Tutorial – 3

Beam – bending exercise using HyperBeam. (Verifying the result with hand calculations).

Problem Definition:-

A 1000mm in length, a solid circular Beam of 100mm diameter is subjected to a bending load of 30000 N in Y-direction and it is fixed at the other end the deflection and stress values has been calculated using the below method.
Then the same problem is solved in HyperMesh and then showing the comparison to both the results that is values of deflection and stress with hand calculation and values of deflection and stress by using the software.

HAND CALCULATION:-

Stress = My/I…….. $(M = PL, I = \pi d^4/64)$

{putting all the required values in above equation}

The magnitude of stress for the defined problem is **305.56 N/mm^2**.

Deflection = $PL^3/3EI$……

{Putting all the required values in above equation}

The magnitude of deflection for the defined problem is **9.7 mm.**

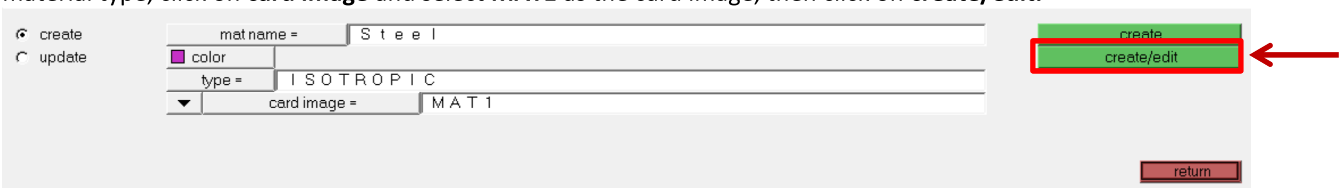
Steps to solve the problem using HyperWorks.

Step 1 :- Open HyperWorks Student Edition.

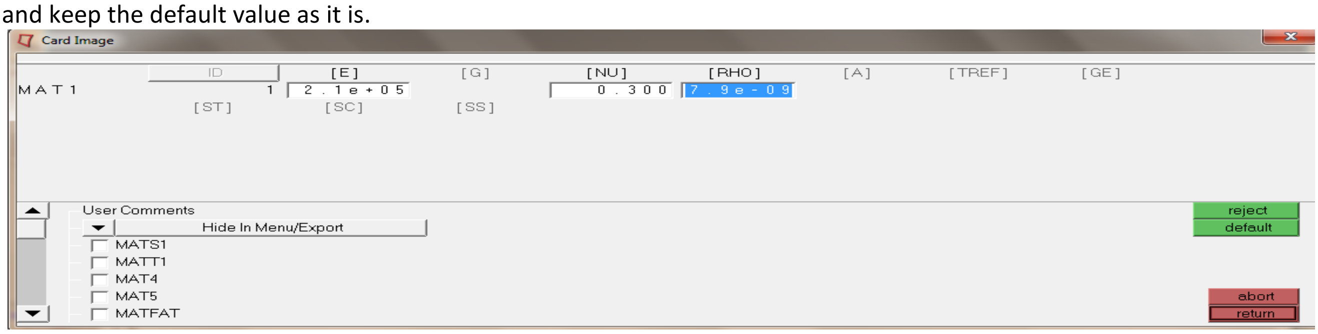
1. To open HyperWorks Student edition follow the given step.
   Start < All Programs < Altair HyperWorks 11.0 Student Edition (32-bit) < HyperWorks.
2. Once the Student edition HyperWorks opens set the User Profile as *RADIOSS (Bulk Data).*
   {if the User Profile Window does not appear, On the tool bar click on *preferences* and from the drop down list select *user profile*}
   This will load all the required information related to solve a linear static analysis problem using **radioss solver**.

Step 2 :- Create Material.

1. Click on the ***Material*** *collector* icon  on the bottom side of your screen.
2. On **mat name** enter **Steel** as name of the material, select colour from the **colour** panel, click on **type** and select **ISOTROPIC** as the material type, click on **card image** and select **MAT1** as the card image, then click on **create/edit.**



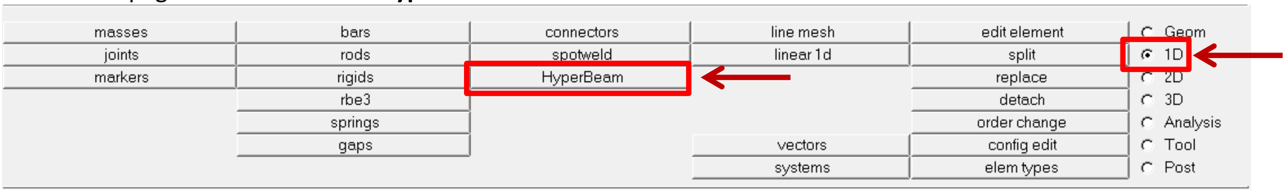
3. A new window will open in which we can add further information about the material properties, here click on [E], [NU] and [RHO], and keep the default value as it is.



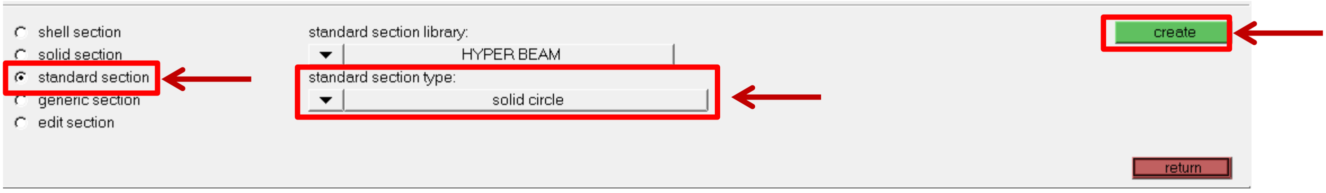
4. Click **return** twice to exit the material collector panel.

