Analysis of Beams Deflection with Finite Elements Method using MATLAB

Muhammed Burak Görmüş

August 2021

1 Introduction

In this document, the MATLAB GUI that is created in order to analyze beams deflection will be discussed. The GUI can find the deflections throughout the length of the beam as well as the reactions at the supports.

The GUI assumes that the beam that is going to be analyzed lies on the x-axis. In other words, while locating the nodes, please put them on the x-axis. Besides, the GUI is prepared such that only vertical shear forces and bending moments can be applied to the structure.

If a more complex geometry needs to be analyzed, please use the Planar Frame Element GUI created. In Planar Frame Element GUI, horizontal and vertical forces as well as bending moments can be applied to the structure. Furthermore, there is no need to place the nodes on the x-axis.

2 Tabs on the GUI

When the GUI is opened, there will be 5 tabs that can be seen at the top of the GUI: HomePage, Manual Analysis, Book Solutions, Material Properties and References.

If you want to analyze a beam structure by entering the node and element information as well as the Boundary Conditions, Manual Analysis tab should be chosen.

If you want to analyze a beam structure that is given on the book of Hibbeler or Beer, you can go to Book Solutions tab, and choose the book and the problem

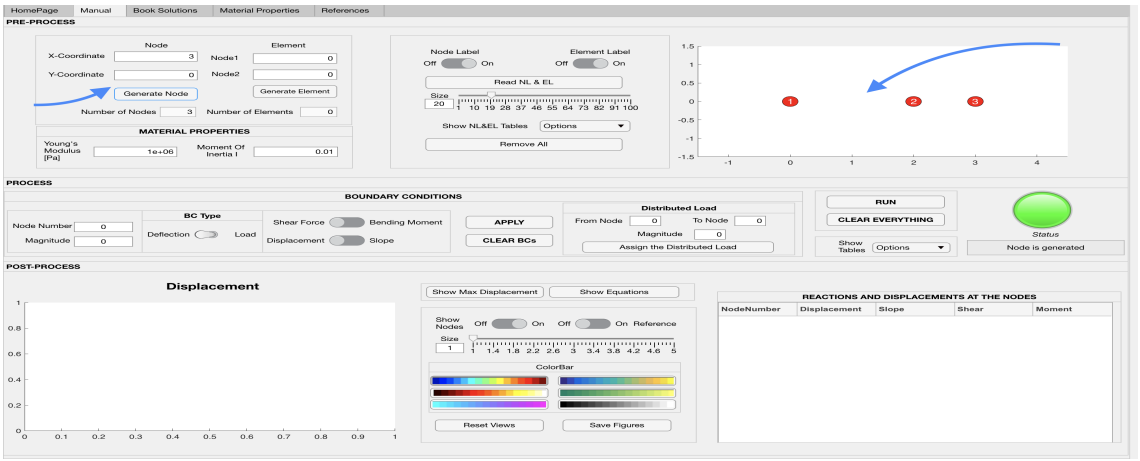
number. After choosing a problem, you will see that the node and element information as well as the boundary conditions are already entered automatically. After that, if you push the RUN button, you can see the results of the corresponding analysis.

3 First Example

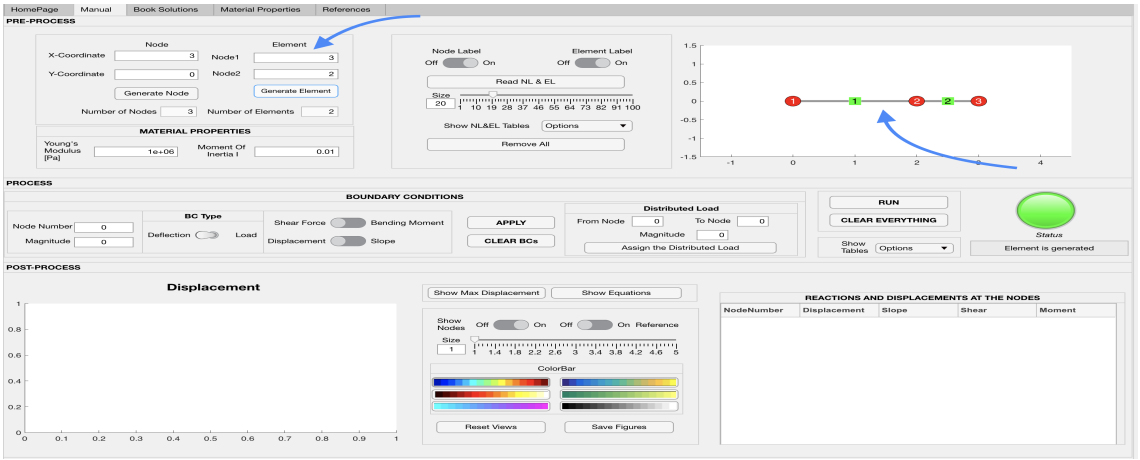
Let's look at the example 12.3 given in Hibbeler 10th Edition [1]. In this problem, we have 3 important points A, B and C. At points A and C, we know deflection will be equal to zero, and at B we apply a shear force having a magnitude of -6000kN.

