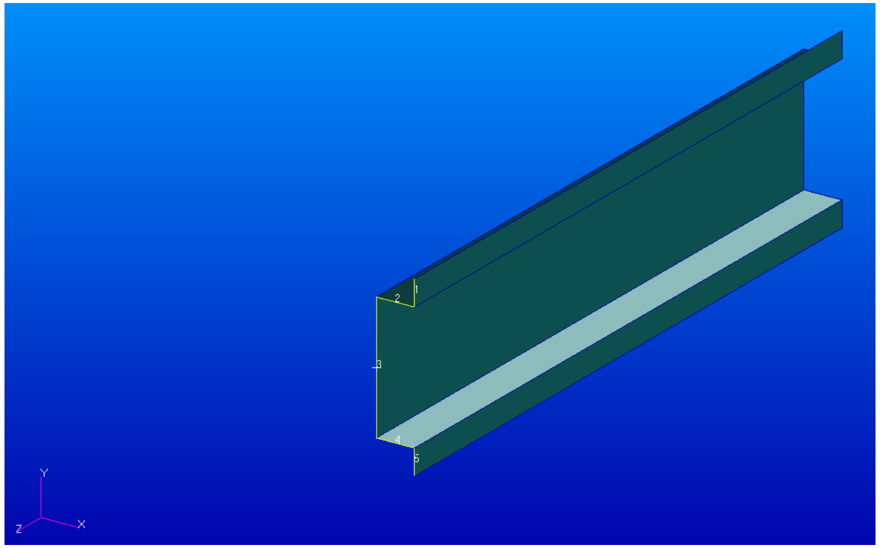
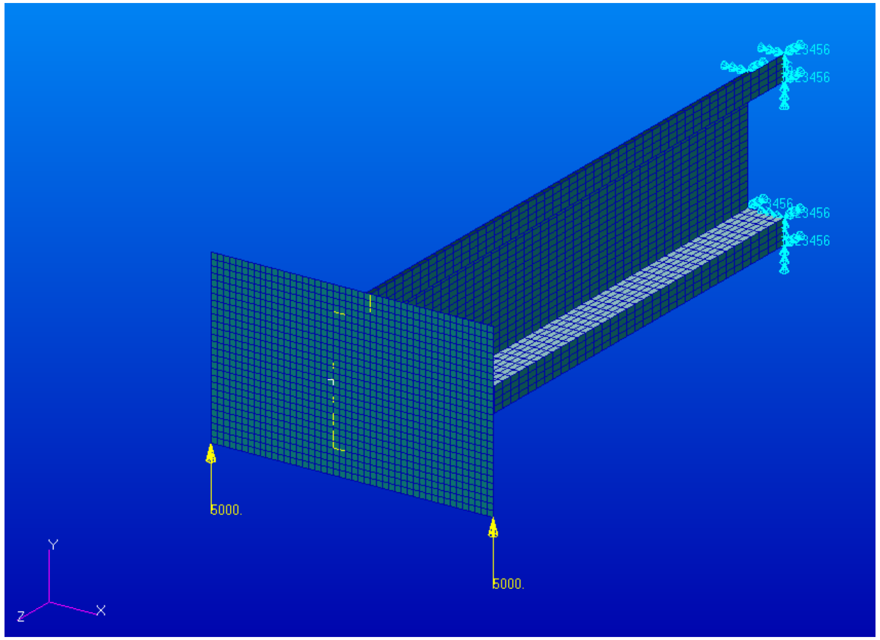
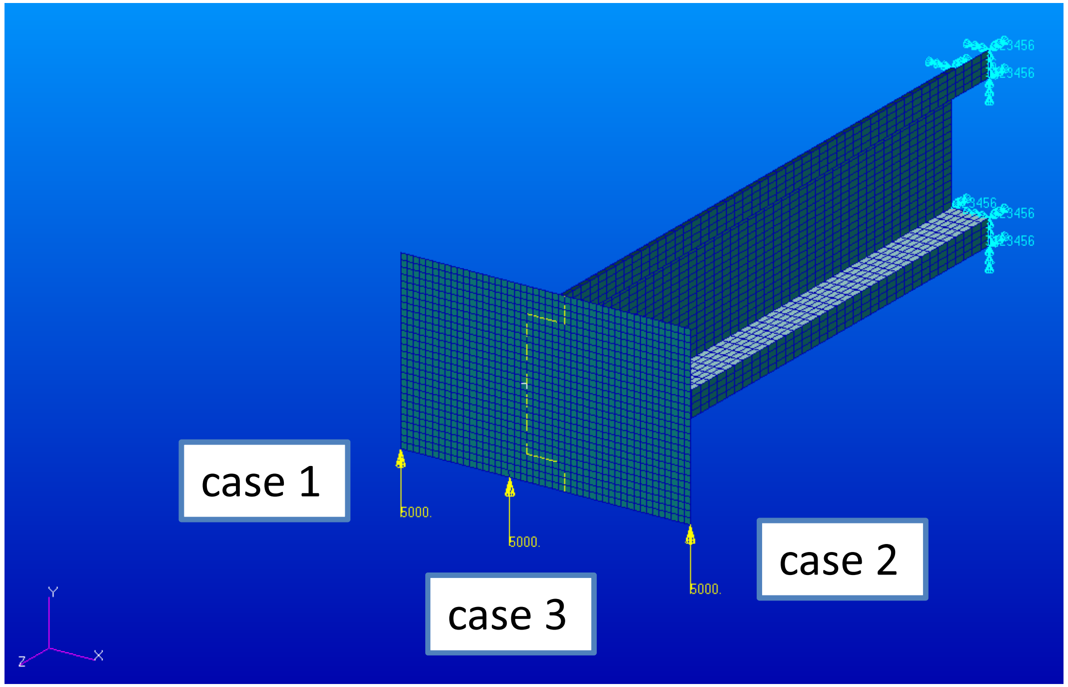
OPEN SECTION THIN-WALLED BEAM







<u>Procedure of the exercise:</u>

1) **First** analysis of the beam's model with **two** load subcases (case 1 and case 2)
2) **Second** analysis of the beam's model with **third** load subcase for which the applied force acts through the shear center (case 3)

PROBLEM DESCRIPTION

In the open thin-walled section a shear load *Sy* is applied through the shear center (S.C.) of the section.

Details of the cross-section shape and the force application point are shown on the right.

Determine:

a) position of the shear center ($X_{S.C.}$)

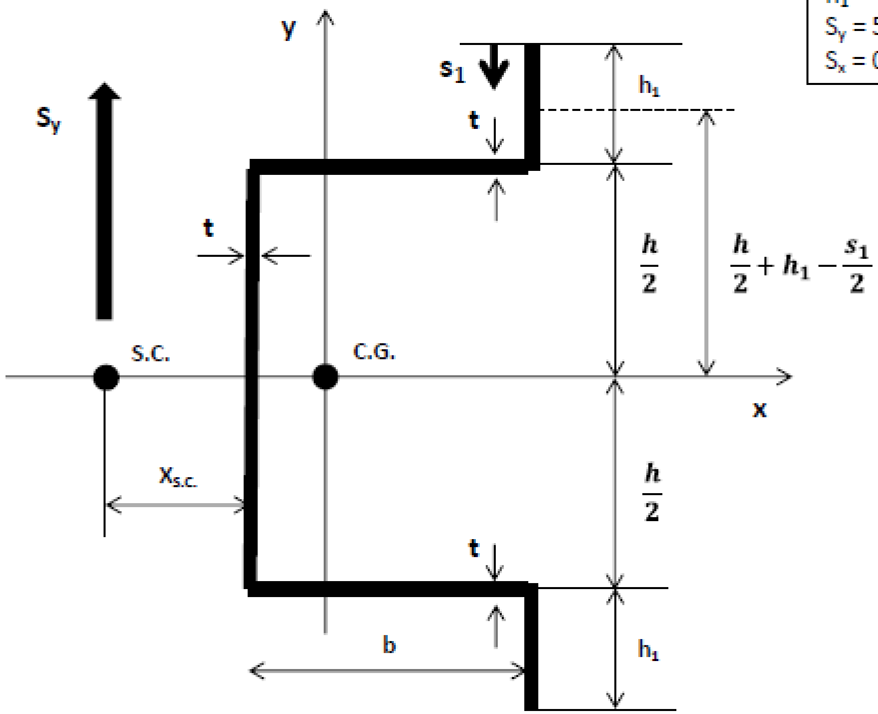
The model of the beam with open thin-walled section will be created. The appropriate load and boundary conditions will be applied to it.

There will be **3 load subcases**:

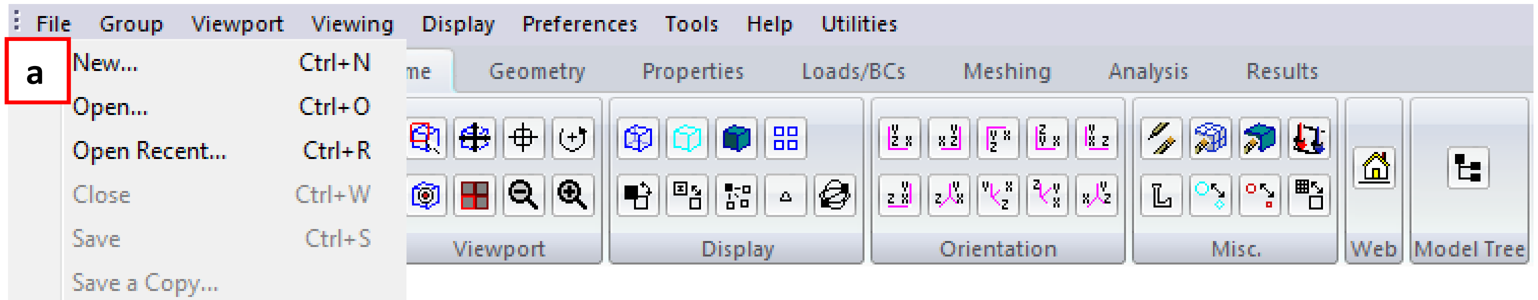
- 1st load subcase
- 2nd load subcase
- 3rd load subcase

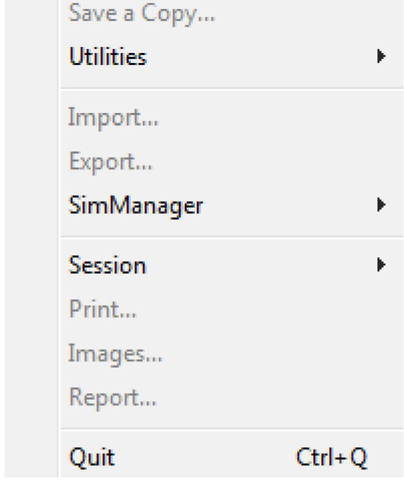
length of the beam = 100 [cm]

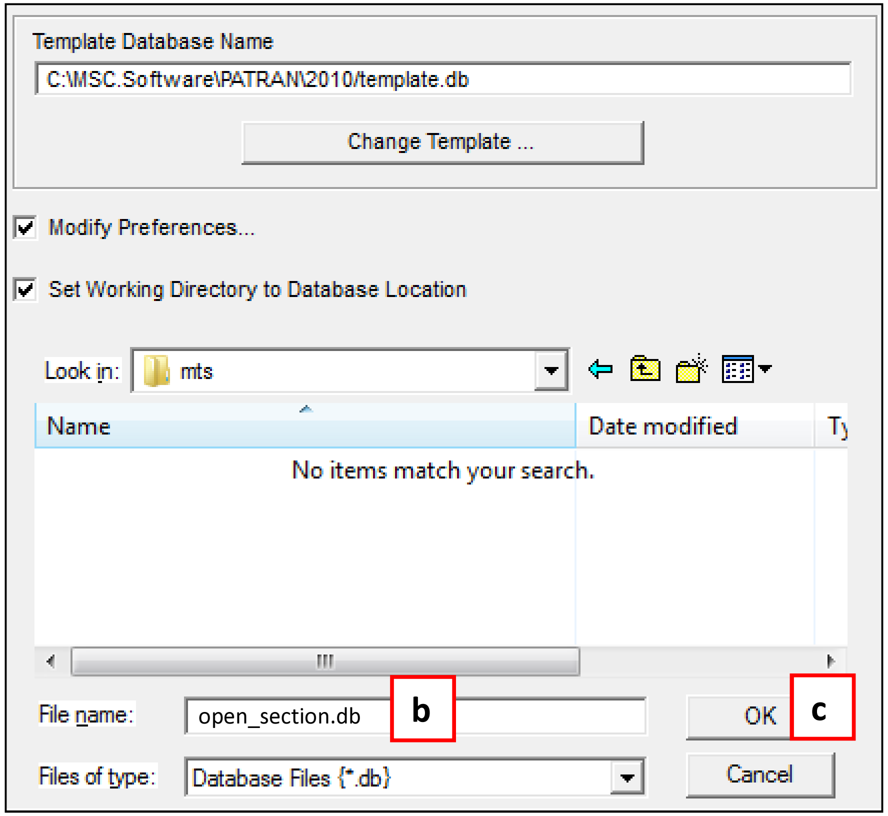
Data:
$t = 0{,}5$ [cm]
$h = 20$ [cm]
$b = 6$ [cm]
$h_1 = 4$ [cm]
$S_y = 5$ [kN]
$S_x = 0$ [N]

