

StructInsight User Manual

The StructInsight software enables one to perform analysis of continuous beams using the stiffness matrix method. Professionals as well as students of structural engineering, architecture, aeronautical, and mechanical engineering would find this software to be very useful in their work.

Obtaining and Running the Software

- Only one file with the name *structinsight.jar* contains the entire software. You can download this file from [the github link here](#). This is an executable Java™ "Jar" file. (Java is a trademark of Oracle Corporation).
- To run the StructInsight program in this Jar file, you would require JRE (Java Runtime Environment) installed on your computer. If the JRE is not already installed, you may install it using the resources available at [this website](#).
- Once the JRE is installed and properly configured, there are two ways to run this program. The easier one is to open your file-explorer, navigate to the folder where you have stored the *structinsight.jar* file, and then double-click that file.
- Sometimes, however, the above method may not work. The alternative is to open the command window/terminal, change to the directory/folder where you have stored the *structinsight.jar* file, and then issue the following command.

`D:\your_folder> java -jar structinsight.jar`

- In either of the methods, the successful execution would be indicated by appearance of the program window on your monitor screen, as shown in the following figure.

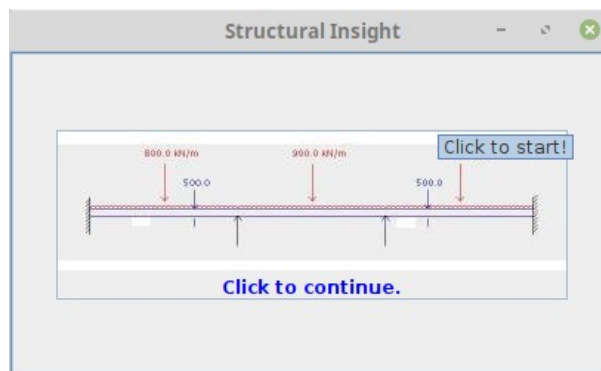


Fig. 1

- Click the continuous-beam icon button in the client area of this window, and you would get another window as shown in the following figure. It is a menu for the various actions we can take using this software.

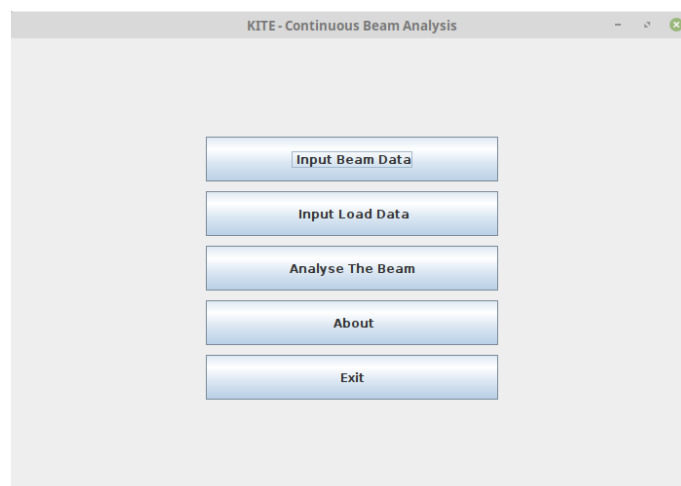


Fig. 2

- For the deeper understanding of the program capabilities, please refer the examples in the next section.
- If you want to study the source code of this program, then you may get it from [here](#).

Usage

We will consider two examples to understand the use of this program to solve our problems of continuous beam analysis. The first one is relatively simple, while the second one deals with slightly more complex beam and loading conditions.

Example 1

Example 2

Example 1:

I recommend to start with sketching the problem you intend to solve, on a plain paper. For the present example, I drew the following sketch-

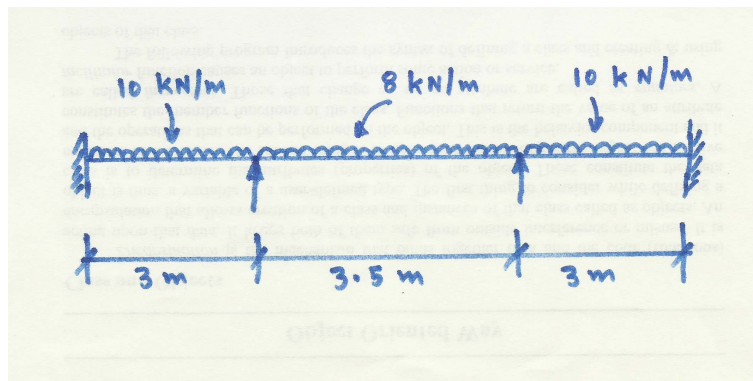


Fig. 3

To feed the data about this problem, we will use the first two options from the menu shown in Fig. 2 above. First click the 'Input Beam Data' button. This will popup the following window-

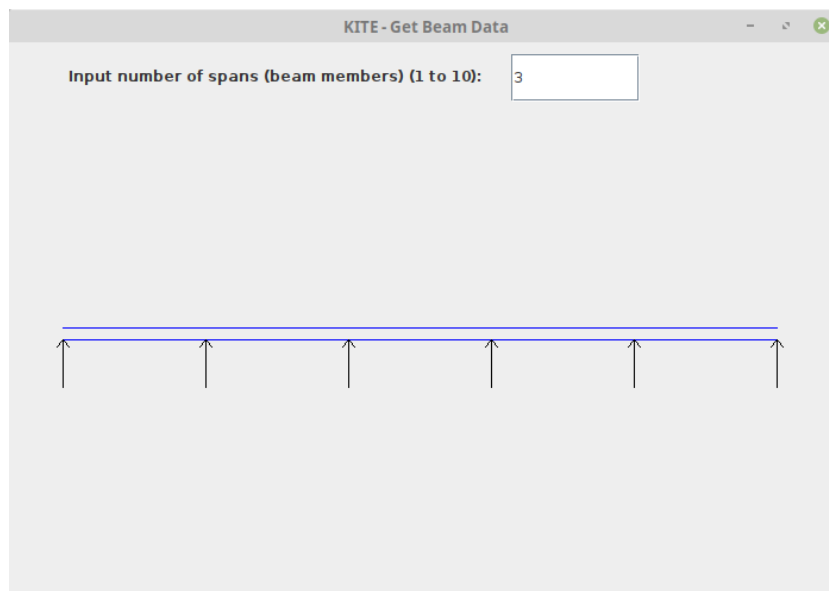


Fig. 4

The default number of spans is 5; but since there are 3 spans in our present example, I rubbed out **5** and typed **3** in the input text field. This '3' is being shown in the above figure. Press ENTER after typing the number of spans, and the appearance of the window will change to -

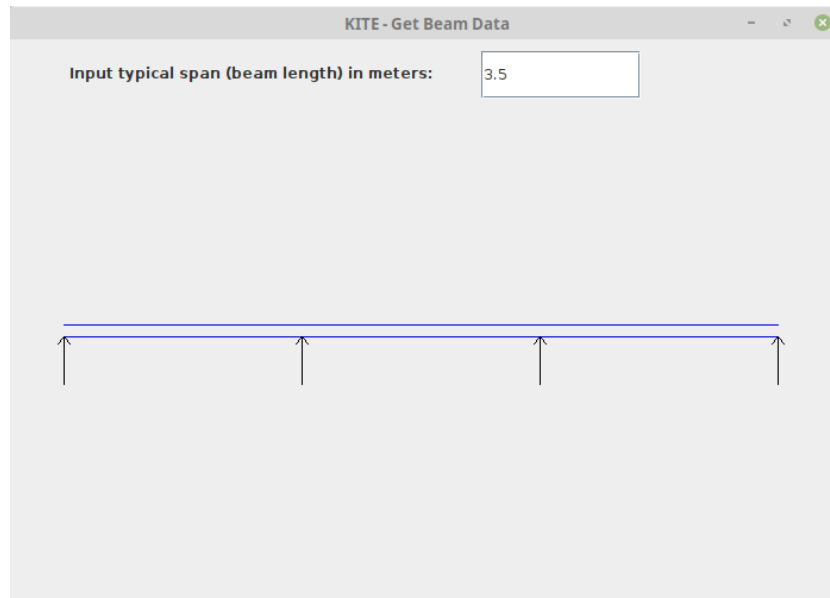


Fig. 5

In our example, there are two spans of 3.0 m length and one span of 3.5 m length, making '3 m' a better candidate to be typical. So, I rubbed out the pre-filled value **3.5** in the input textfield in the above figure and typed **3** there and pressed ENTER. This caused the window to show three new textfields, one under each span, with pre-filled values of **3.0**. I clicked in the textfield under the middle span and changed the value there to **3.5**. This is shown in the following figure.

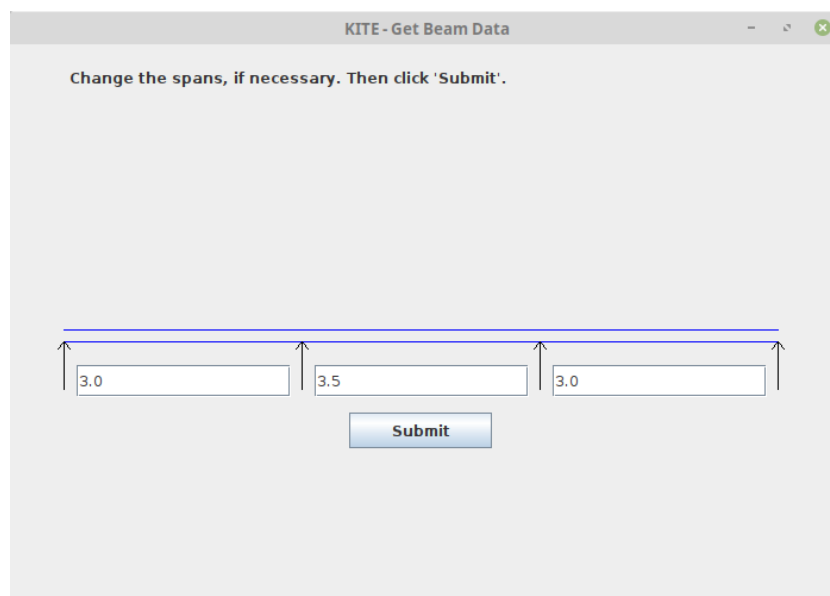


Fig. 6

I then clicked the 'Submit' button, and the appearance of the window changed again. It now asks about modulus of elasticity property of the members.