The coordinate of these points are the coordinates that we need to enter for our nodes. Then, we will connect these nodes, and create our elements.

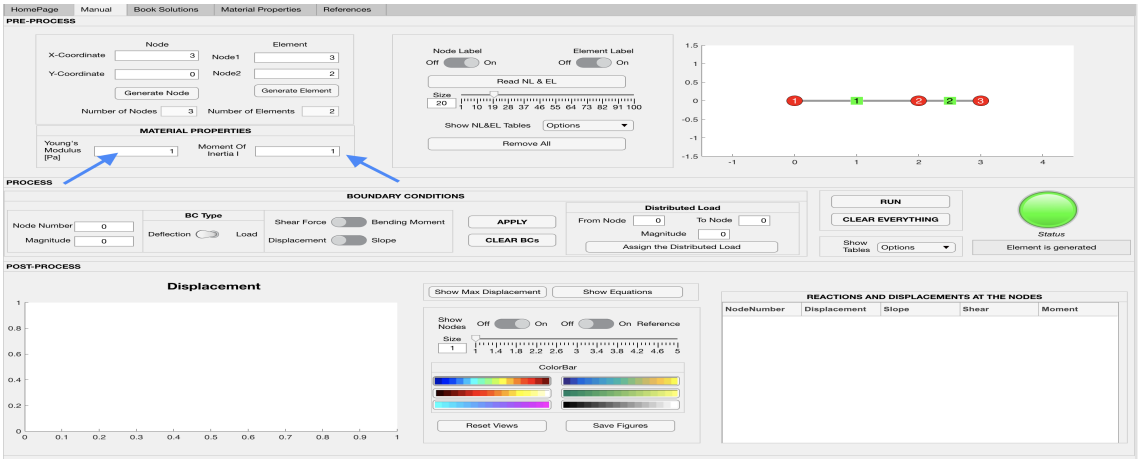
Firstly, we need to enter the coordinates of these points which are (0,0), (2,0) and (3,0) representing A, B and C, respectively.



Then, we need to enter the connectivity information between the nodes, i.e., creating our elements by entering 1,2 and 2,3 to the element section. In detail, our first element is between node1 and node2, and our second element is between node2 and node3.



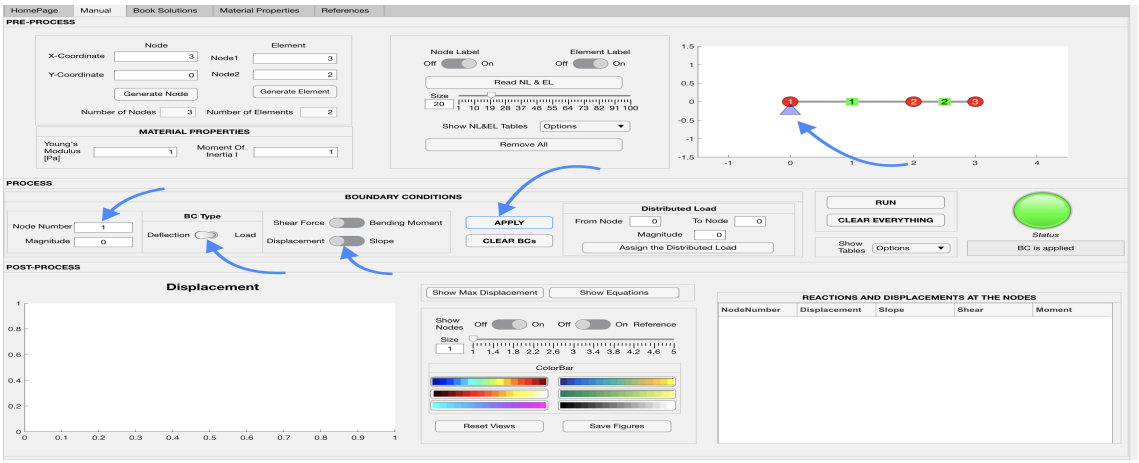
As you can see above, the geometry of the structure is entered. Lastly, we need to enter the values of Youngs Modulus and Moment of Inertia. In this problem, they are chosen as 1.