Step 3 :- Create Circular Cross-Section.(using HyperBeam)

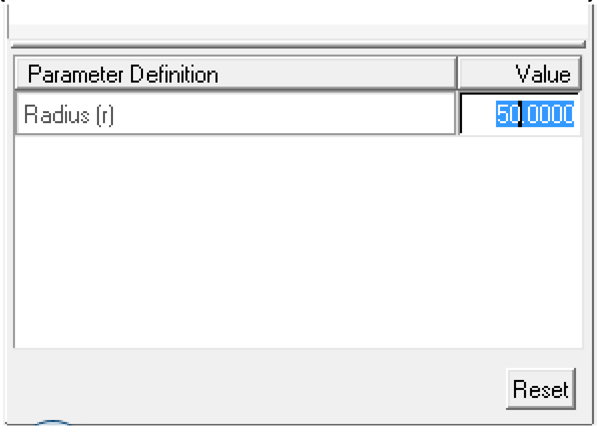
1. Click on **1D** page and then click on **HyperBeam.**



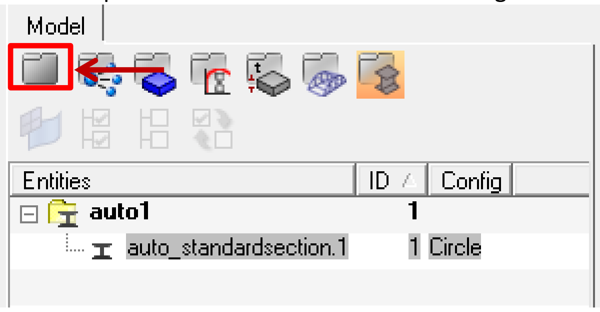
2. Select the **standard section** radio button, below **standard section type:** select **solid circle.** Make sure in **standard section library:** option **HYPER BEAM** is selected. Click **create.**



3. A new HyperBeam window will open. In that window on the bottom left corner of your screen, in **Value** option enter there 50. {This is the value of circular radius of the beam section}



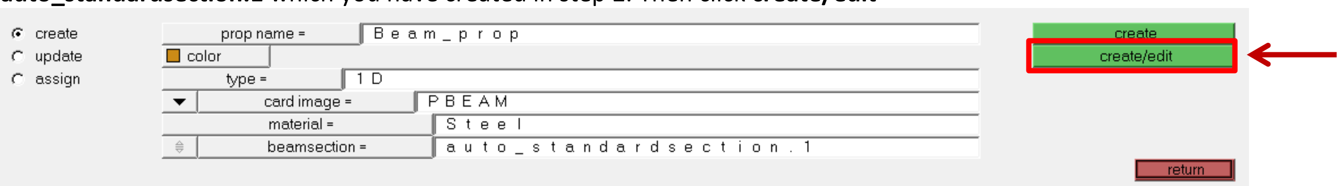
4. On the top left screen click on **model view** to get back to the **HyperWorks** window.



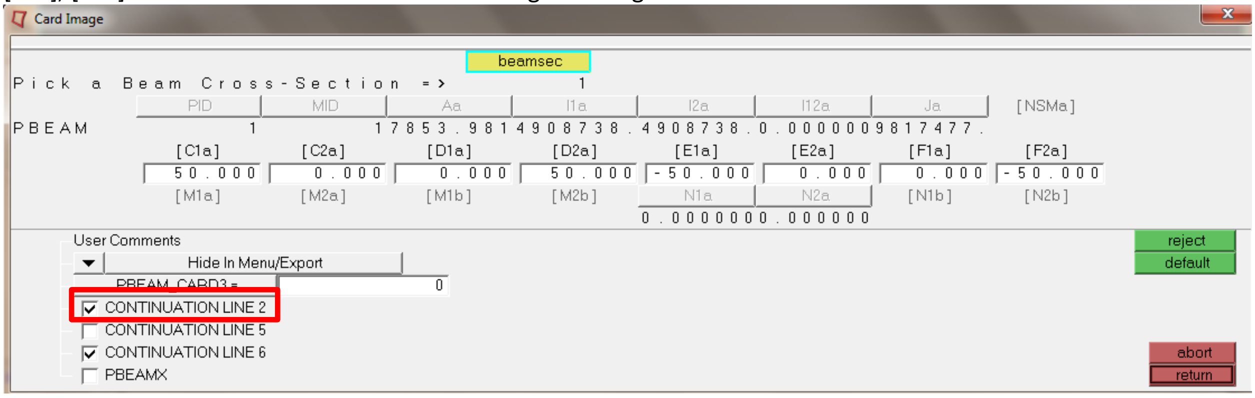
5. Click **return** to exit the **HypeBeam** panel.

Step 4 :- Create Property.

1. Click on the ***Property** collector* icon
2. In **prop name =** enter **Beam_prop** as property name, select a different colour from the **colour** panel, in **type=** select **1D,** in **card image=** select **PBEAM**, click on **material** tab and select **Steel** from the list of material, click on **beamsection** and select **auto_standardsection.1** which you have created in step 2. Then click **create/edit**



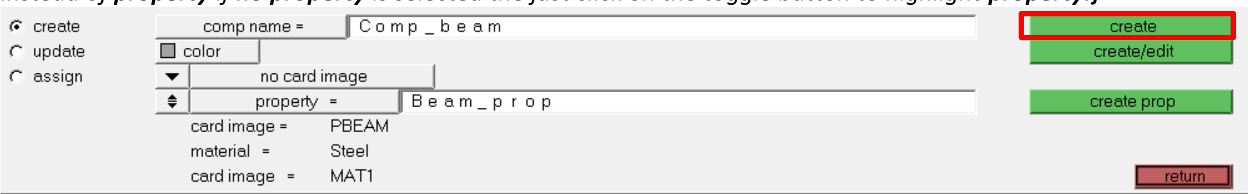
3. A new window will open, check the box in front of **CONTINUATION LINE 2**, now click on [C1a], [C2a], [D1a], [D2a], [E1a], [E2a], [F1a], [F2a]. and enter the values as show in the given image



4. Click **return** twice to exit the property collector panel.

Step 5 :- Create Component.

