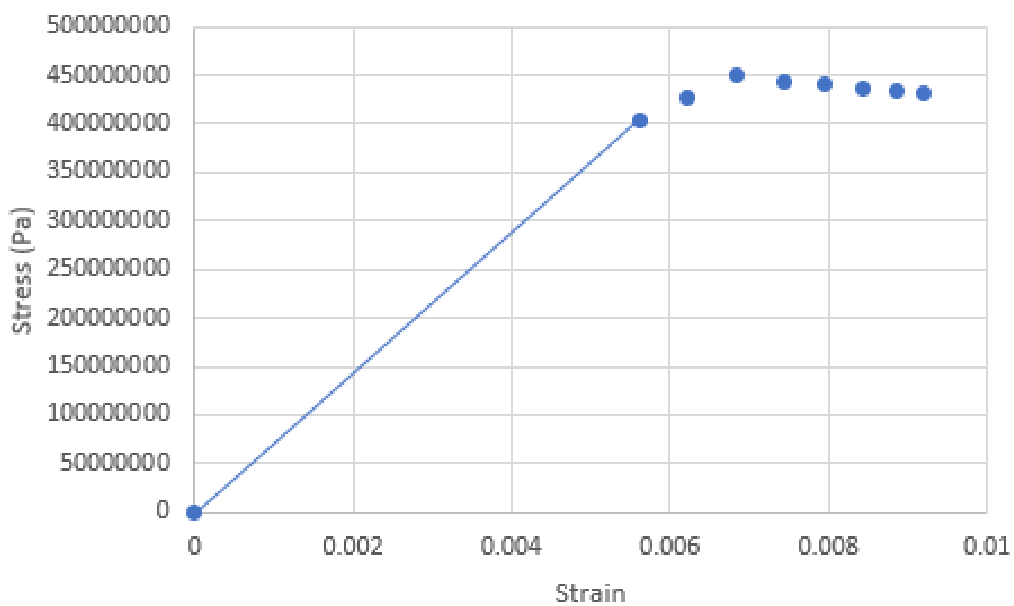


Figure 8c) Stress (Pa) versus strain for test rod after data reduction.

From figure 8c), we find that Young's modulus is 71.9 GPa (the slope of the line) with 4.35% difference from the value 68.9 GPa given by MatWeb https://www.matweb.com/ for Aluminum 6061-T6, see Aluminum 6061-T6; 6061-T651

To determine Young's modulus *E*, enter =E3/D3 in cell F2 and label this column *E* (Pa). To determine percentage error, enter =100*ABS(F2-68900000000)/68900000000 in cell G2 and label this column % diff. The ultimate strength from stress strain diagram is 0.45 GPa and fracture stress is 0.43 GPa.

F. Additional Materials Study

We will now perform a study using the tensile testing rod, but utilizing Brass, Stainless Steel, and Low Carbon Steel as materials for the testing rod.

9. The first step for completing this is to close Ansys Mechanical and navigate to Ansys Workbench.

Click on A2: Engineering Data tab and double click on Explicit Materials. In the Outline of Explicit Materials, click on the plus sign next to SS 304 **(NOT SS-304).**. You will need to add Brass and Low Carbon Steel but that will be accomplished later.

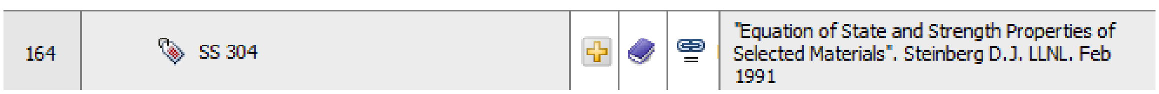
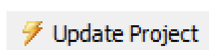


Figure 9a) Materials to Select for Additional Study.

Click on Update Project. 

. Once the project updates, double click on setup in the project schematic. This will open Ansys Mechanical.

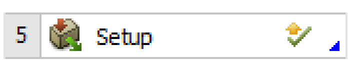


Figure 9b) Setup Option.

If an Ansys Workbench query that says "Upstream data needs to be re-read. Would you like to read the upstream data?" Click No.

