

EGR1010 Lab MATLAB

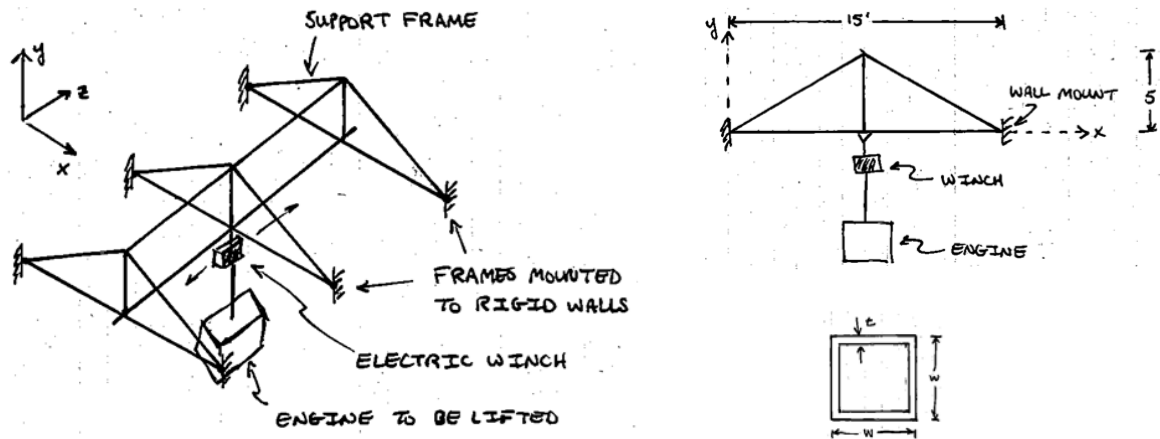
Supplemental Instruction Assignment #2

INSTRUCTIONS:

Complete the following assignment and submit with your laboratory notebook. Use the template provided on the course content page on Pilot. *Please follow the instructions for submission located at the end of this document.*

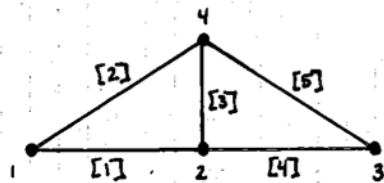
PROBLEM STATEMENT:

You are a co-op student at a local engineering company. The company has been contracted to design an overhead lift system for a truck repair shop used to remove large truck engines from the chassis for repair. A conceptual sketch of the overhead lift design from the lead engineer is shown in the figure below.



The primary load members consist of three support frames that are made from square tubing of wall thickness t and width w . As shown in the front view, the frame is 15 feet wide and 5 feet high. It will be mounted rigidly to the walls of the shop. **The lead engineer gives you the task of determining what size wall thickness tubing should be purchased.** He tells you that for a safe design, the vertical displacement at the center a support frame may not exceed 0.150 inches.

You have no idea how to attack the problem (*after all you're only a freshman anyway!*) so you consult the help of an older co-op student who is studying mechanical engineering. He tells you that you can solve the problem using a technique called finite element analysis (FEA) that he learned in one of his higher-level classes. He gives you the sketch shown in the following figure, which shows the finite element model of the structure. He describes the process of finite element analysis as breaking the larger structure into smaller *elements* that are connected at *nodes*. He tells you it contains 4 nodes and 5 elements. With this discretized representation of the component, you can describe a **system of equations** that can be solved to determine the displacements at the nodes.



The two of you set out to create a MATLAB program that uses finite element analysis to determine the critical displacement (*this particular student goes to a university that doesn't teach MATLAB, so they cannot help with the programming part*).

The program should perform four primary tasks:

- i. take inputs related to the problem
- ii. construct the finite element matrices for the system
- iii. perform finite element analysis to determine the displacement at the critical location
- iv. advise the user whether or not the specified design will fail

Remember that the senior engineer will likely review your program, so be sure to use appropriate variables names!

MATLAB PROCEDURES:

Complete the following procedures in the supplemental template file available in the course content on Pilot. DO NOT alter the formatting of the existing template, but rather simply add in your code where appropriate.

1. Predefine the scalar variables for the dimensions of the tubing (width w and wall thickness t in inches), the elastic modulus E of the material in pounds per square inch (psi), engine weight to be lifted ew in pounds, and the factor of safety FOS (unit-less). Use initial value: $w = 1.0$ in, $t = 0.1$ in, $E = 10 \times 10^6$ psi, $ew = 500$ lb, and $FOS = 3.0$.

2. Compute the cross-sectional area in inches squared of the tubing. Use the equation:

$$A = w^2 - (w - 2t)^2$$

Do NOT suppress command window output.

3. The other student gives you the following table of dimensional information that the finite element model. Create four *row vectors* that correspond to the x and y locations of ends A and B for each element (x_a, y_a, x_b, y_b) . Express the dimensions in *inches*. Suppress command window output.

