ASEN 3112 STRUCTURES

Lab 2 DESCRIPTION

Spring 2020

Version: February 14, 2020

I Summary

The second lab assignment combines computer and experimental work. The goal of this lab is to illustrate the use of experimental measurements and numerical analysis together to gain insight into the mechanical performance of a structural system. It involves the static load test of a 16-bay space (3D) truss in a beam-like support-loading configuration. Measurements are to be compared with computer results obtained using a general-purpose, commercial structural analysis program (ANSYS), which is based on the Finite Element Method (FEM). The FEM model is to be used to quantify the impact of modeling assumptions and uncertainties in geometry, material properties, and measured data.

This document provides general information for this lab. **It does not include tutorials for use of ANSYS, which are provided separately on CANVAS**. Specialized information is provided in the Addenda to this document.

The learning objectives of this lab are to:

- **Experimentally assess** the assumption that the displacement and force measurements of the real-world truss vary linearly with the external loads.
- **Contrast** experimental and FEM-simulated values for the truss displacement and forces.
- **Identify** assumptions made when modeling the real-world truss in FEM software and **analyze** the assumptions' influence on the simulation results.
- **Design, implement**, and **analyze** FEM simulations that provide a more realistic model of the real-world truss.
- **Calculate** and **justify** a factor of safety for the truss.

II Timetable, Groups, and Logistics

II.1 Timetable

This lab spans six weeks (including spring break), with reports due on Thursday, April 2, 2020. The experiment demonstration will be held in the Co-PILOT during normal recitation times on February 20, 2020. Section 011 will meet from 9:00 - 10:20 am, and Section 012 will meet from 2:30 - 3:50 pm. Students must attend the experiment demonstration during their respective recitation times. Attendance to a different section needs prior written approval from the instructors and will only be granted on account of a bona-fide reason, such as a medical emergency. In case of an emergency or unavoidable absence, students should contact the instructors as soon as possible. Attendance is mandatory and will be taken at the start of each demo.

ANSYS tutorials will be given in lieu of recitations on Thursday, March 12, 2020 in the Co-PILOT. Attendance at the ANSYS tutorial session is recommended but not mandatory. The tutorial will take about 50 minutes; thus, each lab/recitation section will be split into two groups for taking the tutorial. The distribution is:

- Section 011: Students with last names starting with letters A to M will attend from 8:30 am to 9:20 am. The rest of the students, letters N to Z will attend from 9:30 am to 10:20 am.
- Section 012: Students with last names starting with letters A to K will attend from 2:30 pm to 3:20 pm. The rest of the students, letters L to Z, will attend from 3:30 pm to 4:20 pm.

Following the ANSYS tutorials, lab groups are expected to perform the computational portion of the lab described below independently. ANSYS is installed on all computers in the PILOT and co-PILOT. These labs are available during the times listed below. The PILOT hours are also available at: `https://www.colorado.edu/aerospace/current-students/manufacturing-shops/pilot-aero-141`. Note that scheduled classes have priority at the workstations.

- PILOT:
  - Monday-Thursday: 8:00 am- 5:00 pm doors open, 5:00-10:00 pm key card access only (given after taking tour). Closed after 10:00 pm to all. Scheduled classes have first priority.
  - Friday: 8:00 am- 6:00 pm. Student staffed 3:00 pm - 6:00 pm. Closed after 6:00 pm to all. Scheduled classes have first priority.
  - Saturday: Closed all day.
  - Sunday: 11:00 am - 9:00 pm key card access only.
- Co-PILOT: open during business hours Monday- Friday, no after-hours access. Scheduled classes have first priority.

There is also a free student version of ANSYS that can be downloaded on personal computers and used for this lab; see `https://www.ansys.com/academic/free-student-products`.

II.2 Groups

Sub-groups of 2-3 students have been selected in recitation, and these will be combined into lab group of 5-6 students, all in the same recitation. If you did not choose a sub-group in recitation you will be assigned to a group. Groups will be posted to Canvas before the experiment demonstration on February 20.