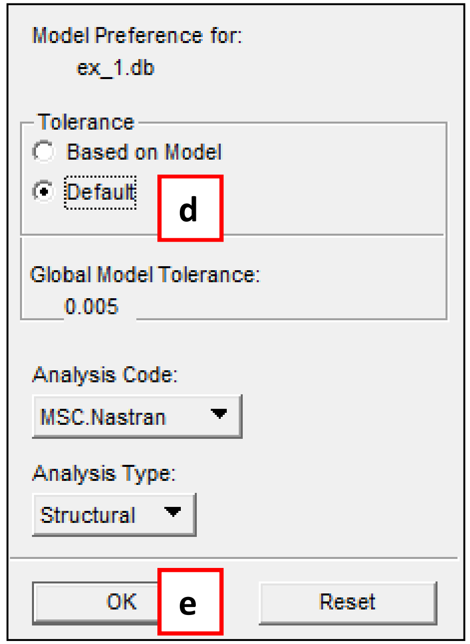


Units: mm, N, MPa









Create a new database:

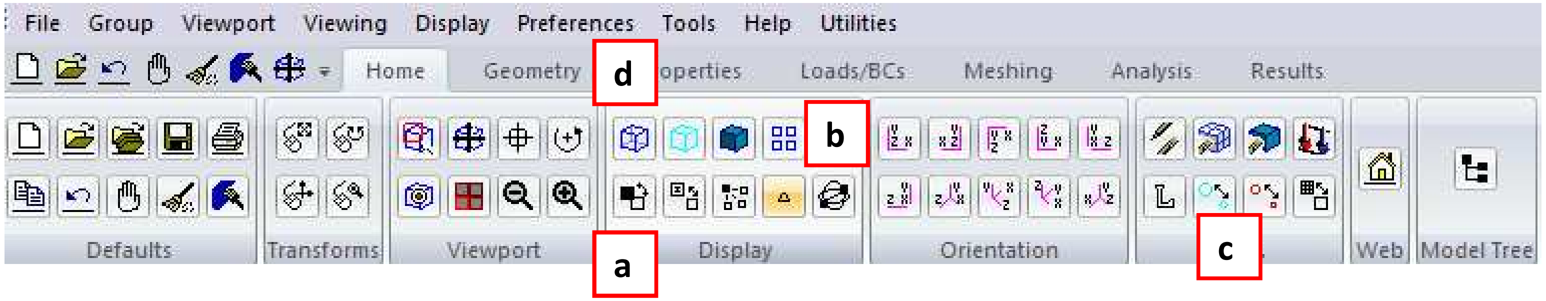
a. **File / New...**
b. Enter **open_section.db** as the File name
c. Click **OK**
d. Select **Default**
e. Click **OK**

OPEN SECTION THIN-WALLED BEAM

A **white background** of **all** figures is **obligatory**.

OPEN SECTION THIN-WALLED BEAM

GEOMETRY CREATION



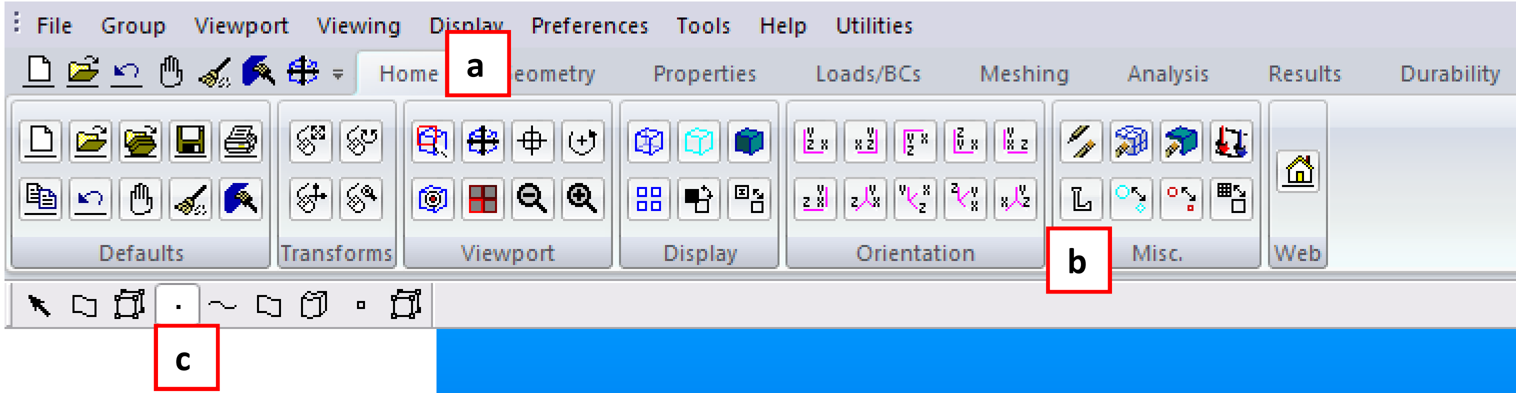
a. Change *Background Color* to **White** (click on the **Cycle Background** icon)
b. Click on the **Front view** icon
c. Click on the **Point size** icon

Create geometry points using coordinates from **Table 1**:

d. Click on **Geometry** tab and choose **Create/Point/XYZ** from right menu
e. Uncheck **Auto Execute**
f. Enter coordinates of the first point **[60 140 0]** in *Point Coordinate List*
g. Click **Apply**
h. Repeat steps d÷g for the rest of the points

Table 1. Geometry points coordinates

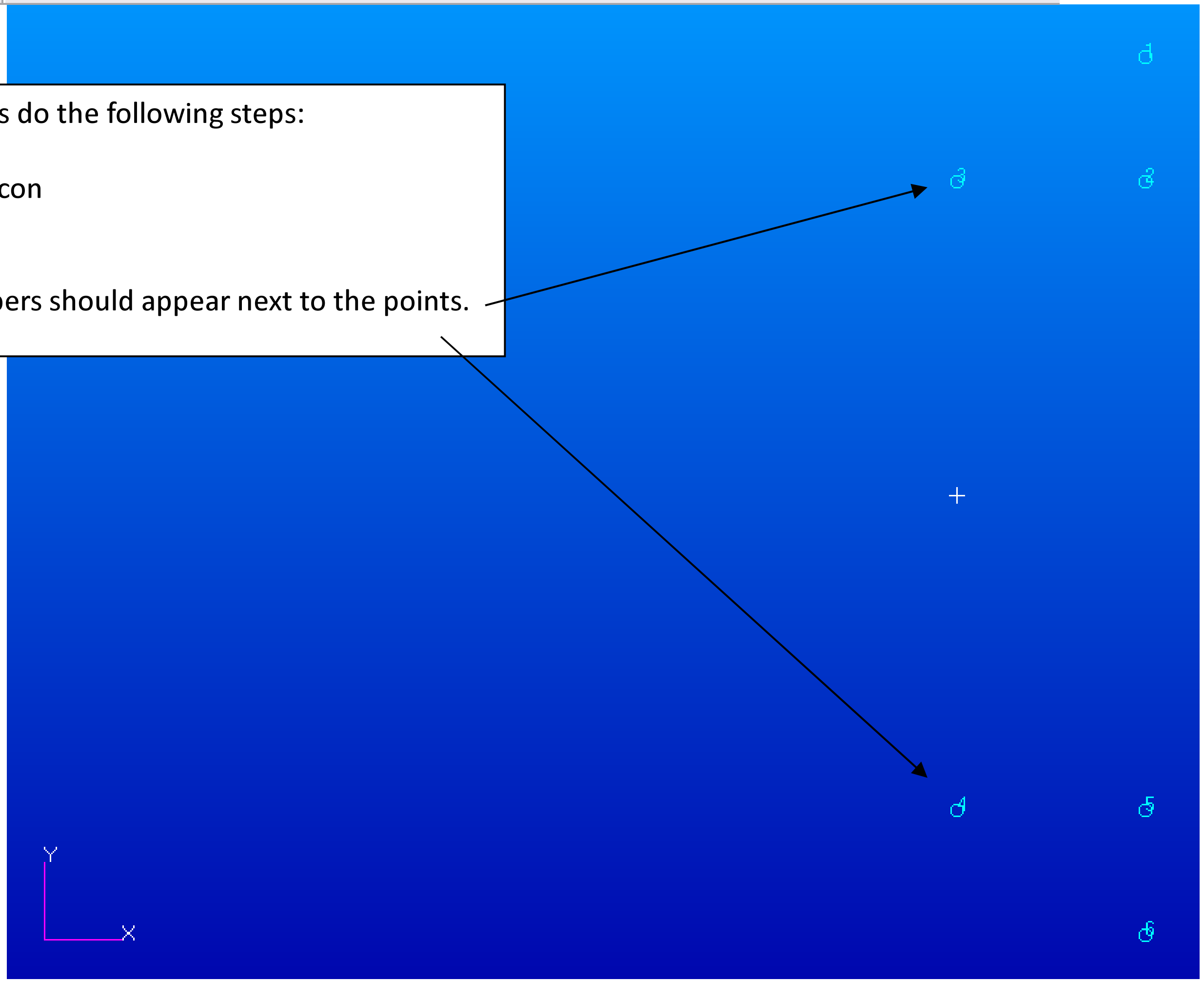
No.	X	Y	Z
1	60	140	0
2	60	100	0
3	0	100	0
4	0	-100	0
5	60	-100	0
6	60	-140	0

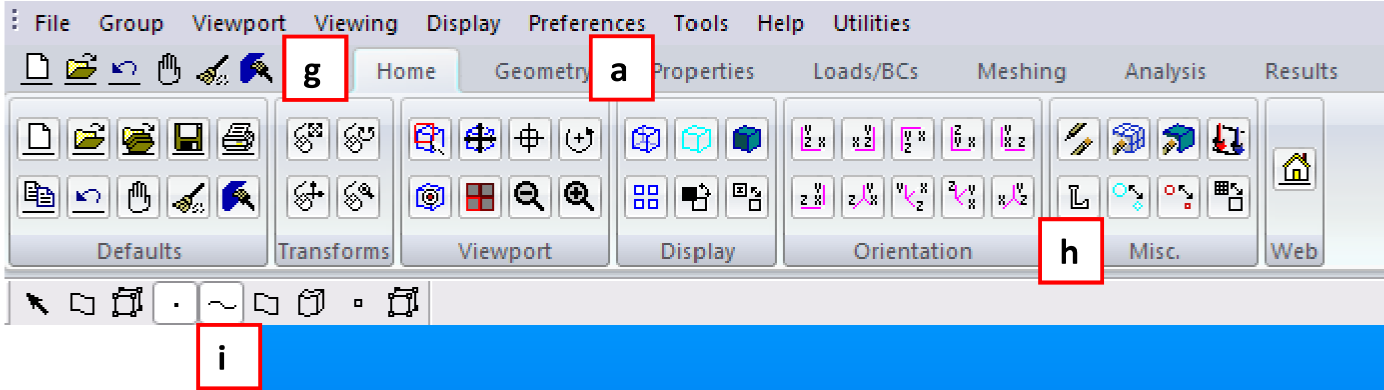


In order to see the points labels do the following steps:

a. Click on **Home** tab
b. Click on the **Label Control** icon
c. Click on the **Point** icon

Then, the labels with the numbers should appear next to the points.





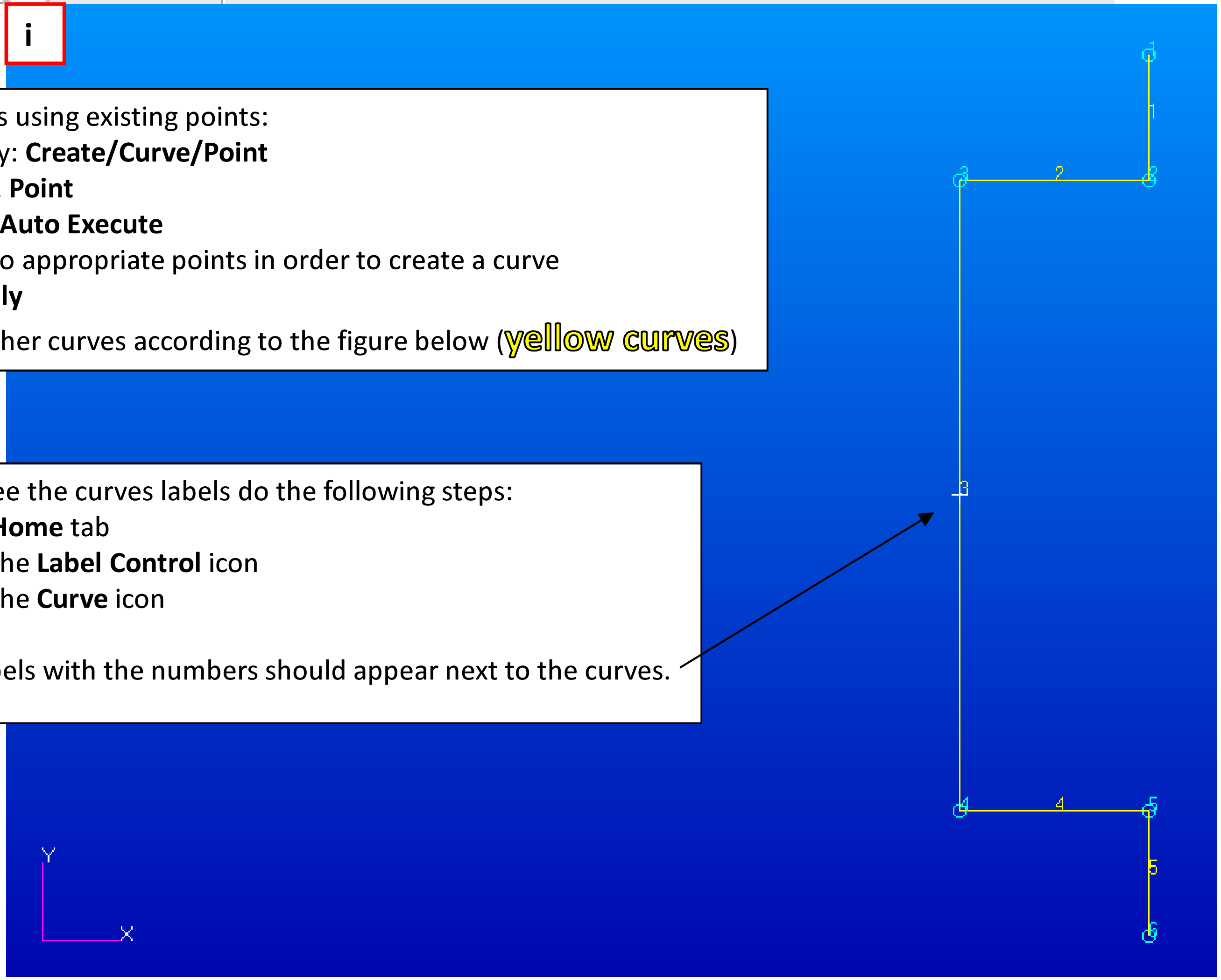
Create curves using existing points:

a. Geometry: **Create/Curve/Point**
b. Option: **2 Point**
c. Uncheck **Auto Execute**
d. Select two appropriate points in order to create a curve
e. Click **Apply**
f. Create other curves according to the figure below (yellow curves)

In order to see the curves labels do the following steps:

g. Click on **Home** tab
h. Click on the **Label Control** icon
i. Click on the **Curve** icon

Then, the labels with the numbers should appear next to the curves.