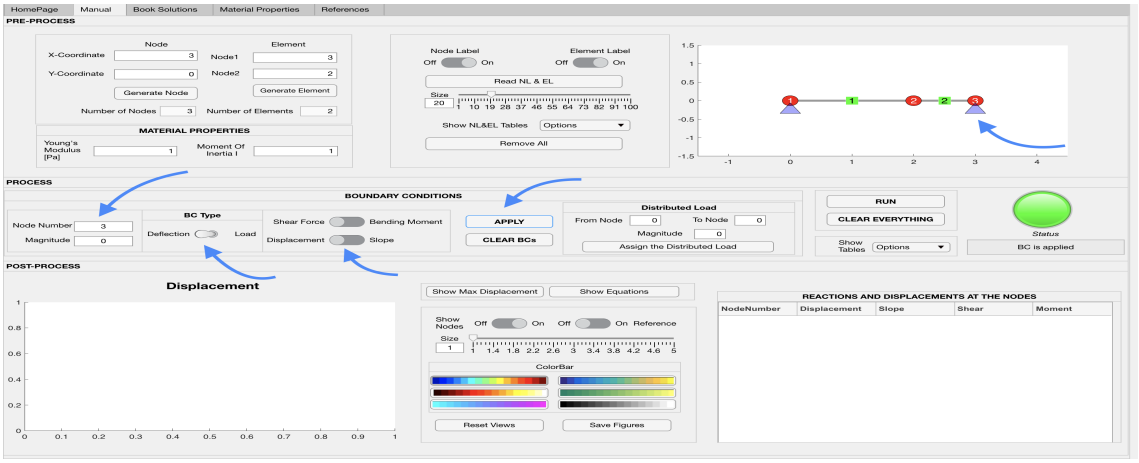
Now, it is time to assign the boundary conditions. As it is known, at the pin (point A - node1) and at the roller (point C - node3), the vertical displacement should be equal to zero.

If we want to enter an information related to displacement and slope, we should choose Deflection for the BCType. If a shear force or bending moment need to be assigned, we should choose Load for the BCType.

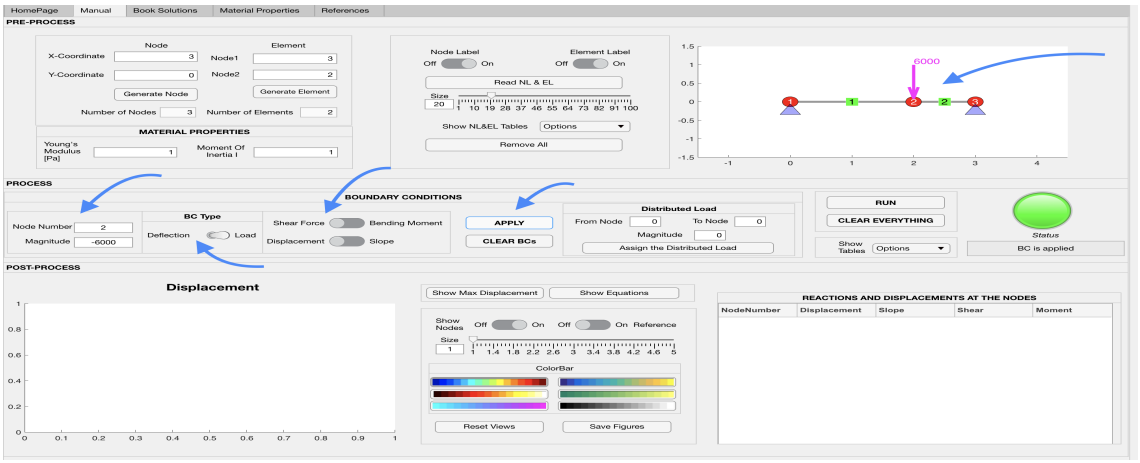
At node number 1, as said, the vertical displacement is zero. Therefore, we enter the following information. As a result, on the upper right figure, a triangular shape at the bottom of node 1 can be seen indicating it is fixed in the y direction.
Node Number: 1
Magnitude: 0
BC Type: Deflection
Displacement



