StructInsight (Python) User Manual

Welcome to *StructInsight*, an easy-to-use structural analysis software with intuitive input mechanism. The current version of the software deals with only one type of structure and that is *Continuous Beam*. The analysis is performed 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. Earlier, this software was written in *Java* programming language (available on github). However, it is observed that from the last several years *Python* programming language is becoming quite popular among software developers. This surge in popularity and widespread adoption among software developers, is attributed particularly to *Python*'s robust suite of tools and libraries tailored for cutting-edge domains of Computer Science like Artificial Intelligence (AI) and Machine Learning (ML). *StructInsight* was, therefore, rewritten in *Python* with the belief that the students of the above referred core engineering branches would find it:

- a resource to hone their Python programming skill from the comfort of their familiar domain.
- a solid stepping stone for building software to analyse and design other types of structures.
- a brain-storming mechanism giving them ideas for bringing AI/ML to their own field.

Obtaining and Running the Software

- To run this software, your computer system should have *Python* environment installed. If it is not already installed, you can easily install it from the Internet. There are several resources available to download and install the *Python*. One such resource is the official Python download site. If you are new to this process, then search the Internet and you would find hundreds of tutorials and videos explaining the process. Here is a link to one of these tutorials: Python 3 Installation and Setup Guide. Please note that the *StructInsight* software was developed on *Python 3.8* and so any version greater than or equal to 3.8 would be acceptable.

- With proper *Python* environment available, installing the *StructInsight* software is very easy! It is available on the Internet in PyPI repository, with package name continuousbeam. To install it, just issue the following command from the terminal.

  $ pip install continuousbeam ↵

- This command will install the latest *continuousbeam* package in your *Python* environment. This package is dependent on two packages, viz., *continuousbeam-backend* and *numpy*. The above command automatically adds these two packages to your *Python* environment. So in future, for any reason, if you want to remove the *StructInsight* software from your system, you will have to issue the following three commands from the terminal.

  $ pip uninstall continuousbeam ↵

  $ pip uninstall continuousbeam-backend ↵

  $ pip uninstall numpy ↵

- Once installed, you can invoke this program from any folder (directory) on your *Python* environment. Use the command in one of the following two formats to run the program.

  $ python -m continuousbeam.contibeam_main ↵

  or

  $ contibeam ↵

- The successful execution of the command would be indicated by appearance of the program window on your monitor screen, as shown in the following figure. The window provides the menu for various actions we can take using this software.
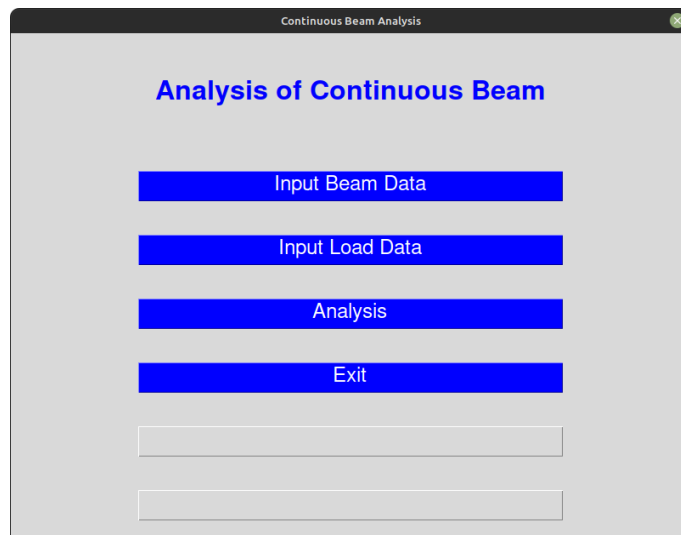
Fig. 1

- For 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 obtain it from <u>here</u>.

Usage

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

<u>Example 1</u>

<u>Example 2</u>

Example 1:

I recommend to start with sketching the problem you itend to solve, on a plain paper. It acts as an easy reference while inputting the data. For the present example, I drew the following sketch-
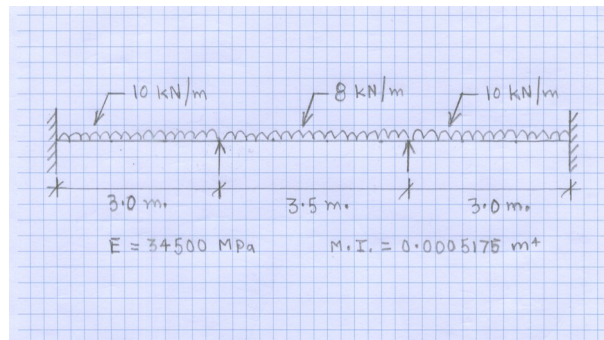
Fig. 2

To feed the data about this problem, we will use the first two options from the menu shown in Fig. 1 above. First click the 'Input Beam Data' button. This will popup the following window-
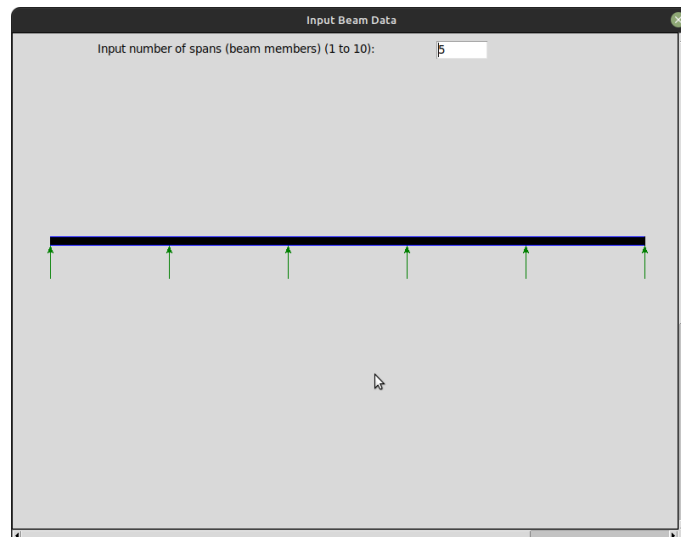
Fig. 3

The software considers 5 as the default number of spans; but since the continuous beam in our present example has 3 spans, I rubbed out 5 and typed 3 in the input text field. This '3' is being shown in the following figure.
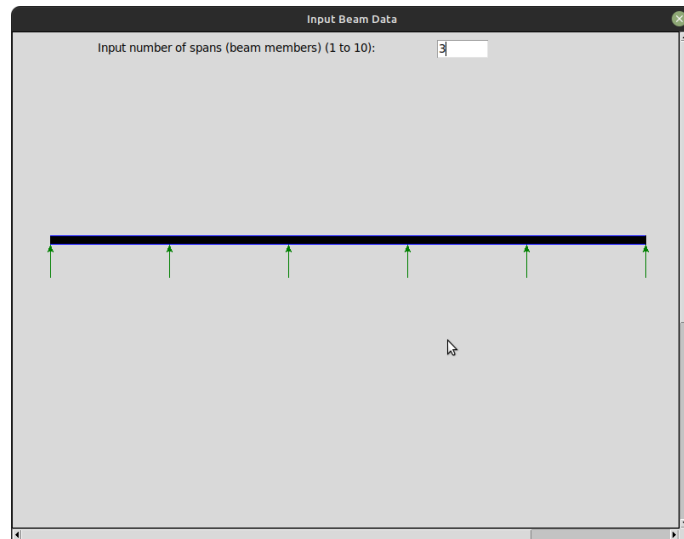
Fig. 4

 Press ENTER after typing the number of spans (3 in present case), and the appearance of the window will change to -
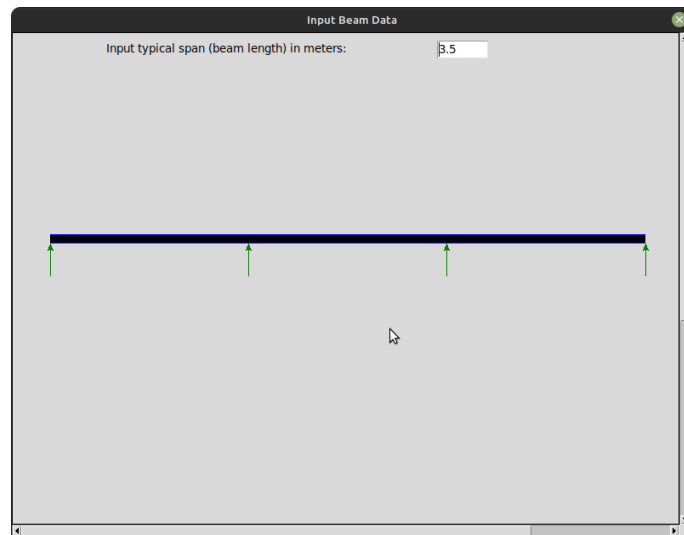
Fig. 5