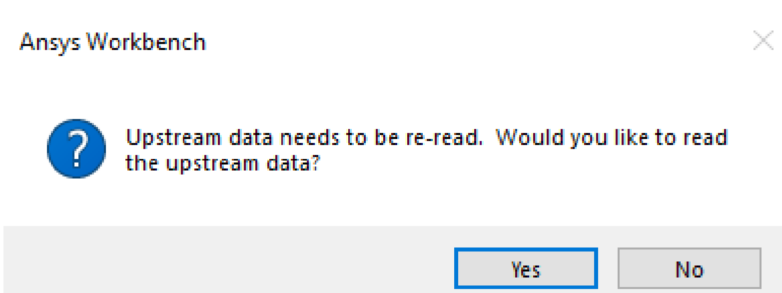


Figure 9c) Possible Query when starting Ansys Mechanical.

Verify that the new materials are available under the Materials tab in the outline. If not, right-click on Materials, and type in the name of the missing material(s) in the Engineering Data: Material View and add them using the menu. This is how brass will be added. Type Brass, C36000 into the search bar. Hover over the brass option and click the plus sign to add the material. To add Low Carbon Steel, type Low Alloy Steel, 300M (low carbon) and click on the plus sign.

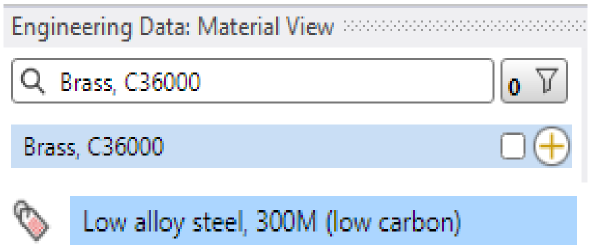


Figure 9d) Additional way to add materials.

Complete the same steps for the 6061 aluminum with the other three materials to build the stress strain curves. To complete this reassign the material by clicking on Correct Rod-FreeParts|Fillet1 under the geometry tab in the outline. Click on Assignment in Details of "Correct Rod-FreeParts|Fillet1" and assign the new material and click solve. Then perform the above steps for the three new materials.

G. Theory

10. We define stress as $\sigma = \frac{F}{A}$ where F (N) is the force and A (mm^2) is the cross-sectional area of the test specimen. Strain is defined as $\varepsilon = \frac{\Delta L}{L}$ where ΔL (mm) is the change in length and L (mm) is the original length of the test section of the test specimen.

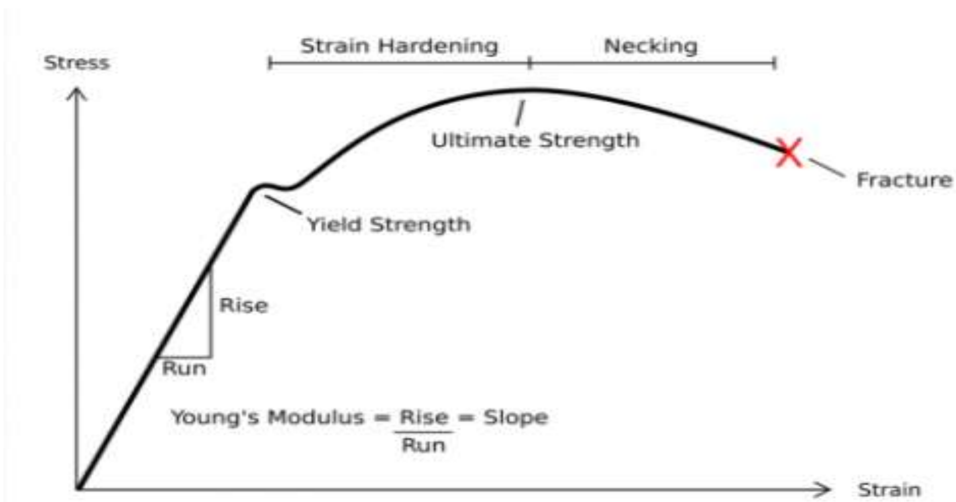


Figure 10 Stress – strain diagram for Aluminum

H**. Reference**

1. Pham, Q and Trinh, H. T., Simulation and Experimental Test in Tensile Behavior of Austenitic Stainless Steels, Advances in Materials, 8(3): 108-111, 2019.

Appendix 3: Table with Project Cost for Tensile Testing Project

Vendor	**Item Description**	**Item# or Model#**	**Price**
University	Specimens	N/A	$500.00
Ebay.com	Extensometer	N/A	$175.00
Micro-Measurement	Strain Gages and Equipment for Application	250UWA and M-Bond Equipment	$500
Total			**$1175.00**