As in Lab 1, each group should select a leader (the Group Leader, or GL) who will have the following responsibilities:

- Divide tasks to be accomplished in writing the report (e.g. writing specific sections, analyzing data and producing necessary plots/figures).

- Compile and edit the final report (ensure consistency between sections and make sure that other member's contributions are satisfactory).
- Provide internal deadlines to group members so that the lab report project stays on schedule.
- Keep a record of delegated tasks, internal deadlines, and confirmations from team members. This record can simply be a thread of emails between the group leader and group members. This will not be turned in but will be used by the TAs to resolve any disputes about participation scores.
- Provide a participation report for your group with a brief summary of each group member's tasks, contributions, and performance as a group member. It is the group leader's responsibility to organize a peer evaluation process to determine each group member's contribution grade; see Section IV.3. Note, the group leader should not assign a participation score without the input of the entire group.

Group leaders do not have to write their own section of the report (though they can if they want to). Group members are responsible for timely communication with the group leader. If a student is assigned a task to complete with a deadline, the student should confirm that she/he will do so. If the student does not agree to the task or has difficulty with it and needs more time or help with the task, this should also be communicated to the group leader (well before the deadline).

II.3 Lab Reports

Each group must prepare and submit one copy of the report to Gradescope by 1:00 pm on Thursday, April 2, 2020. Instructions for preparing this report are given in Section IV of this document.

III Test Article and Model Description

III.1 Test Article

A photo of the 16-bay truss structure is shown in Figure 1. **SI units are ultimately used** although some dimensions and forces are initially expressed in English units. The truss consists of 16 cubic bays with joint-to-joint distances of 0.250 m (250 mm). The struts are fabricated from extruded 6061-T6 Aluminum tube stock as illustrated in Figure 2. The cross section dimensions are $d = 3/8$ in $= 9.525$ mm and $t = 1/16$ in $= 1.587$ mm. The elastic modulus for 6061-T6 Aluminum typically ranges between 69 and 70 GPa.

Applied loads and supports are illustrated in Figure 4. The button load cells used to measure the reaction loads at either end of the truss allow truss ends to rotate and therefore provide nominally pinned and roller boundary conditions. A total load of $P = 50$ lbs $= 222.4$ N will be applied with shot bags at the midspan of the truss. This can be modeled as two point loads of $P/2$ applied on the two top joints at midspan.

The internal force across one of the longeron struts on the lower side of the truss near the midspan will be measured by an inline load cell. [1] Vertical displacements will be also measured at the

[1] This is likely to be that going from node 68 to node 69 in the finite element model shown in Figure 3(b), but please check the actual set up in your experiment. Node numbers in your ANSYS model may differ. Replacing a

Figure 1: Photograph of the 16-bay truss test article.

midspan using a Linear Variable Differential Transformer (LVDT). When performing the experiment, make careful notes on the location of each of these measurements relative to the pattern of diagonals of the truss.

III.2 Note on Truss Fabrication

The joint-to-joint dimensions are **metric** because the original trusses, fabricated in 1992, were supposed to be a scaled model of the main truss components of the original International Space Station (ISS) design of 1991. This design was later modified. To reduce costs, however, struts were fabricated from commercial aluminum stock, cut and precisely machined at both ends to fit into the joint attachments. Ball joints and strut fittings were custom made at the Physics shop.

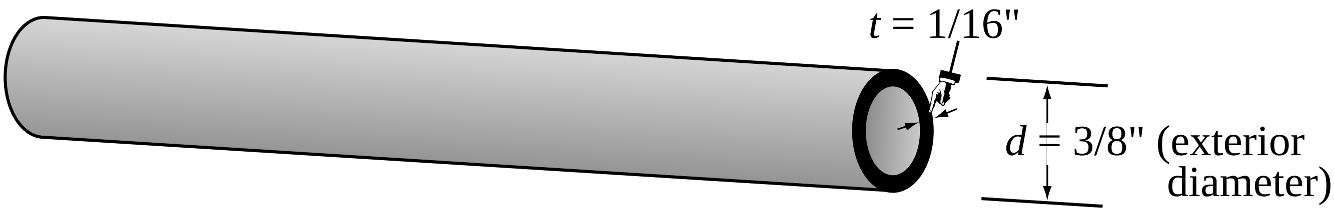

Figure 2: Truss member (strut) cross section schematics. End fitting details (threads and joint adapters) not shown.