Create surfaces using existing curves:

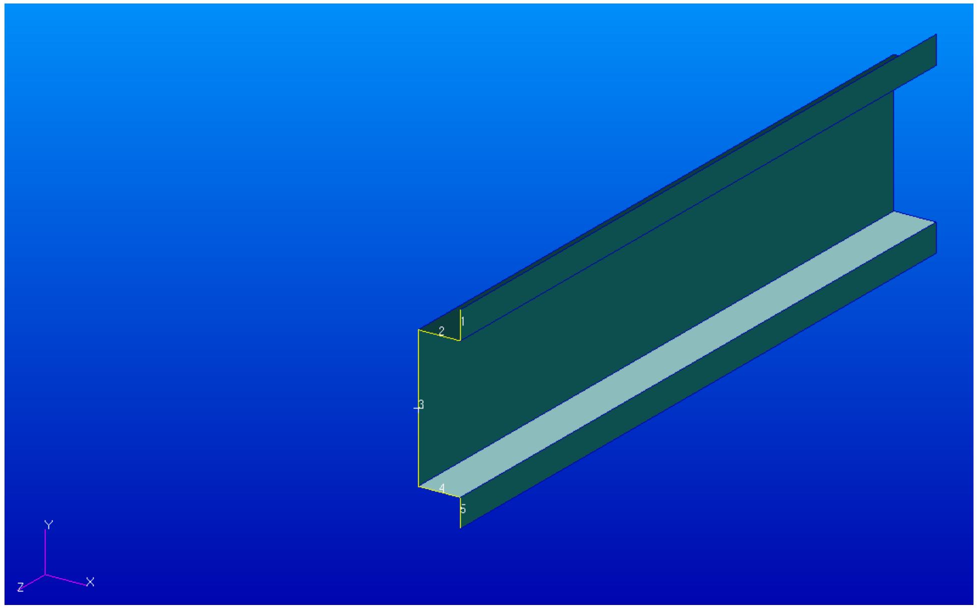
a. Geometry: **Create/Surface/Extrude**
b. Translation Vector: **<0 0 -1000>**
c. Uncheck **Auto Execute**
d. Click on the **Curve List** panel
e. Select all curves in order to create surfaces
f. Click **Apply**

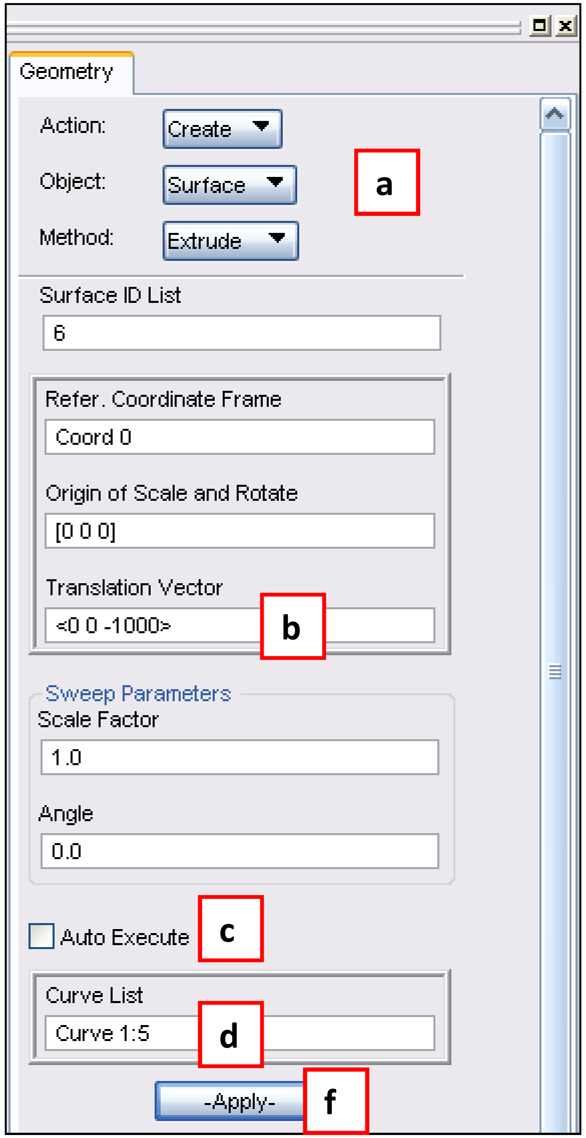
In **Home tab**:

g. Click on the **Iso 1 view** icon
h. Click on the **Fit view** icon
i. Click on the **Smooth shaded** icon

Save the figure of the geometrical model (<u>remember about white background</u>):

j. File/Images…
k. Choose Image Format: **JPEG**
l. Click **Apply**



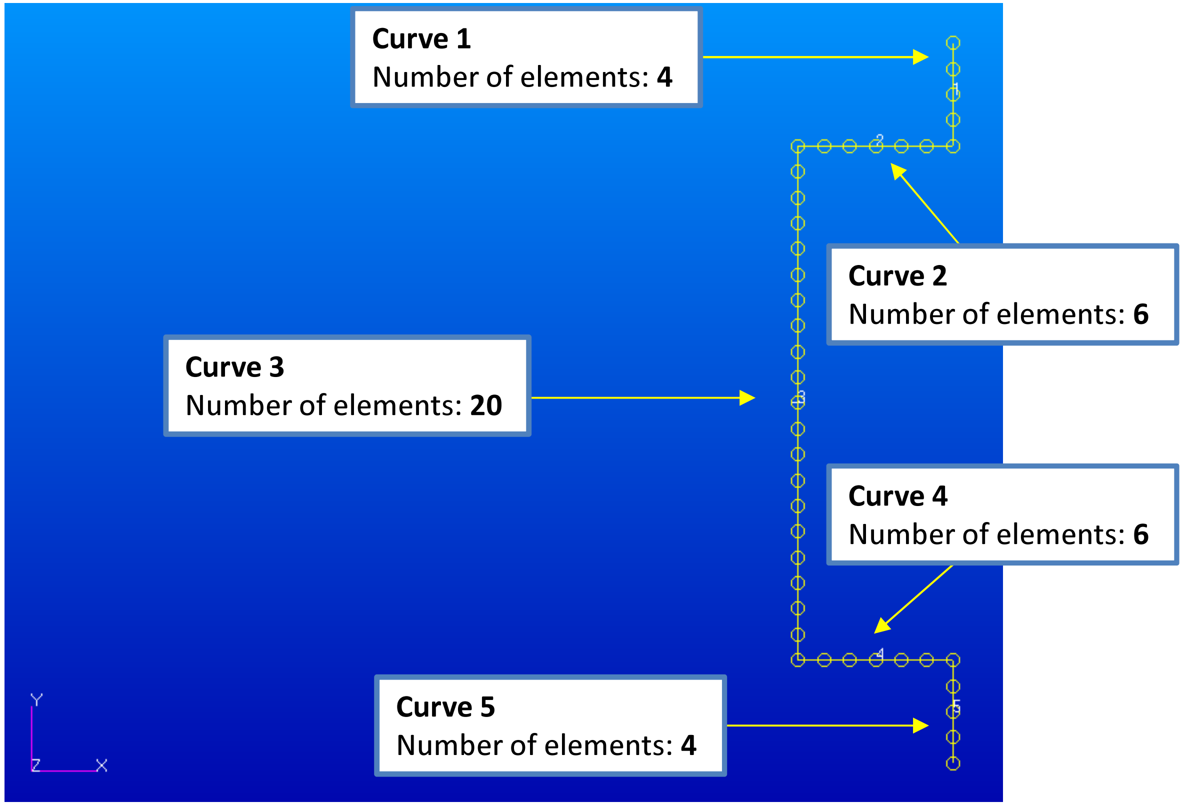


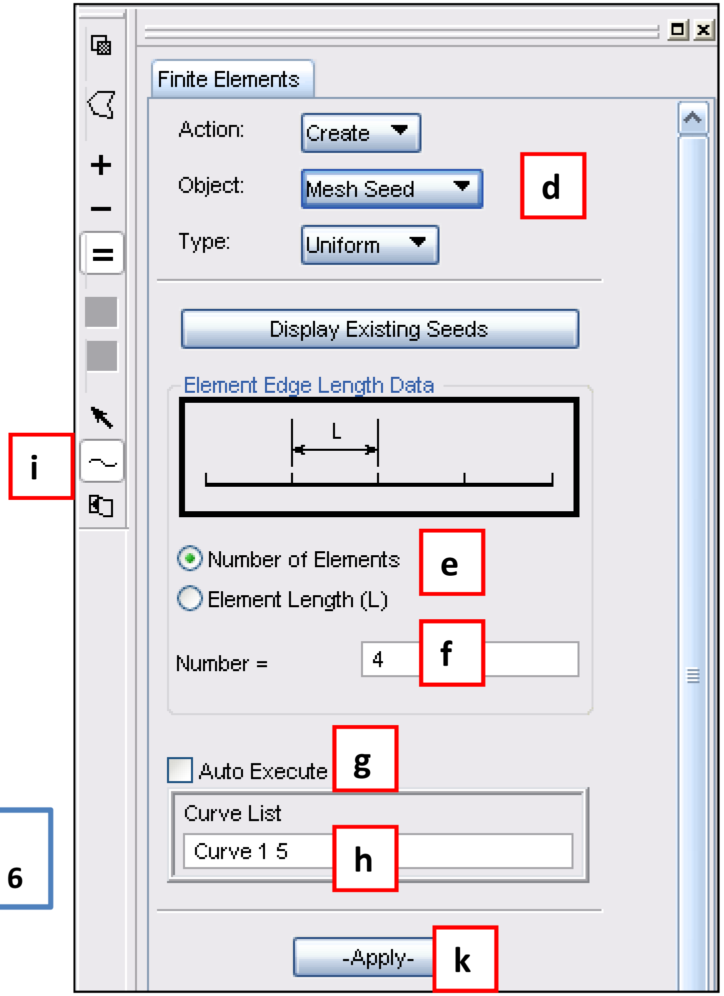
OPEN SECTION THIN-WALLED BEAM

MESH CREATION

In **Home tab**:
a. Click on the **Front view** icon
b. Click on the **Fit view** icon

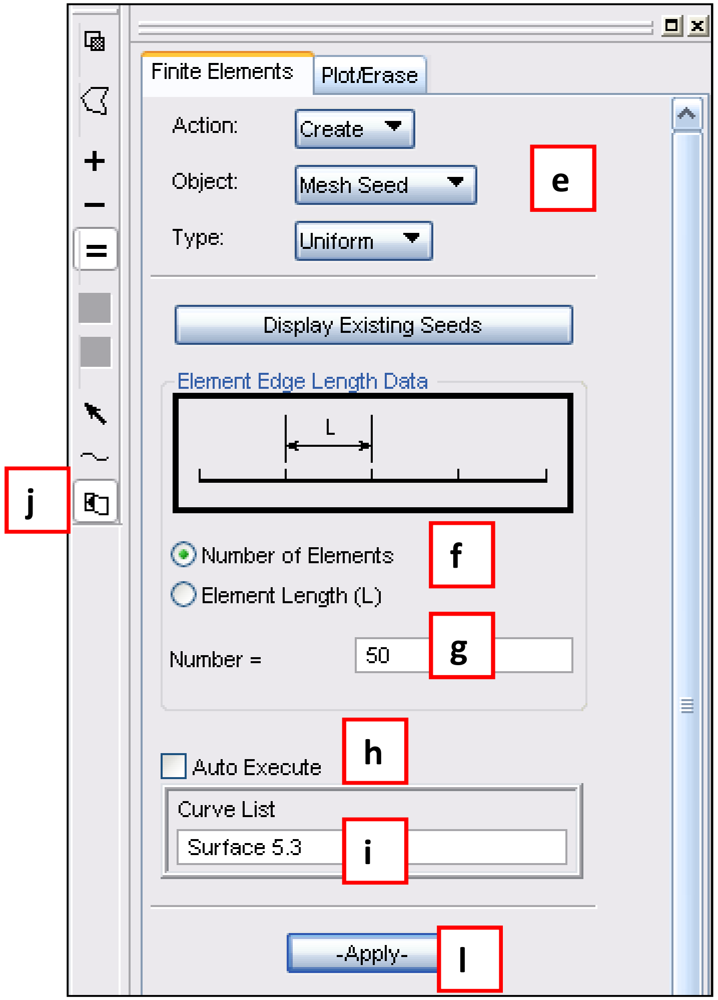
Create mesh seeds:
c. Click on the **Meshing** tab
d. Meshing: **Create/Mesh Seed/Uniform**
e. Select **Number of Elements**
f. Enter **4** as the Number (of elements)
g. Uncheck **Auto Execute**
h. Click on **Curve List** panel
i. Select **Curve** icon
j. Select two shorter vertical curves: **Curve 1** and **Curve 5** (holding ***Shift*** button)
k. Click **Apply**
l. Repeat steps f÷j for the rest of the curves according to the figure below:

