INDIAN INSTITUTE OF TECHNOLOGY, KANPUR

SURGE-2022

Project-Report

Calibration of stiffness of members of truss-type structure using Finite Element and Artificial Neural Network Model

Submitted by

Dheeraj Bana

SURGE No. : 2230435
Department of Civil Engineering
IIT Kanpur

Under the Guidance of

Dr. Suparno Mukhopadhyay

Department of Civil Engineering
IIT Kanpur

CERTIFICATE

This is to certify that the project entitled "**Calibration of stiffness of members of truss-type structure using Finite Element and Artificial Neural Network Model**" submitted by Dheeraj Bana(2230435) as a part of Summer Undergraduate Research and Graduate Excellence 2022 offered by the Indian Institute of Technology, Kanpur, is a Bonafede record of the work done by him under my guidance and supervision at the Indian Institute of Technology, Kanpur from May 13, 2022, to July 14, 2022.

Suparno Mukhopadhyay

Dr. Suparno Mukhopadhyay
Assistant Professor
Department of Civil Engineering
Indian Institute of Technology Kanpur

Abstract

Global warming, climate change, and extreme weather(floods) are damaging our infrastructures badly. So, it’s now very critical to provide proper maintenance to our infrastructures to increase their life cycle. Structural Health Monitoring is a new interdisciplinary research field that provides new cost-effective solutions to maintain the good health of our infrastructures. In this project, we focus on truss structures that are widely used in bridges(for example Howrah bridge). Finite Element modeling is done for the truss structures to check stability and calculate the stiffness of different members of a simple truss structure using nodal displacements. To get elements properties from nodal displacements is an Inverse Problem. An Artificial Neural Network model is also proposed to calculate the stiffness of members and the results are compared with the results of the Finite Element model. All analysis is done under static loading conditions and can be done under dynamic loading conditions also which is not the focus of this project.

Keywords: Artificial Neural Network; Finite Element modeling; Inverse Problem; Truss structures; Stiffness calibration

---

1 Introduction

The Finite Element Method (FEM) is a computational tool and is highly used in all civil engineering applications. Also, with the developments in data processing power and sensing technologies, it is evident that in the coming years, there would be high uses of machine learning (ML) techniques in the civil engineering field. There have already been some uses of ML in geotechnical and structural engineering across the world. In this project, the applications of ML in civil structural health monitoring are proposed with Finite Element modeling of structures. The failure of any civil engineering costs millions. So, it's very important to monitor these structures, to avoid failure because of poor maintenance. There are some manual ways to monitor the health of structures, but they are not much effective due to the subjectivity and the time and labor involved. So instead the data that is measured automatically by structural sensors, along with new advanced techniques like ML, data science, and FEM can be used to monitor the health of the structures. In this project, the focus is mainly on bridges, as they are important structures and play a very critical role in our everyday life. Nowadays, there are frequent failures, while not catastrophic, some parts of the bridge fail and stop working; hence, it's very important to monitor the health of these bridge structures.

A truss-type bridge model is used in this project. All members of Truss structures are either under compression or in tension. They support external loads by going under axial elongation or shortening. The basic stable truss structure is a three-member structure in which all members are connected and joints don't take any moment means they can freely rotate. Now if the above basic truss structure of three members is not provided in any infrastructure that is built from truss structures then it will be locally unstable as under external load some members can rotate freely. Also if boundary constraints are not provided appropriately then the structure will be globally unstable. To check both the local and global stability of the structure, a FE(Finite Element) model simulation of the truss structure can be used. In FE modeling, the global stiffness matrix after imposing boundary conditions will be singular if the truss structure is unstable. It means either boundary constraints are not provided appropriately or a basic three-member truss structure is not provided(or some diagonal members are missing). Eigenvalues can be used to check if the stiffness matrix is singular or not. If the magnitude of some of the eigenvalues is significantly less than others then the global stiffness matrix is singular and has to check the boundary constraints and member's connectivity. Now, assume that the truss structure is locally and globally stable. As with time, the E(modulus of elasticity) reduces due to deterioration from environmental effects, sudden extreme loadings, etc. So it will reduce the stiffness of members. So, it’s very critical to find the modulus of elasticity (E) of members/elements from time to time for estimating the strength of members.