The software is asking the length of beam members that can be considered as typical length. 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. This is shown in the following figure.
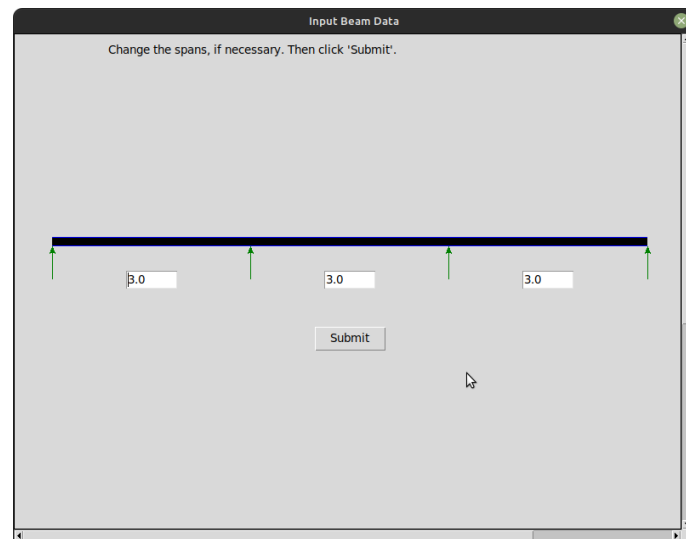
Fig. 6

I clicked in the textfield under the middle span and changed the value there to 3.5. 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.
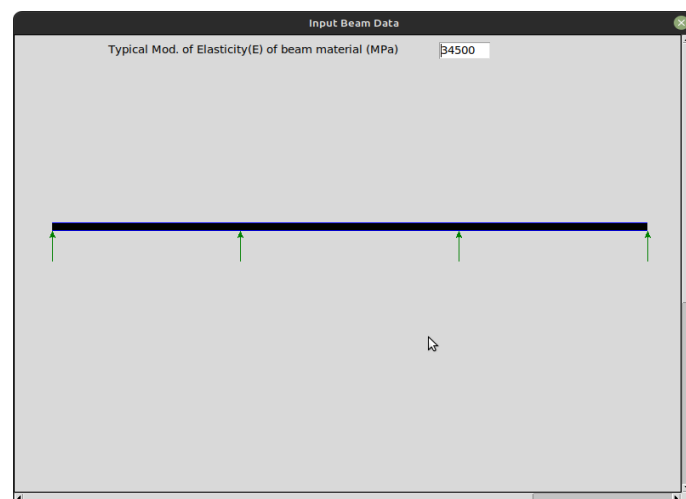

Fig. 7

I kept the default value 34500 MPa (modulus of elasticity of typical concrete used in India) and pressed ENTER. The window changed to -
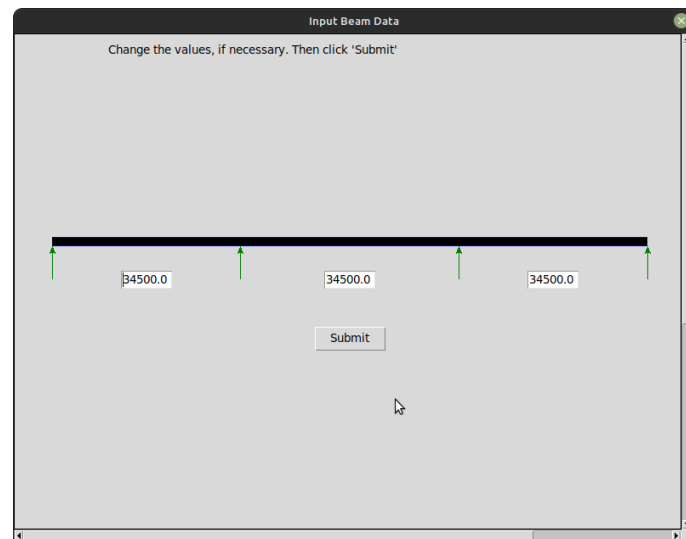
Fig. 8

I simply clicked the 'Submit' button, and the window changed to enquire about moment of inertia properties of the members.
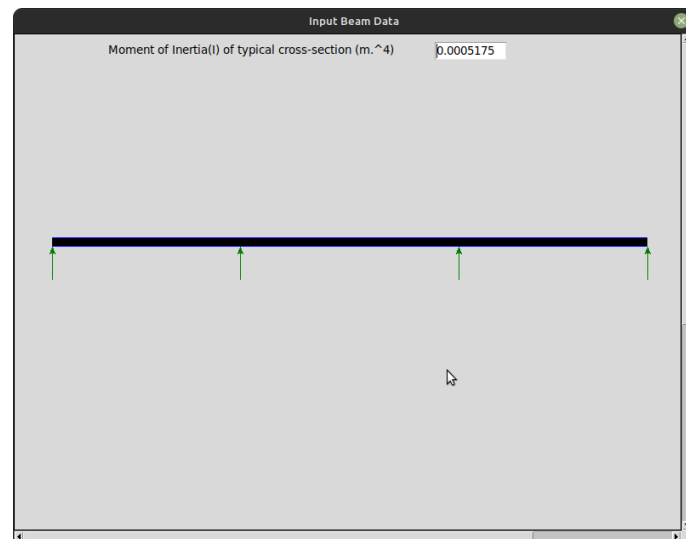
Fig. 9

I kept the pre-filled value 0.0005175 m⁴ (M.I. of section 230X300) and pressed ENTER to get the window as shown in the following figure, which asks for M.I. of individual beam members.
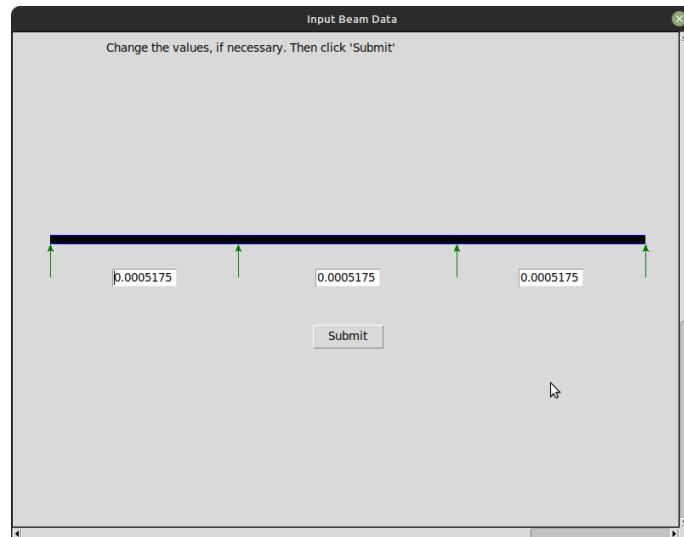
Fig. 10

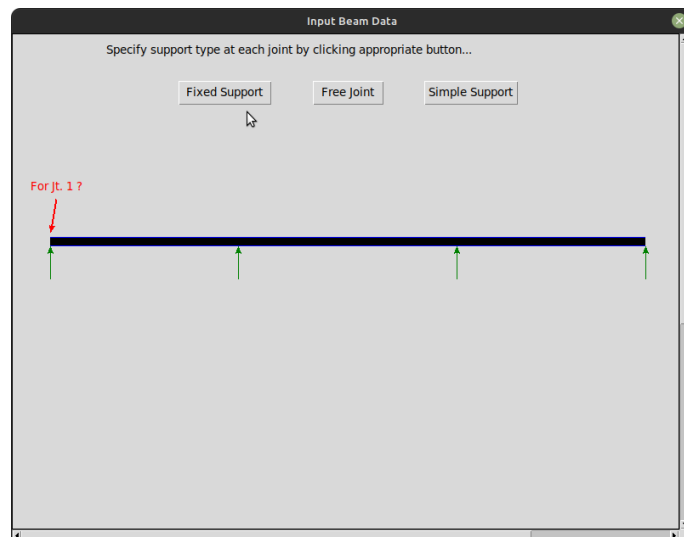Here, I clicked the 'Submit' button, to get new appearance in the window, as shown in the following figure -