Fig. 7

I kept the default value **34500** MPa (modulus of elasticity of typical concrete used in India) and pressed ENTER. The window changed to -

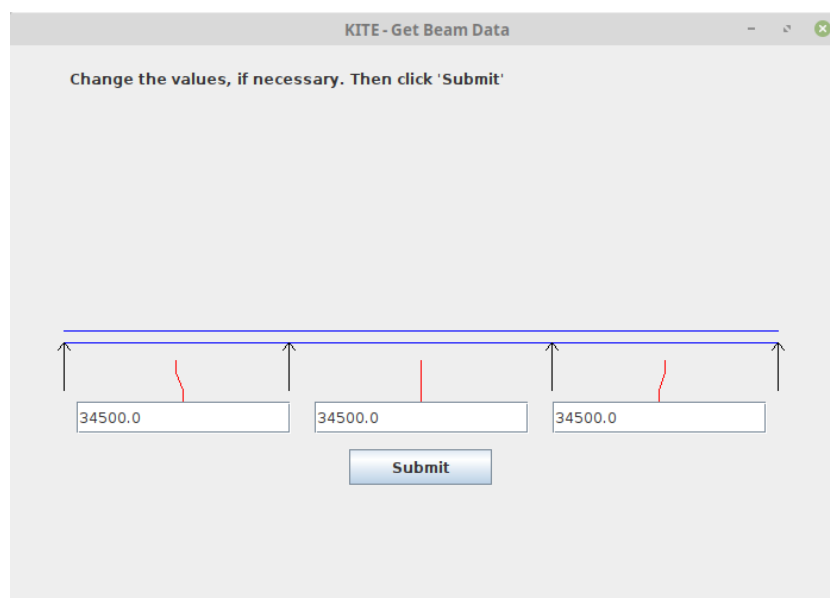


Fig. 8

I simply clicked the 'Submit' button, and the window changed to enquire about moment of inertia properties of the members.

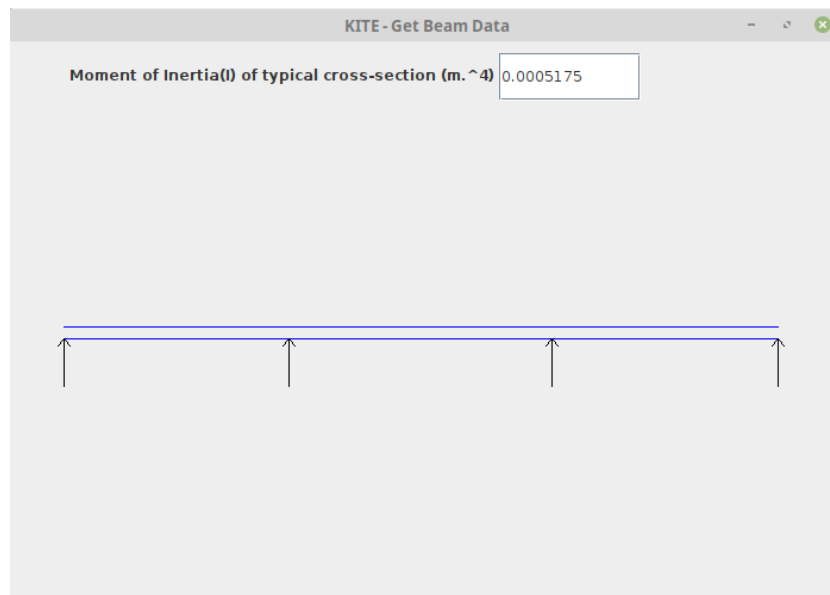


Fig. 9

I kept the pre-filled value 0.0005175 m^4 (M.I. of section 230X300) and pressed ENTER.

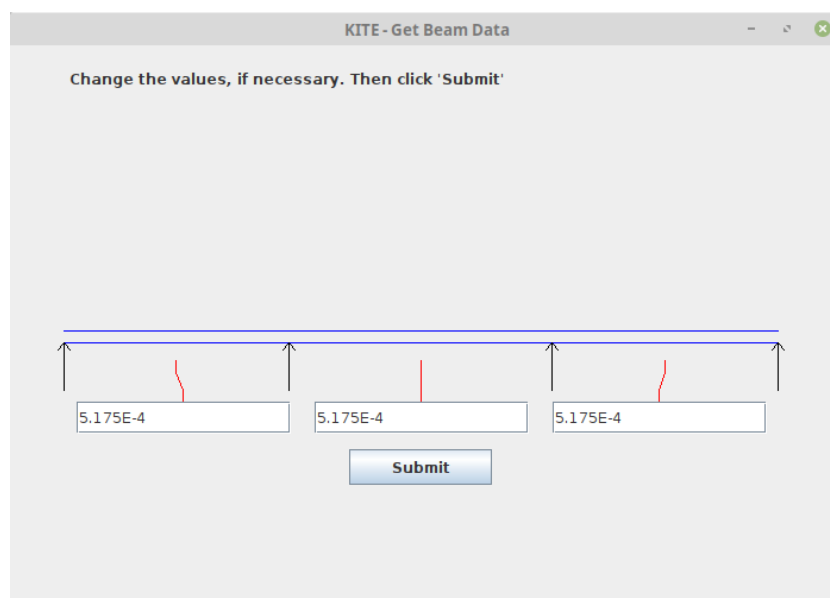


Fig. 10

Here, I clicked the 'Submit' button, to get new appearance in the window, as shown in the following figure -

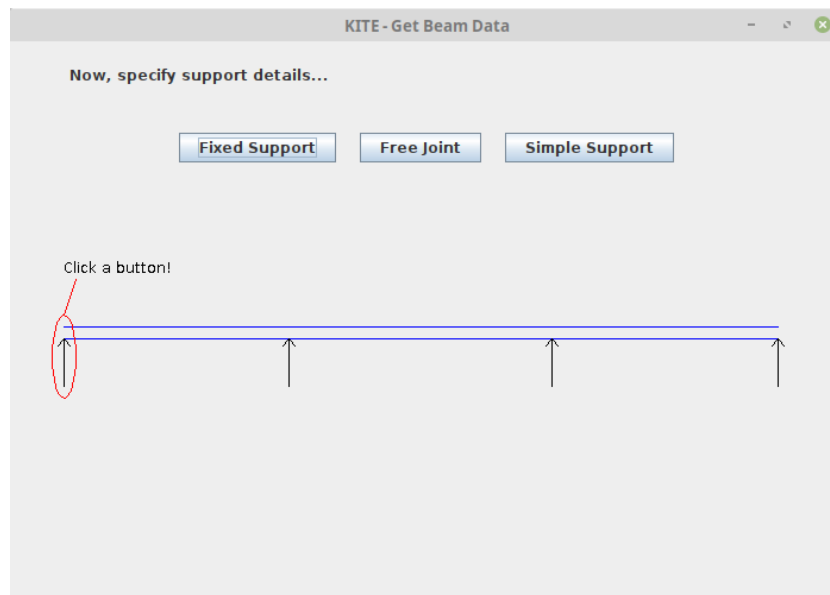


Fig. 11

I clicked the 'Fixed Support' button for the left-most joint (high-lighted in the above figure) and once more for the right-most joint (high-lighted in the following figure).

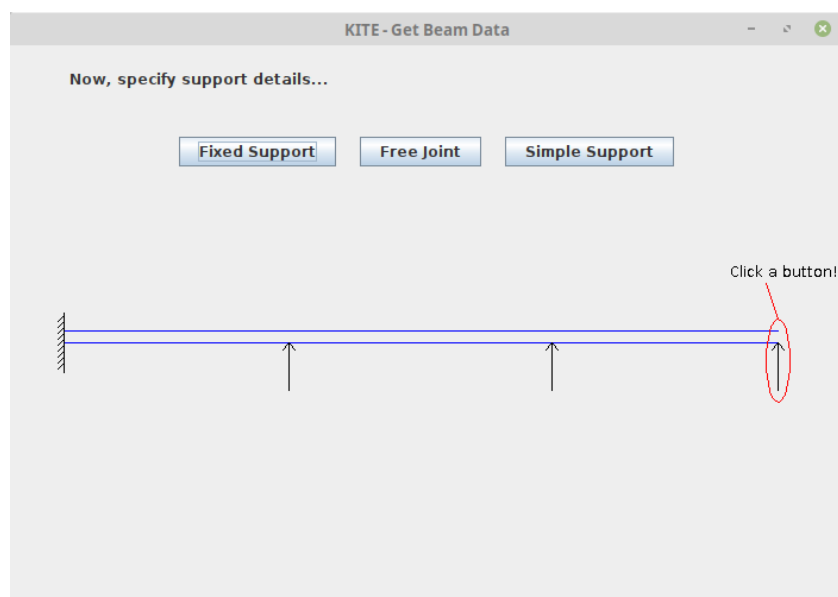


Fig. 12

For the remaining two joints (one of which is high-lighted in the following figure), I clicked on 'Simple Support' button.