Finding the E of different elements from nodal displacements under given load conditions is called Inverse Problem. It is subject to some static loading, and analyze this structure by FEM.
Nodal displacements are recorded and used to calculate the stiffness of different members by using the FE model and the ANN model. The results are compared with each other for both cases. This truss structure can be subjected to some dynamic loading also and vibrations can be measured and used for estimation of the stiffness of different members but this is not discussed here.

2 Finite Element Modeling

The finite Elements Method(FEM) is a very powerful numerical tool that is used a lot in civil engineering in modeling complex problems and big problems. Analyzing a big bridge like the Howrah Bridge will be a very cumbersome and time taking process if we do it by hand but it can be done easily using FEM. In this project, the truss structure is analyzed using the Direct Stiffness Method. The truss structure has 20 members and 10 joints. All truss members are modeled using 2D - truss elements. Each node(joint) has two degrees of freedom in the x and y directions. Element stiffness equation in the local coordinate is determined first and then it is transformed into a global coordinate system using a transformation matrix. Then it is assembled for all global degrees of freedom using the gather operator and boundary conditions can be imposed using any one of the Partition approach, Deletion, and Penalty approaches. It is defined as follows:

$$KU = F \tag{1}$$

Where K is the global stiffness matrix after imposing boundary conditions, U is the nodal displacement vector, and F is the known external force vector.
If boundary conditions are imposed using the Partition approach then the dimensions of K will be 17x17, U will be 17x1, and F will be 17x1.

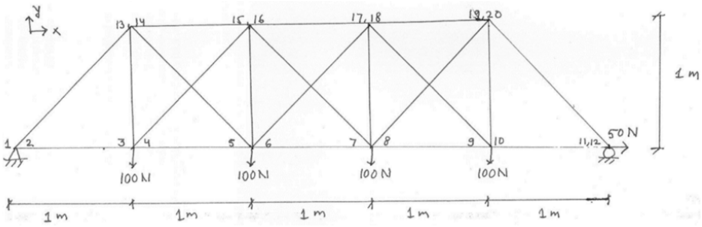

Fig. 1. Truss Structure

Nodal Forces corresponding to the different degrees of freedom (DoFs) shown in Fig. 1. are as follows:
(Please note that the first DoF at each node corresponds to the x-directional DoF.)

Table 1

DoF	F(N)	DoF	F(N)	DoF	F(N)	DoF	F(N)
1	0	6	-100	11	50	16	0
2	0	7	0	12	0	17	0
3	0	8	-100	13	0	18	0
4	-100	9	0	14	0	19	0
5	0	10	-100	15	0	20	0

For known external loading conditions, there are two cases possible as follows:

1. K is a function of E(Stiffness of an element), L(length of an element), and A(area of an element). If the geometry and stiffness of all members are known then we can find nodal displacements as follows:
   $$U = K\backslash F \tag{2}$$
   '\' is the backslash command in MATLAB

2. If the stiffness of members is unknown then K will be unknown and if we know the nodal displacements vector then we can find the stiffness of members as it will be an Inverse Problem.

2.1 Solving for unknown nodal displacements

If we change E, L, or A, then K will change and nodal displacements will be different and can be found very easily using equation (2). Now assuming that geometry is constant, then different sets of stiffness of members will give different sets of nodal displacements under the same external loading condition.