Element	x_a (ft)	y_a (ft)	x_b (ft)	y_b (ft)
1	0.0	0.0	7.5	0.0
2	0.0	0.0	7.5	5.0
3	7.5	0.0	7.5	5.0
4	7.5	0.0	15.0	0.0
5	15.0	0.0	7.5	5.0

4. Compute the length of each element using element-by-element math (should be only one line of code to determine ALL lengths). Use the equation:

$$L = \sqrt{(x_b - x_a)^2 + (y_b - y_a)^2}$$

Do NOT suppress command window output.

5. The other student then tells you that the finite element matrices for each element are given as:

$$k_1 = \frac{AE}{L_1} \begin{bmatrix} 1 & 0 & -1 & 0 \\ 0 & 0 & 0 & 0 \\ -1 & 0 & 1 & 0 \\ 0 & 0 & 0 & 0 \end{bmatrix}, \quad k_2 = \frac{AE}{L_2} \begin{bmatrix} 0.6923 & 0.4615 & -0.6923 & -0.4615 \\ 0.4615 & 0.3077 & -0.4615 & -0.3077 \\ -0.6923 & -0.4615 & 0.6923 & 0.4615 \\ -0.4615 & -0.3077 & 0.4615 & 0.3077 \end{bmatrix}$$

$$k_3 = \frac{AE}{L_3} \begin{bmatrix} 0 & 0 & 0 & 0 \\ 0 & 1 & 0 & -1 \\ 0 & 0 & 0 & 0 \\ 0 & -1 & 0 & 1 \end{bmatrix}, \quad k_4 = \frac{AE}{L_4} \begin{bmatrix} 1 & 0 & -1 & 0 \\ 0 & 0 & 0 & 0 \\ -1 & 0 & 1 & 0 \\ 0 & 0 & 0 & 0 \end{bmatrix}$$

$$k_5 = \frac{AE}{L_5} \begin{bmatrix} 0.6923 & -0.4615 & -0.6923 & 0.4615 \\ -0.4615 & 0.3077 & 0.4615 & -0.3077 \\ -0.6923 & 0.4615 & 0.6923 & -0.4615 \\ 0.4615 & -0.3077 & -0.4615 & 0.3077 \end{bmatrix}$$

Make five 4x4 matrices (k_1 through k_5) using the relations above. Suppress the command window output.

6. Use the `zeros()` function to create an 8x8 matrix full of zeroes called K_g . This is the global stiffness matrix for the finite element system. Suppress command window output.

7. The other student now tells you that each element matrix must be added into specific rows and columns in the global matrix. He says that element 1 should be added rows/columns [1,2,3,4]. Element 2 to [1,2,7,8]. Element 3 to [3,4,7,8]. Element 4 to [3,4,5,6]. Finally, element 5 to [5,6,7,8]. This process is known as assembling the matrix.

For example for the 1st element: $K_g([1,2,3,4],[1,2,3,4]) = K_g([1,2,3,4],[1,2,3,4]) + k_1$

Do NOT suppress output for the last element addition (k_5). Suppress all other command window output.

8. Now the global force vector must be created for the finite element system. Create an 8x1 vector named F_g that contains all zeroes except for the 4th entry. This entry is equal to the force applied to the structure. In this case it is equal to the negative of the factor of safety times the engine weight:

$$Force = -FOS * ew$$

9. The older student now says the finite element system must be reduced by applying the boundary conditions. He says since the model is fixed at nodes 1 and 3, that columns/rows [1,2,5,6] must be removed from stiffness matrix and that rows [1,2,5,6] must be removed from the force vector. Create a reduced stiffness matrix (4x4) by retaining only rows/columns [3,4,7,8] of the global stiffness matrix and a reduced force vector (4x1) by retaining only rows [3,4,7,8]. Do NOT suppress command window output.

10. Now solve the finite element system by using backwards division. Left divide the reduced stiffness matrix into the reduced force vector. The resulting vector contains the reduced displacement vector (the displacement in the x & y directions of nodes 2 and 4). Name this vector Ur and do NOT suppress command window output.

11. The critical displacement in the y-direction at node 2 is located in the second entry in the reduced displacement vector. Take the absolute value of this quantity and set it equal to a new variable called $crit_disp$.

CODE VALIDATION: for $E = 10 \times 10^6$, $w = 1$, $t = 0.1$, $FOS = 3$, and $ew = 500$ gives a $crit_disp = 0.0982$

12. Use your program to determine the minimum wall thickness of 1 inch square tubing ($w = 1.0$ inches) that is appropriate to lift a Cummins diesel from a semi truck. The dry weight of these engines is approximately 1200 pounds. Use a factor of safety of 2.0 and assume the tubing is aluminum ($E = 10 \times 10^6$ psi). To do this change the initial values in part 1 and rerun your program to not exceed $crit_disp = 0.0982$.

SUBMISSION:

Publish your code as a pdf and turn it into your lab TA.