Fig. 11

I clicked the 'Fixed Support' button for the left-most joint (currently being pointed to, as in the above figure). Upon this, the software drew the usual symbol for a fixed support at that joint and the red pointer shifted to the next joint. At this moment, the window appeared as shown in the following figure.
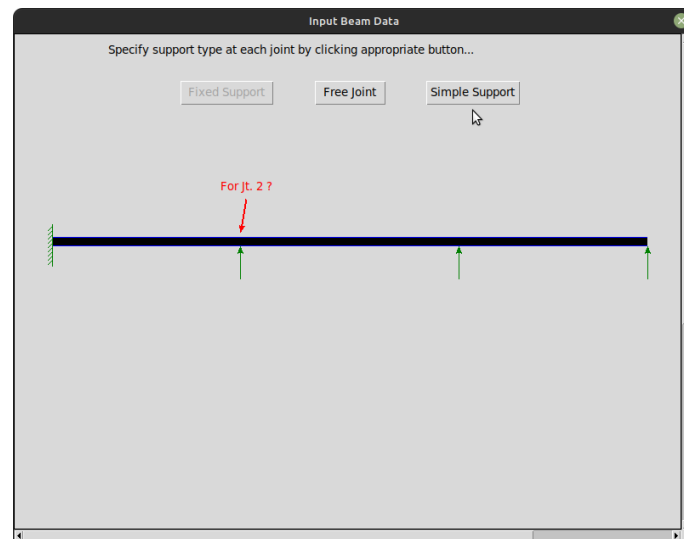
Fig. 12

I clicked the 'Simple Support' button for joint #2, again the same button for joint #3, and for the last joint I clicked the 'Fixed Support' button. After specifying the type of support for the last joint, the window appeared as show in the following figure. If you have made any error while specifying the support types, click the 'Edit' button and the program will ask you to specify supports at all the joints again. If you are satisfied with the specified support types, click the 'Exit' button. On clicking the 'Exit' button, the beam data entry module gets over and you will return to the menu as shown in Fig. 1 earlier.
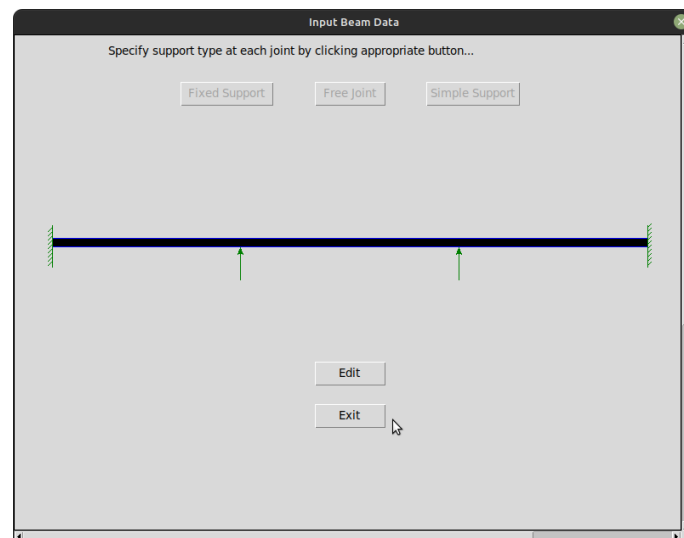

Fig. 13

Having defined the beam geometry, we are now ready to input the loading details. To specify loads, click the 'Input Load Data' button in the menu. 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
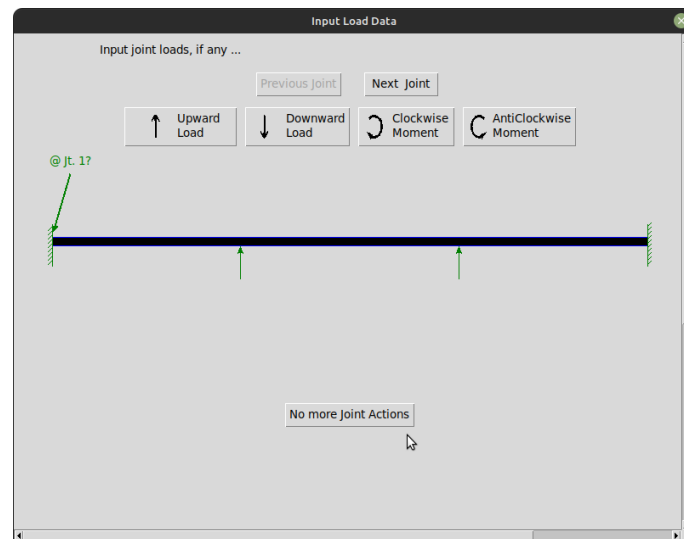
Fig. 14

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.
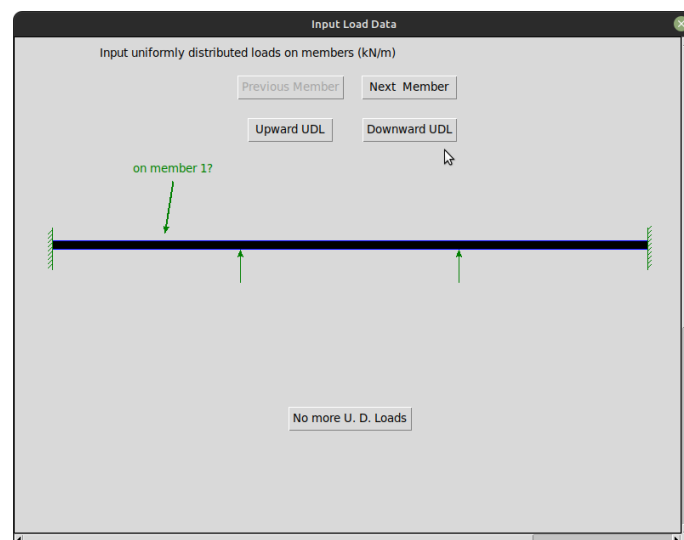

Fig. 15

Here, I clicked the 'Downward udl' button which caused a dialog box to pop up, as shown in the following figure. 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).
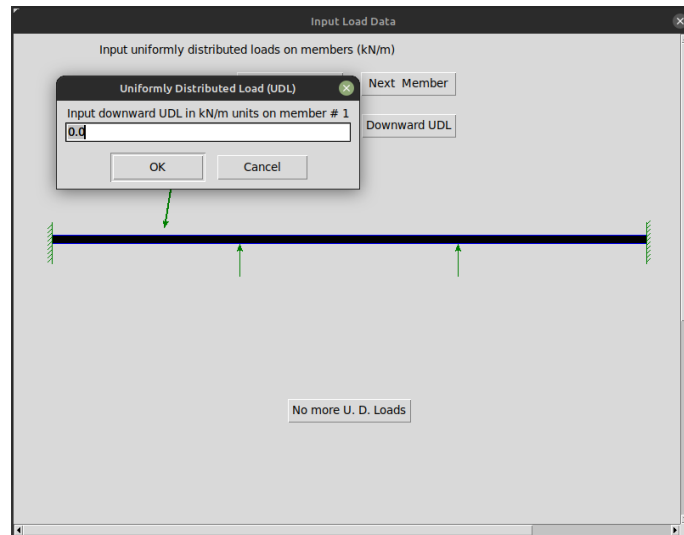
Fig. 16