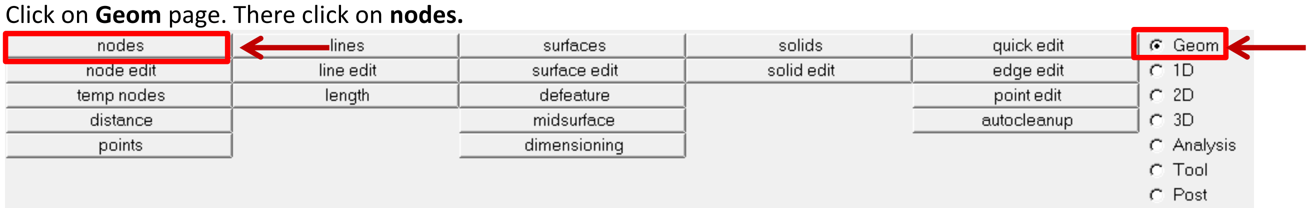
1. Click on Component collector icon as shown  .
2. Click on **create** radio button. Enter **Comp_beam** as the **comp name = ,** select a different colour from the colour panel. Click on the **property=** option and select **Beam_prop** as the property type which you have created in step 3. *{Note: - If in component window instead of **property** if **no property** is selected the just click on the toggle button to highlight **property.**}*



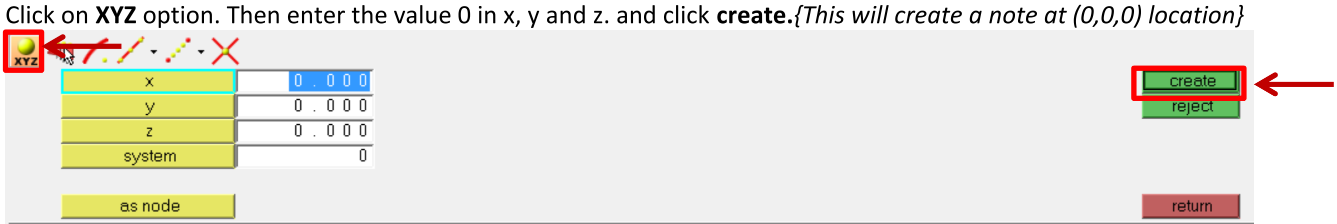
3. Click **create,** to create component.
4. Click **return**, to exit the component collector.

Step 6 :- Create Nodes.

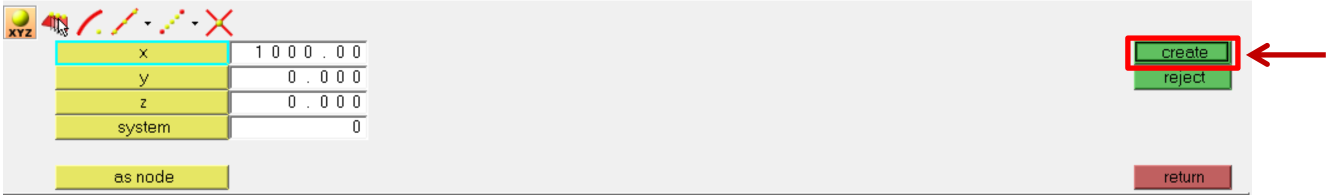
1. Click on **Geom** page. There click on **nodes.**



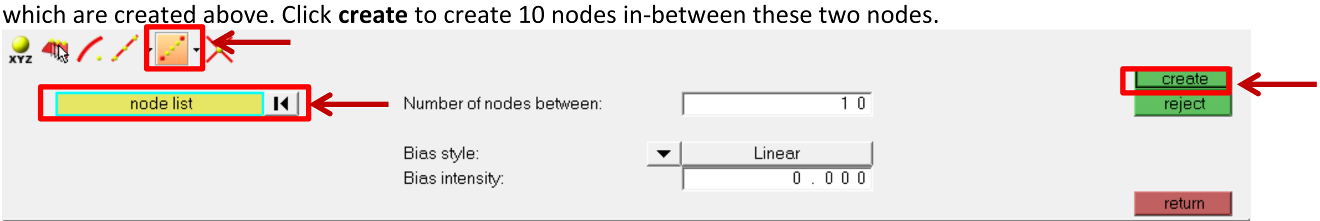
2. Click on **XYZ** option. Then enter the value 0 in x, y and z. and click **create.***{This will create a note at (0,0,0) location}*



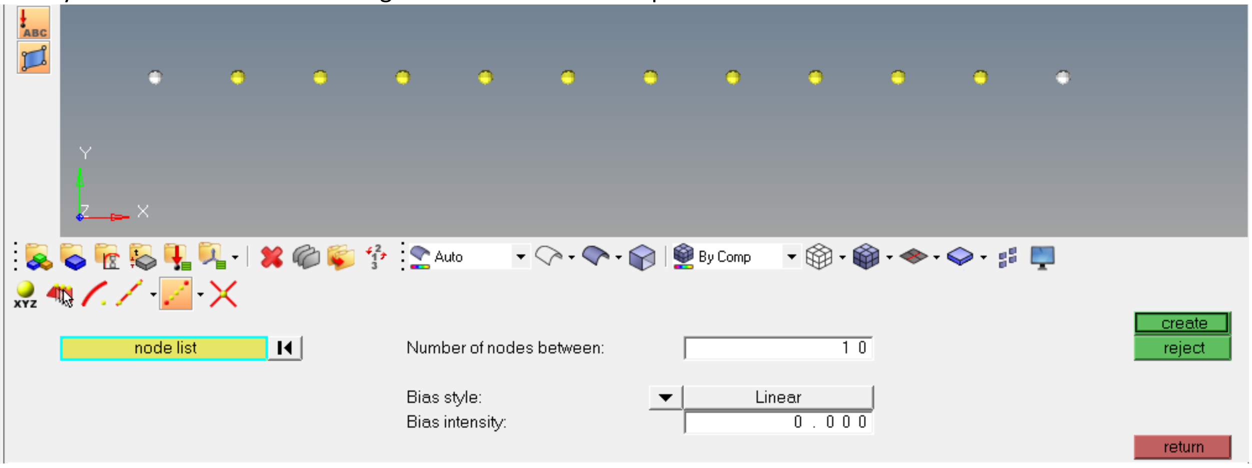
3. Then enter 1000 in x and 0 in y and z. click **create.***{ This will create a note at (1000,0,0) location}*



4. Two nodes have a distance of 1000mm is created.
5. Then click on **interpolate node** option, on **Number of nodes between:** enter 10, set the **Bias style:** as **Linear**, select the two nodes which are created above. Click **create** to create 10 nodes in-between these two nodes.