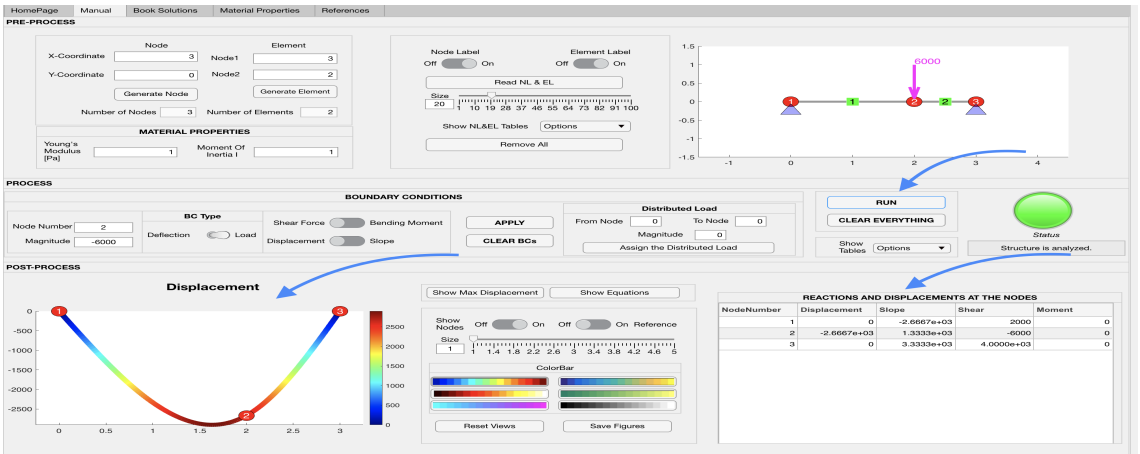
At node number 3, as said, the vertical displacement is zero. Therefore, we follow the same procedure as we followed for node number 1.
Node Number: 2
Magnitude: 0
BC Type: Deflection
Displacement



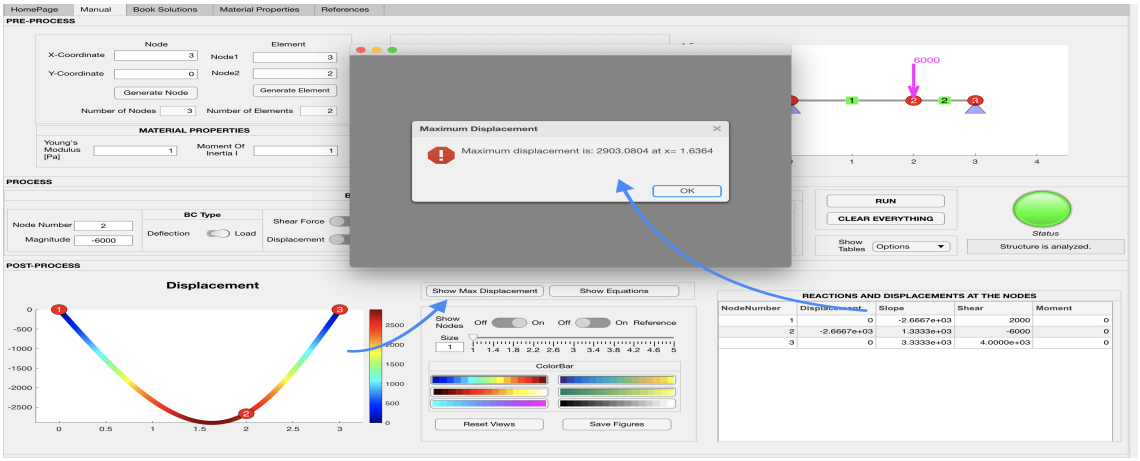
The last boundary condition is the shear force having a magnitude of -6000N applied on the node 2 (point B). In order to assign this BC, we enter the following information.
Node Number: 2
Magnitude: -6000
BC Type: Load
Shear Force