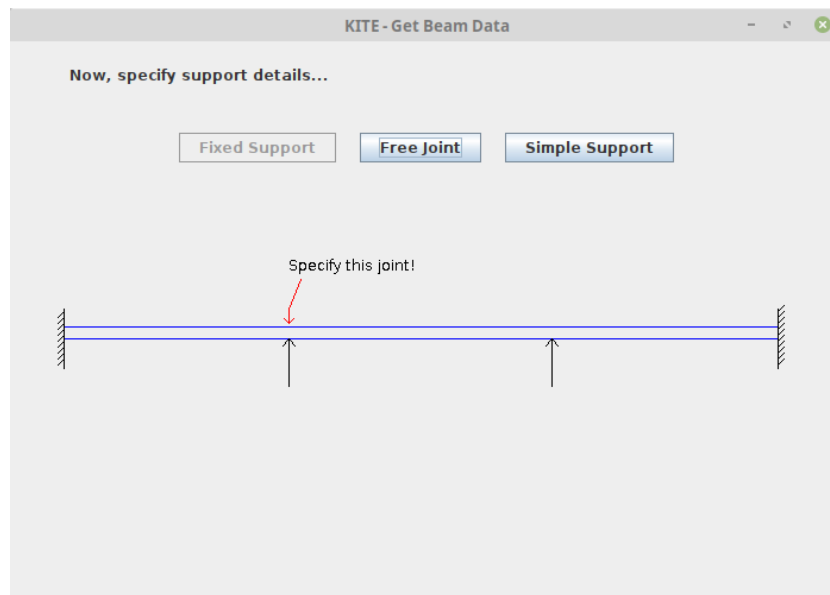


Fig. 13

Beam data entry module gets over here, and the appearance of the window is as shown in the following figure. Click the 'Exit' button here, causing this window to get closed and return to the window showing the main menu (Fig. 2). If any error occurred during the beam data input, you may again click the 'Input Beam Data' button in the main menu and specify the correct data.

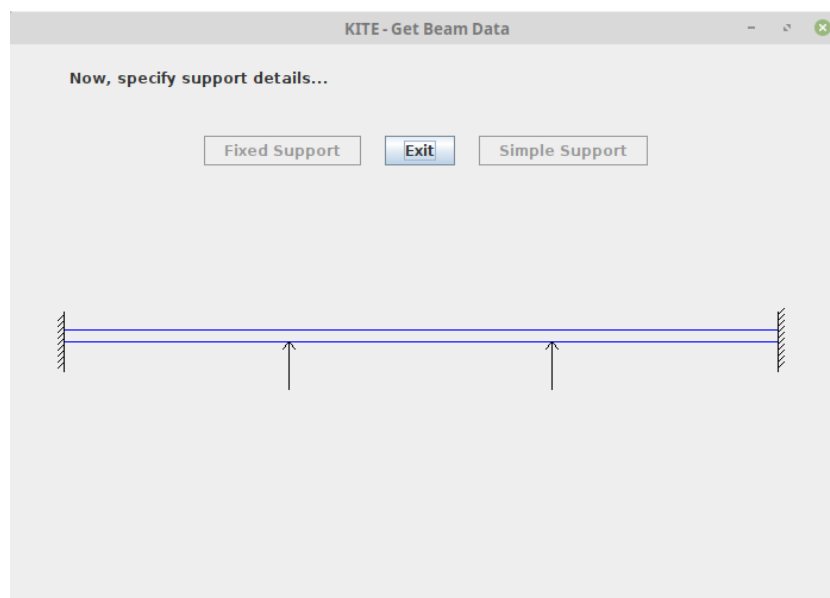


Fig. 14

Having defined the beam geometry, we are now ready to input the loading details. Click the 'Input Load Data' button in the main menu, to specify loads. A window will appear as shown in the following figure. We can specify three types of loads, viz.,

- Joint actions (both - vertical loads & moments)
- Uniformly distributed loads
- Point loads (vertical) on the members of the continuous beam

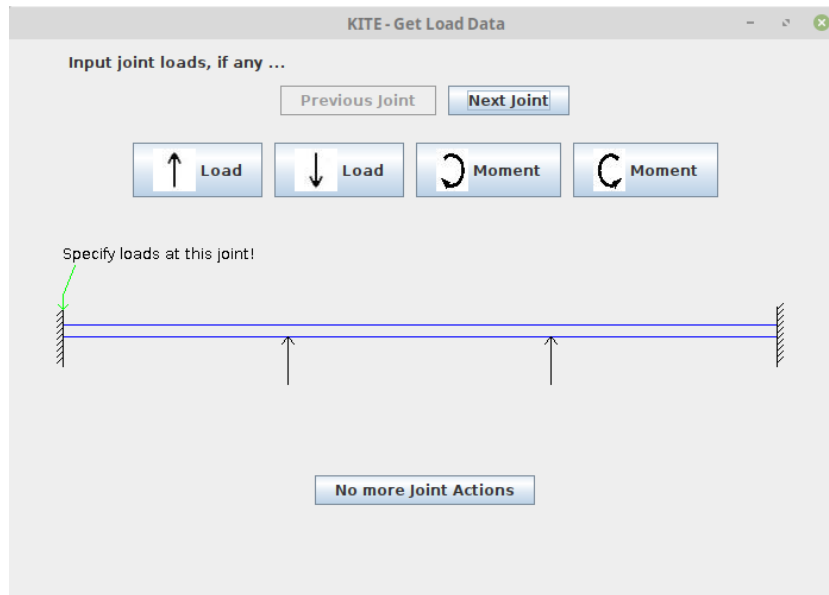


Fig. 15

In the present example, there are no joint loads; so I clicked the 'No more Joint Actions' button and the window changed to the one shown in the following figure.

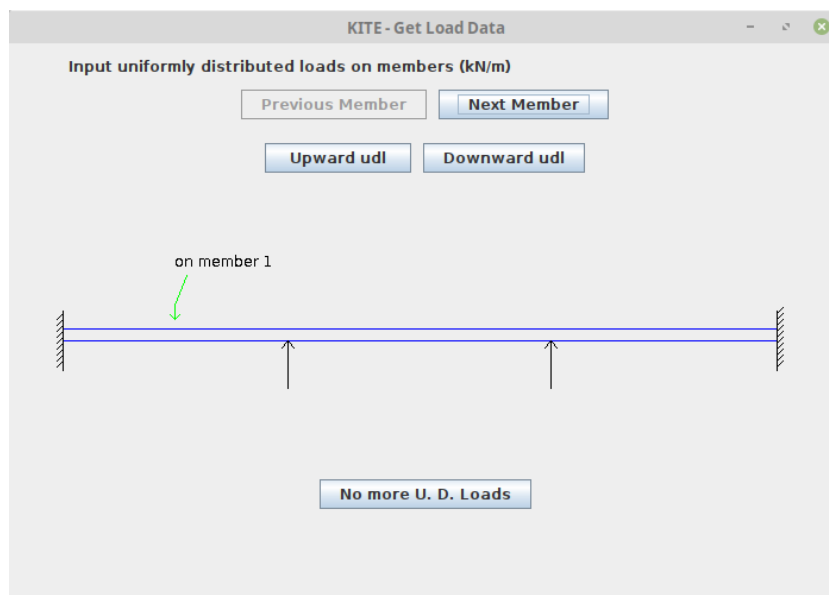


Fig. 16

Here, I clicked the 'Downward udl' button which caused a dialog box, as shown in the following figure, to pop up. I typed there 10, the value of udl in kN/m units on the first member of the present example and then pressed ENTER (you may click 'OK' button also).

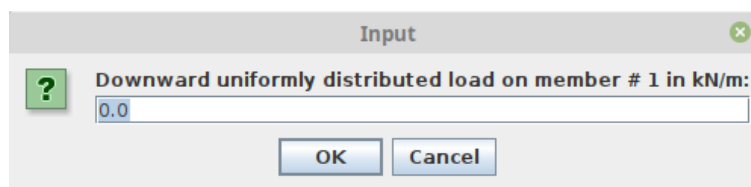


Fig. 17

Then I clicked 'Next Member' button to specify udl on the second member. The appearance of the window was as shown in the following figure.

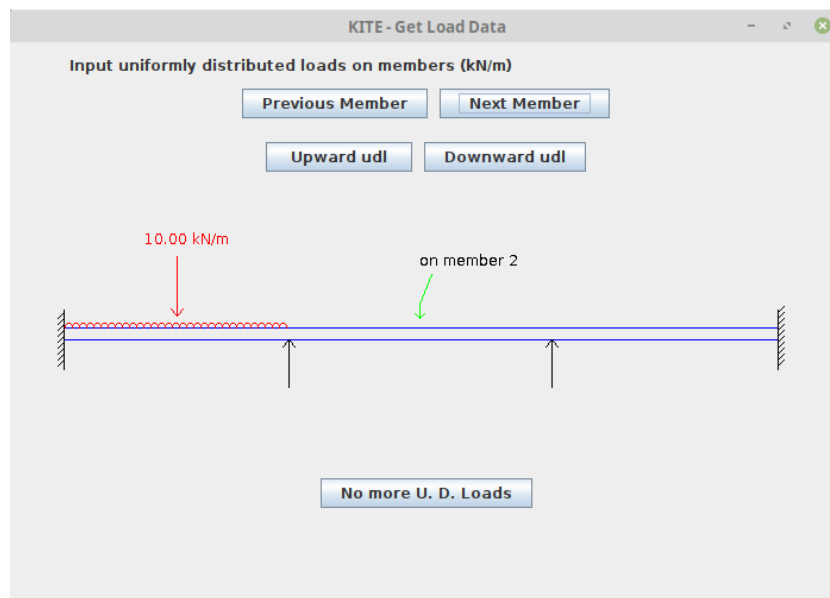


Fig. 18

By similar procedure, I specified the udl of 8 kN/m on the second member and 10 kN/m on the third member. At this stage, the window looked like the one shown in the following figure.