III.3 The FEM Structural Model: Crosschecking with ANSYS

Using a truss finite element model implemented in *Mathematica* the ANSYS results were crosschecked. The full 16-bay truss is modeled via bar elements and is pictured in Figure 3(a). Node numbering details are shown in Figure 3(b) (The ANSYS node number will likely be different since the model generation methods are not the same). A typical (repeating) two-bay unit is illustrated in Figure 3(c) — the full model (in both *Mathematica* and ANSYS) is generated by repeating this unit seven times.

The physical units are metric: millimeter (mm) for lengths, Newton (N) for forces and MPa=N/mm^2 for stresses (and elastic moduli). Note that although the member cross section dimensions (d and t in Figure 2) are given originally in English units (because struts were fabricated from tube stock for the reason noted in Section 3.2 above) those should be converted to mm.

It is recommended to place the origin or coordinates (x, y, z) as shown in Figure 3(a) so that all node coordinates are non-negative. Node 1 is placed at this origin. All node locations are defined by the repeating-bay dimension $L_b = 250$ mm; see Figure 3(b). Unlike the *Mathematica* model, ANSYS does not require element numbers to be specified by the user — those are internally assigned by ANSYS in the order in which they are defined.

For the purposes of this lab we will not explicitly model the details of the mechanics of the node assemblies, for example threads and joint fittings. This potential source of modeling error, however, should be considered when comparing experimental and computational results and performing uncertainty analysis.

standard strut by the inline cell will alter somewhat the axial stiffness of that member. Since we do not have data on its effective stiffness, for the computer model one should assume a standard strut and consider uncertainty in the strut stiffness when performing uncertainty analysis.

