Analysis of Planar Truss Structures with Finite Elements Method using MATLAB

Muhammed Burak Görmüş

September 2021

1 Introduction

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

The GUI assumes that the elements that is going to be analyzed can be only subjected to the vertical and horizontal forces. It means that using this GUI, we can analyze two force members, and it is assumed that the structure will not be subjected to the moments.

If a more complex geometry needs to be analyzed, please use the Planar Frame Element GUI created if it is suitable. In Planar Frame Element GUI, horizontal and vertical forces as well as bending moments can be applied to the structure.

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.

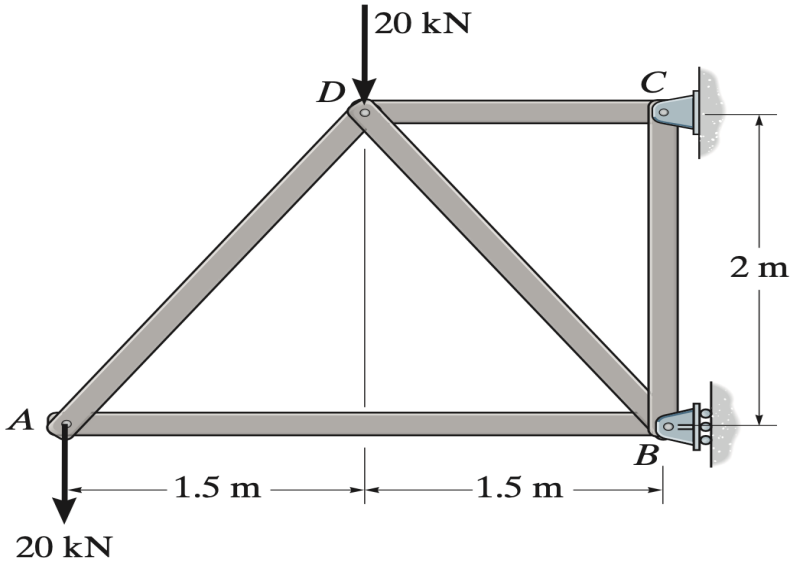
If you want to analyze a structure by entering the node and element information as well as the Boundary Conditions, Manual Analysis tab should be chosen.

If you want to analyze a 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 An Example

Let's look at the example 14.84 given in Hibbeler 10th Edition [1].

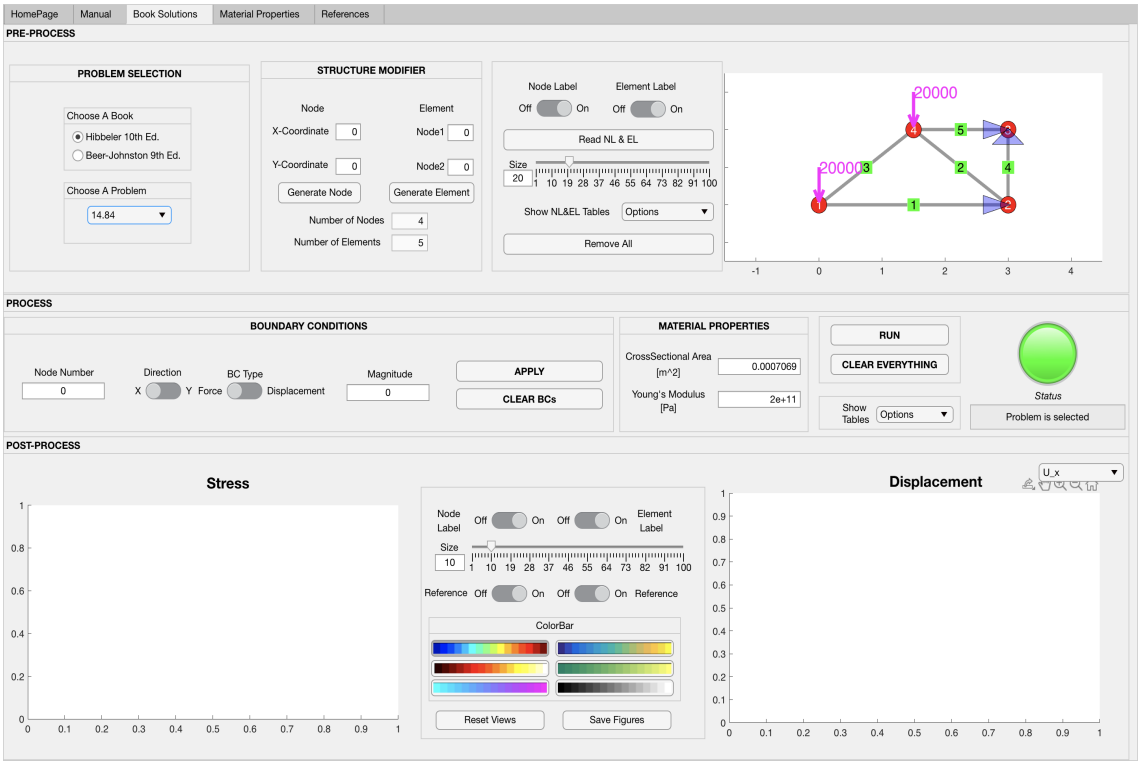
***14–84. Determine the vertical displacement of joint *D*. The truss is made from A992 steel rods having a diameter of 30 mm.**



In this problem, we have 4 important points A, B,C and D.