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. 17

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.
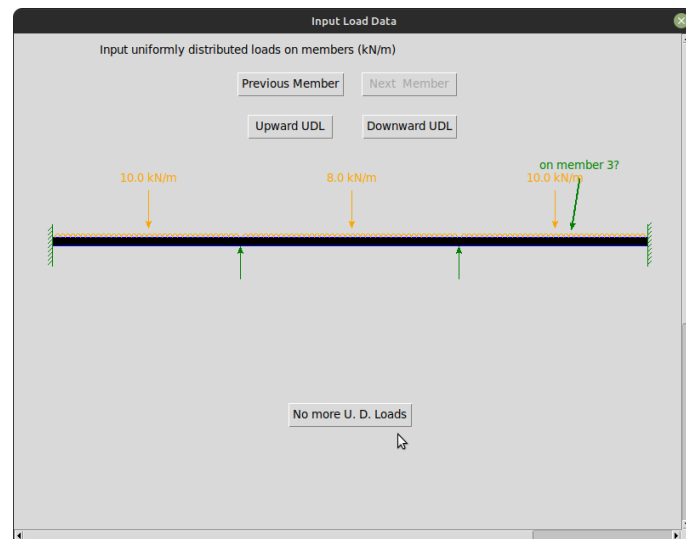
Fig. 18

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.
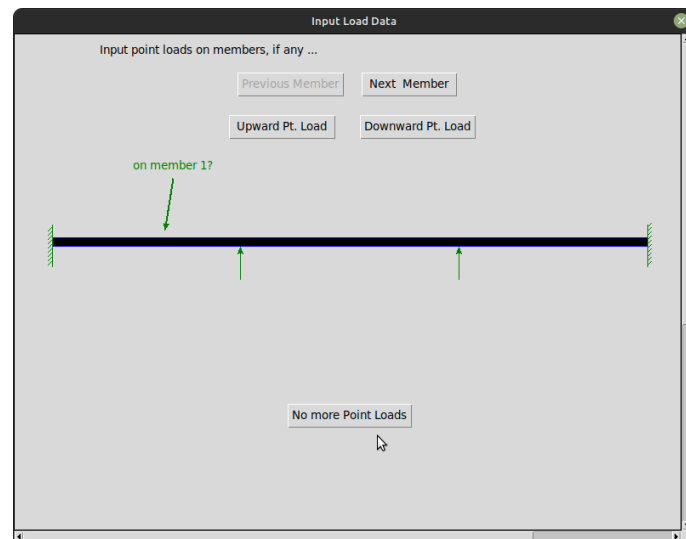

Fig. 19

As there are no point loads in the present example, I clicked the 'No more Point Loads' button, which caused this window to close and the menu window as shown in Fig. 1 earlier to reappear.

Now we are ready for the analysis. So, click 'Analysis' button in the menu (Fig. 1) and a window as shown in the following figure pops up.
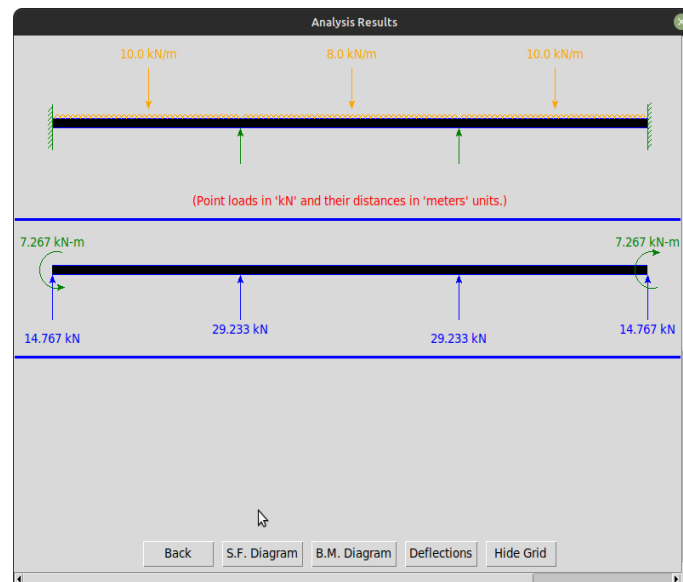
Fig. 20

The loading diagram and the joint reactions diagram are shown by default. To view shear force diagram, click the 'S. F. Diagram' button. The view on the window would change to the one shown in the following figure.
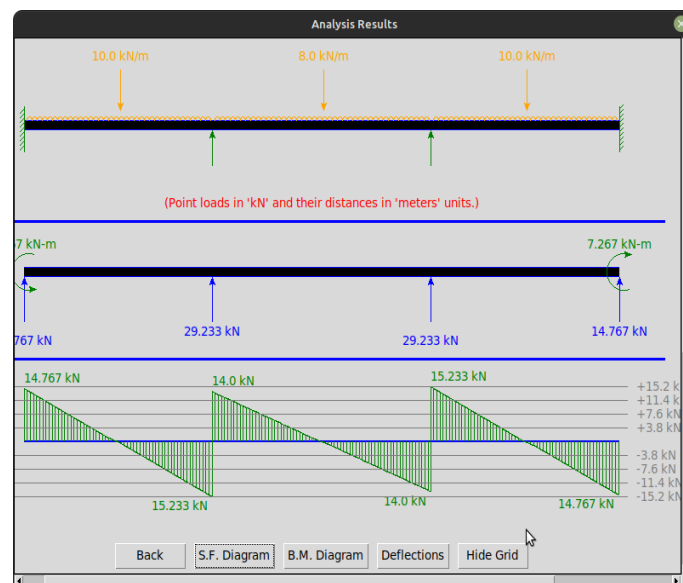

Fig. 21

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. The grid lines are helpful to estimate shear forces at other sections. The numeric value at the right end of each grid line indicates the shear force value represented by that line. If you want more accurate values of shear forces (and bending moments, and slope-deflections) at all these sections, then refer the '*structinsight_results.txt*' file, a plain-text file automatically created by the *analysis* module in the directory (folder) from where you invoked the *structInsight* program. On the other hand, if you find these grid lines an unnecessary distraction or clutter, then click the 'Hide Grid' button. This will cause the grid lines to disappear and the view will be as shown in the following figure.
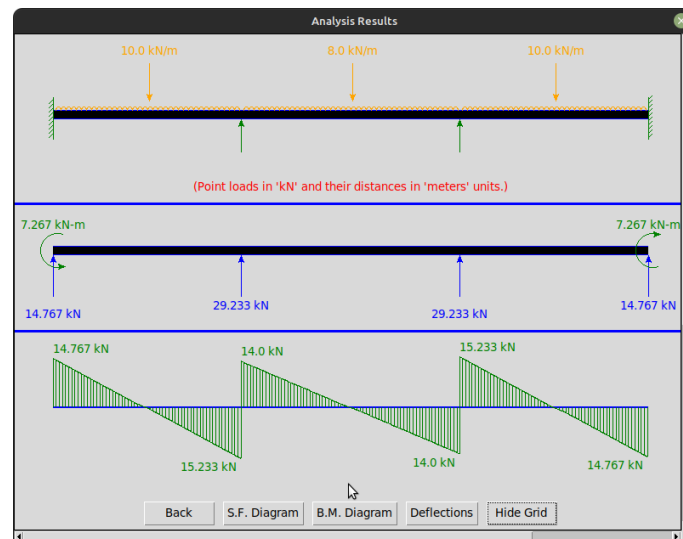
Fig. 22

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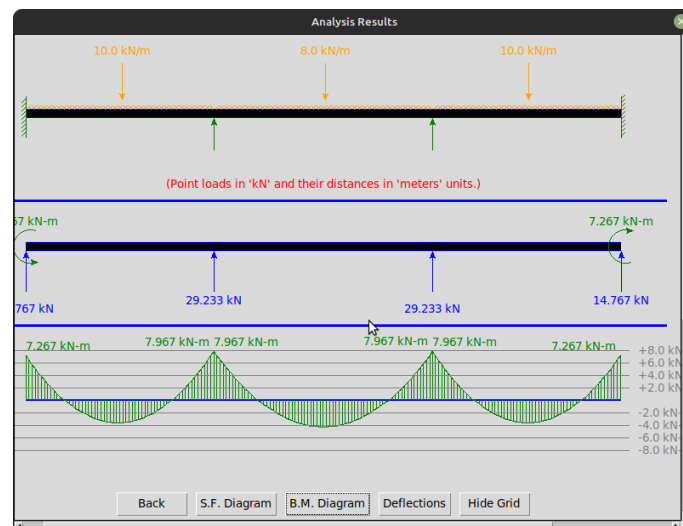

Fig. 23

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. Here also, you can click the 'Hide Grid' button to make the grid lines disappear. Again clicking the 'B.M. Diagram' button would bring back those grid lines.