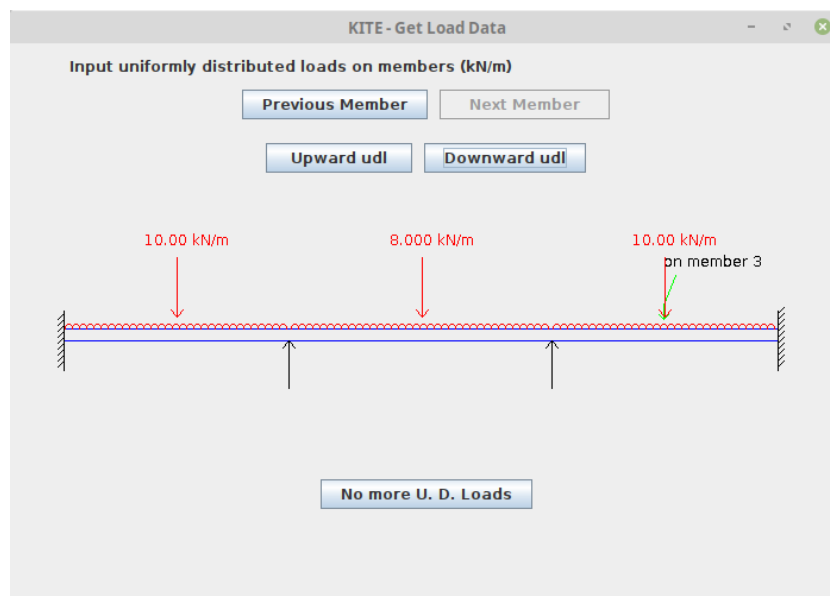


Fig. 19

I then clicked the 'No more U. D. Loads' button and the window got ready for accepting the third type of loads. Its appearance changed to the one shown in the following figure.

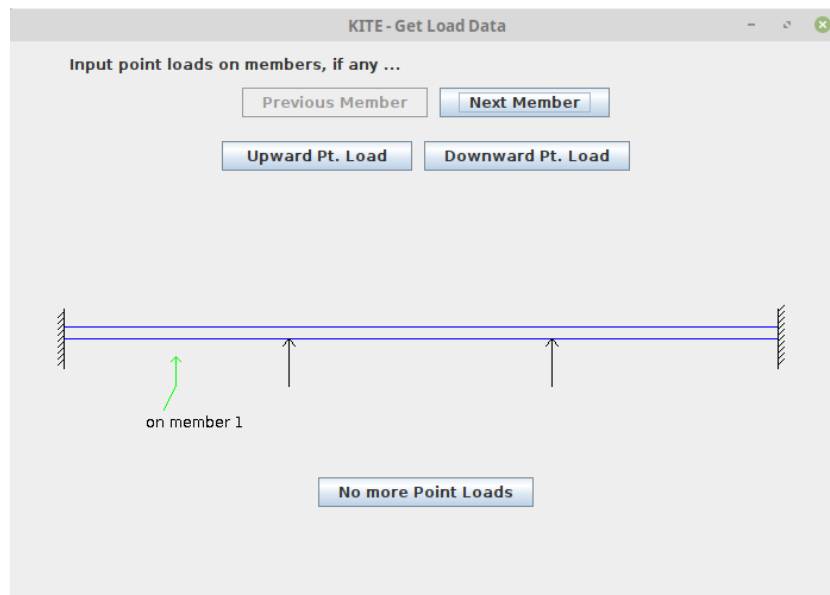


Fig. 20

As there are no point loads in the present example, I clicked the 'No more Point Loads' button, closing this window and returning to the one showing the main menu.

Now we are ready for the analysis. So, click 'Analyse The Beam' button in the main menu (Fig. 2) and a window as shown in the following figure pops up.

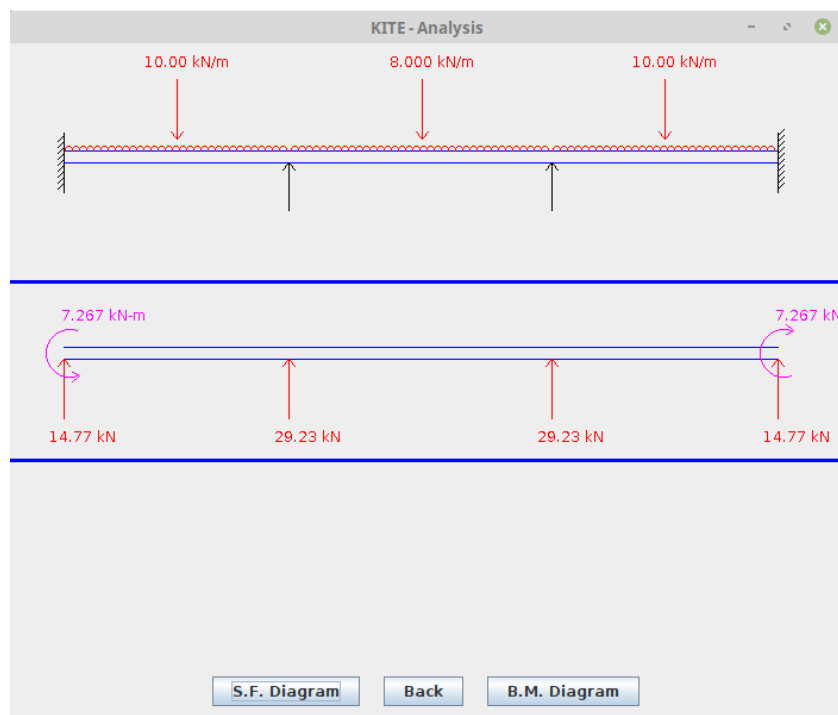


Fig. 21

The loading diagram & the joint reactions diagram are shown by default. To view shear force diagram, I clicked the 'S. F. Diagram' button. The view on the window changed to the one shown in the following figure.

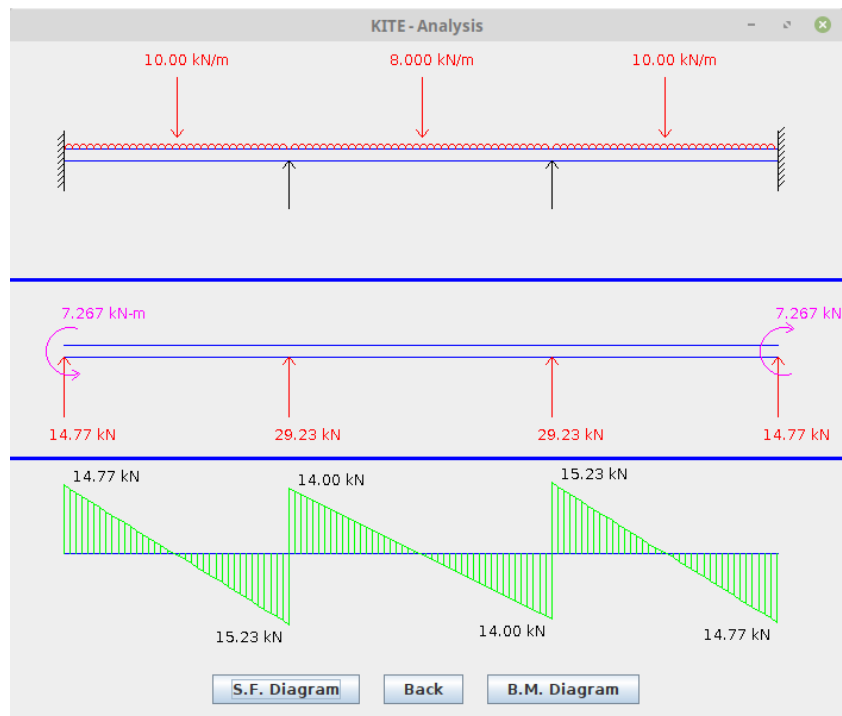


Fig. 22

The diagram shows the pattern of shear force changes at various section along the length of the continuous beam and also shows the shear force values at all the joints.

To view bending moment diagram, click the 'B. M. Diagram' button. The window view changes to the one shown in the following figure.

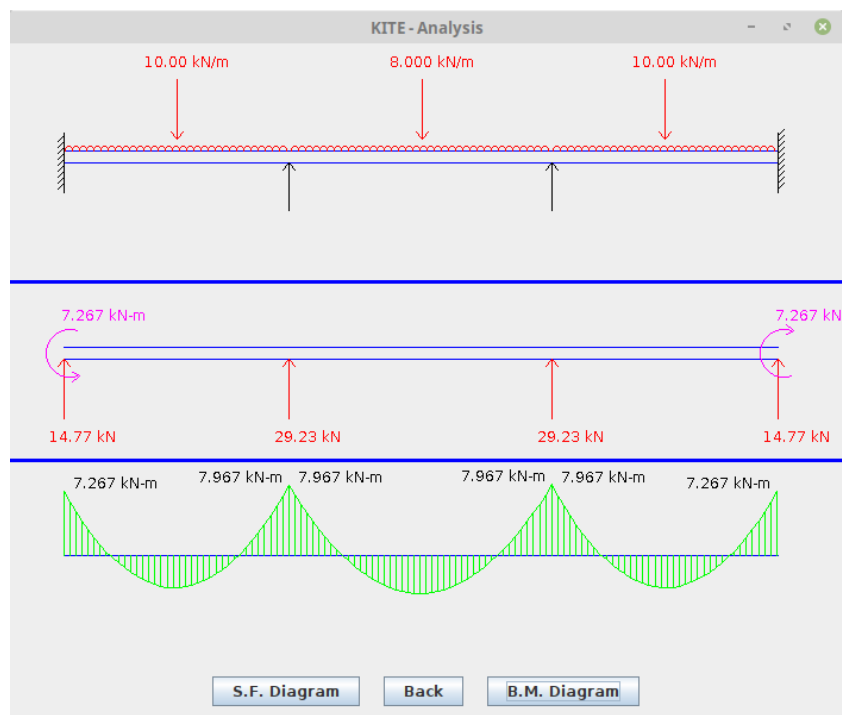


Fig. 23

The figure shows the pattern of bending moment changes along length of the beam and also values of bending moments at all the four joints.