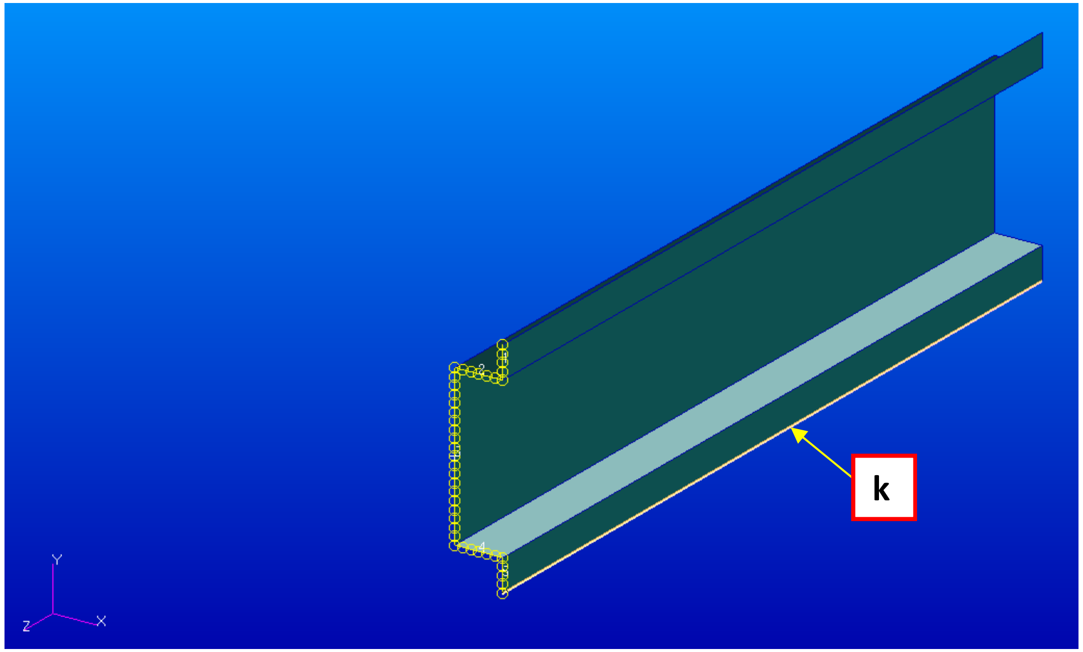




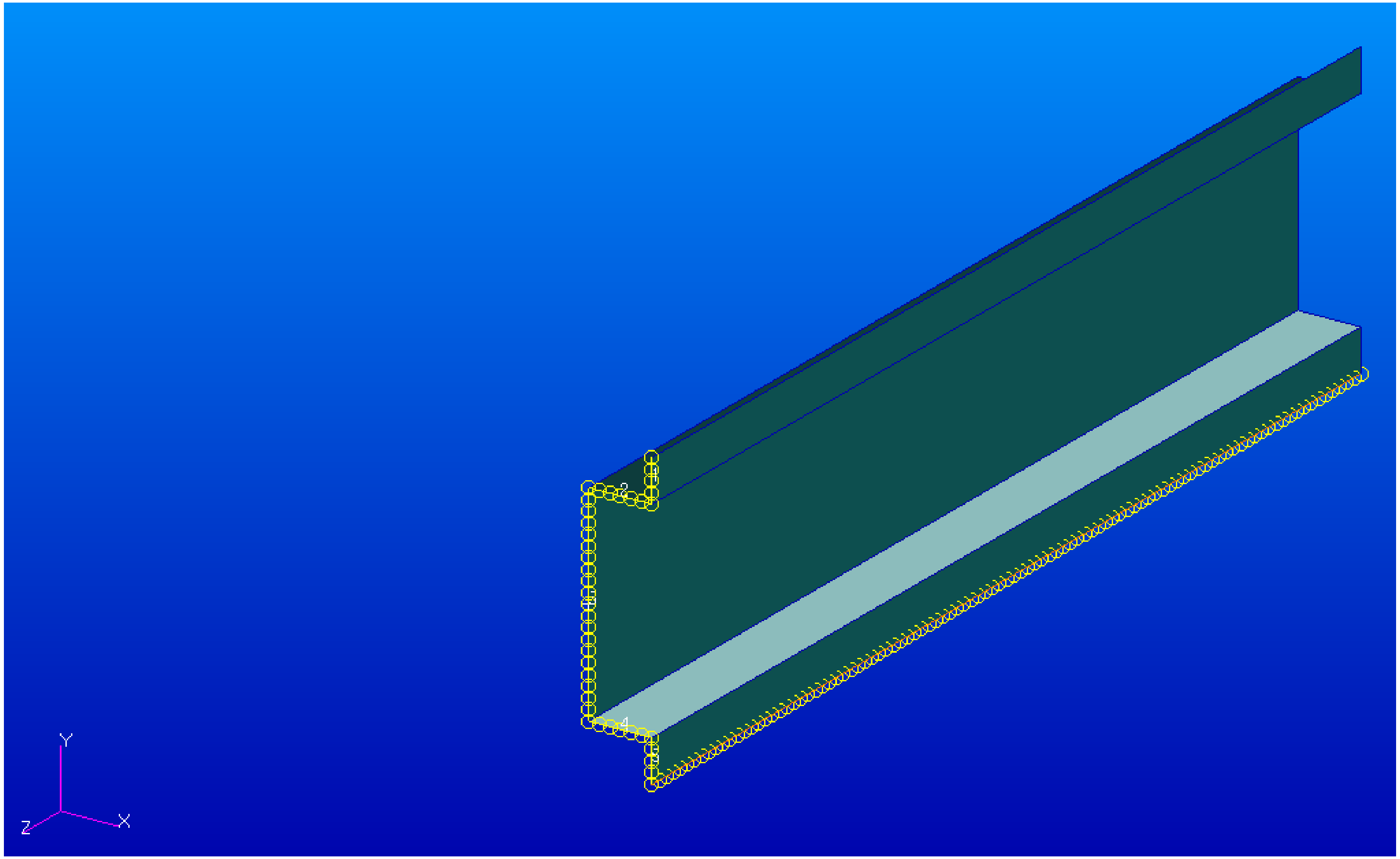
In **Home tab**:
a. Click on the **Iso 1 view** icon
b. Click on the **Fit view** icon
c. Click on the **Smooth shaded** icon

Create mesh seeds on one edge along beam's length:
d. Click on the **Meshing** tab
e. Meshing: **Create/Mesh Seed/Uniform**
f. Select **Number of Elements**
g. Enter **50** as the Number (of elements)
h. Uncheck **Auto Execute**
i. Click on **Curve List** panel
j. Select **Edge** icon
k. Select one edge along beam's length
l. Click **Apply**



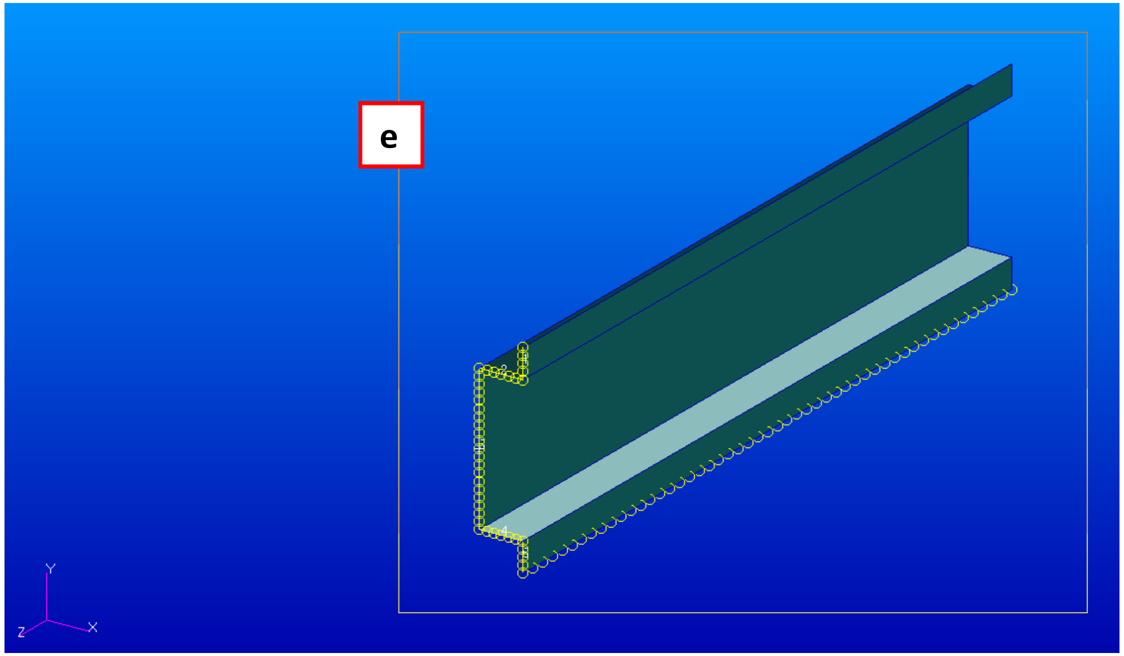


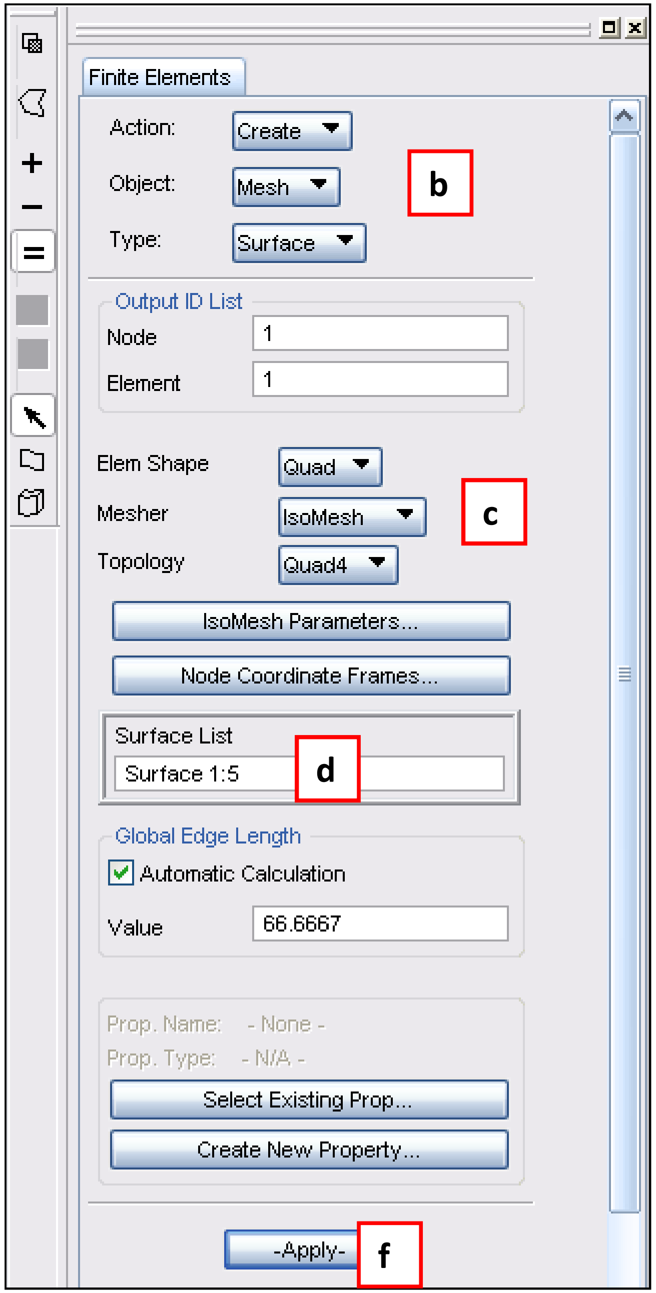
The following figure shows the geometrical model with the mesh seeds.