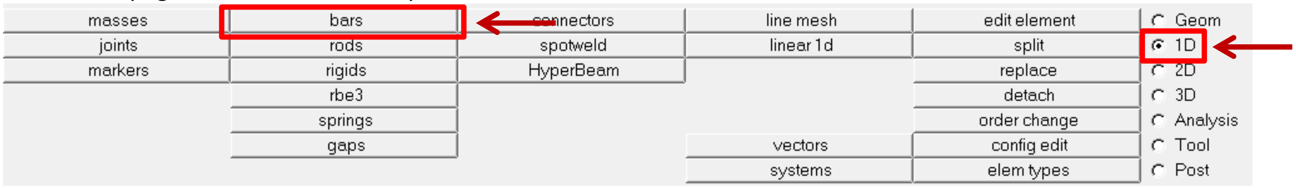
6. Once you click create the software generated 10 nodes at equal distance. As shown below.



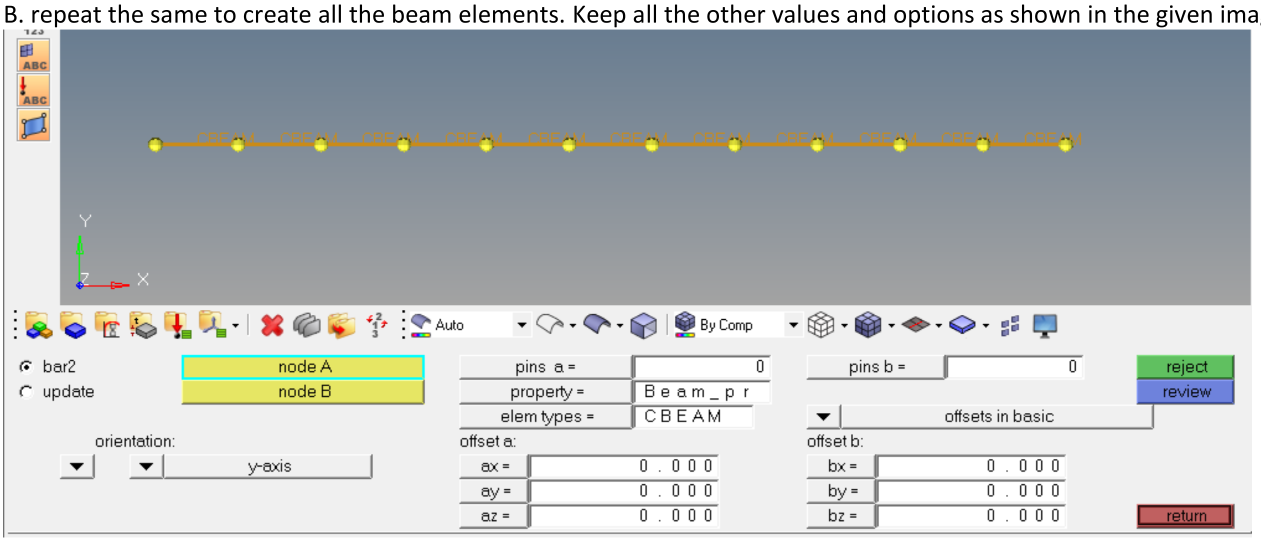
7. Click **return**.

Step 7 :- Create Beam Elements.

1. Click on **1D** page, in this click on **bars** panel.



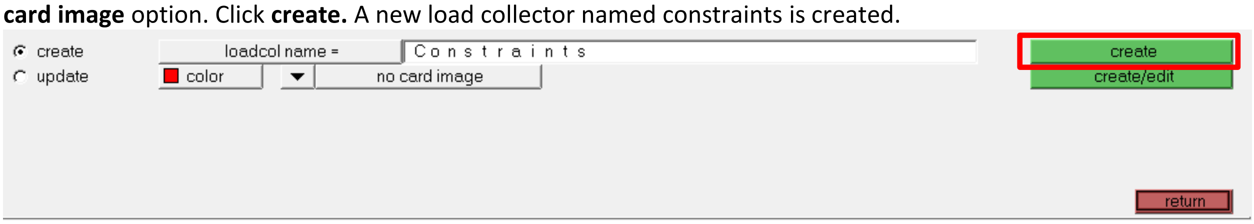
2. In this select **bar2** radio button, In this panel click on the **property** option and select **Beam_prop** click on **elem types=** and select **CBEAM** set the **orientation** as **y-axis.** Click on **node A** and select the 1st node on the screen click on **node B** and select the 2nd node. A beam element automatically created. Similarly create another beam element by clicking on 2nd and 3rd node as node A and node B. repeat the same to create all the beam elements. Keep all the other values and options as shown in the given image.



3. Click **return** to exit the **bar** panel.

Step 8 :- Create and apply constrains.

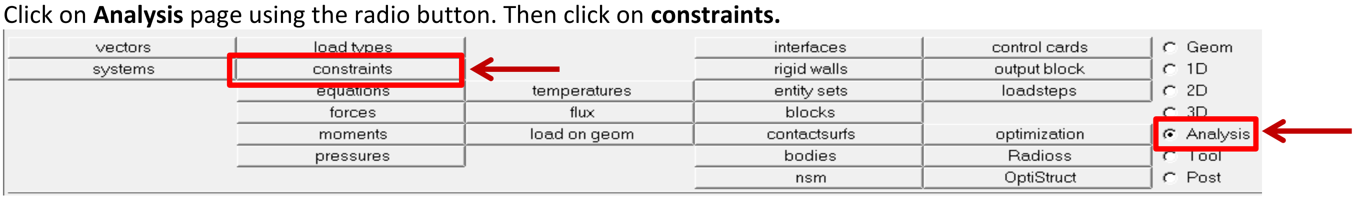
1. Click on **load collector** icon as shown
2. Go to **create** subpanel. Enter **Constraints** in **loadcol name=**, select the different colour from the colour option. Toggle down to **no card image** option. Click **create.** A new load collector named constraints is created.



