

1 332 Laboratories

January 11, 2021

Contents

1 332 Laboratories	1
L1-C: Cantilever Beam	1
L3-C: Stress Concentrator	5
L3-E: Mechanical Testing of Materials - Lab and Final Project	14

L1-C: Finite Element Modeling of a Uniformly Loaded Cantilever Beam

The embodiment of stresses and strains in a cantilever beam is well known from solid mechanics. A simple but powerful theory used to explain this behavior is [Euler-Bernoulli beam theory](#), or E-B theory, a model that, while extremely useful, is limited. For example, it does not account for large or plastic deformation, transverse shear strain, or Poisson contraction. You have likely encountered this theory in CIV_ENG 216, if you've taken the course.

In this exercise, you will use the finite element method (FEM) to model a rectangular cantilever beam under load. You will use COMSOL to explore the stresses and strains present in the beam, compare your model to E-B theory, and use engineering principles to optimize the cantilever design under operational constraint.

Outcomes

- Develop basic FEM models (defining geometries, loads, boundary conditions, and meshes) using COMSOL.
- Use computational solvers to calculate and subsequently visualize stress and displacement fields.
- Compare computational and analytical results, noting the strengths and weaknesses of each model.
- Use your results to adapt the cantilever design to achieve a performance tolerance using reasonable constraints.

Directions

Use COMSOL to model a cantilever beam loaded on its top surface by a load P_z , as shown in [Figure 1.1](#), below. Step-by-step directions are supplied in the walk-through handout.

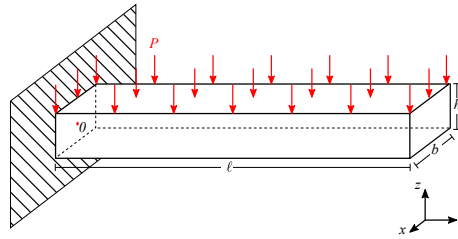


Figure 1.1: A cantilever beam with a uniform load on the top surface. Dimensions not to scale.

The beam is made of a solid, linear-elastic material with various mechanical properties: elastic modulus, Poisson’s ratio, and yield strength. Its dimensions are length L , height h and base b . The beam is loaded with a uniform pressure of P_z on the surface. We will model the cantilever beam to be affixed to the wall at one end (displacements of nodes at $x = 0$ are $u = v = w = 0$). We define our origin (point 0) as the point at the beam’s centroid and the interface with the wall.

Using COMSOL, we’ll first use linear hexahedral elements with “normal” resolution) to solve for, visualize, and interpret the displacement and stress fields within the beam. We’ll then compare these values to E-B theory. Finally, you will use parametric design techniques to design a stiff, lightweight composite beam.

Refer to the walk-through provided in class to construct your model and perform the computational analysis. Don’t hesitate to ask questions if you get stuck.

Exploration of the Model

Work in groups to compose a coherent, narrative, journal-style report of your results. Work through the following **Parts** to guide your investigation of the model.

- Part 1: Describe (using both words and figures) the u , v , and w (these are expressions u , y , and w in COMSOL) displacement fields present in the deformed beam. Explain the sources of the features of these fields and their pertinent features.
- Part 2: Describe (using both words and figures) the σ_{yy} (`solid.sy`) and von Mises (σ_v , `solid.mises`) stress fields present in the deformed beam. Explain the similarities and differences between these fields. Do you expect this beam to yield?¹

¹It’s my understand that you’ll be seeing yield criterion in later lectures. To determine whether the beam will yield, find if the maximum von Mises stress ($\sigma_v > \sigma_y$). This will shift you into the plastic regime of the stress-strain curve.