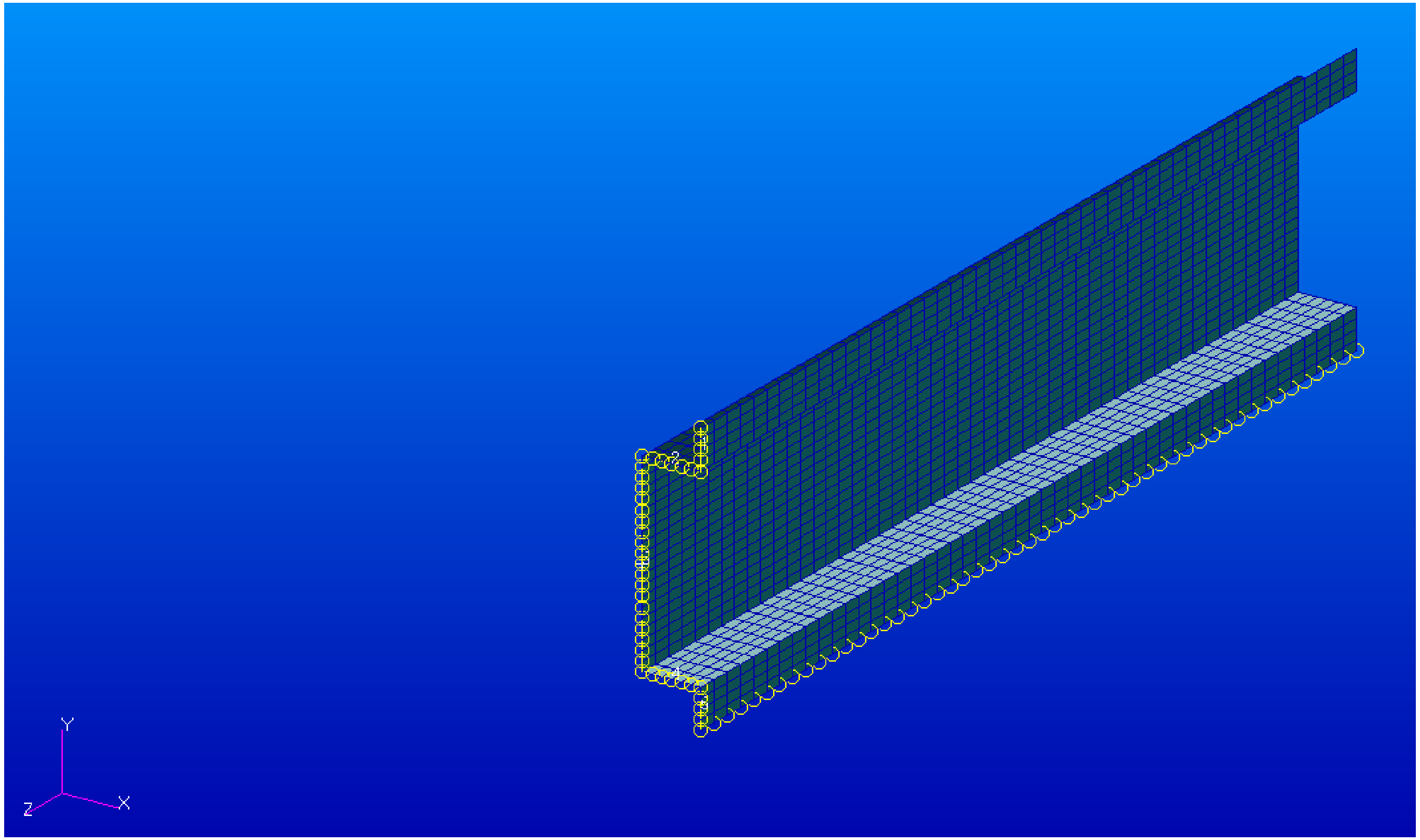
Create mesh:

a. Click on the **Meshing** tab
b. Meshing: **Create/Mesh/Surface**
c. Elem Shape: **Quad**
   Mesher: **IsoMesh**
   Topology: **Quad4**
d. Click on **Surface List** panel
e. Select all surfaces
f. Click **Apply**





The following figure shows the FE model beam model with the mesh seeds.



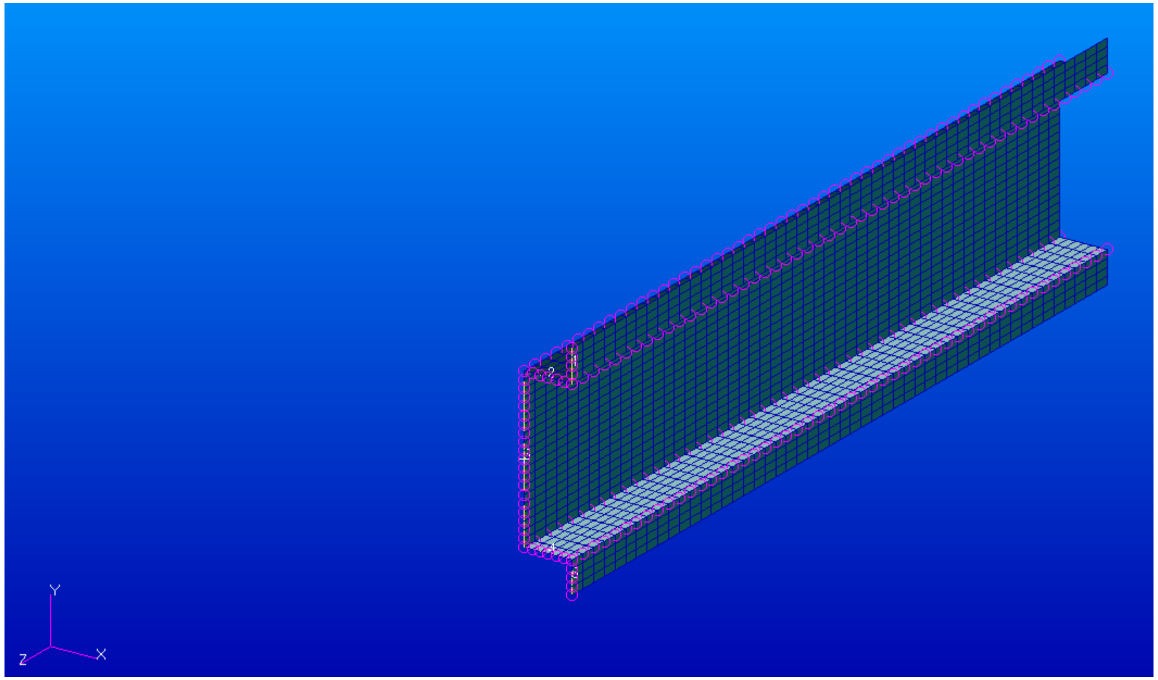
Delete the duplicate nodes:

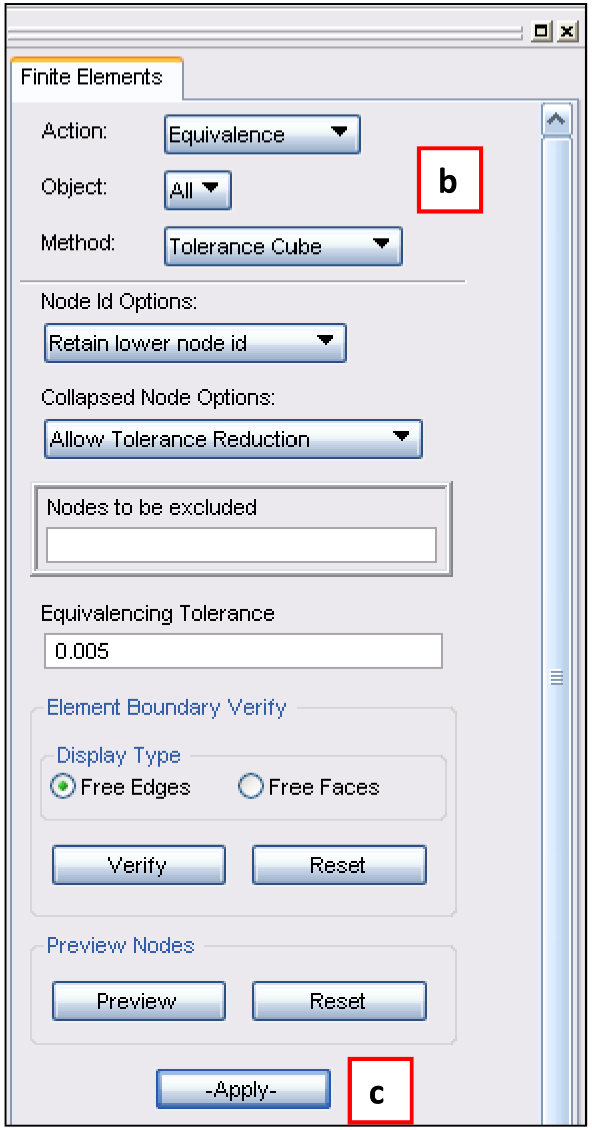
a. Click on **Meshing** tab

b. Equivalence/All/Tolerance Cube

c. Click **Apply**

The figure below shows the FE model of the beam after deletion of the duplicate nodes.



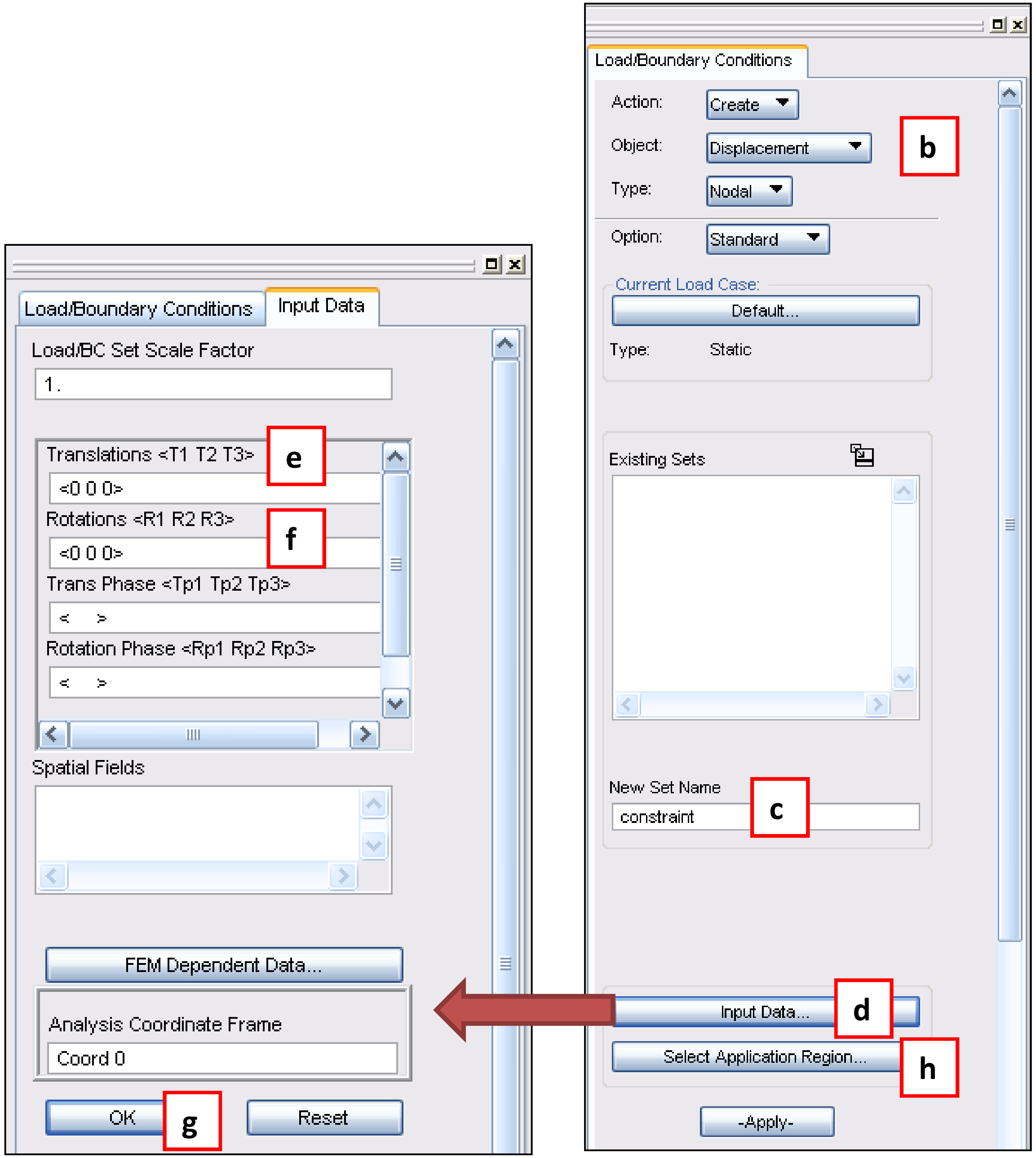


OPEN SECTION THIN-WALLED BEAM

FIXING OF THE MODEL

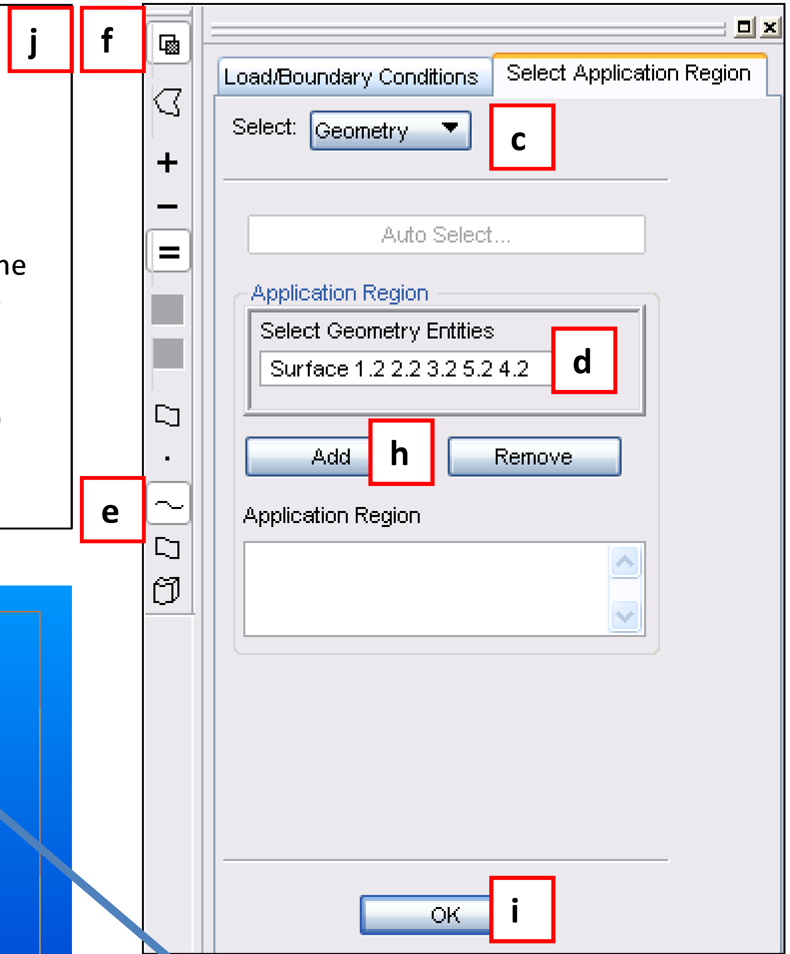
Apply the boundary conditions:

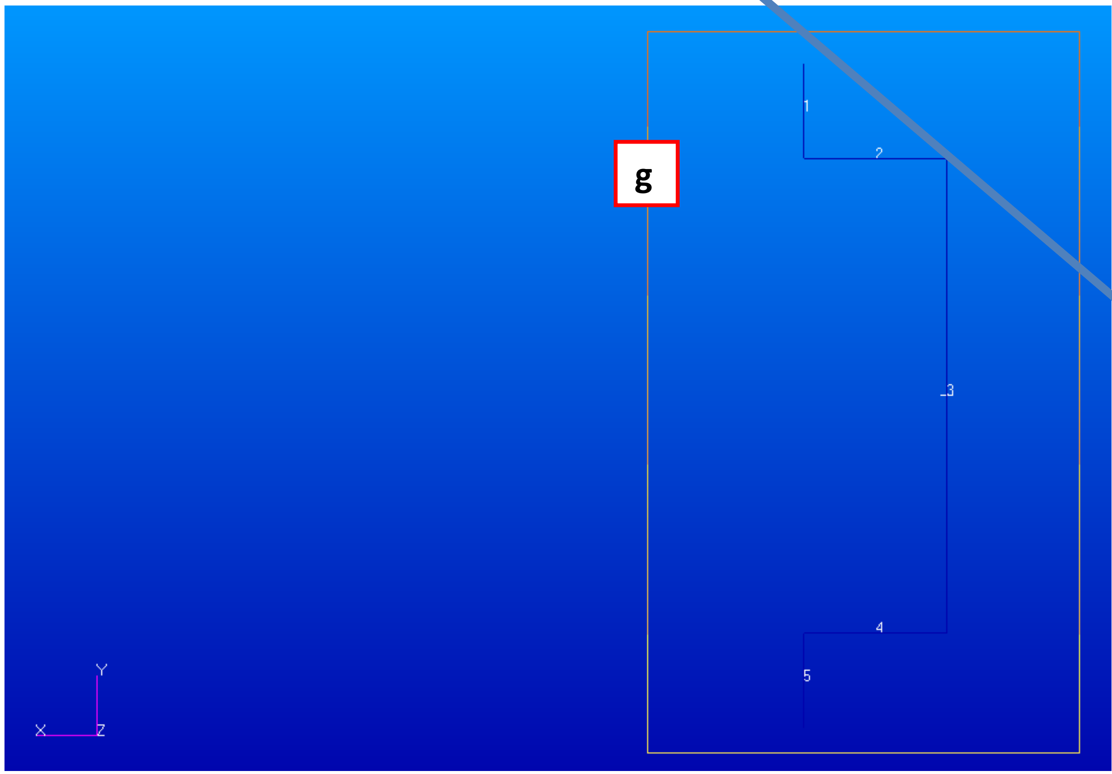
a. Go to **Loads/BCs** tab
b. Create/Displacement/Nodal
c. Enter **constraint** as the New Set Name
d. Click **Input Data…**
e. Enter **<0,0,0>** for the Translations
f. Enter **<0,0,0>** for the Rotations
g. Click **OK**
h. Click **Select Application Region…**

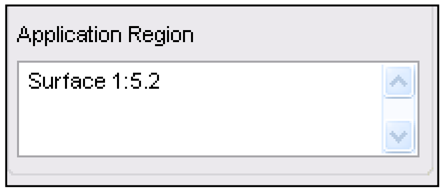


In **Home tab**:

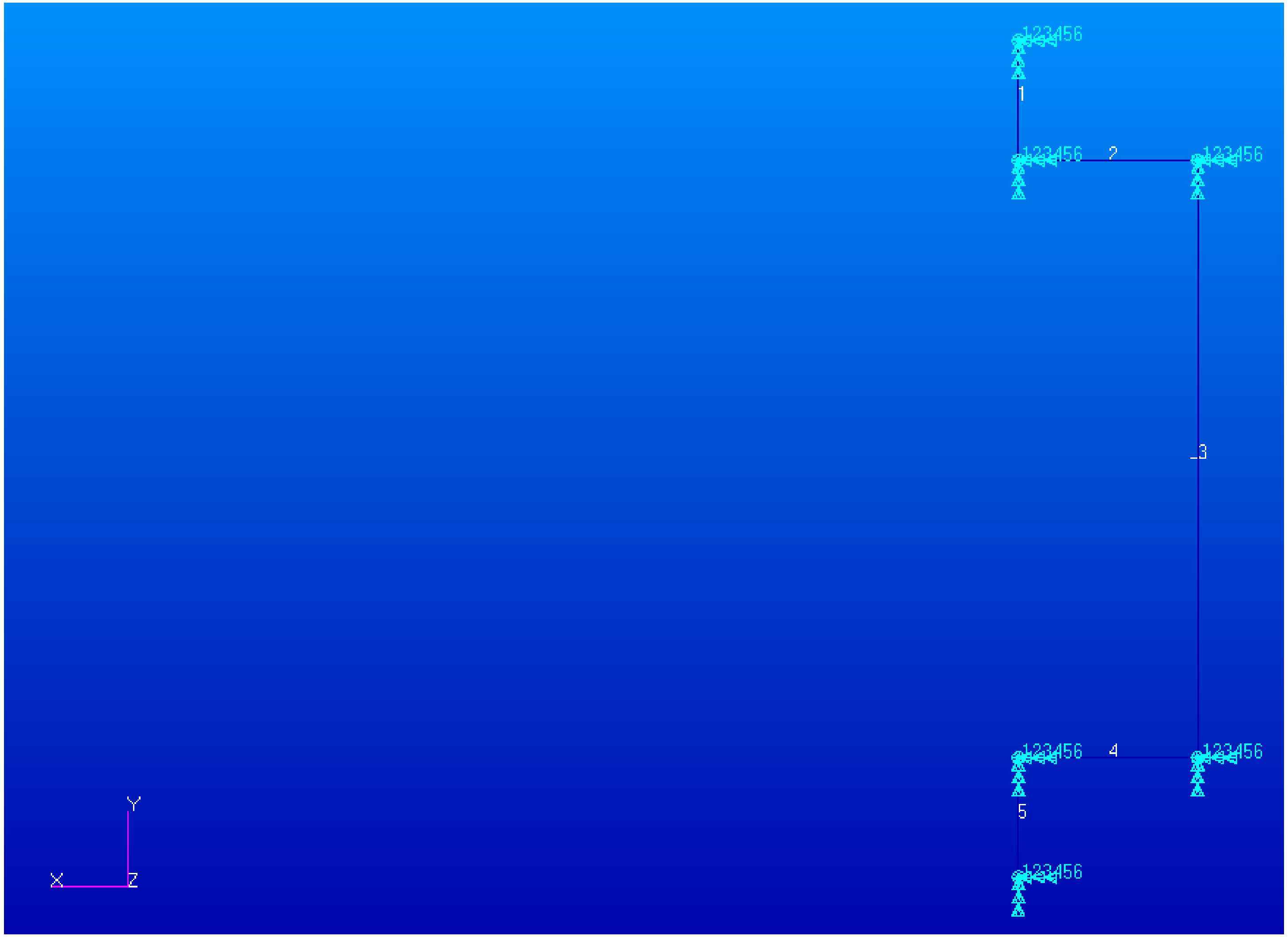
a. Click on the **Rear view** icon
b. Click on the **Fit view** icon
c. Select **Geometry**
d. Click on the **Select Geometry Entities** panel
e. Select **Curve or Edge** icon
f. Select **Visible entities only** icon and click **OK** when the new window with the following *Message* appears *„Acknowledgement requested from application Graphics Manager…"*
g. Select all visible edges of surfaces
h. Click **Add** (after this step the following data is shown in *Application Region*)
i. Click **OK**
j. <u>Uncheck</u> **Visible entities only** icon







The following figure shows the fixing of the beam's model (rear view).



OPEN SECTION THIN-WALLED BEAM

CREATION OF THE ADDITIONAL SURFACE AND ITS MESHING

The additional surface is created for the possibility of the load application.
The applied load will generate bending and torsion of the beam simultaneously.

Creation of the additional surface:

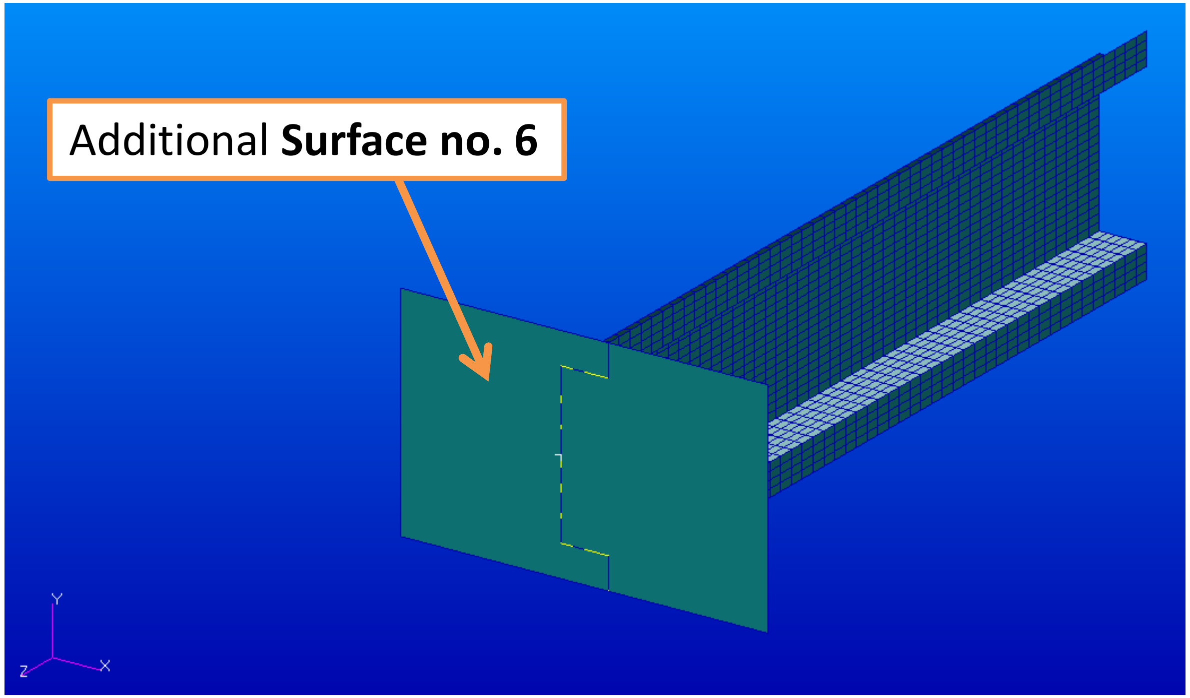
In **Home tab**:

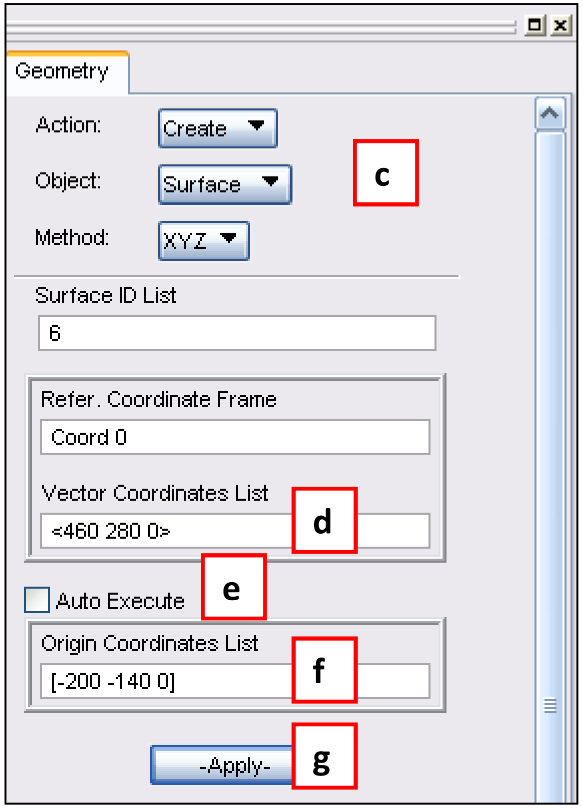
a. Click on the **Iso 1 view** icon
b. Click on the **Fit view** icon