If you want to see the deformation pattern of the beam, then click the 'Deflections' button. And the window would change to the one as shown in the following figure.
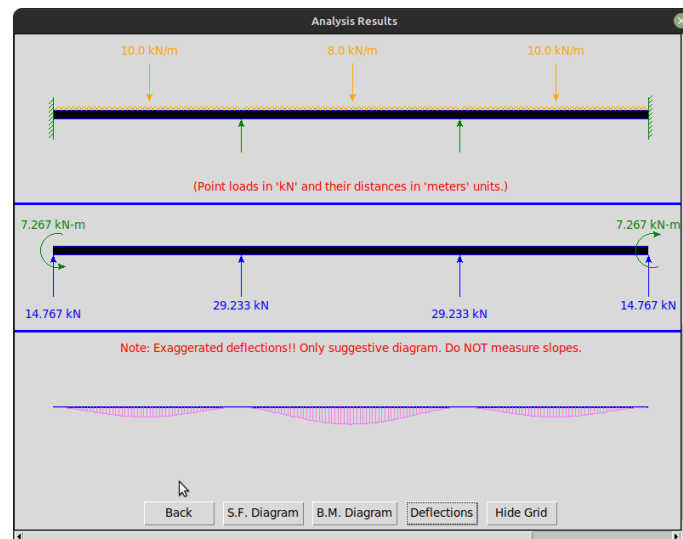
Fig. 24

Please note that the scale for drawing the deformations (Y-direction scale) is much larger than the scale for drawing the length of the beam (X-direction scale). Because of this slopes are not correctly represented in the above diagram. Even deflections can not be estimated from this diagram. As mentioned before, get the actual values of slopes and deflections from the '*structinsight_results.txt*' file, a plain-text file automatically created by the *analysis* module in the directory (folder) from where you invoked the *structInsight* program.

If your study of these diagrams is complete, click the 'Back' button and you would return to the continuous beam module menu as shown in Fig. 1 earlier. To exit the program and return to the Operating System prompt, click the 'Exit' button, and a warning window as shown in the following figure would popup.
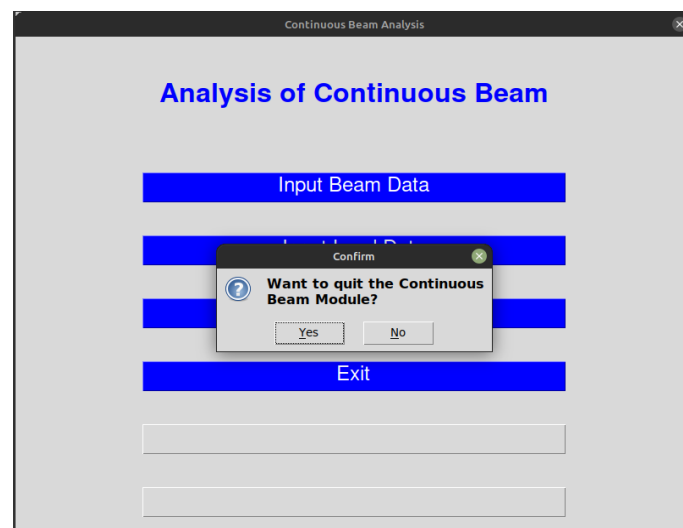

Fig. 25

If you click the 'No' button here, the software would understand that you have clicked the 'Exit' button by mistake, and it would hide the popup window, exposing the menu window for you to continue working with the program. However, if you click the 'yes' button then the program would take it as your confirmation to quit and would destroy both the windows taking you back to the OS prompt.

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 itend to solve. For this example, my sketch was -
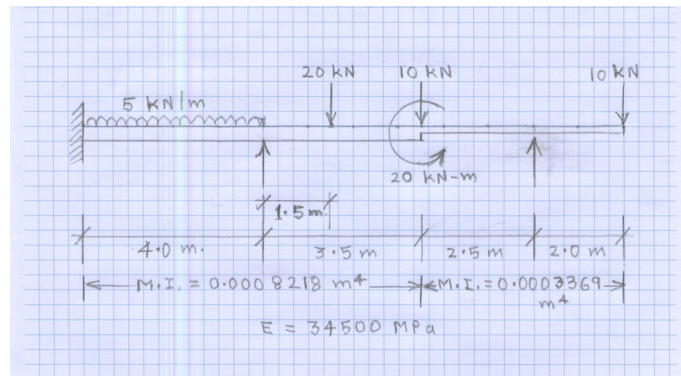
Fig. 26

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. 1), 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.
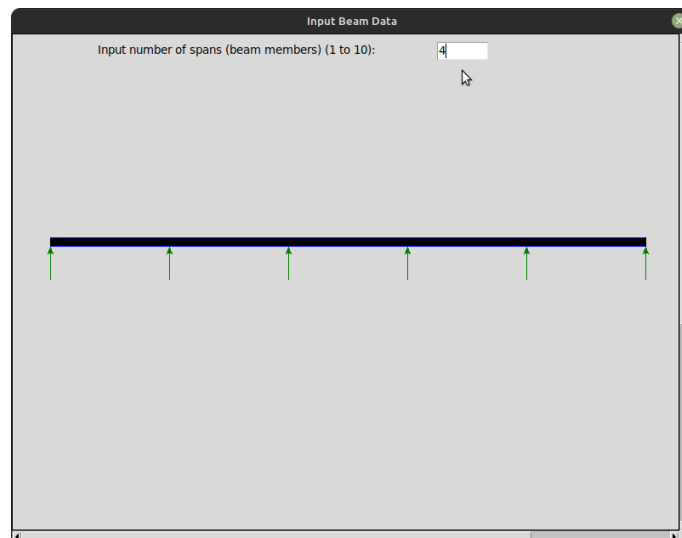
Fig. 27

The next thing the program asks is, the typical length, as shown in the following figure.
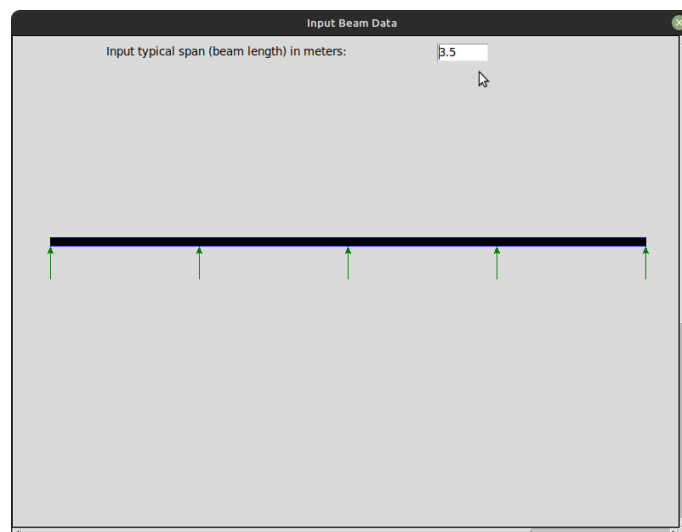

Fig. 28

In the present example, all the four beam members have different lengths; so, I just pressed ENTER to accept the given default typical length. This transformed the window to display four text boxes to input lengths, one each under the four beam members. There I typed spans 4, 3.5, 2.5, and 2 in the consecutive input boxes, as shown in the following figure.