- Part 3: Plot and discuss the FEM-computed and the Euler-Bernoulli displacement as a function of distance from the fixed end (y -direction). The beam deflection in the z -direction as a function of distance is

$$w(y) = \frac{\omega y^2}{24EI} (y^2 + 6\ell^2 - 4\ell y) \quad (1.1)$$

where $\omega = F/\ell$ in this equation is the applied force divided by the length of the beam ℓ . The beam and $I = \frac{bh^3}{12}$ is the second moment of inertia. Run this model for both linear shape functions and quadratic Lagrange shape functions. What do you observe in each case compared to the analytical result?

- Part 4: Plot and discuss the FEM-computed and the Euler-Bernoulli axial stresses (σ_{yy} , or `solid.sy` in COMSOL) in the tensile region of the beam in the y -direction. Use quadratic Lagrange elements. The analytical expression for the axial stress, σ_{yy} , at the topmost xy -face of the beam is:

$$\sigma_{yy}(y) = \frac{\omega(\ell - y)^2}{2Z}, \quad (1.2)$$

where $Z = \frac{bh^2}{6}$ is the section modulus.

- Part 5: Referring to your results in Parts 3 and 4, identify any major deviations between the FEM results and the results from Euler-Bernoulli theory. Identify two possible sources reasons for any derivations you may see. Hint: Both E-B theory *and* your FEM model may have shortcomings. Consider them both.
- Part 6: You are charged designing this beam for a use in an engineering application and you want to reduce its weight as much as possible without resulting in yield (while accounting for a safety factor) and limiting overall displacement. You decide to replace a portion of the beam with a more compliant material. Based on your earlier results, design a new composite beam which compiles to these restrictions. Describe your engineering approach and your final design, as well as how much you reduced the weight of the component.

• **Part 6 Hints:**

1. Use Quadratic Lagrange elements. Find this under `Solid Mechanics` → `Discretization` → `Quadratic Lagrange`
2. Remember the *layers* option you encountered during geometrical construction — this is a great area to explore. You are, of course, free to consider other geometries.
3. Look at the von Mises stress field. Where can you tolerate a more compliant material?

4. You may have to remesh or re-apply boundary conditions.
5. Parametrization studies (Study \rightarrow Parametric Sweep) are the best way to approach this problem. Figuring out a good optimization algorithm may save you time and may provide you with the best answer. Google is your friend here.
6. You can analyze values from a parameter sweep in the Results tab. Check this out:
<https://www.comsol.com/video/postprocessing-results-parametric-sweep>
7. It is very easy to copy and modify an COMSOL model!

MAT SCI 332 FEM Report Guidelines

Your formal report should contain:

1. A brief introduction describing your work and providing context for your study.
2. A methods section including all information necessary for the reader (assume they know how to operate COMSOL).
3. A results/discussion section in which you describe *relevant* results, analyze the strengths and shortcomings of your FEM results and how they compare to E-B theory, and present your newly designed composite beam.
4. A brief conclusion summarizing results and generalize the importance of your study.

I am looking for accuracy and deliberation in your modeling and thoughtful consideration of the results. I am not particularly concerned the length of your report, but about 5 pages of text (not including figures) is typically sufficient. As is true for any report you turn in, your work should be concise, readable, well cited, possess clear, informative figures. For technical writing, I strongly recommend the use \LaTeX for good typesetting, figure presentation, and equation writing. [Overleaf](#) is a powerful online \LaTeX editor that you can use for co-authoring in small groups.

L3-C: FEM of a Stress Concentrator

In this assignment, you'll assess the stress fields in a simulated section of the of an aluminum (Young's modulus $E = 70$ GPa, Poisson's ratio $\nu = 0.2$) ASTM tensile specimen. Here we investigate the stress concentration effects of a 5 mm diameter (d) hole.

You will construct the finite element problem in Fig 1.2, below. In COMSOL, define your origin to be center of the stress concentrator (the hole). The experimental sample can be adequately approximated by approximating the specimen as a 2D plate of length length (l) \times width (w) \times thickness (t) of 60 mm \times 20 mm \times 3.2 mm. We will take advantage of specimen symmetry and Saint-Venant's Principle and further reduce our model to a single quadrant of the rectangular section of the gauge (dimensions shown in Fig. 1.2). We will assume a plane-stress condition and operate in 2D. We'll investigate the stress distribution near the hole during ramping a load of $\sigma_{App} = 0$ MPa to 135MPa and back to 0 MPa.