Example - For $E = 2 \times 10^7$ N/m², $A = 2.5 \times 10^{-3}$ m² for all elements, nodal displacements from the equation (2) are given in Table 2 as follows:

Table 2
Nodal displacements from FE model

DoF	Disp(m)	DoF	Disp(m)	DoF	Disp(m)	DoF	Disp(m)
1	0	6	-0.0400	11	0.0222	16	-0.0398
2	0	7	0.0138	12	0	17	0.0086
3	0.0040	8	-0.0400	13	0.0180	18	-0.0398
4	-0.0291	9	0.0182	14	-0.0271	19	0.0042
5	0.0084	10	-0.0291	15	0.0136	20	-0.0271

Table 3
Recorded/measured Nodal displacements

DoF	Disp(m)	DoF	Disp(m)	DoF	Disp(m)	DoF	Disp(m)
1	0	6	-0.0683	11	0.0294	16	-0.0672
2	0	7	0.0183	12	0	17	0.009
3	0.005	8	-0.0683	13	0.0303	18	-0.0672
4	-0.0478	9	0.0244	14	-0.0454	19	-0.0009
5	0.011	10	-0.0478	15	0.0204	20	-0.0454

2.2 Estimation of stiffness of members

If the stiffness of members are unknown then global stiffness matrix K also will be unknown. Now if nodal displacements are known then the stiffness of members can be found using the same FE model for given external loading conditions by treating it as an inverse problem. Nodal displacements can be obtained through images or video recordings and using the FE model, the stiffness of different members can be calculated. The measured nodal displacements corresponding to the different degrees of freedom (DoFs) shown in Fig. 1. are in Table 3 (Please note that the first DoF at each node corresponds to the x directional DoF.) A maximum of 17 unknowns can be solved in the inverse problem using the FE model in this case(Static Loading) as only 17 equations are there. So, assume that (i) all top chord members have the same E (= E^1), (ii) all bottom chord members have the same E (= E^2), (iii) all vertical members have the same E (= E^3), and (iv) all diagonal members have the same E (= E^4). These will be as follows:

Table 4
Stiffness of members from FE model

Stiffness	In 10^7 N/m²
E^1(Top members)	1.9562
E^2(Bottom members)	0.9815
E^3(vertical members)	2.4724
E^4(diagonal members)	1.4634

3 Artificial Neural Network Model

ANN can capture the non-linear relationship between input and output data easily and has many applications in geotechnical and structural engineering. In geotechnical engineering, researchers are exploring the applications of ANN for the calibration of soil parameters. In this project, ANN is used for calibration of stiffness of members of a truss - structure. A minimum and maximum range is taken for stiffness values and using the *rand* MATLAB function, different values can be taken for stiffness of top, bottom, vertical, and diagonal members. 500 sets are taken for stiffness of members. For these sets, nodal displacements can be calculated using the FE model that is built above for the inverse problem. Nodal displacements are taken as an input and stiffness of members is taken as output. One hidden layer is used in the ANN model and the number of hidden neurons can be given as follows:

$$m \leq 4\sqrt{(n(l + 3) + 1)} \tag{3}$$

where m is the number of neurons in the hidden layer, n is the number of neurons in the input layer, and l is the number of neurons in the output layer.

In our case,
$n = 17$; $l = 4$

The *tansig* (MATLAB function) is the activation function for neurons on the hidden layer. The *purelin* (MATLAB function) is the activation function for neurons on the output layer, and the training function is the *trainlm* (MATLAB, 2009), which is a function that implements the LM method in MATLAB. ANN model is trained using the training set(taken as 70% of the dataset) and is validated on the validation set(taken as 30% of the dataset).

Now, the model can be used to calibrate the stiffness of members by providing nodal displacements as input to the model. Nodal Displacements that are given in Table 3 are taken as an input and given to the ANN model and the output by the model is given in Table 5 as follows:

Table 5
Stiffness of members from ANN model

Stiffness	In 10^7 N/m²
E^1(Top members)	1.9994
E^2(Bottom members)	0.9951
E^3(vertical members)	2.5032
E^4(diagonal members)	1.4984

4 Results

Stiffness of members from the ANN model given in Table 5 are almost similar to the results from FEM given in Table 4. Also, The computation time for FEM is less than the ANN model.

5 Conclusion

This report proposed an inversion type of method based on the Finite Element and Neural Network Model for a truss-type structure.

Acknowledgments

I will thank IIT Kanpur for giving me an opportunity to do a SURGE research internship under the supervision of Dr. Suparno Mukhopadhyay.

References

https://www.sciencedirect.com/science/article/pii/S2467967419300443?via%3Dihub