3. Click **return.** *{This will create a load collector name constrained}.*

To apply constraints on the model follow the step shown below....

4. Click on **Analysis** page using the radio button. Then click on **constraints.**



5. Click on **create** radio button. Click on **nodes** and select the node as shown in the image. Select the **load types =** as **SPC**, make sure all the other values are as shown in the below image. Click **create**.

dof 1, dof 2 and dof3 means translation along x-axis, y-axis and z-axis. And dof 4, dof 5 and dof 6 means rotation about x-axis, y-axis and z-axis respectively. If any dof is checked that means it is fixed and if any dof is unchecked that means that the component is free to translate or rotate in that particular dof.

6. Click **return.**

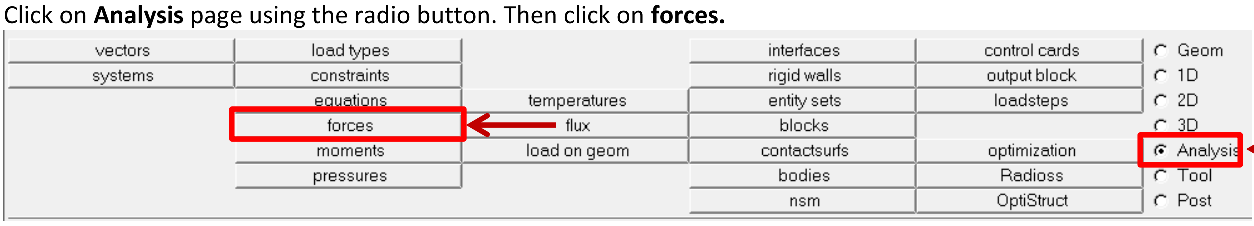
Step 9 :- Create and apply force.

1. Click on **load collector** icon as shown
2. Go to **create** subpanel. Enter **Force** in **loadcol name=**, select the different colour from the colour option. Toggle down to **no card image** option. Click **create.** A new load collector named **Force** is created.

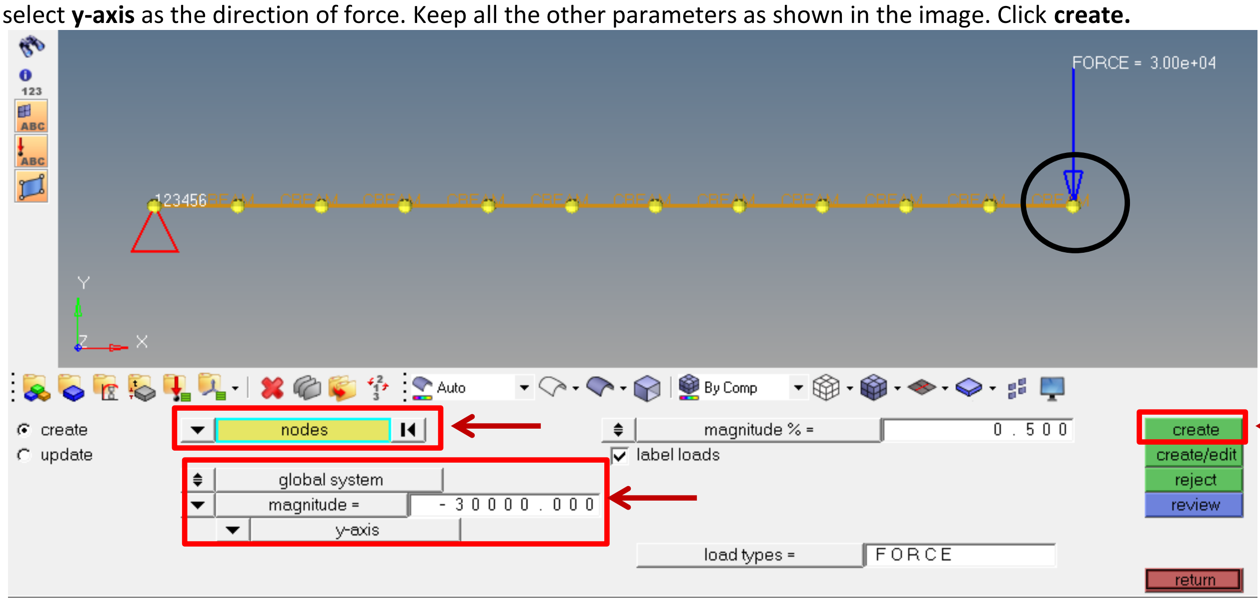
3. Click **return.**

To apply Forces on the model follow the process shown below.

4. Click on **Analysis** page using the radio button. Then click on **forces.**



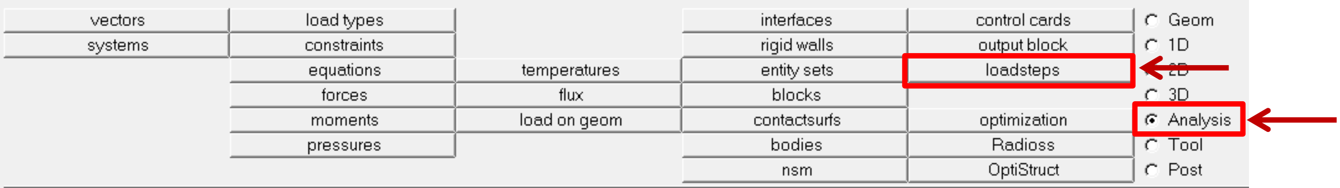
5. Click on the **nodes** and select the node as shown. Make sure the toggle is set to **global system,** Enter **-30000** in **magnitude=**, and select **y-axis** as the direction of force. Keep all the other parameters as shown in the image. Click **create.**



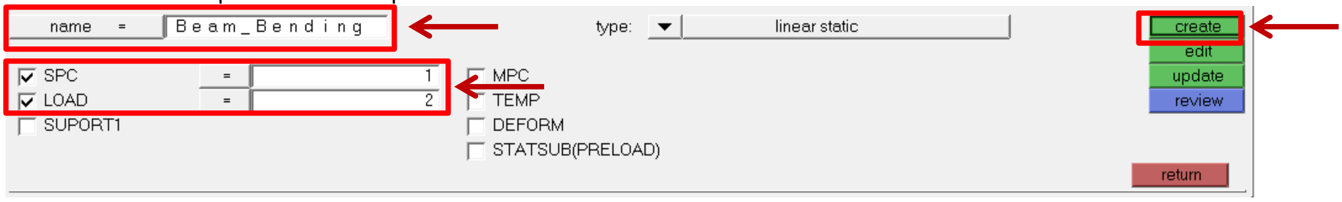
6. Click **return.**

Step 10 :- Create Loadstep.