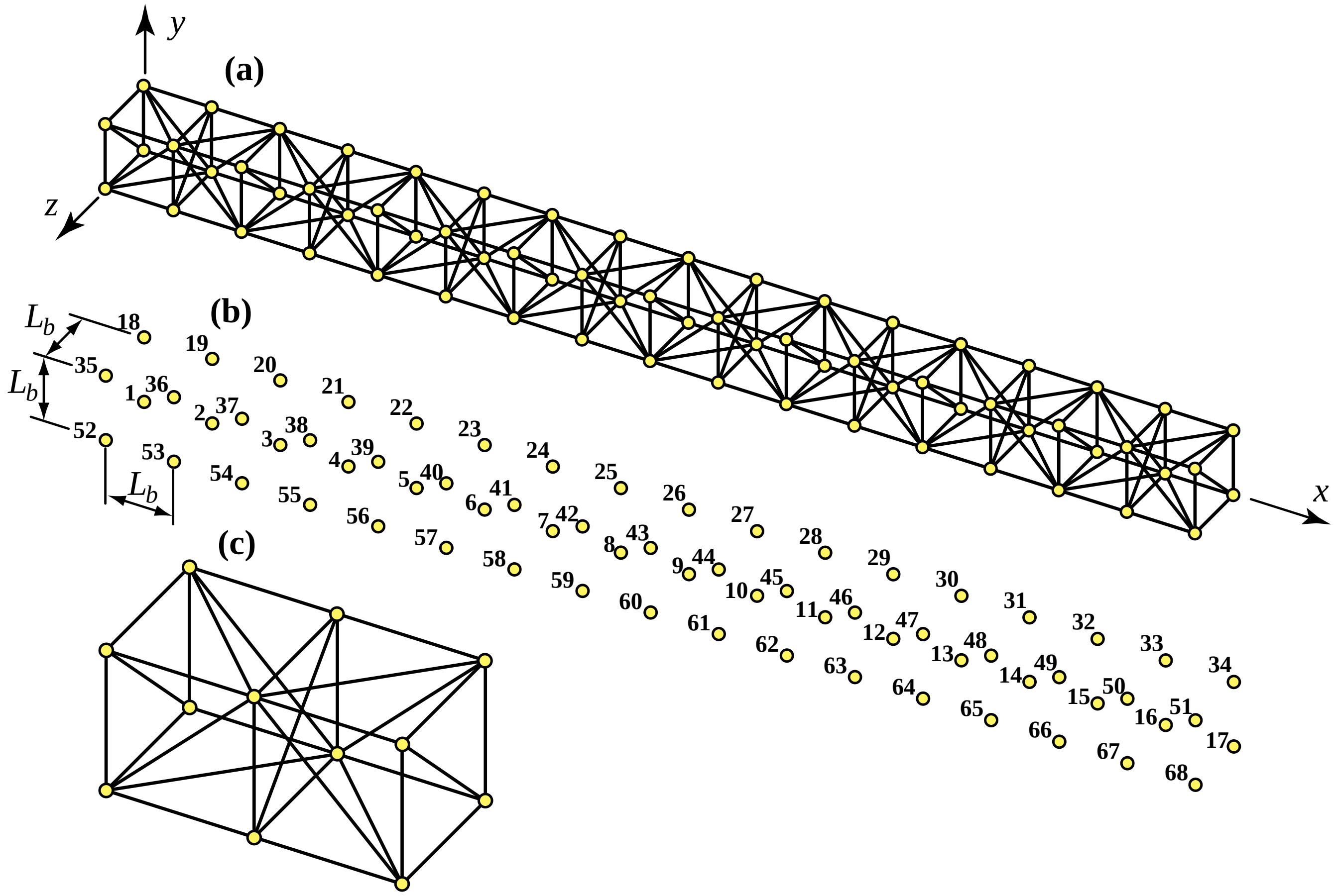

Figure 3: Schematics of the 16-bay space truss FEM model built with *Mathematica: (a) 16-bay truss showing members and nodes; (b) node numbering detail(the numbering for the ANSYS model will likely be different); (c) repeating 2-bay unit used to generate the full truss.*

III.4 Loads and Supports

The applied loads and support conditions are schematically shown in Figure 4. As previously noted, the expected maximum total load to be applied at midspan (nodes 29 and 49) is $P = 50$ lbs $= 222.4$ N. This load is physically applied using shot-put bags of 5 and 10 lbs, in any increments up to the maximum total load (i.e. time 1 - Load = 0 lbs, Time 2 - Load = 5 lbs, ... Time N - Load = 50 lbs). The truss is placed on load cell support plates at both ends.

For the finite element model it is recommended to fix the left-support nodes in all three (x, y, z) directions. The right-support nodes are fixed in the transverse (y, z) directions but allowed to move in the longitudinal (x) direction, as illustrated in Figure 4. This modeling decision approximates a friction-less support and tries to avoid buildup of spurious axial forces in the FEM model. However, friction may cause real axial forces. Thus, the influence of assumptions for modeling the boundary conditions should be studied in the uncertainty analysis.

III.5 What To Compute

In the computer lab you should record the set of node displacements and internal (axial) member forces computed by ANSYS under a load of $P = 50$ lbs $= 222.4$ N. From this set of results, you can

