Joshua Allessio

0806188

CE420 Computational Methods of Structural Analysis Final Project

5/7/2023

**Task 1:**

In this project, a MATLAB code was created to analyze any 3D truss using the structural stiffness method. In the folder, it is titled, “Task1\_3DTruss\_Analysis.” The analysis also uses the file, “stiff\_spacetruss.m”.

**Input:**

In order for the code to work, it needs a particular input format. This input format consists of four text files saved in the same folder as both of the MATLAB files. The first text file, “joints.txt” labels each joint and degree of freedom (DOF) of the truss. It has seven columns with one row for each joint and takes the form:

|  |  |  |  |  |  |  |
| --- | --- | --- | --- | --- | --- | --- |
| Joint number | X Coordinate | Y Coordinate | Z Coordinate | X DOF Number | Y DOF Number | Z DOF Number |

It is important that the DOF numbering system is consistent with that found in “Kassimali: Matrix Analysis of Structures.” The next text file, always titled “members.txt” labels the members and takes the form:

|  |  |  |  |  |
| --- | --- | --- | --- | --- |
| Member Number | Beginning Joint | End Joint | E (ksi) | A (in2) |

For this file, it is arbitrary which joint is the beginning or end of any member, though it is easiest to always choose the joint with the lower number as the beginning and higher number as the end. This way it is much less likely to accidentally double count a member. E is Young’s Modulus of Elasticity, commonly 29,000ksi, and A is the cross-sectional area of the member.

The third file, titled ‘supports.txt’ tells the program which DOF’s relate to the supports of the truss and do not move. It has one column, with each row containing only the DOF’s relating to the truss’ supports.

|  |
| --- |
| Constrained DOF Number |

The fourth file must be titled ‘loads.txt’ and it specifies external loads. It takes the form:

|  |  |  |  |
| --- | --- | --- | --- |
| Joint Number | Px | Py | Pz |

Where Px, Py, and Pz represent loads applied in the global X, Y, and Z dimensions, respectively. It is important that all of the loads use a consistent sign convention for the chosen coordinate system orientation.

**Output:**

The output of the code provides all joint displacements, reaction forces, and axial member forces for the given truss and loading conditions. The joint displacements can be found in the vector ‘d.’ It has a height of NDOF, and the displacements, in inches, correspond in order from 1 to NDOF, for each DOF.

The support reactions measured in kips are found in vector ‘R.’ R has a height of NR, and each row of R corresponds with each row of ‘supports.txt.’

Member axial forces are measured in kips and stored in vector ‘F.’ F has a height of NM, and each row corresponds to each row of ‘members.txt.’ Positive values represent tension forces, and negative values represent compression forces.

**Task 2:**

Another thing we can do with the structural stiffness method in MATLAB is create influence lines to display how truss members bear load under changing load conditions. For the truss shown below, we are interested in finding out the maximum compressive and tensile force in members DL and DE due to a 4KN load being placed anywhere along member AF. To begin, we will create influence lines for member DL and member DE as a unit load moves across member AF.

Diagram

Description automatically generated

Figure 1: Truss Diagram

In the attached file titled “Task2.m,” a ‘for’ loop moves a unit load from joint B to C to D to E and takes advantage of the file “truss\_analysis.m” to calculate the axial forces in members DE and DL for each loading case and then store them in vectors. The code then plots the influence lines for the two members. The influence lines are shown in the figure below.

Chart, line chart

Description automatically generated

Figure 2: Influence Lines for Truss Member DE and DL

Tabulated versions of the influence lines can also be found in the workspace of the MATLAB file, saved in the variables ‘IL\_DE’ and ‘IL\_DL.’ They can also be seen in the figure below.

|  |  |  |  |
| --- | --- | --- | --- |
| Unit Load | Location along AF (m) | DE Axial Force (unit) | DL Axial Force (unit) |
| A | 0 | 0 | 0 |
| B | 2.49 | 1.58 | 0.82 |
| C | 4.5 | 2.86 | 1.48 |
| D | 7.5 | 4.76 | 2.47 |
| E | 9.51 | 1.42 | 0.033 |
| F | 12 | 0 | 0 |

Table 1: Tabulated Influence Lines

Finally, determining the maximum tensile and compressive forces in each member from a 4 kilonewton load is very easy to do with the use of influence lines. It is as simple as taking the maximum and minimum values from table 1 and multiplying them by 4 KN.

The chosen sign convention for tension and compression labels tension forces as positive, and compression forces as negative. We can see in each of the influence lines that the axial forces in both members are always 0 or greater than 0. This means that placing a load on the bottom chord of this truss will never cause members DE and DL to be in compression. The maximum compressive force for both members is 0. The maximum tensile force for member DL is 9.88KN, and the maximum tensile force for member DE is 19.04 KN.

**Task 3:**

For this task, we are asked to calculate the reaction forces and determine which column has the greatest axial force for the given loading in the frame shown below.

A picture containing chart

Description automatically generated

Figure 3:Wind-Loaded 2D Frame

For a frame this large, it is beneficial to define an analytical model before beginning calculations. This model can be seen below in figure 4.

Calendar

Description automatically generated with medium confidenceA picture containing diagram

Description automatically generated

Figure 4: Analytical Model for Task 3 Frame

The model is split into two to counter congestion in the diagram. On the left is the member numbering, and on the right is the joint and DOF numbering. Using this model, the frame was converted into text files for MATLAB using a very similar convention to that described for task 1. All of these files are stored inside the “Task 3” folder alongside the code used to execute the analysis. The vertical reaction forces are stored inside the variable “R\_y” in the file “Task3\_PlaneFrame.m”. They can also be seen tabulated below.

|  |  |
| --- | --- |
| Column | Reaction (kip) |
| B1 | 62.48 |
| B2 | 23.52 |
| B3 | -3.11 |
| B4 | 13.09 |
| B5 | 28 |
| B6 | -1.26 |
| B7 | -122.76 |

**Task 4:**

In this part, we used the structural stiffness method with MATLAB to calculate joint displacements for the structure in the figure shown below.

**Diagram

Description automatically generated**

Figure 3: Task 4 Structure for Analysis

For this structure, we are instructed to model cantilever beam as a three member frame, and the supporting springs as truss members which are pin-connected to the ground. The chosen analytical model for the structure is shown below.

A white board with writing on it

Description automatically generated with medium confidence

Figure 4: Task 4 Analytical Model

For this task, the goal was to find out how much the springs compress. Since joints 1, 2, and 3 are supports and do not move, the amount the springs compress can be found as the vertical displacement of joints 5, 6 and 7; AKA DOF’s 5, 8, and 11. In the attached MATLAB file called “Task4.m”, we analyzed the structure and found structure stiffness matrix [S], and it’s load vector {P}, in order to calculate its joint displacement vector {d}. This is stored as ‘d’ in the workspace when you run the code, and the relevant results are summarized in the table below.

|  |  |
| --- | --- |
| Member number | Spring Compression (inches) |
| 4 | 0.0003 |
| 5 | 0.0007 |
| 6 | 0.0012 |