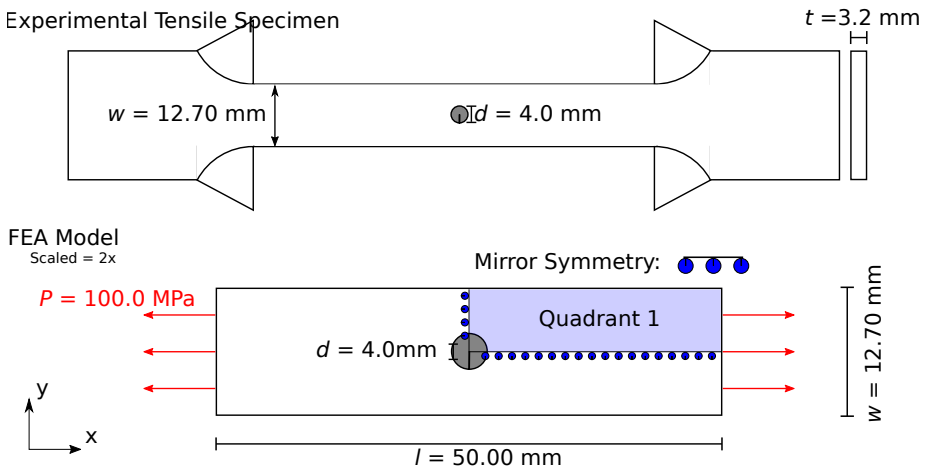


Figure 1.2: (Top) The ASTM dogbone specimen with a $d = 5$ mm hole drilled through the center of the gauge. (Bottom) The finite element model limits the calculation to the center of the gauge and takes advantage of mirror symmetries.

Part I

Use COMSOL and the walk-through to formulate the model in Fig. 1.2 using a 2D representation. Note that plane stress conditions (the principle stress in the z -direction is zero) applies when the plate thickness is small compared to the other plate dimensions, as is the case here. We seek to find the stress

distribution, and in particular, the magnitudes and positions of the maximum and minimum normal stresses, $\sigma_{xx(max)}$ and $\sigma_{yy(min)}$ respectively.

- Initially, mesh the model using linear, free triangular elements and predefined “normal size” elements (minimum element size should be ~ 0.01 mm by default). Solve the problem. At the maximum applied load: $t = 1.1$ and $\sigma_{App} = 135$ MPa briefly describe — in figures and words — the:

- Overall displacement field (`solid.disp`)
- σ_{xx} stress field (`solid.sx`)
- σ_{yy} stress field (`solid.sy`)

If you smooth or scale your result for presentation purposes, be clear about this in your report.

Be sure to consider:

- Where does the maximum value of $\sigma_{xx(max)}$ occur? Report the ratio $\sigma_{xx(max)}/\sigma_{App}$.
 - Where does the minimum (largest negative/compressive) value of $\sigma_{yy(min)}$ occur? Report the ratio $\sigma_{yy(min)}/\sigma_{App}$.
 - Our specimen is a decent approximation of a stress concentrator in an thin, infinite plate under uniaxial far-field tension. In an ideal “infinite plate”, maximum $\sigma_{xx(max)} = 3\sigma_{App}$ and $\sigma_{yy(min)} = -1\sigma_{App}$. How does our FEM analysis compare?
- In addition to the meshing parameters employed in Part 1(a), explore the parameters listed below:
 - Total number of elements
 - the estimated time and virtual memory required to complete the job
 - $\sigma_{xx(max)}$
 - $\sigma_{yy(min)}$.
 - Element type: linear triangular Min/max element size: 4/5
 - Element type: linear triangular Min/max element size: 0.4/0.5
 - Element type: linear triangular Min/max element size: 1/2
 - Element type: linear triangular Min/max element size: 0.1/0.2
 - Element type: linear triangular Min/max element size: 0.8/0.9
 - Element type: linear triangular Min/max element size: 0.04/0.05²

In your report, provide two plots, one of $\sigma_{xx(max)}/\sigma_{App}$ vs. total number of elements and one of $\sigma_{yy(min)}/\sigma_{App}$ vs. total number of elements. Comment on the trends.