Example 2:

Before using the program, keep your problem statement ready. In my opinion, the most convenient way to do that is to draw a sketch for the problem you intend to solve. For this second example, my sketch was -

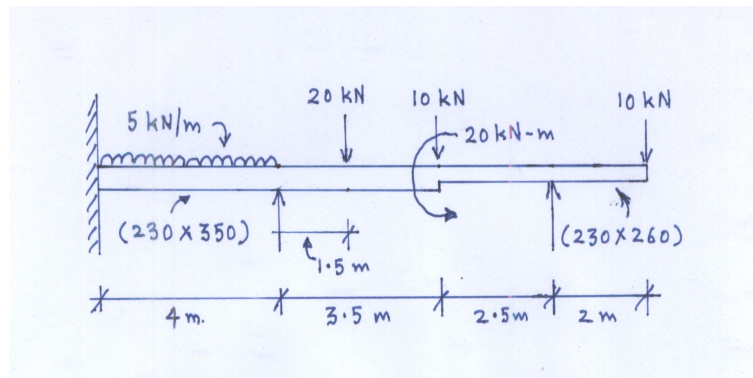


Fig. 24

In this example, there are three beam members. But, as the middle member consists of two parts with different sizes, we will consider a joint at the point where the member changes its size. Due to this the continuous beam now is made up of 4 members.

So, in the first window that pops up after clicking the 'Input Beam Data' button in the main menu (see Fig. 2), answer the question 'Input number of spans (beam members) (1 to 10):' by **4**, since we have four members in this example, as discussed above.

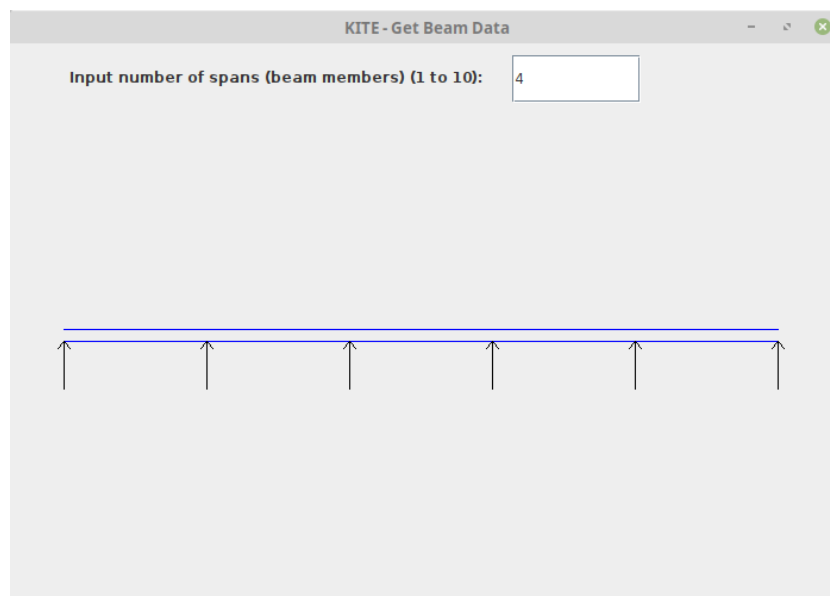


Fig. 25

When the program comes to the stage as shown in the window in the following figure, type spans **4**, **3.5**, **2.5**, and **2** in the consecutive input boxes.

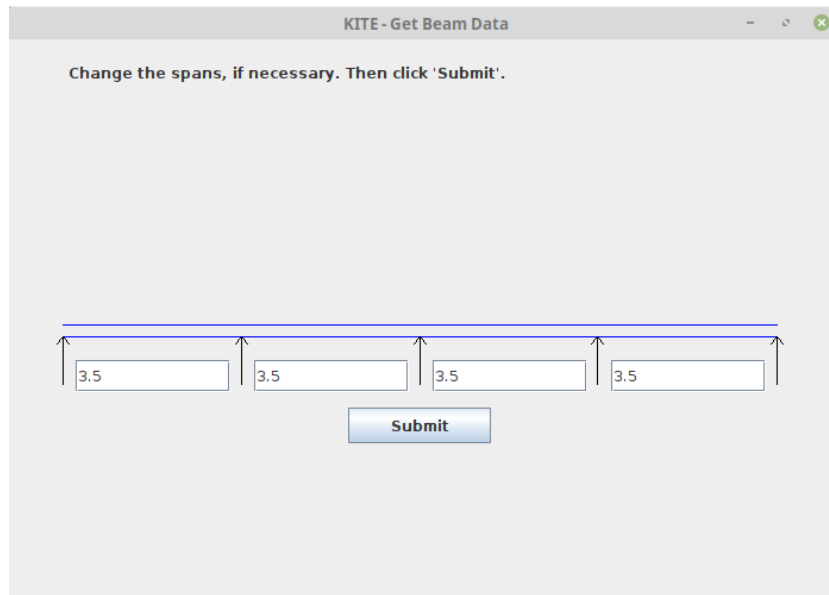


Fig. 26

After typing those values, the window would appear as shown in the following figure.

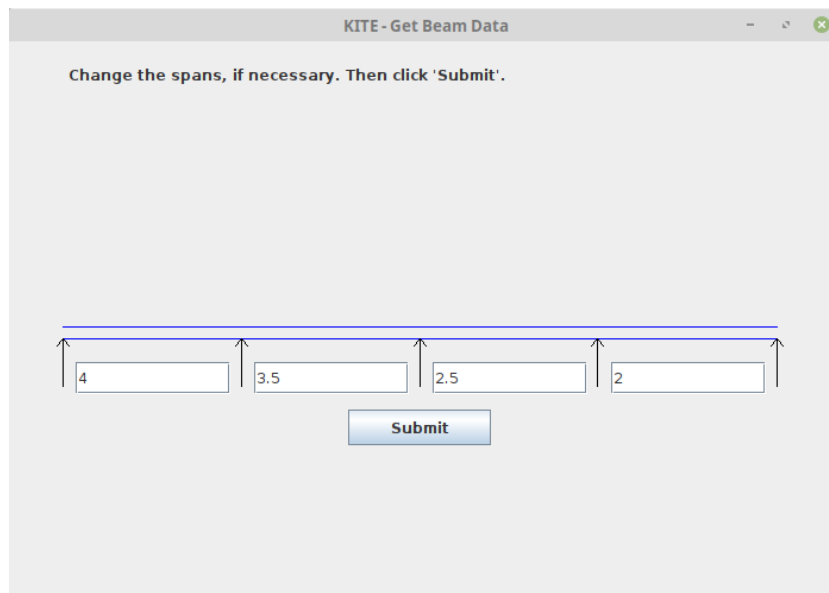


Fig. 27

Click the 'Submit' button and the window as shown in the following figure would appear. As all members of the beam are of the same material, retain the default value of modulus of elasticity (34500 MPa), by clicking the 'Submit' button in this window also.

KITE - Get Beam Data

Change the values, if necessary. Then click 'Submit'

Diagram showing a horizontal beam with four vertical arrows pointing upwards. Below each arrow is an input box containing the value 34500.0. A red line connects each arrow to its corresponding input box. A Submit button is located below the input boxes.

Fig. 28

When the program asks you about typical moment of inertia value, type **0.0008218** (M.I. of 230X350 section in m^4 units) and then press ENTER. The window would now appear as shown in the following figure.

KITE - Get Beam Data

Change the values, if necessary. Then click 'Submit'

Diagram showing a horizontal beam with four vertical arrows pointing upwards. Below each arrow is an input box containing the value 8.218E-4. A red line connects each arrow to its corresponding input box. A Submit button is located below the input boxes.

Fig. 29

Change the values in the last two input boxes to **0.0003369** (M.I. of 230X260 section in m^4 units). The window would now appear as shown in the following figure.

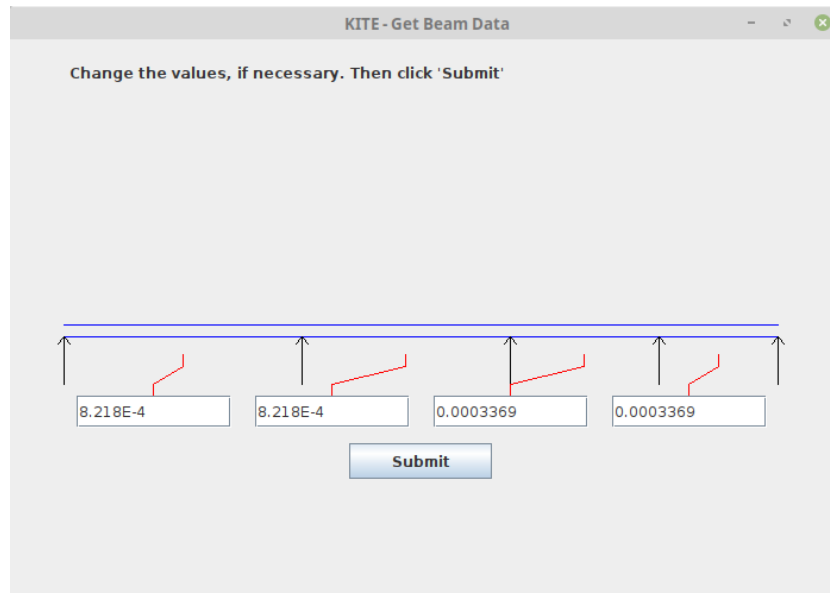


Fig. 30

After clicking the 'Submit' button in the above figure, the program would ask about support conditions. Click the button titled 'Fixed Support' for the first joint and 'Free' for the last joint. When the marker arrow is on the second, third, and the fourth joint, click buttons titled 'Simple Support', 'Free', and again 'Simple Support' respectively. Some stages during this interaction are shown in the following four figures.