Fig. 29

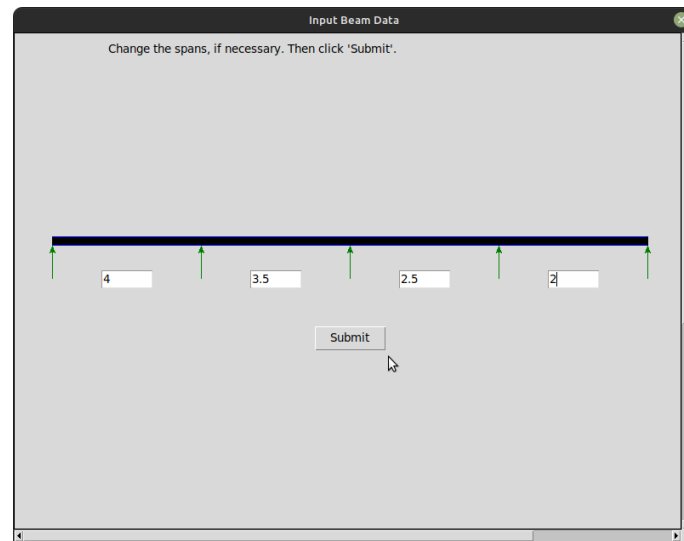Click the 'Submit' button and next the program asks for the typical modulus of elasticity. Press ENTER there and the window as shown in the following figure would appear. As all members of the beam are of the same material, retain the default value of modulus of elasticity (34500 MPa), by clicking the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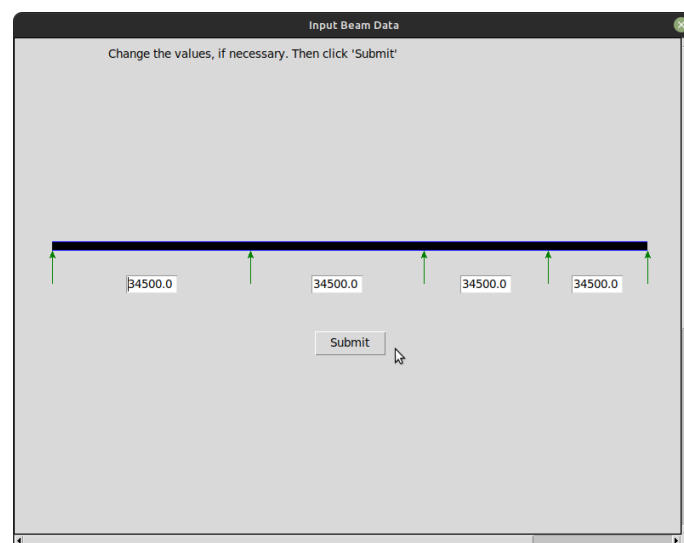

Fig. 30

When the program asks you about typical moment of inertia value, type 0.0008218 (M.I. of 230X350 section in m^4 units). Just to ensure that the values can be input in scientific notation also, I had input the value as shown in the following figure.
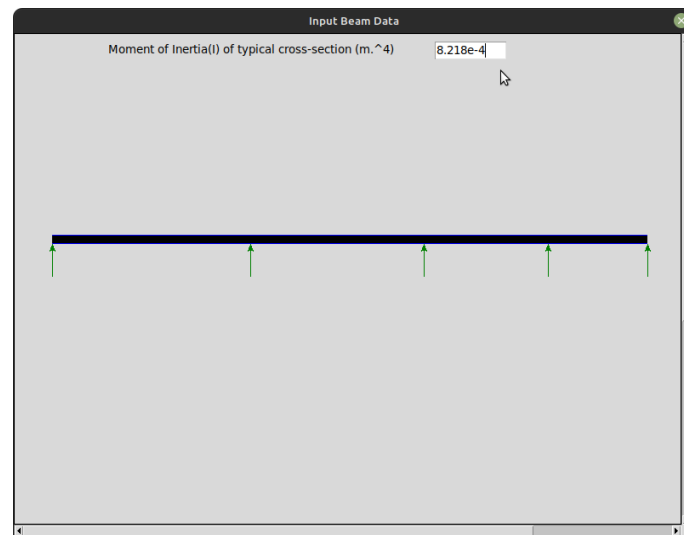
Fig. 31

On pressing ENTER after typing the typical moment of inertia, the window view would change; now, it would show four text boxes, one each under every span and each pre-filled with the value of typical moment of inertia that we input in the previous step. As required by our example (ref Fig. 26), 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.
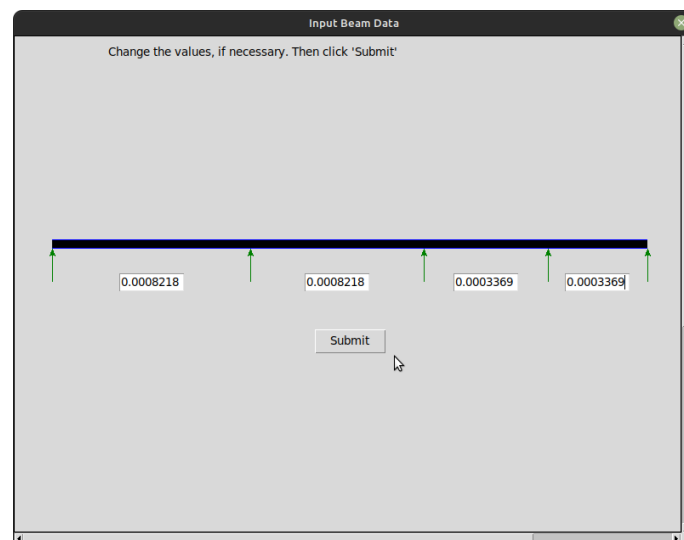

Fig. 32

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 three figures.
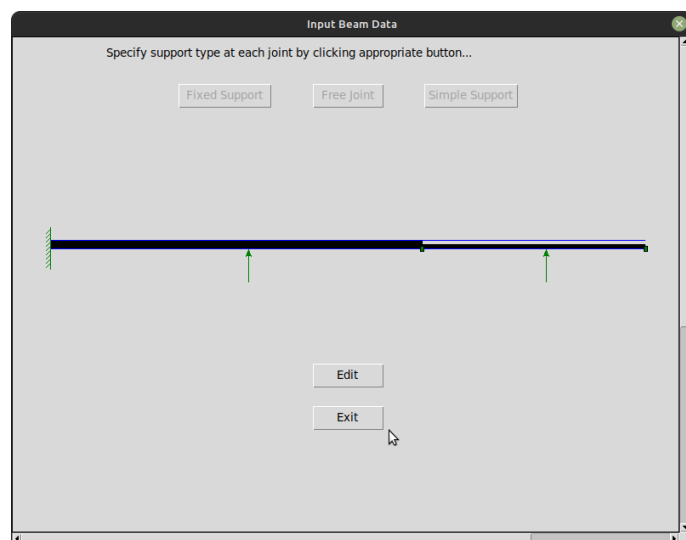
Fig. 33

Fig. 34

Fig. 35

After this click the 'Exit' button to close the current window and return to the menu window (Fig. 1).

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 as shown in the following figure. An arrow marker would be pointing to the first joint.
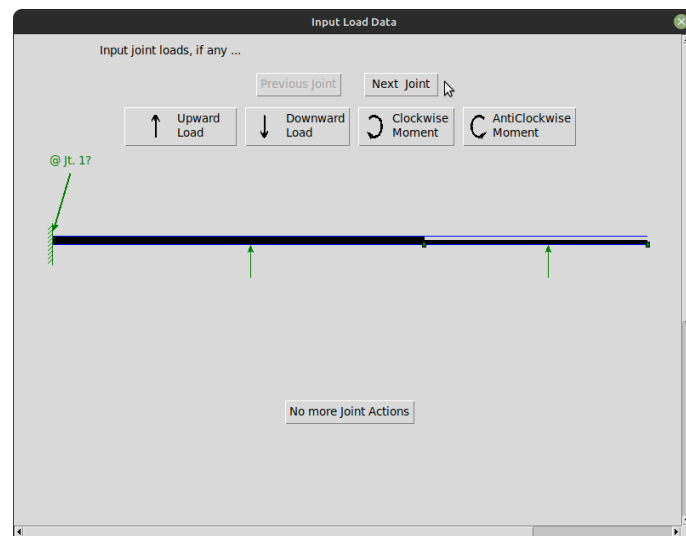