3. For a finite-width plate of some thickness, t , the empirical solution for the maximum stress (Roark's Formulas for Stress and Strain, 8th Ed.), $\sigma_{xx(max)}$ is

$$\sigma_{xx(max)} = K_t \sigma_{nom}$$

where

$$K_t = 3.000 - 3.140\left(\frac{d}{w}\right) + 3.667\left(\frac{d}{w}\right)^2 - 1.527\left(\frac{d}{w}\right)^3$$

is the stress concentration factor and

$$\sigma_{nom} = \frac{\sigma_{App} w}{(w - d)}$$

is the nominal stress due to the reduced cross-sectional area that arises from the defect.

- Using these equations, find the analytical value for the maximum stress, $\sigma_{xx(max)}$.
- Now, create an efficient mesh (i.e., as few domain elements as possible) using the knowledge gained from your preliminary meshes and mesh refinement techniques to converge your solution to within 1.5% of the analytical solution.

Challenge Details: Turn in your .mph file for this mesh.

- Use your coarse result to inform your meshing. There are numerous tactics. Explore:
 - Partitioning:** creating sub-domains with different element densities. To do this, navigate to the **Geometry** tab and create new shapes and use **Partition Object** to create sub-domains.
 - Change Mesh Sizes:** you can define the size of the mesh in various sub-domains by adding different sizes in each domain. Add **Sizes** under **Mesh 1** to try this.
 - Meshing Algorithms:** you can control mesh densities by employing various meshing algorithms such as free-quad, mapping, or boundary layers. To do this, navigate to the **Mesh** tab and explore these possibilities.
 - Mesh Distribution:** you can define nodal density functions using **Distribution** controls. Add these to a meshed domain and control **Number of Elements** to define node distributions. Press **F1** when hovering above the **Distribution** button for full descriptions
- Meshes must be convergent. It is possible to get a result for an unconverged mesh that is in the 1.5% of the target value without being converges.

- You must use linear shape functions: i.e. Solid Mechanics
→ Discretization (you will need to activate this, like you did in the previous lab) → Displacement Field → Linear.
- Algorithmic mesh refining is not allowed.
- The group with the most efficient mesh (fewest elements) will receive 2 *e.c. points on their assignment*.
- If you beat my mesh, you get 4 *e.c.* (I'm undefeated, but some students have come close!).

Part II

The analysis in Part 1 yields stress concentrations higher than the yield strength of our material (~243MPa). This makes a linear-elastic model insufficient to correctly describe the stress and strain state.

1. Enable plasticity. Rerun the model using linear triangular elements and 0.09/0.1 mm min/max size for your elements.
2. Show plots of σ_{xx} , σ_{yy} , and effective plastic stain (solid.epε) — a measure of permanent plastic deformation at relevant time steps.
3. What do you conclude about the effect plasticity has on stress concentrations after yielding occurs?

Step-by-Step Directions

The model for the formulation of stress concentrator is a bit complex. To formulate the basic model (you will adapt it), follow the step-by-step directions below.

Open COMSOL. Select the Model Wizard
Model Wizard

1. In the Model Wizard, click on 2D.
2. In the Select Physics Tree, select Structural Mechanics → Solid Mechanics (solid)
3. Click Add
4. Click Study
5. In the select Study tree, select Preset Studies → Stationary
6. 6. Click Done

Discretization

First, we need to set up the element type. Quadratic elements, while powerful, are not instructive here. We'll use linear elements.