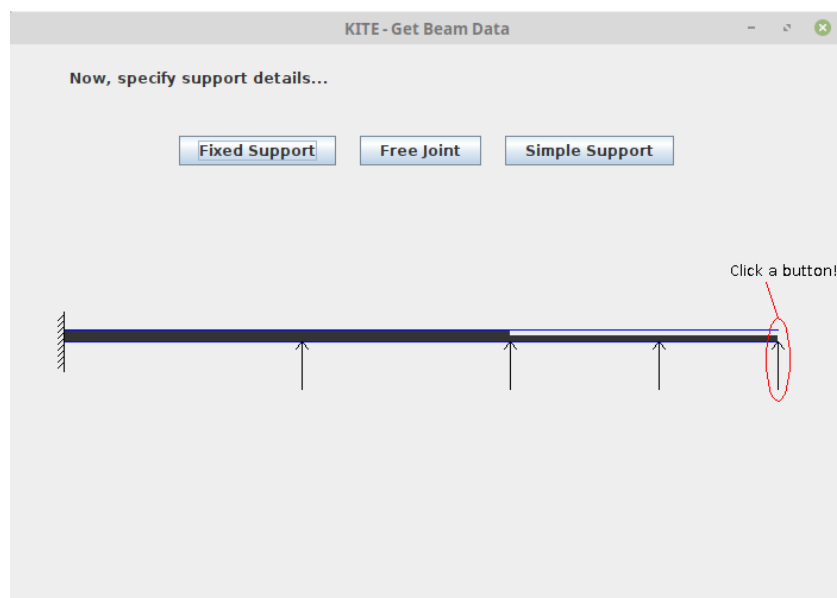


Fig. 31

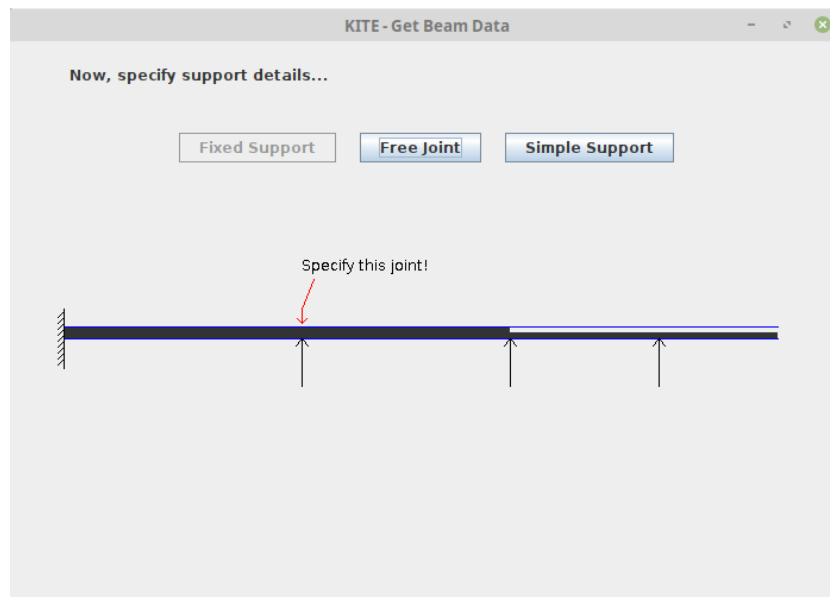


Fig. 32

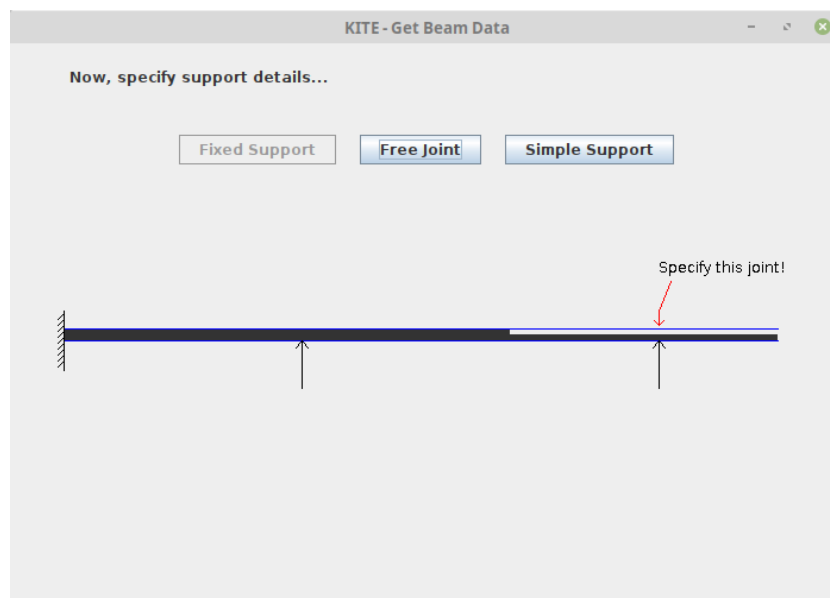


Fig. 33

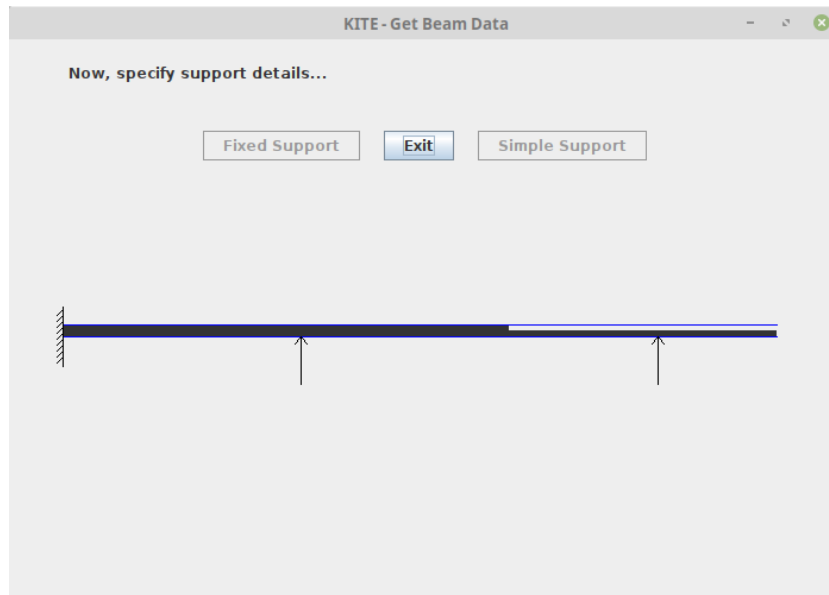


Fig. 34

After this click the 'Exit' button to close the current window and return to the menu window (Fig. 2).

We have to specify loads on the beam now. So, click 'Input Load Data' button in the menu window. This will open a new window. An arrow marker would be pointing to the first joint. By clicking the 'Next Joint' button twice, it has been taken to the third joint and the resultant appearance is depicted in the following figure.

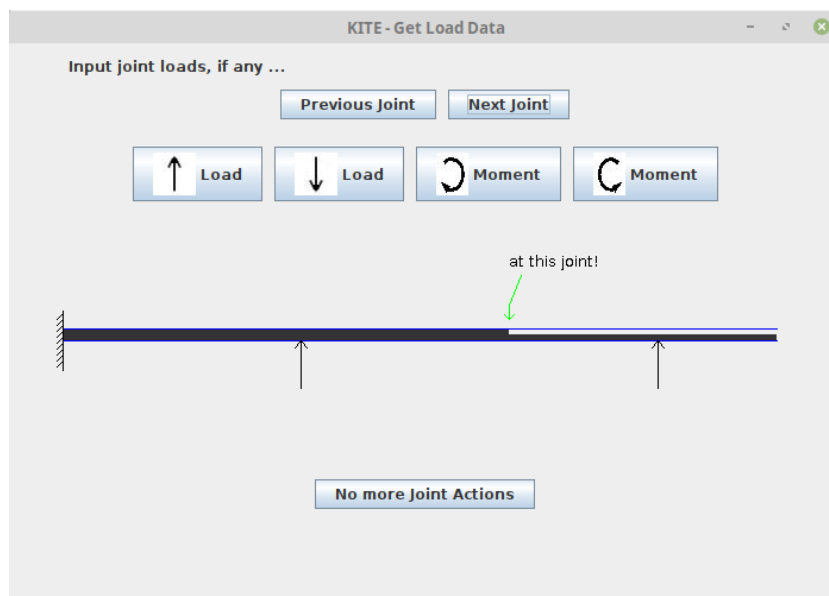


Fig. 35

Here, click the button showing downward load icon. An input dialog box would pop up, as shown in the following figure. Type 10 (the value of vertical load at this joint, in our present example) and press ENTER.