Fig. 36

By clicking the 'Next Joint' button twice, take it to the third joint and the resultant appearance would be as depicted in the following figure.
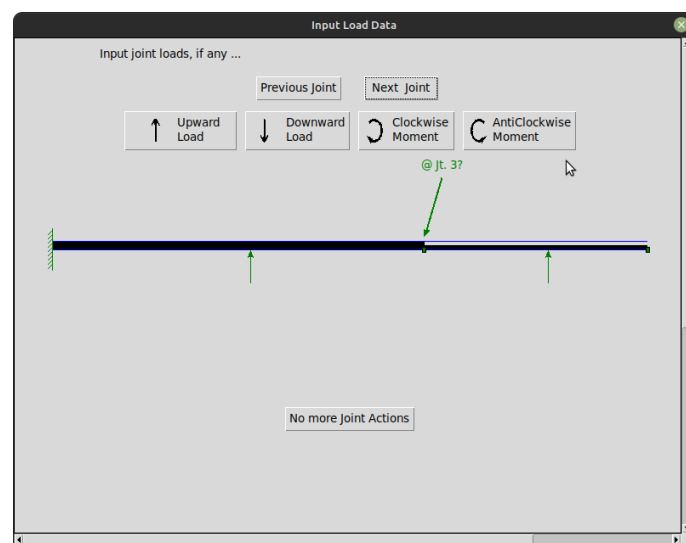

Fig. 37

Here, click the 'AntiClockwise Moment' button. An input dialog box would pop up, as shown in the following figure. Type 20 (the value of joint moment in our example) in the input box and press ENTER (or click the 'OK' button in the dialog box).
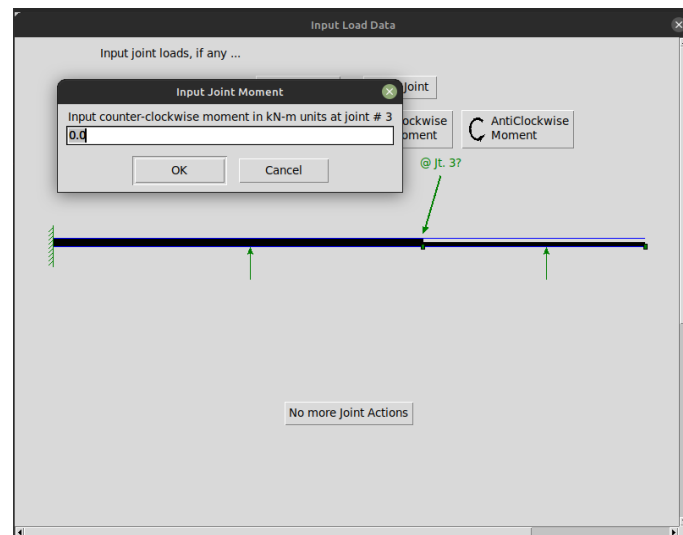
Fig. 38

The dialog box in the previous figure will vanish. Next, click the 'Downward Load' button and another input dialog box would pop up. Type 10, the value of vertical load at this joint, in our present example. The dialog box would look at this stage, as shown in the figure below.
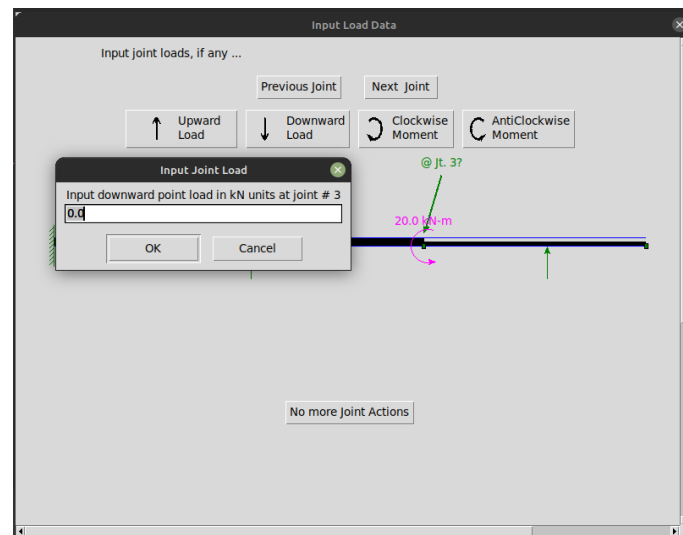

Fig. 39

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. 40

Once again, click the 'Downward Load' button; in the resultant input dialog box, type 10, and press ENTER. After this, the window should appear as shown in the next figure.
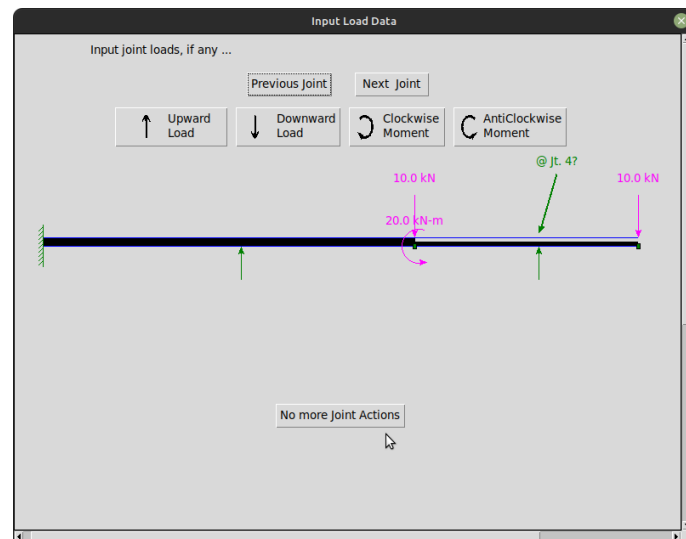

Fig. 41

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. In our example, there is a udl only on the first member. So, click the 'Downward UDL' button; a dialog box to accept the input for the udl would popup like the one shown in the following figure.
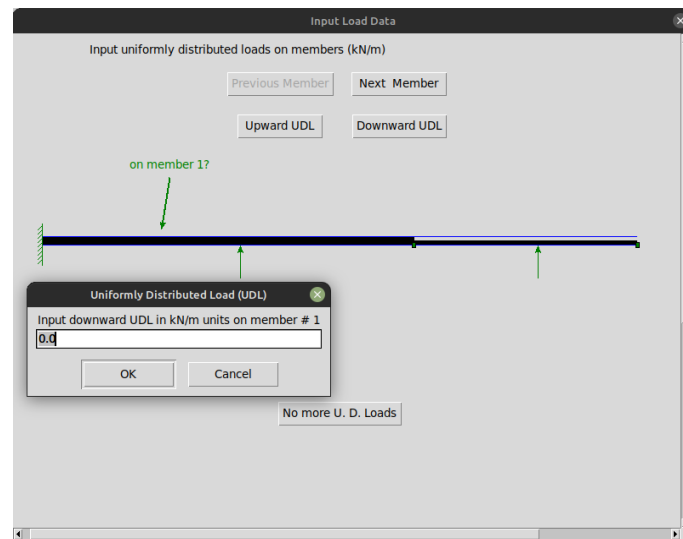
Fig. 42

In that input dialog box, type 5, and press ENTER. The resultant window will be as shown below.

Fig. 43

Click the button labeled 'No more U.D. Loads' and next get ready to specify the point loads on the members. The pointer would be pointing to the first member. In our case, there is only one member point load and that is on the second member. So, by clicking the 'Next Member' button, move the pointer to the second member. Then click the 'Downward Pt. Load' button and an input dialog box would popup. The overall appearance would be as shown in the following figure.
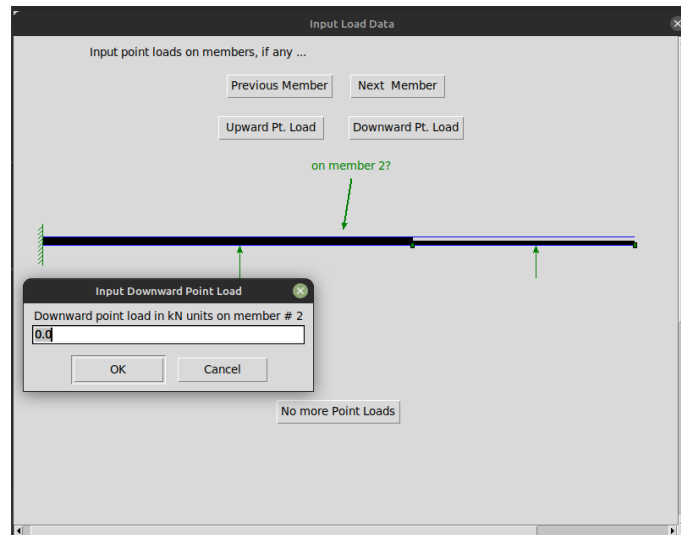
Fig. 44