1. Click on **Analysis** Page, there click on **loadsteps**.



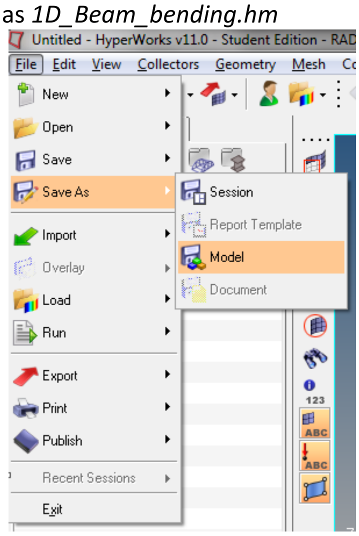
2. In **name=** enter **Beam_Bending.** Check the box before **SPC** and **LOAD**, click on blank window in front of **SPC** and select **Constraints** from the list of load collectors. Similarly click on the blank window in front of **LOAD** and select **Force** from the list. Keep the **type:** as **linear static**. Keep all the other options as shown. Click **create.**



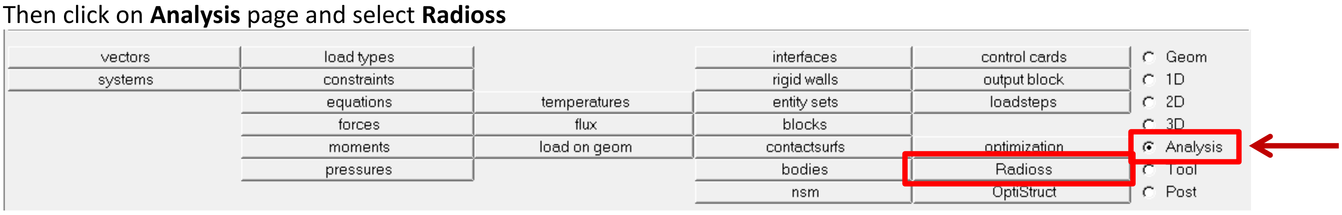
3. Click **return.**

Step 11 :- Save the Model and Run the Analysis.

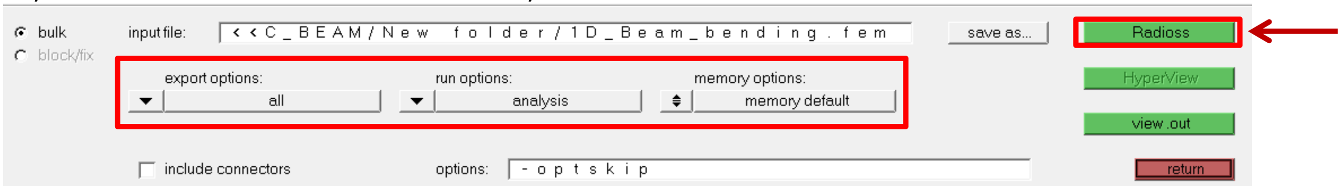
1. On the tool bar panel (on the top of your screen), click on **File**, go to **save as** and click on **model**. Create a folder and save the file as *1D_Beam_bending.hm*



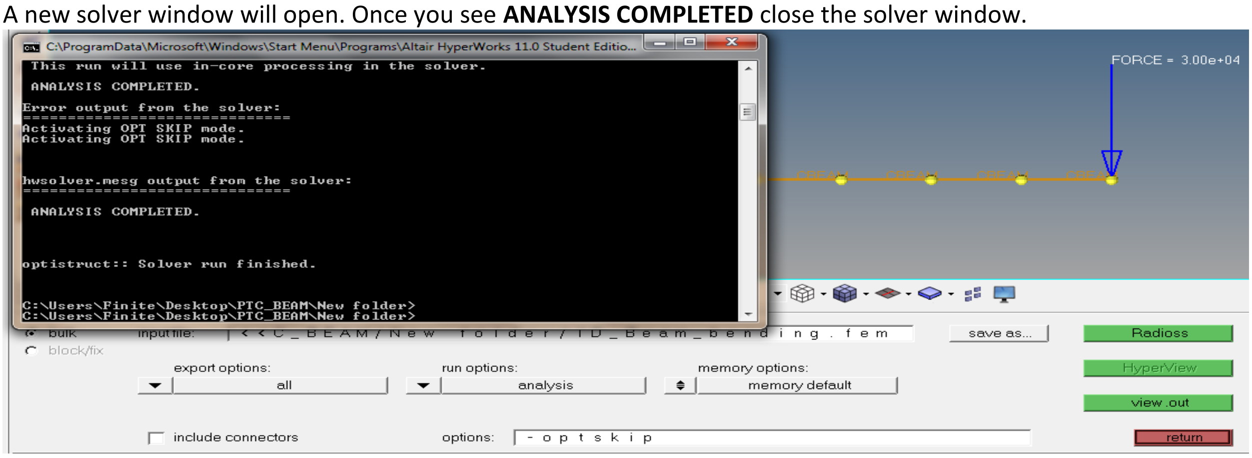
2. Then click on **Analysis** page and select **Radioss**



3. Set the **export options:** to all, **run options:** to analysis, **memory options:** to memory default. Click on **save as** to save the **.fem** file of your model. Click on **Radioss** to start the Analysis.



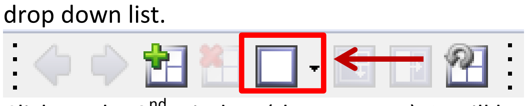
4. A new solver window will open. Once you see **ANALYSIS COMPLETED** close the solver window.