1. Make sure you can see discretization options. Go to the eyeball-looking thin under Model Builder and ensure Discretization is enabled.
2. Click on Solid Mechanics (*solid*).
3. Navigate to Discretization and change the Displacement field to Linear.

GEOMETRY 1

1. In the Model Builder window, under Component 1 (comp 1) click Geometry I
2. In the Settings window for Geometry, locate the Units section.
3. From the Unit Length list, choose mm
 - (a) Rectangle I (r1)
 - (b) Right-click on the Geometry branch, select rectangle.
 - (c) In the Settings window for the rectangle, locate the Size and Shape section.
 - (d) For Width, type 25.
 - (e) For Height, type 6.35.
 - (f) Build Selected
 - (g) Is a corner of the rectangle at the origin?

Circle I (c1)

- (a) Right-click on the Geometry branch, select circle.
- (b) In the Settings window for the circle, locate the Size and Shape section.
- (c) For the Radius, type 2.
- (d) Is the circle's center at the origin (0,0)? It should be.
- (e) Build Selected

Difference I (dif1)

- (a) In the Geometry toolbar, select "Booleans and Partitions" and choose Difference.
- (b) Select the rectangle only.
- (c) Active the Objects to subtract "active" toggle button.

- (d) Select the circle only.
- (e) Build All Objects

We're going to cycle a load on the tensile specimen. To do so, we need to define an auxiliary sweep to prescribe applied loads at different timesteps. By first, we need to initialize the time step parameter.

Parameters

1. In the Home toolbar, click Parameters
2. In the Setting window for Parameters, locate the Parameters section and enter:

Name	Expression	Value	Description
para	0	0	Load parameter

Alright, now we need to define the load function. We'll use an interpolation function to prescribe a linear ramp from 0 MP to 100 MP from "time" 0 to 1.1, and then unload from "time" $t = 1.1$ to 2.2.

Interpolation I (loadfunc)

1. On the Home toolbar, click Functions and choose Local→Interpolation
2. In the Settings window for interpolation locate the Definition section
3. In the Function name text field, type loadfunc
4. In the table below, enter the following settings:

t	f(t)
0	0
1.1	100
2.2	0

5. Locate the Units section and define Argument as 1 Function as MPa.
6. Plot this function. This is a graph of "time" vs applied force/unit area. What does this mean?

SOLID MECHANICS (SOLID)

1. In the Model Builder window, under Component 1, navigate to Solid Mechanics.
2. In the Settings window, locate the 2D Approximation section.
3. From the list, choose Plane stress³

³We assume the stress vector is zero perpendicular to the plane. This is appropriate for flat plates in which the load is applied within the plane.

4. Locate the thickness section. Define d to be 3.2[mm]. Note, include the unit — for some reason COMSOL has hard-coded in m for this field.

Linear Elastic Material 1

1. In the Model Builder window, expand the Solid Mechanics node, then click Linear Elastic Material 1.
2. On the Physics toolbar, click Attributes and choose Plasticity (and check out all the other cool properties you can model!!).

MATERIALS Material 1

Lets create our own material. Note that, in principle, we could use reroved values from experiment here, but you have to take into account various considerations, which we won't consider here. Instead, we'll use the isotropic hardening model.

1. In the Model Builder window, under Component 1, right-click Materials and choose Blank Material
2. In the Settings window for Material, locate the Materials Content Section
3. Enter the following setting in the table:

Property	Name	Value	Unit	Property Group
Young's modulus	E	70e9	Pa	Basic
Poisson's ratio	ν	0.2	1	Basic
Density	ρ	7850	kg/m ³	Basic
Initial Yield Stress	σ_y	243e6	Pa	Elastoplastic material model
Isotropic tangent modulus	E_t	2.2e9	Pa	Elastoplastic material model

4. In the Model Builder window, under Component 1, under Solid Mechanics, under Linear Elasticity 1, right-click Plasticity 1 and choose Disable. (You'll re-enable this later.)