In **Geometry tab**:

c. Choose **Create/Surface/XYZ** from right menu
d. Enter **<460 280 0>** as Vector Coordinates List
e. Uncheck **Auto Execute**
f. Enter **[-200 -140 0]** as Origin Coordinates List
g. Click **Apply**

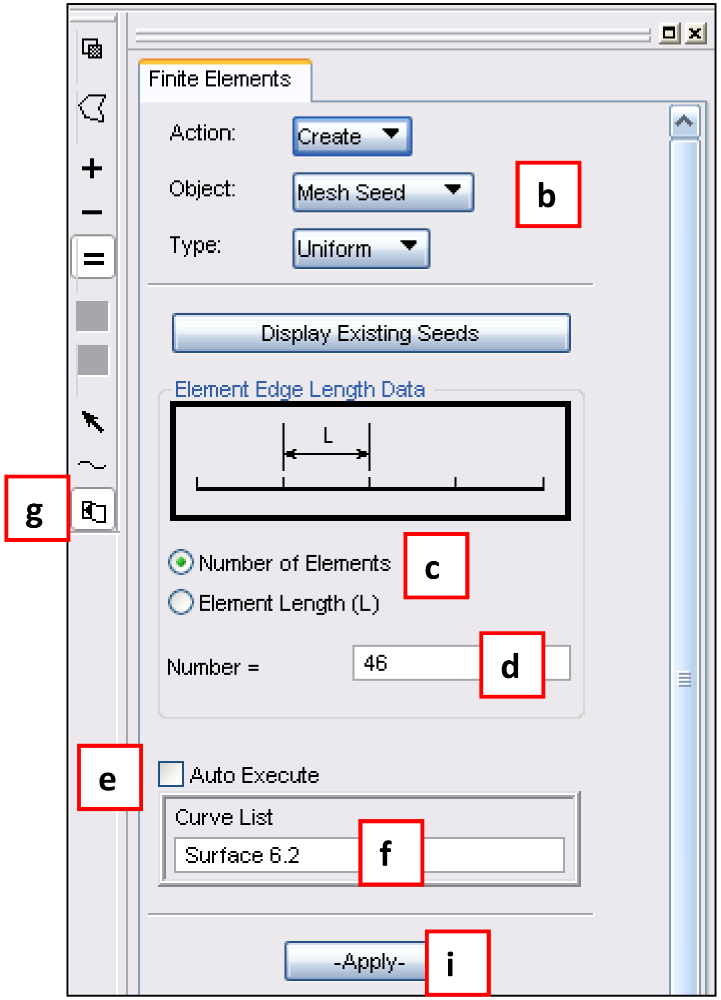
The figure below shows the model after creation of the additional surface.

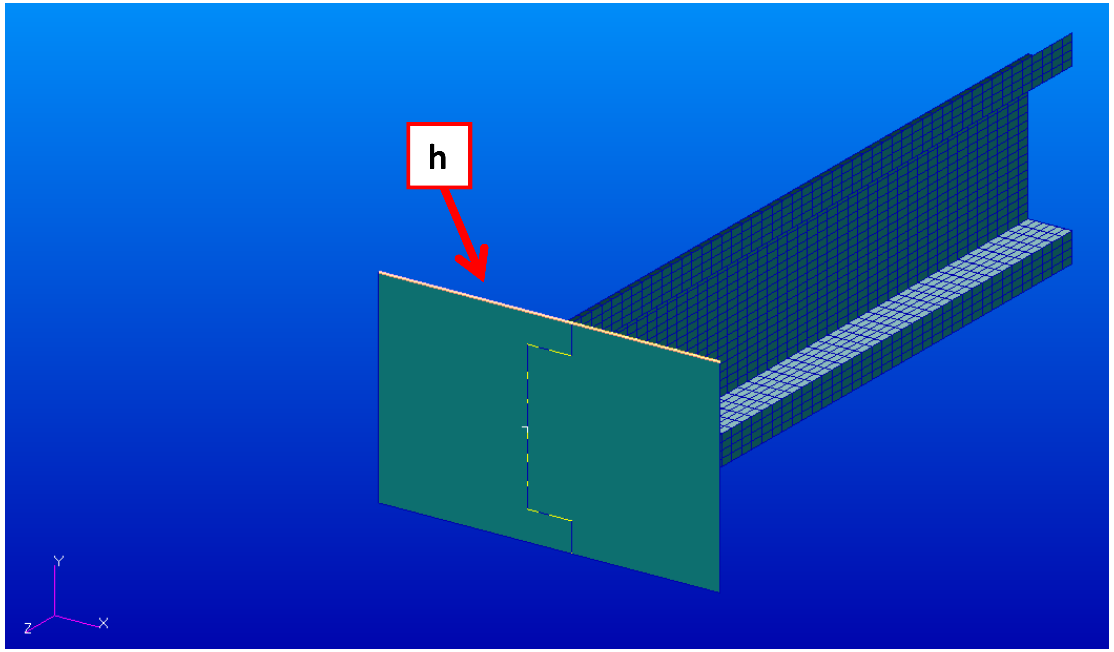




Create mesh seeds on additional surface's edges:

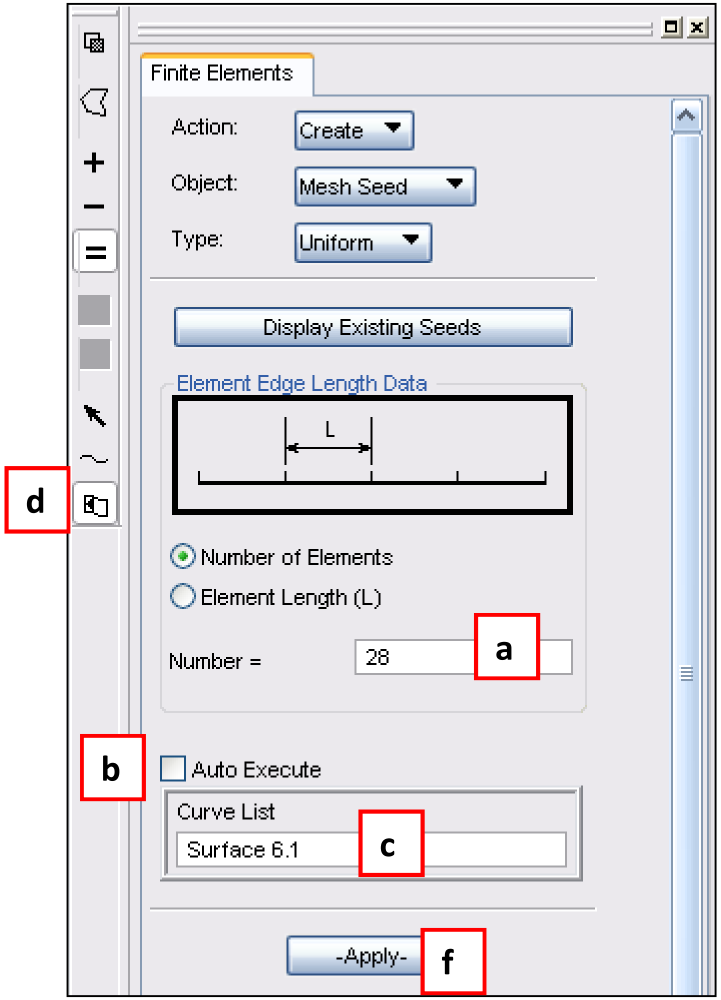
a. Click on the **Meshing** tab
b. Meshing: **Create/Mesh Seed/Uniform**
c. Select **Number of Elements**
d. Enter **46** as the Number (of elements)
e. Uncheck **Auto Execute**
f. Click on **Curve List** panel
g. Select **Edge** icon
h. Select **longer** edge of the additional surface no. 6
i. Click **Apply**

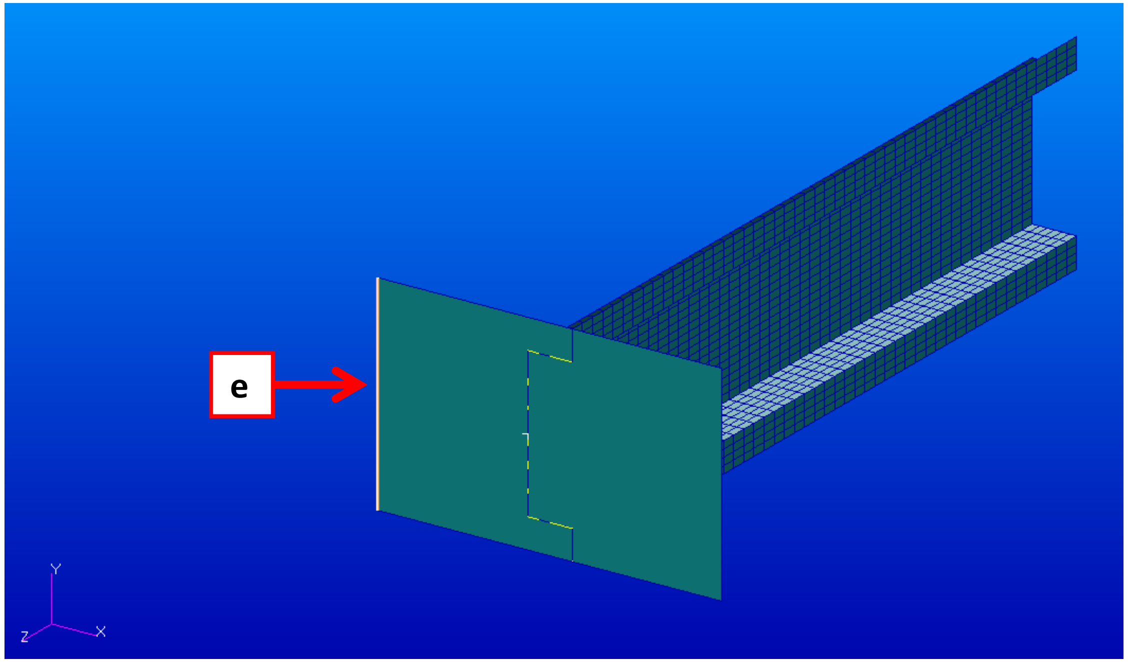




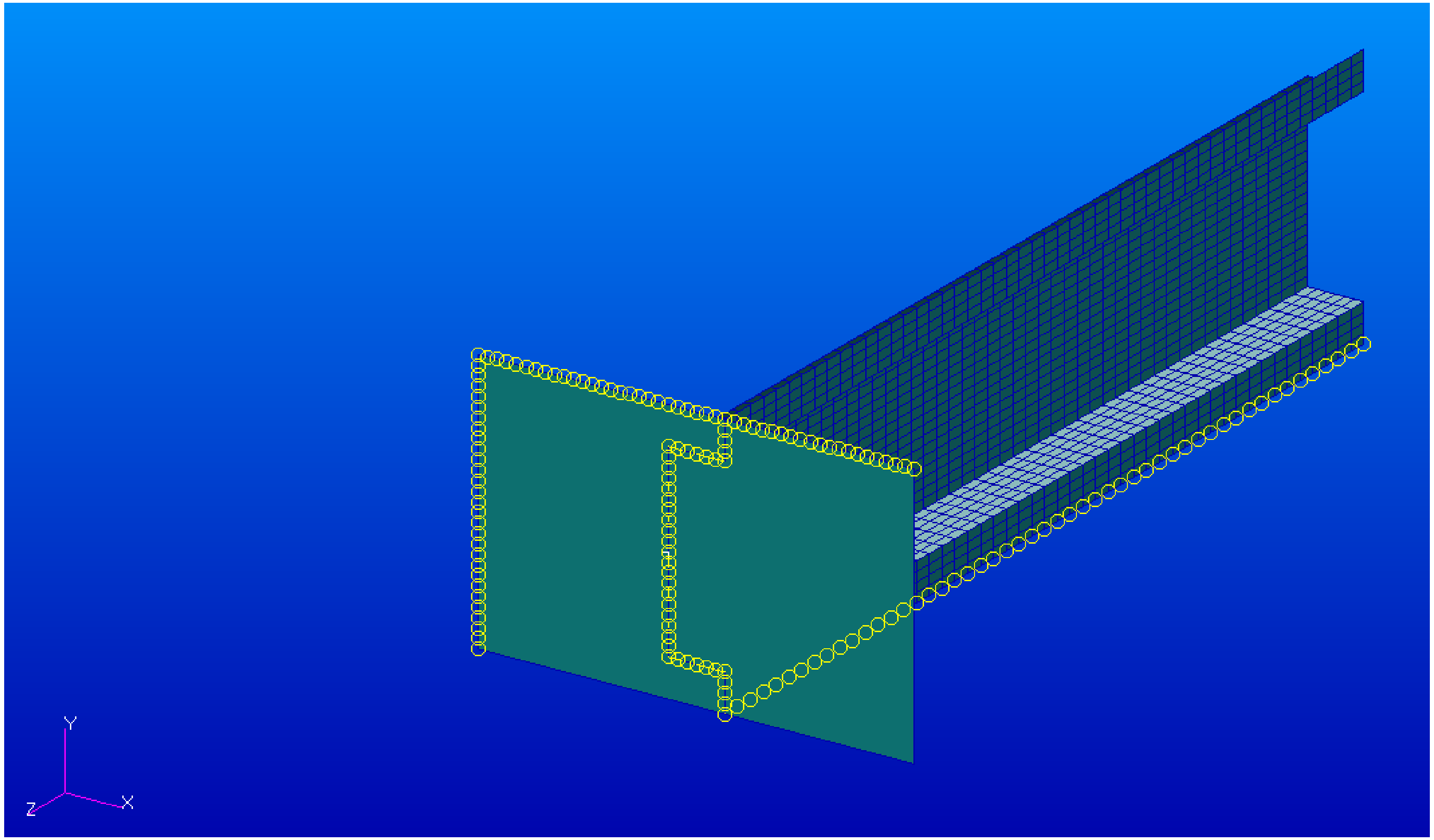
Create mesh seeds on additional surface's edges:

a. Enter **28** as the Number (of elements)
b. Uncheck **Auto Execute**
c. Click on **Curve List** panel
d. Select **Edge** icon
e. Select **shorter** edge of the additional surface no. 6
f. Click **Apply**