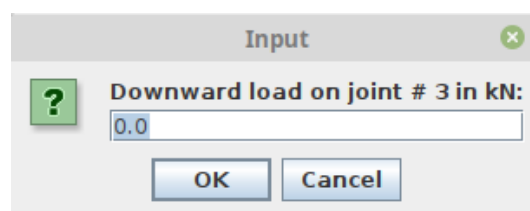


Fig. 36

The dialog box in the previous figure will vanish. Next, click the button having anti-clockwise moment icon on its face and another input dialog box would pop up. Type **20** (the value of joint moment in our example) in the input box. The dialog box would look at this stage, as shown in the figure below.

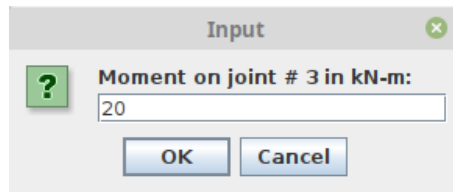


Fig. 37

Click the 'OK' button and the input dialog box would vanish. To give the last joint load in the present example, click the 'Next Joint' button twice; this would bring the marker arrow on the last joint. The window would appear as shown below.

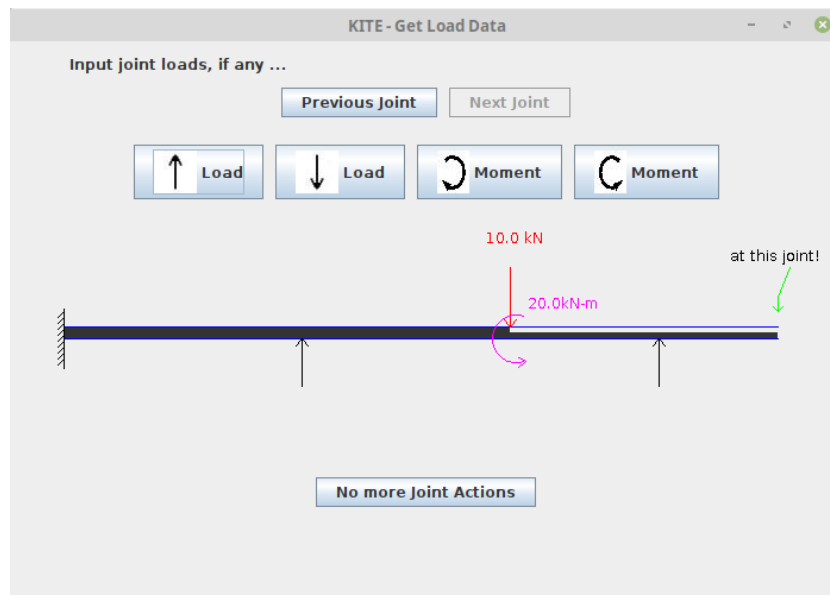


Fig. 38

Once again, click the button having downward load icon on its face; in the resultant input dialog box, type **10**, and press ENTER. After this, the window should appear as shown in the next figure.

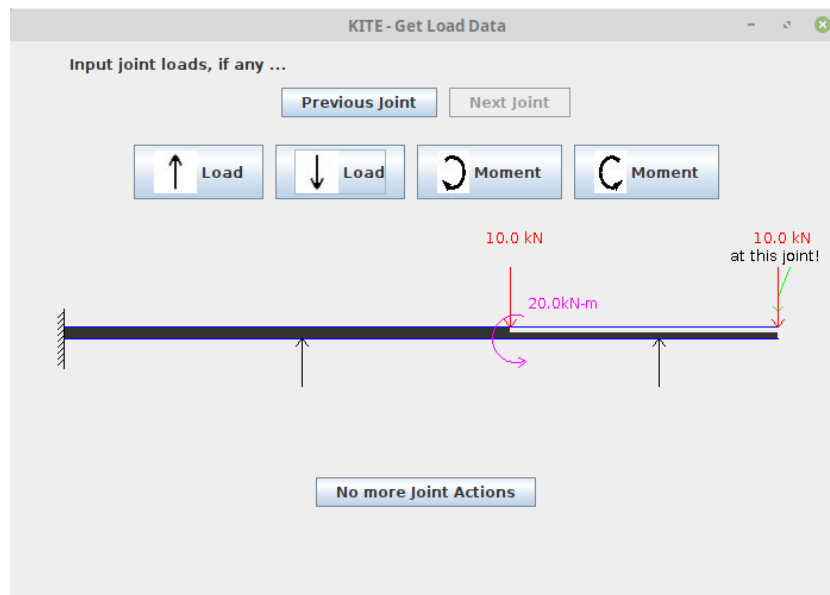


Fig. 39

Click the button labeled 'No more Joint Actions' and the program gets ready to accept data about uniformly distributed loads (u.d.l.) acting on the members. The window would look like the one shown in the following figure.

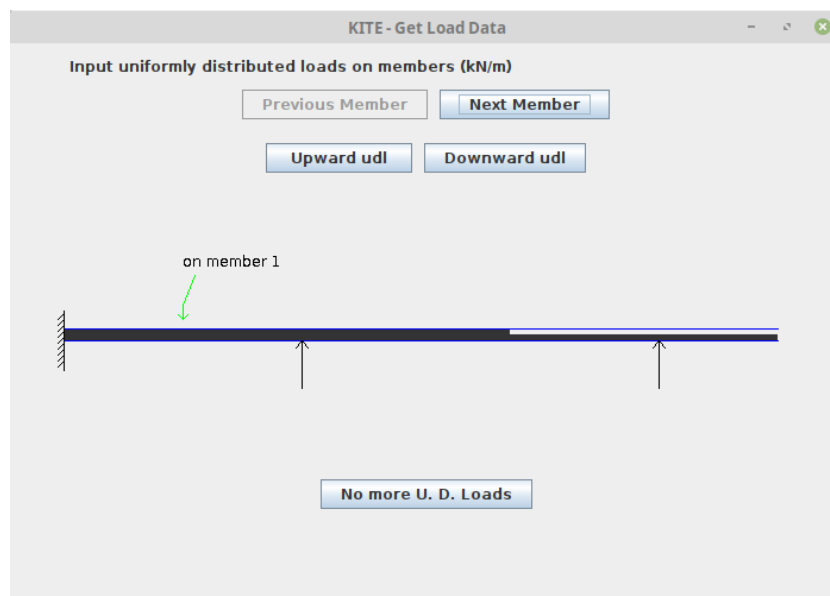


Fig. 40

In our example, there is a udl only on the first member. So, click the 'Downward udl' button; in the ensuing input dialog box, type 5, and press ENTER. The resultant window will be as shown below.

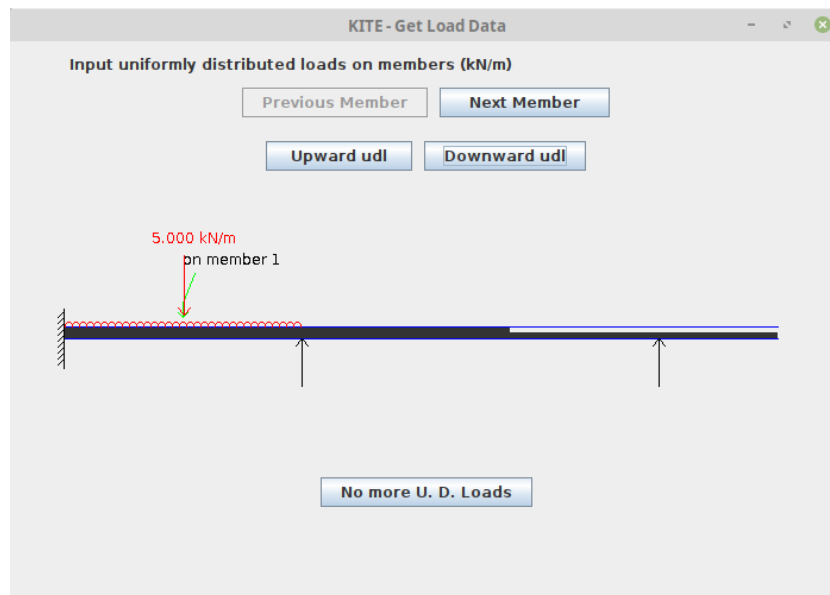


Fig. 41

Click the button labeled 'No more U.D. Loads' and next get ready to specify the point loads on the members. In our case, there is only one member point load. So, by clicking the 'Next Member' button, move to the second member. The window would appear as shown in the following figure.

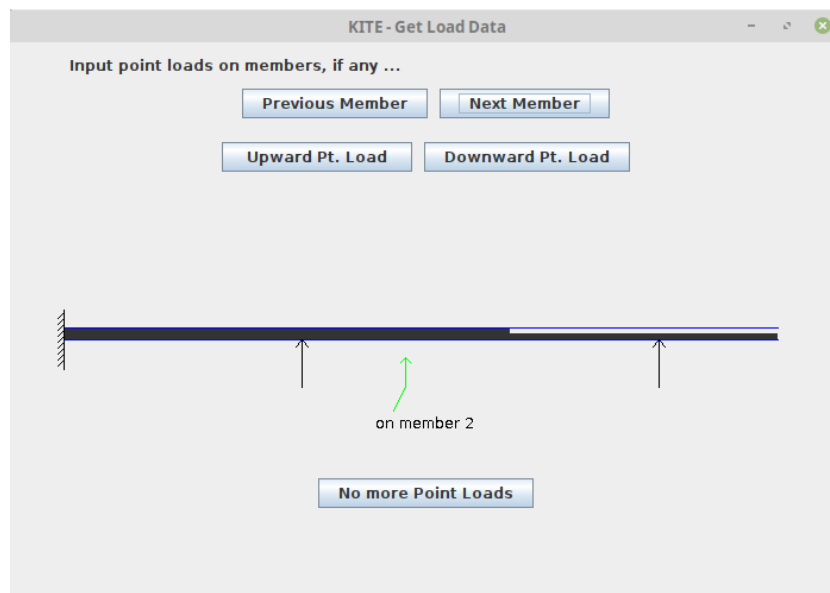


Fig. 42

Click the 'Downward Pt. Load' button and an input dialog box as shown in the following figure would appear.