SOLID MECHANICS (SOLID)

1. In the Physics toolbar, click Boundaries and choose Symmetry. (You have to decide on these yourself.)
2. Select the correct mirror symmetry boundaries. Test different boundary models if you aren't sure what this will do.
3. In the Physics toolbar, click Boundaries and choose Boundary Load.

4. Select the correct load boundary. (you have to decide on these yourself.)
Not — the load is mirrored but the magnitude is defined by your defined load: that is, the total load on the system is 100MPa.
5. In the Settings window for the Boundary Load, locate the Force section.
6. Specify the F_A vector as

loadfunc(para)	x
0	y

para will be defined later.

SOLID MECHANICS (SOLID)

1. Under Component 1, right click Mesh 1 and choose Free Triangular.
2. Click Build All.

STUDY I

Here, we'll set up an auxiliary sweep of the para parameter.

1. In the Model Builder window, expand the Study 1 node and click Step 1: Stationary.
2. In the Settings window for Stationary, click to expand the Study extension section.
3. Locate the Study Extensions section. Select the Auxiliary sweep check box.
4. Click Add (the blue + sign below the table). The parameter para will be initialized.
5. In the table, enter the following values (Dont mess up the syntax, or it will turn red):

Parameter name	Parameter value listed
para	0 range (0.44, 0.05, 0.59) range (0.06, 0.05, 1.09) range (1.1, 0.2, 1.9) range (1.95, 0.05, 2.2)

6. This defines a list of numbers ("times") for which calculations will be made. For example range (0.44, 0.05, 0.59) makes the list 0.44, 0.49, 0.54, 0.59.
7. In the Model Builder window, click Study 1
8. In the Setting window for Study, type Isotropic Hardening in the Label text field.

9. On the Home toolbar, click Compute .
10. Boom! (Hopefully?)
11. You may see an all-green component. Look at what you are plotting (note the $\text{para}(26) = 2.2$ value). This is the last step in the load function. Why would this be green at this point? (Look at the legend.)
12. Look at some other para values. Or, create an animation!

L3-E: Mechanical Testing of Materials - Lab and Final Project

Overview

Description

Testing of materials to acquire mechanical properties is a critical skill for the materials scientist or engineer. Assessing the mechanical properties of a material is a necessary tool in determining whether a material is suitable for its expected application. Since this lab will be the basis for your final project, it will be treated as a structured research assignment, consisting of a white paper proposal, material testing, data analysis, and presentation of your results (both written and oral).

Objectives

- Research and identify the test protocols necessary to investigate your material.
- Gain experience with mechanical testing of a material or set of materials.
- Determine mechanical properties from the analysis of tensile, compression, indentation, bending, and/or impact experiments.
- Assess if the results from experiments to answer the question proposed in your white paper.
- Present your results through both written and verbal communication.

Materials

Each lab group (3 students per group) is requested to source their own materials. These can be everyday materials (coins, cellphone components, automotive components, earplugs, food items etc.), or materials from your research groups. However, if you do not have samples available through a research group or are finding selection of another material challenging, there are some options available from groups in the MSE department. These specimens are limited, so let us know as soon as possible of your interest in using a sample set by emailing [Gwen](#) to claim that sample.*

- Epoxy samples - Shull group
- Thin film coatings - Chung group
- Bio-inspired samples - Joester group
- Metal Foams (McMaster)**
- Porous Conductive Carbon (McMaster)**

- Miscellaneous tensile or bending samples from CLAMMP (metals, polymers, and wood being the most common).*** Please contact [Carla Shute](#) for more details.

*If your entire group is unable to participate in the in-person component of labs, please email Gwen and Dr. Shull as soon as possible to discuss potential options for a virtual final project.