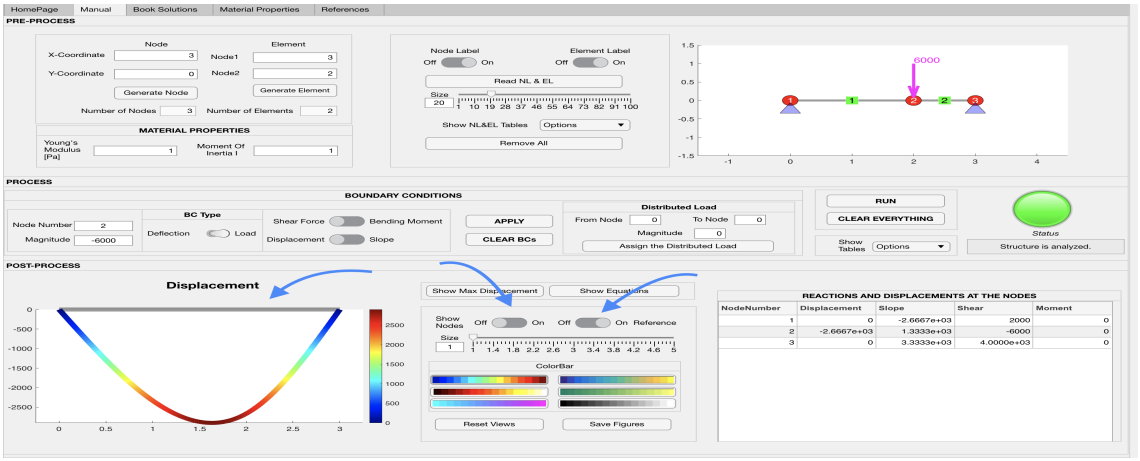
After we enter the nodes and BCs, we can push the RUN button. As a result, the shape of the beam can be seen on the displacement figure. Besides, on the right, the reaction forces and displacement information given on the table.



If you want to see the maximum deformation and its location, you can push the Show Max Displacement button. Pushing Show Equations yields the deflection equations for each element.



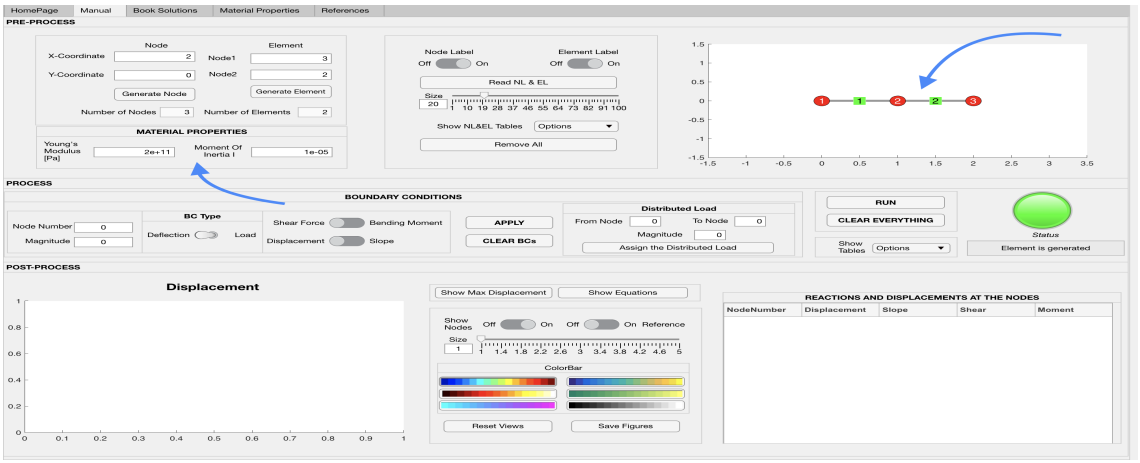
If you want to see the undeformed configuration as well as the deformed configuration, please turn on the Reference Switch. If you also want to see the nodes on the deformed beam, you can turn on the Show Nodes Switch.



Lastly, if you want to examine the deflection throughout the beam with different color variations, you may change the color using ColorBar Section.