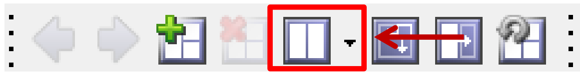


5. Click **return** to exit the Radioss panel.

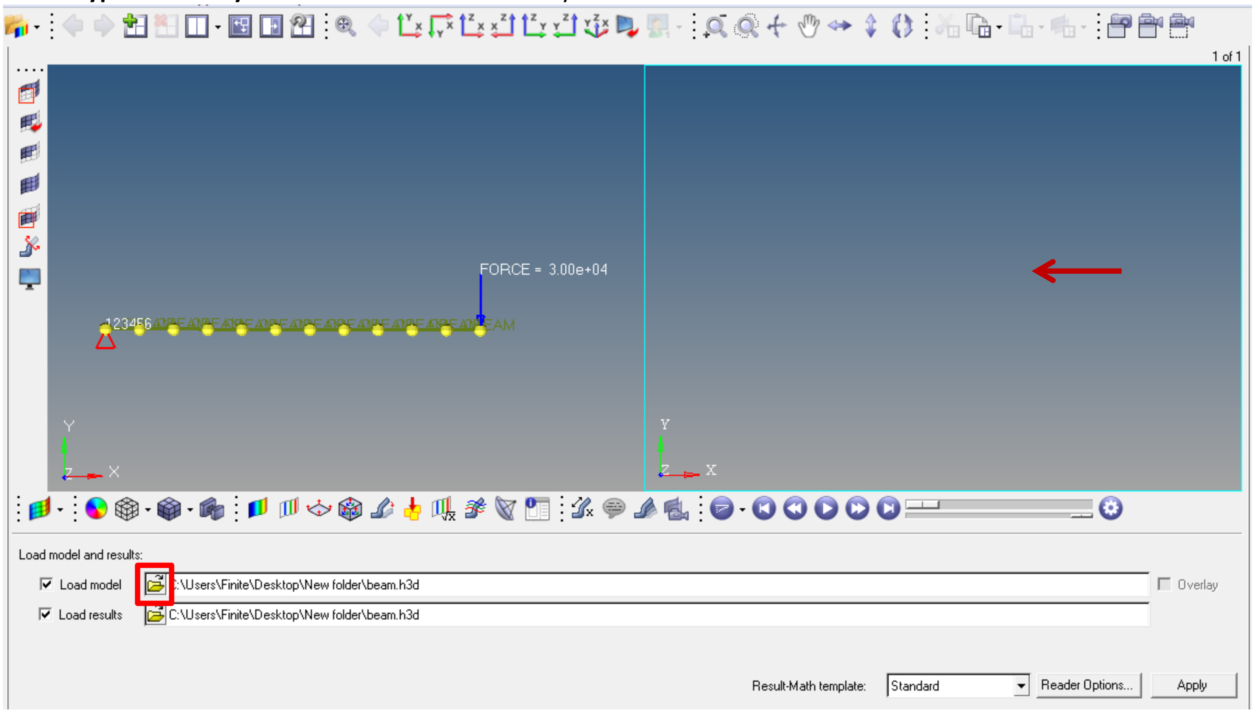
Step 12 :- View the displacement and stress results using HyperView.

1. Click on the small arrow below the **Page Window Layout** option *{on the top of your screen}*. And select the two window option from the drop down list.