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.
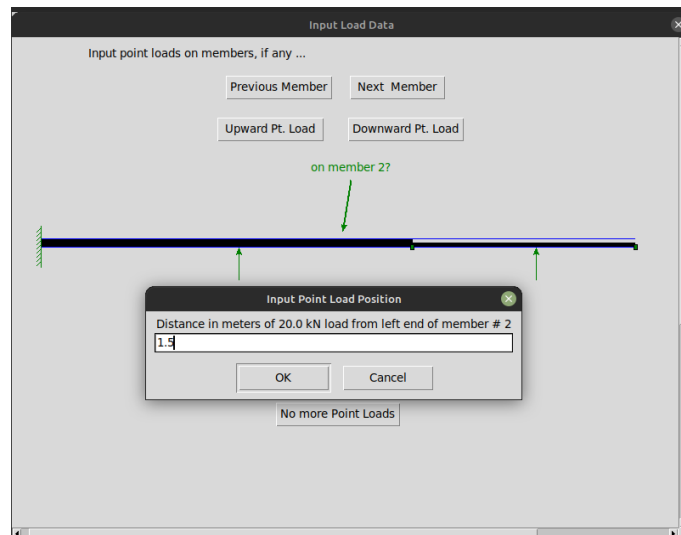

Fig. 45

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.
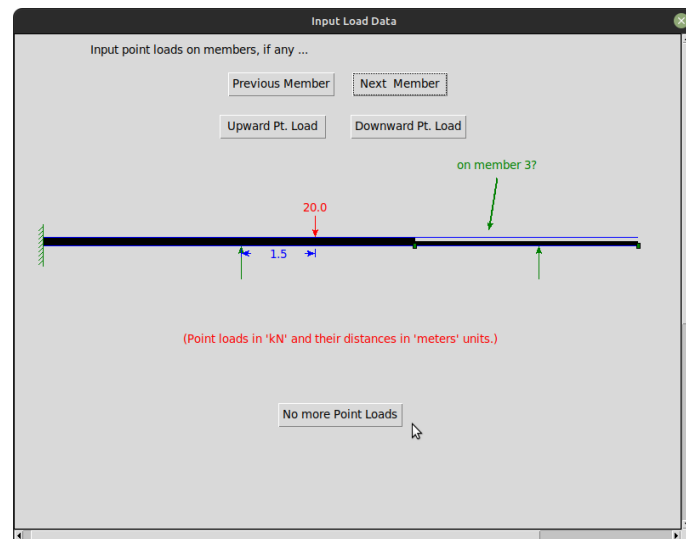
Fig. 46

As there are no more point loads, click the 'No more Point loads' button to return to the menu (Fig. 1) window.

To perform the desired analysis, click 'Analysis' 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.
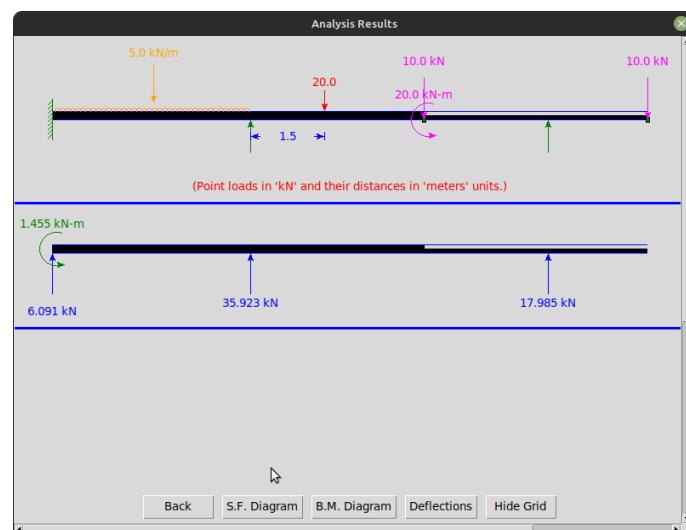

Fig. 47

On clicking the 'S.F. Diagram' button, the shear force diagram will be displayed. The window would appear as shown in the following figure.
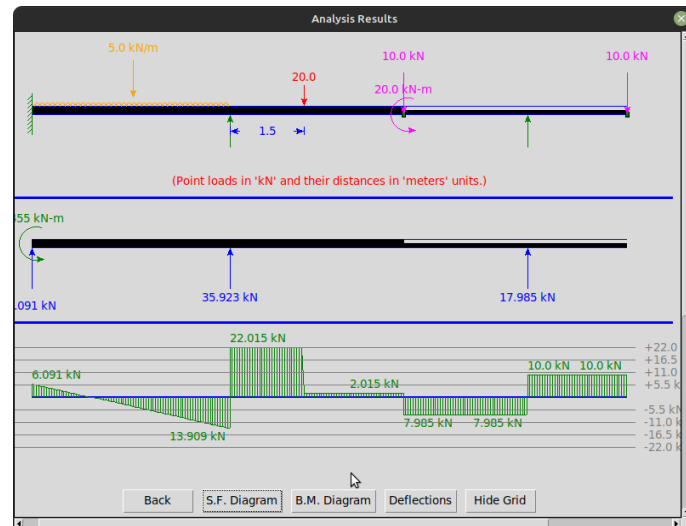
Fig. 48

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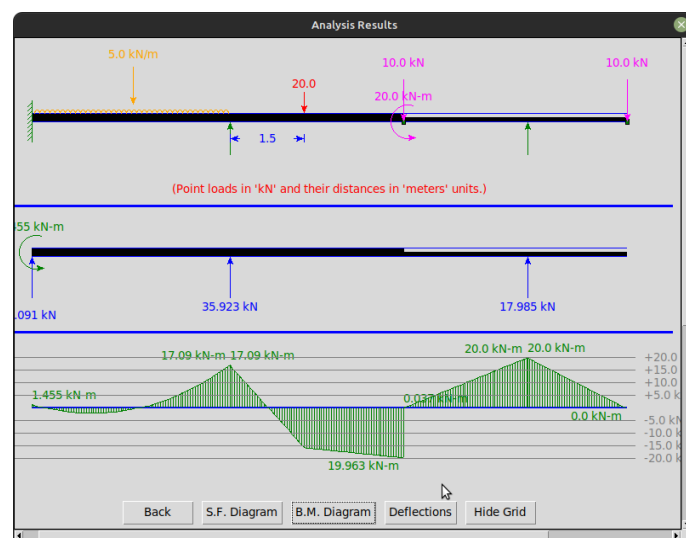

Fig. 49

To view the deformation pattern, click the 'Deflections' button and the window appearance would be as shown in the following figure.
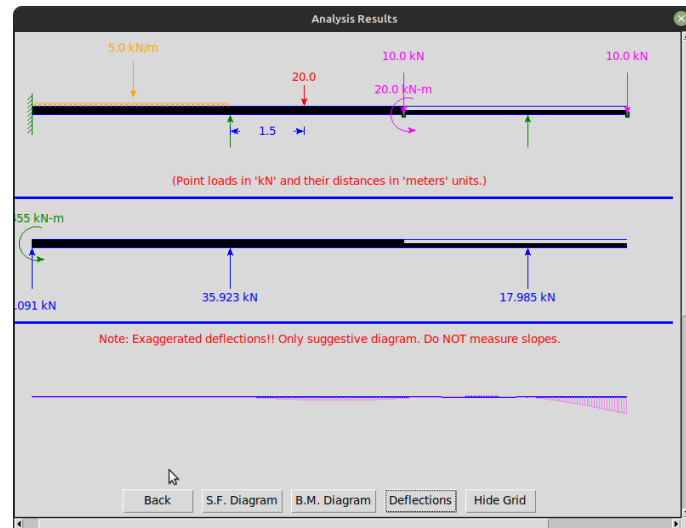
Fig. 50

Clicking the 'Back' button closes this window and returns to the menu window (Fig. 1). Clicking the 'Exit' button in the menu window, displays the warning message as explained in 'Example 1'. On confirming by clicking the 'Yes' button, it closes that window and exits the program.

- - - O O O - - -