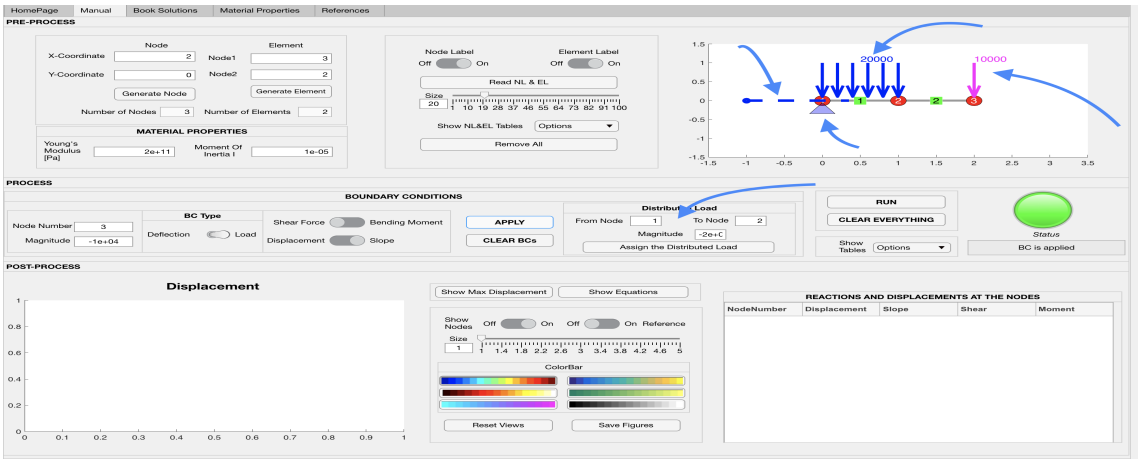
4 Second Example

Let's look at the problem F12-10 given in Hibbeler 10th Edition [1]. In this problem, we have 3 important points (nodes). The first node is (0,0); the second node is (1,0); the third node is (2,0). After entering the nodes, we should create our elements and complete the geometry. Further, the magnitude of E and I should be entered.

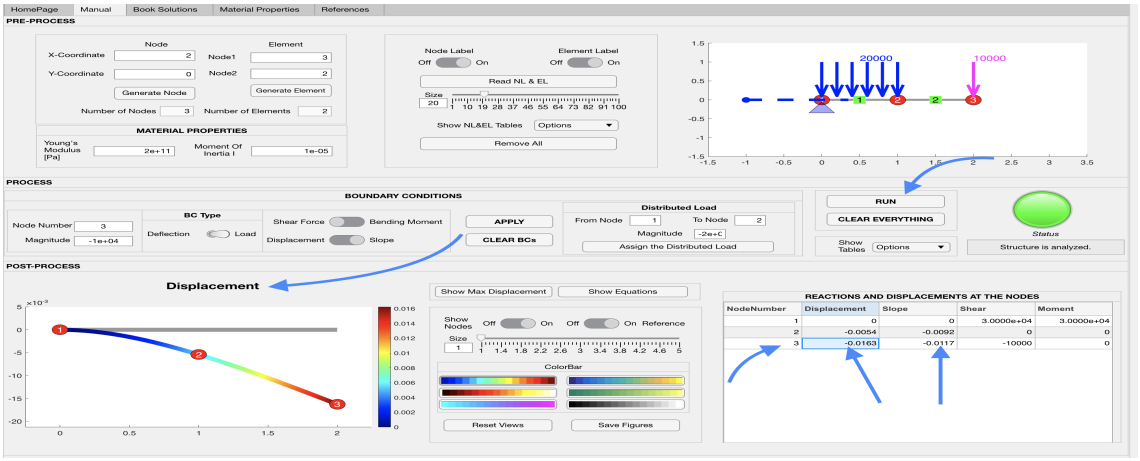


Now, it is time to enter the BCs for the problem.
BC1; Node Number:1; Magnitude:0; BC Type:Deflection ; Displacement
BC1; Node Number:1; Magnitude:0; BC Type:Deflection ; Slope
BC1; Node Number:3; Magnitude:-10000; BC Type:Load ; Shear Force

Lastly, we need to enter the distributed load from node1 to node2 having a magnitude of -20kN. In order to do this, we enter the following information.
From Node: 1 To Node: 2 Magnitude: -20000



After giving the BCs, we push the RUN button and analyze the structure.



5 References

[1] R. C. Hibbeler and K. B. Yap, in Mechanics of materials, Harlow: Pearson, 2018.