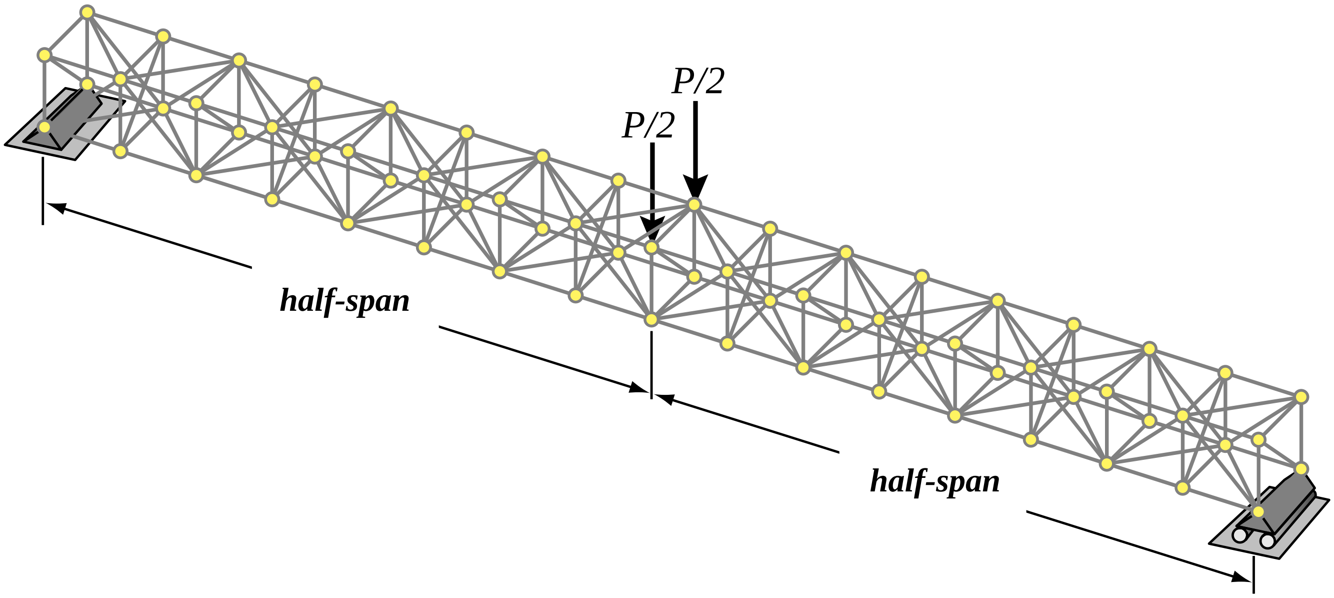

Figure 4: Applied loads and support conditions. These are idealizations of the real loads and supports: the shot-put-applied loads actually span two bays, whereas the fixed and rolling simple-support conditions idealize the actual supports.

select appropriate quantities to compare with measurements. Note: To compare predictions with a different load P, a simple scaling is sufficient because the FEM truss behavior is assumed to be exactly linear: if the load is doubled, displacements and internal forces are doubled.

III.6 Equivalent Beam Calculation

As a quick analytical check, compare the maximum ANSYS y-deflection with the midspan deflection furnished by the so-called *equivalent beam* model. This model treats the truss as beam where the moment of inertia is computed via parallel axis theorem considering only the four longeron-strut cross sections taken with respect to the neutral axis of the equivalent beam. The moment of inertia of the individual struts about their individual bending axes, i.e. the member self-inertia, is ignored. This highly simplified model ignores the presence of diagonal members. The elastic modulus is that of 6061-T6 aluminum alloy. Expect that this beam models provides a fairly good estimation of the midspan deflection (within $\approx$ 10-15%) of the more accurate FEM results.

III.7 What To Measure

Several sets of measurements will be taken. The midspan deflection of the truss will be recorded with an LVDT device. The internal force in a longeron member adjacent to midspan (that is element joining nodes 58-59 in the *Mathematica* model, but node numbers may be different in the ANSYS model) will be measured through an in-line load cell. Readings on the support load cells will also be recorded as described in Addendum III. All measurements should be converted to mm and N as necessary for comparison with the ANSYS results.

IV Writing A Report

IV.1 Organization

Reports are due in PDF form to Gradescope by Thursday, April 2 at 1:00 pm.
It must include:

- **Title Page**: Describes Lab, lists the name of the team members and identifies the group leader.
- **Results:** The results should address the questions in Section IV.2.
- **Appendix - Participation report.** More details on how the grade of individual group members is calculated can be found in Section IV.3.

IV.2 Report Content

The report will be graded on both technical content and presentation. Regarding the content, instead of an open-ended report, you should process the experimental data and computational results to address the specific questions posted below. Regarding presentation, make sure that you follow these guidelines:

- All plots should be readable. This includes using different color or line styles and a suitable font size for the axis labels and all other text in the plot. The range of both axes should be chosen to focus on the region of interest (i.e., the data).
- Show your work, including equations used and partial results.
- All results should be presented with appropriate units.
- Be quantitative when comparing results. Use percentage of error or deviation. Refer back to predicted error or variance when applicable.