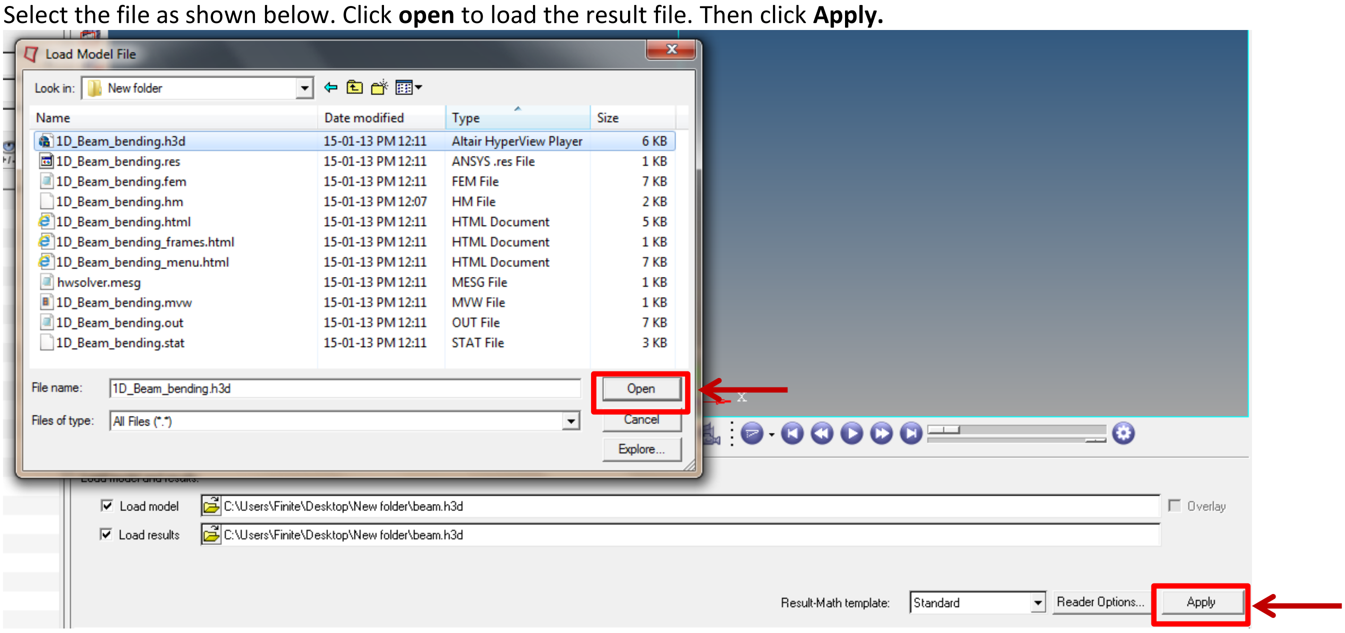




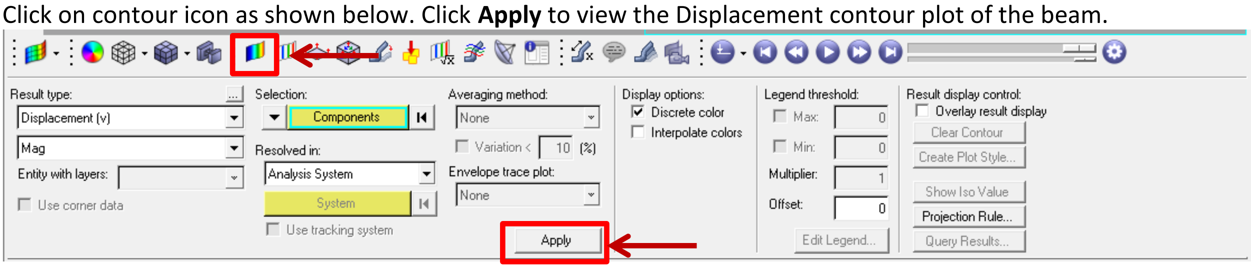
2. Click on the 2nd window (the new one). It will load HyperView, click on the folder icon in front of **Load model** and open the **.h3d** or **Altair HyperView Player** file from the folder where you have save the model before the **radioss** run.



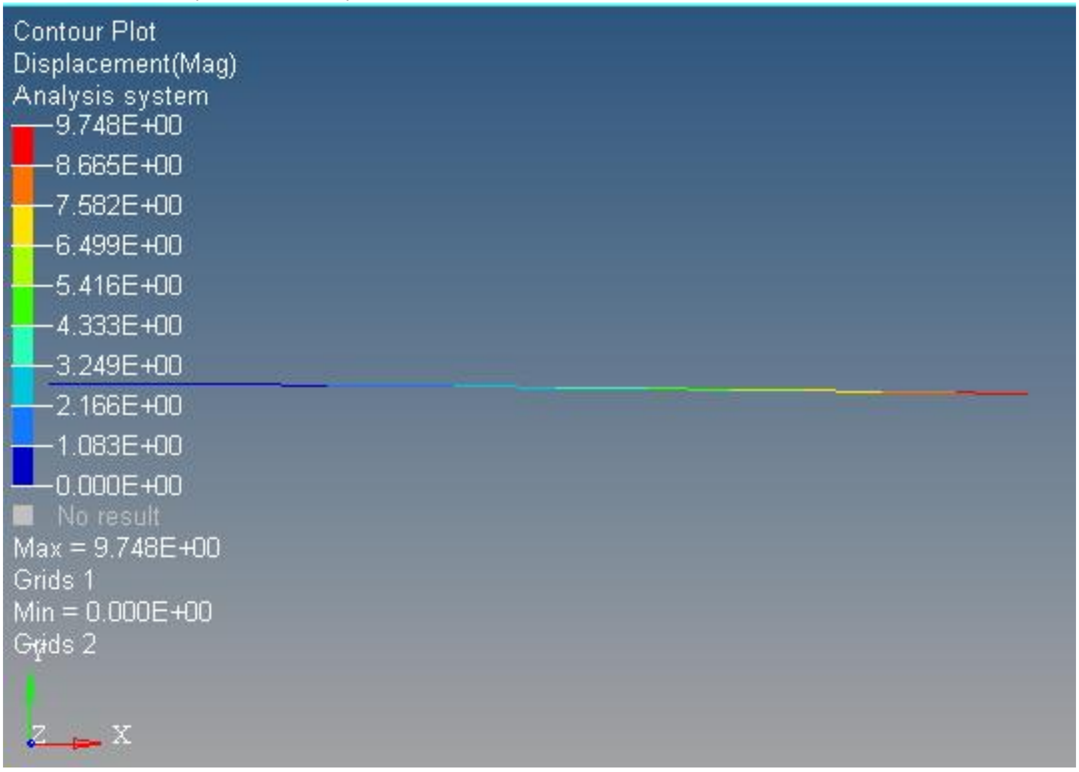
3. Select the file as shown below. Click **open** to load the result file. Then click **Apply.**



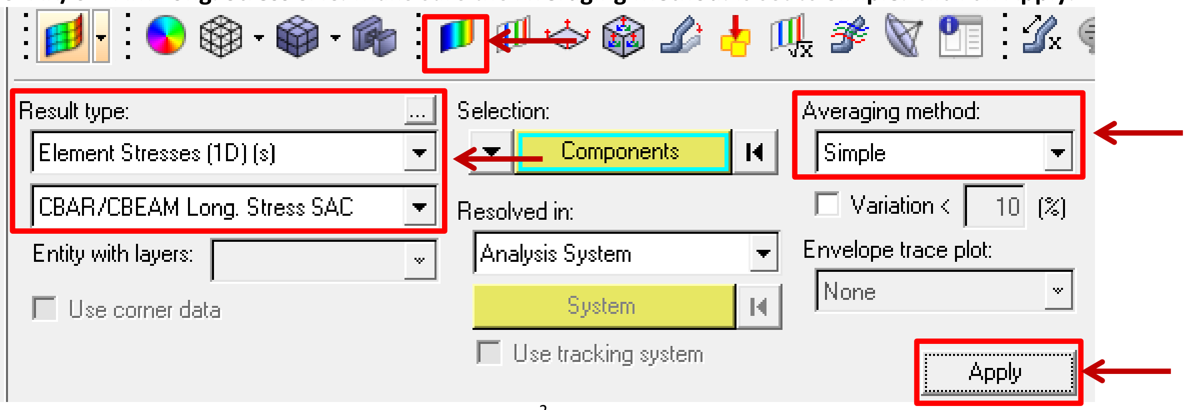
4. Click on contour icon as shown below. Click **Apply** to view the Displacement contour plot of the beam.



5. The maximum displacement as per the software is 9.7 mm.



6. Again click on **contour** then in **Result type:** select **Element Stresses (1D) (s)** from the drop down list, then below that select **CBAR/CBEAM Long. Stress SAC.** Make sure the **Averaging Method:** is set to **Simple.** Click on **Apply.**



7. The maximum Stress for the Beam is 305.6 N/mm^2.