**These samples need to be purchased and require as much lead time as possible.

***If you would like to use these samples, please let Gwen and Carla know with the material class you're interested in.

Instrumentation

In order to help us prepare the in person component of the lab, you are expected to propose which tests and ASTM standard(s) would be most useful to probe the mechanical properties of your material. The Sintech 20G instrument in CLAMMP can perform tensile, bending and compression measurements. Charpy impact tests will be performed in CLAMMP, but require a specific sample geometry for the experiment. Indentation tests will be performed in [MatCI](#) using the Struers Duramin 5. Depending on the type of material and sample geometry, you will need to use between 1 and 3 different instruments to fully investigate your material. If you have specific questions about a test or an ASTM standard, please reach out to [Carla Shute](#).

Timeline and Assignment Expectations

White Paper

The white paper is due **Jan. 29th, 2021 at 11:59 pm** as a *Canvas* submission. The white paper should provide the following information:

- Who is in your lab group
- What sample you are planning to test
- Significance of the material (i.e., what is its application)
- What research project do you propose (material property, variation between samples, effect of solvent, characterization of unknown samples)
- What instrumentation you will need to determine the mechanical properties
- ASTM standard(s) that you will need for your instrumentation
- Availability for testing over the data collection period (February 8-19)

Your white paper should be between 1/2 and 1 page in length.

Lab Data Collection

In person data collection will occur from **Feb. 8-19**. Due to Covid safety protocols, only 1-2 students will be able to participate in a lab session at a time. Please be prepared to take detailed notes about your experimental setup, data collection, and any preliminary findings from the tests. These will be important to share with your other group members and for your final report.

Status Update

To help track progress on your final projects and provide guidance for your data analysis, a **Status Update is due on February 26th, 2021 at 11:59 pm** as a *Canvas* submission. The status update should include:

- Your sample material
- The experiments you ran on your material (instrumentation and relevant protocol information)
- Preliminary data analysis (i.e., stress-strain curves for tensile testing, hardness and modulus data for indentation tests)
 - All figures and tables shown in this section should be labeled with proper units and axes!
- Any questions you may have about your data

This portion of the final project is an informal write up to ensure that you are on track for finishing your project by the end of the quarter and should be between 1 and 5 pages in length, depending on how many figures you need to include for your data analysis.

Final Report

The final report is due on March 19th by 9 am as a *Canvas* submission. The report should be a formal report 9-12 pages in length and include the following sections:

- Background - what is your material, how is it used, and why is it important? Include citations in this section.
- Experimental methods - What instrumentation did you use to test your material? What were the experimental parameters and protocols used? Provide enough detail so that a knowledgeable researcher could reproduce your work.
- Data analysis - Present the data from your experiments, providing the mechanical properties of interest for your sample.

- Discussion - What were significant findings from your data? Did you have to make any assumptions in your data analysis (i.e., account for compliance of the instrument)? How complete is your dataset? Is there another test that could be performed and provide more information about the mechanical properties of the material?
- Conclusion - based on your results, is your material well suited for its given application? Are there any improvements that can be made to the material? If you were working with unknowns, were you able to successfully identify your samples?

Final Presentation

The final presentations will take place on **Friday, March 19th, 2021 from 9-11 am**. Presentations should be between 1 and 12 minutes in length and cover the following information:

- Your material and the projected application of the material
- How you investigated your material - instrumentation and ASTM standard(s)
- Overview of results - were they what you expected?
- Significance of the results - summarize your conclusion

Everyone in the group must present during the course of the presentation. Please ensure your labels and text on figures and tables are legible and that any information not directly from your results is cited.

Assessment

Each portion of the final project will be assessed for completeness (addressing all components for each section) and connection to the course material. Towards the end of the quarter, self assessments will be distributed to allow you to each provide feedback about your and your partners' contributions to the project during the quarter.

Best-practices Reminders

- Please use SI units.
- Appropriately caption and label all figures and tables.
- Please cite any sources.
- Upload your reports to *Canvas* as a .pdf.