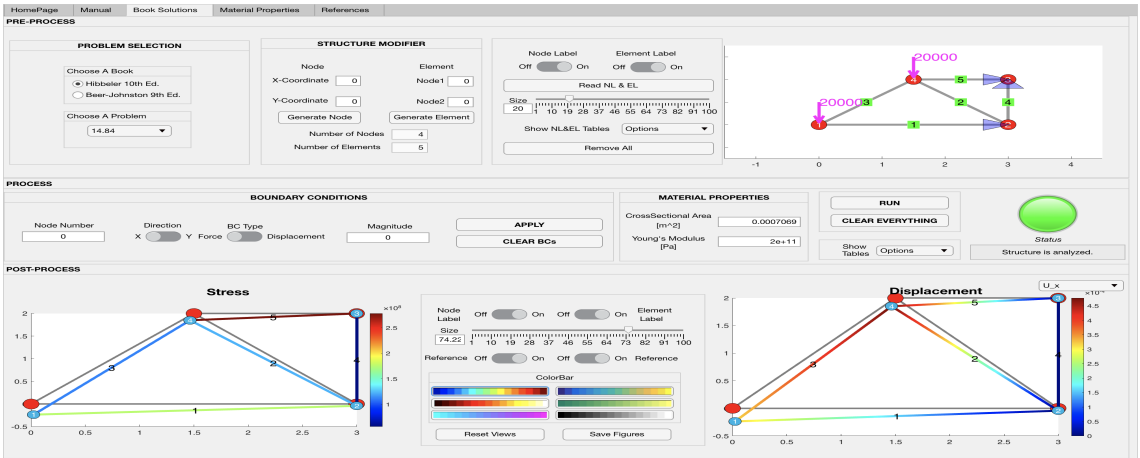
At points A and D, we apply vertical forces having a magnitude of 20kN. At point C, there is a pin which means that the displacement along x and y axes will be equal to zero. Lastly, there is a roller at B and the displacement along the x axis will be zero.

If we use the Book Solutions Tab, and choose our book and problem number, the geometry and the boundary conditions will be already assigned as shown below.



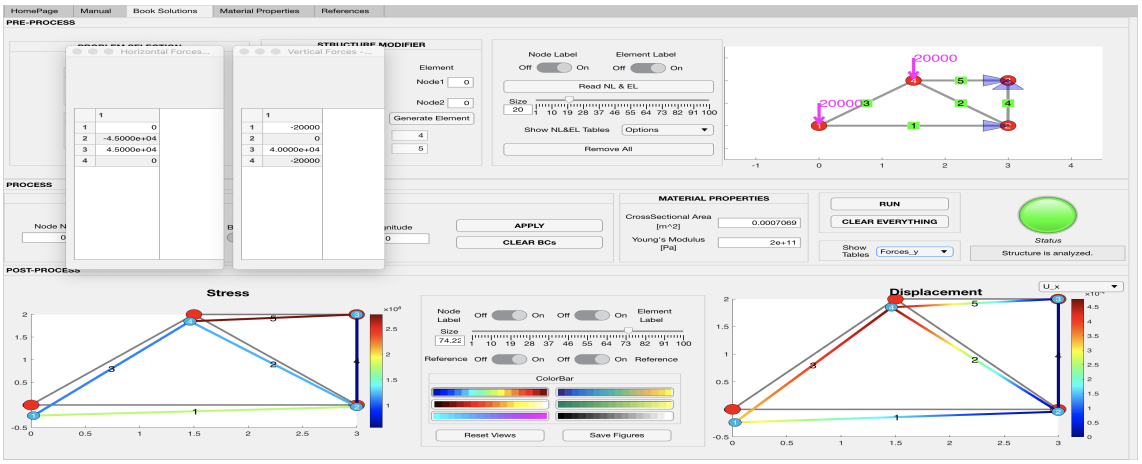
As you can examine, blue rectangles indicate the displacement constraints; and forces are also shown on the figure at the right top on the GUI.

After choosing our problem, the only thing needed to be done is to push the RUN button and analyze the structure. As a result, we can obtain the stress for each element and displacement values for the structure.



As it can be observed, the GUI allows us to examine the stress and displacement values in each direction; x and y.

The GUI also allows us to examine the reaction forces at the nodes 2 and 3 as shown below.



4 References

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