IV.2.1 Question 1: Experimental Results

- Provide plots of the displacement and internal and reaction force measurements.
- Use linear regression and uncertainty analysis to verify the linearity of the measurements versus external load magnitude.

IV.2.2 Question 2: Analytical and FEM Results

- Which simplifications have you made when modeling the truss by your finite element model. Consider actual versus idealized internal forces and moments in joints and struts, as well as boundary conditions.
- Provide plots of the displacement and internal forces and reaction forces obtained by ANSYS simulations. Plot the undeformed versus deformed truss. Provide a plot of the internal forces in the bars.

- Compare your finite element results against the results of an equivalent beam model; see Section III.6. Discuss the reasons for the discrepancies between the FEM and equivalent beam model results.

IV.2.3 Question 3: Uncertainty Analysis

- Compare the experimental measurements and FEM results. Show plots that illustrate the differences between measurements and FEM predictions. Provide *quantitative* measures of the differences.

- Discuss potential sources that may cause the differences between experimental measurements and FEM results. To provide a *quantitative* assessment of these differences, analyze the influence of the following uncertainties by performing additional FEM simulations. Present your results in form of tables and/or graphs.

  - Imperfect joints, such as free-play, may reduce the effective stiffness of the bars.
  - Free-play in the in-line load cell may also reduce the effective stiffness of the bar containing the in-line load cell.
  - Friction at supported joints may require considering different boundary conditions.
  - Manufacturing imperfections may change the relative position of the joints and the cross-section of the struts.
  - The bar model assumes a uniform axial stress distribution in the struts; thus, the struts have no internal bending moments and the joints are modeled as perfect pin-joints. Consider that the joints can transmit bending moments and the struts provide bending stiffness, i.e., act as beams.

  Discuss additional sources that may impact the measurements. Be as quantitative as possible.

- Based on your uncertainty quantification results, what factor(s) of safety would you use to design a truss similar to the 16-bay truss used here. Assume that failure is due to the stresses in the struts exceeding the material strength and the midspan deflections exceeding an allowable limit. You may assume specific maximum/minimum values and/or ranges of values for the imperfections considered above, such as the variation in cross-sectional areas. You may ignore uncertainties in operational conditions, such as the magnitude of the applied mechanical load or thermal loading.

IV.3 Individual Contribution Evaluation and Deductions for No-Show

The group leader submits along with the report a separate participation report for your group with a brief summary of each group member's tasks, contributions, and performance as a group member. **Please make sure to include this as an Appendix in the report**.

The performance of each group member is rated with a "contribution factor" on a scale of 0 to 100%. A score of 100% indicates that the member contributed the expected share to the experiment and to the preparation of the report. The scores are normally assigned by *peer evaluation*, using the same procedures followed in the ASEN 200x sophomore courses. The group leader is responsible for administering the peer evaluation, tabulating and submitting the results of said peer evaluation.

The individual score will be equal to:

$$\text{Individual score} = \text{Group score} \times \frac{100 + \text{Contribution factor}}{200} \tag{1}$$

For example, if the group receives an overall score of 88.75 and the individual received a "contributing factor" of 90%, the individual score is $88.75 \times (100 + 90)/200 = 84.31$.

ALL students in the group must attend the experiment demonstration on Thursday, February 20, 2020. **A no-show at the experiment demos, without justification, will be penalized by a deduction of 50% from the final individual score score.** Students that miss the experimental demo on account of a bona-fide reason, such as a medical emergency or unavoidable absence, should contact instructors or TAs as soon as possible.

Addendum I. Report Grading

The score assigned to the lab report includes technical content (75%) and presentation (25%). This is a more detailed breakdown of the weights:

Category	Points	Score	Contribution
Technical content			
Question 1	15		
Question 2	25		
Question 3	35		
Presentation			
Plots	10		
Grammar, style & spelling	10		
Formatting	5		
Total	100		(overall score)

The points shown are the maximum number of points awarded for each category. The final score of each team member is then calculated following the procedure detailed in Section IV.3.