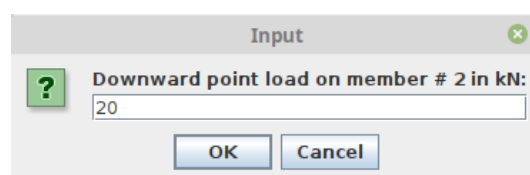


Fig. 43

Specify the point load value of **20** kN there and click the 'OK' button. This would cause the above dialog box to vanish and automatically another dialog box, as shown in the following figure, to appear.

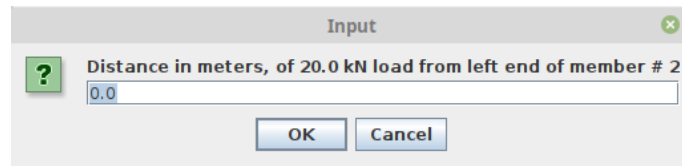


Fig. 44

Here, type **1.5**, the distance of the point load from the left end of the second member and press ENTER. The dialog box would go away and the window would look like the one shown in the following figure.

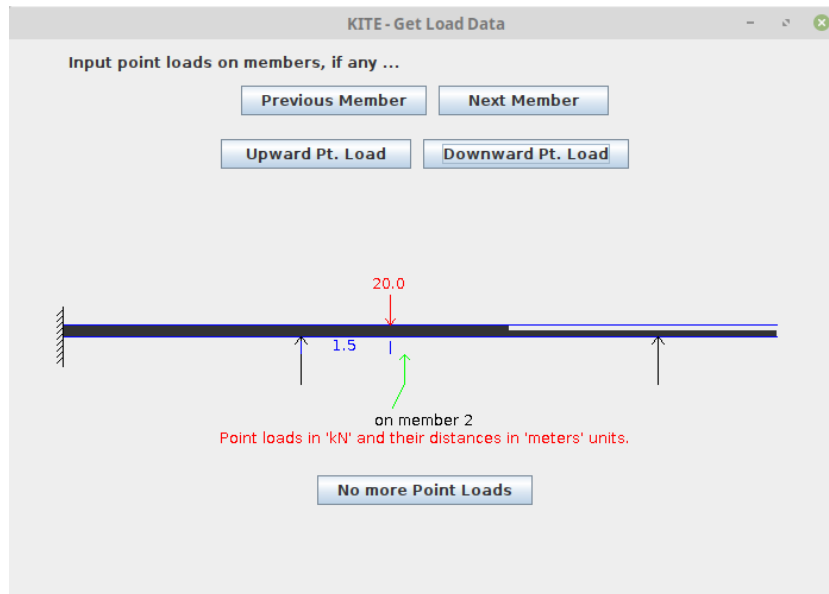


Fig. 45

As there are no more point loads, click the 'No more Point loads' button to return to the menu (Fig. 2) window.

Now to perform the desired analysis, click 'Analyse The Beam' button in the main menu and a window as shown in the following figure becomes visible. It is showing the load diagram and the joint reactions diagrams.

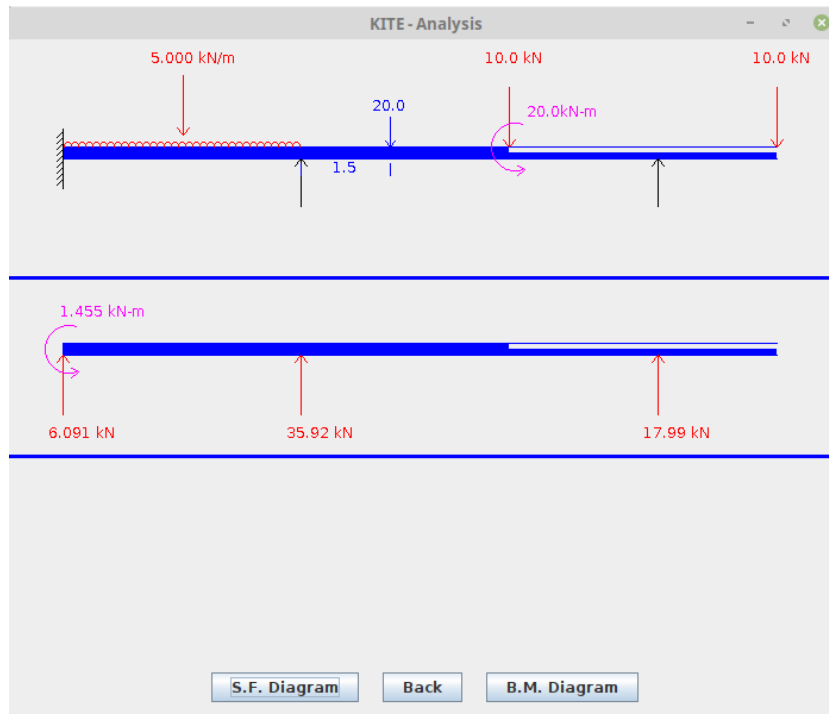


Fig. 46

On clicking the 'S.F.Diagram' button, the shear force diagram will be displayed. The window would appear as shown in the following figure.

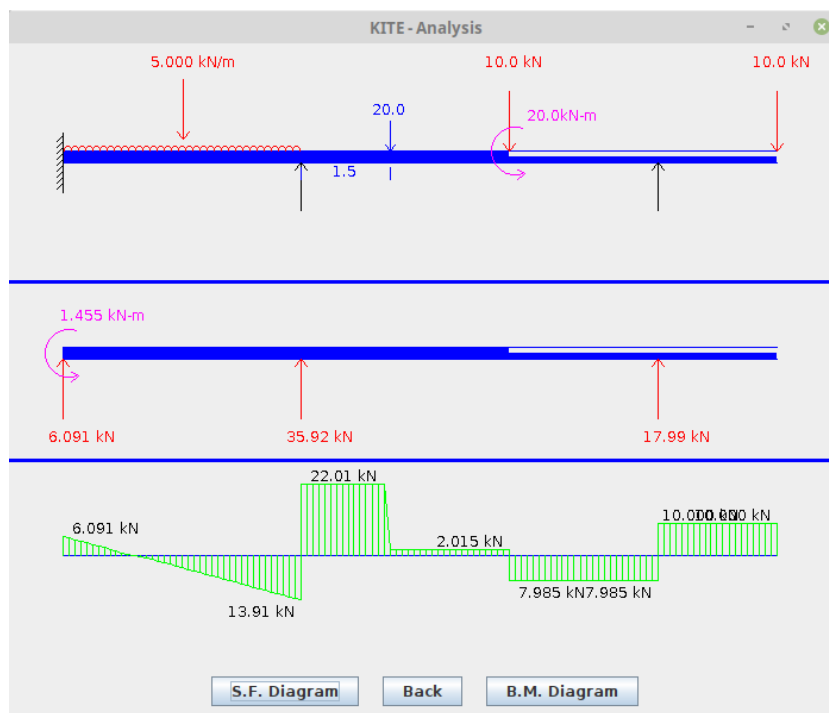


Fig. 47

To see the bending moment diagram, click the 'B.M.Diagram' button. The window would now look as shown below.

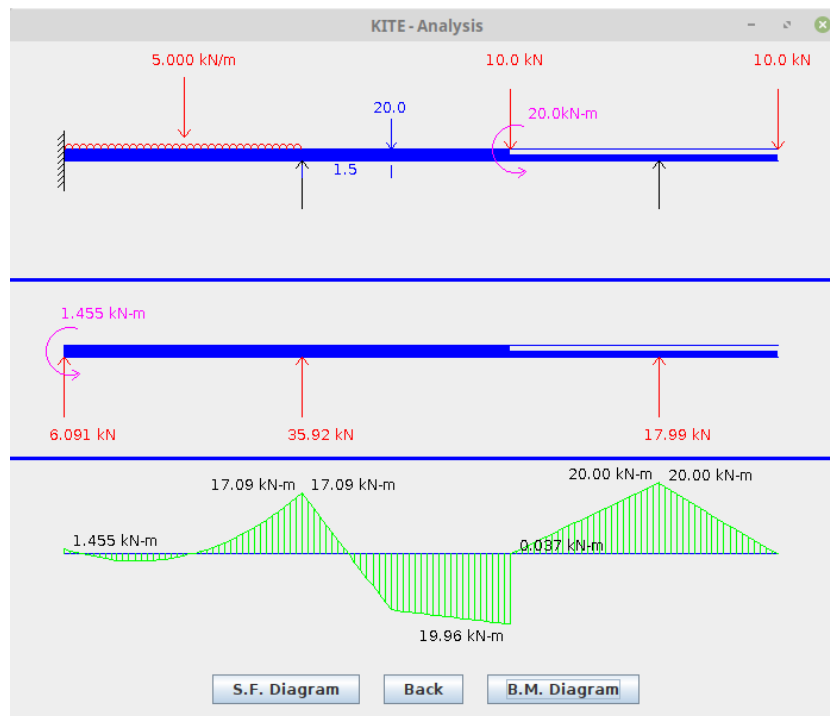


Fig. 48

Clicking the 'Back' button closes this window and returns to the menu window. Clicking the 'Exit' button in the menu window, closes that window and exits the program.

--- 000 ---