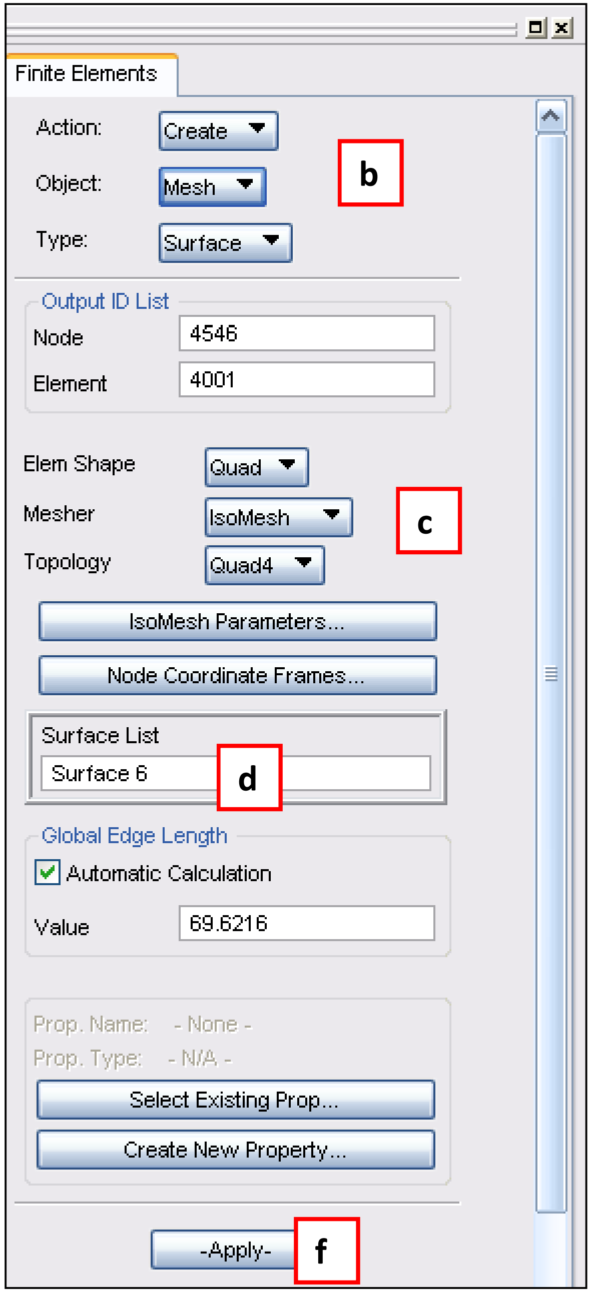
The following figure shows the beam's model with all mesh seeds including the mesh seeds on the additional surface.

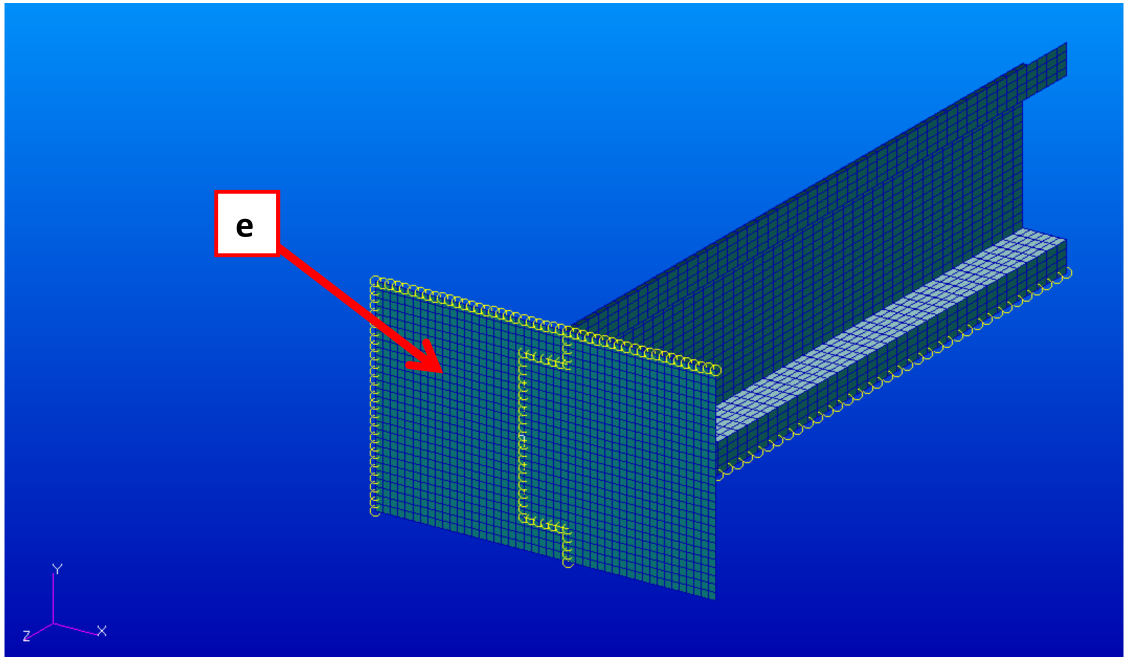


Create mesh:

a. Click on the **Meshing** tab
b. Meshing: **Create/Mesh/Surface**
c. Elem Shape: **Quad**
   Mesher: **IsoMesh**
   Topology: **Quad4**
d. Click on **Surface List** panel
e. Select the additional surface: **Surface no. 6**
f. Click **Apply**

The figure below shows the meshed additional surface.





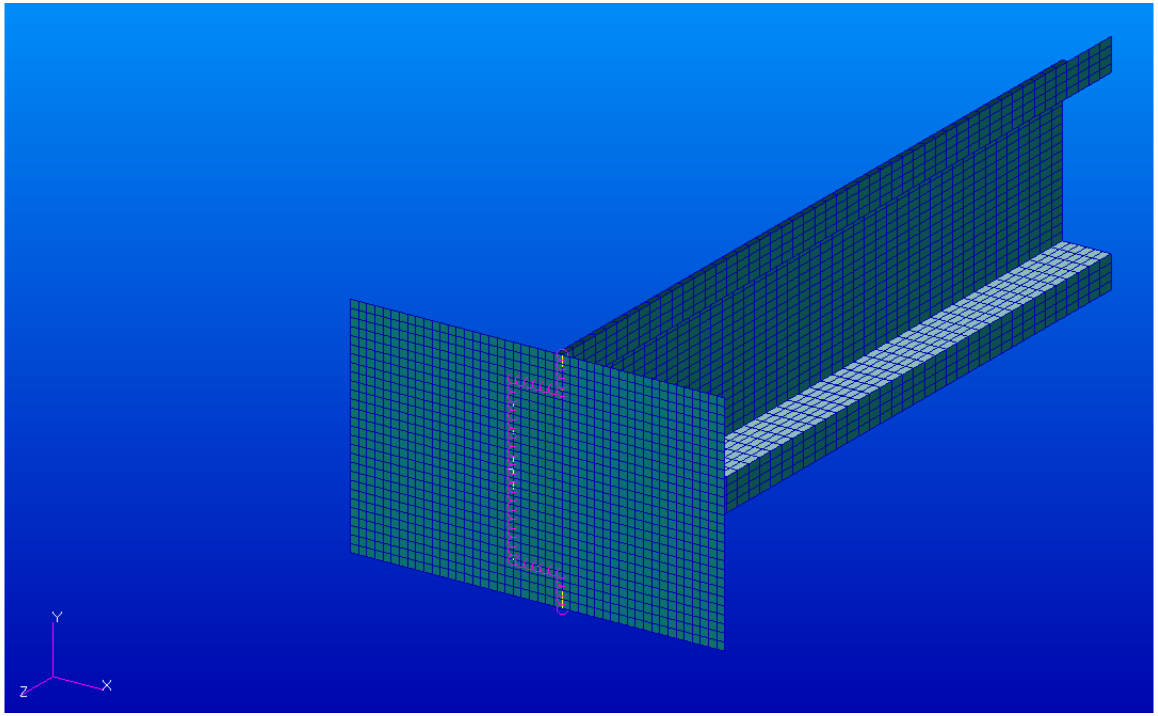
Delete the duplicate nodes:

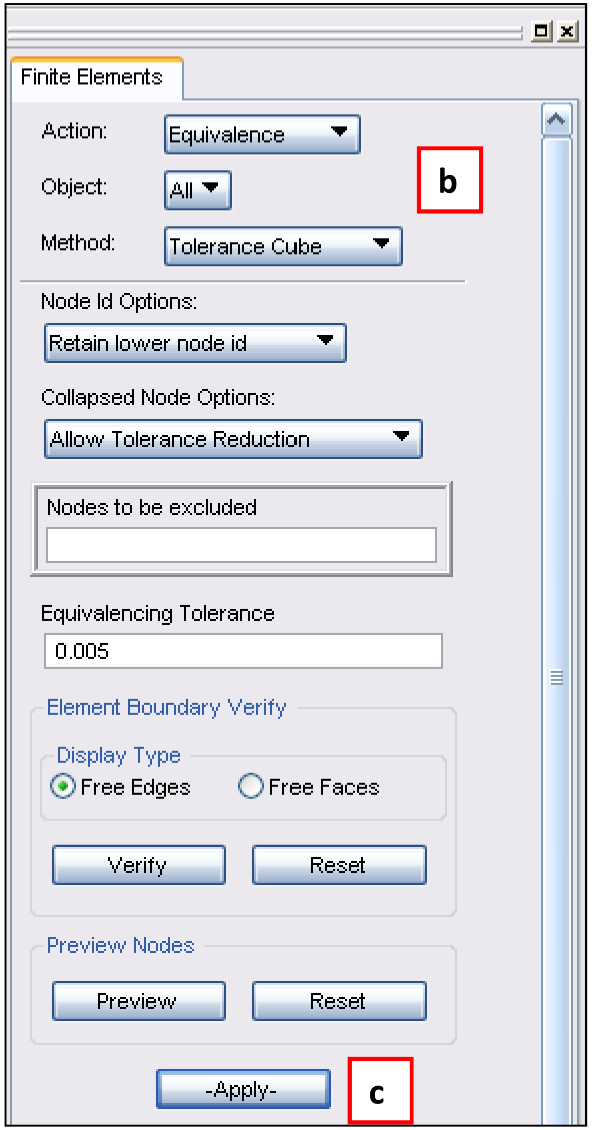
a. Click on **Meshing** tab

b. Equivalence/All/Tolerance Cube

c. Click **Apply**

The figure below shows the FE model of the beam with the additional surface after deletion of the duplicate nodes.



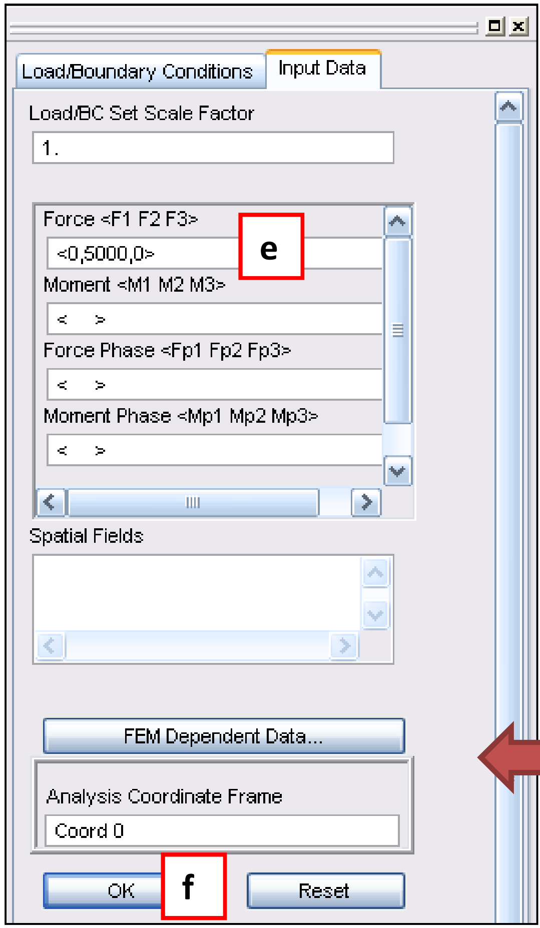


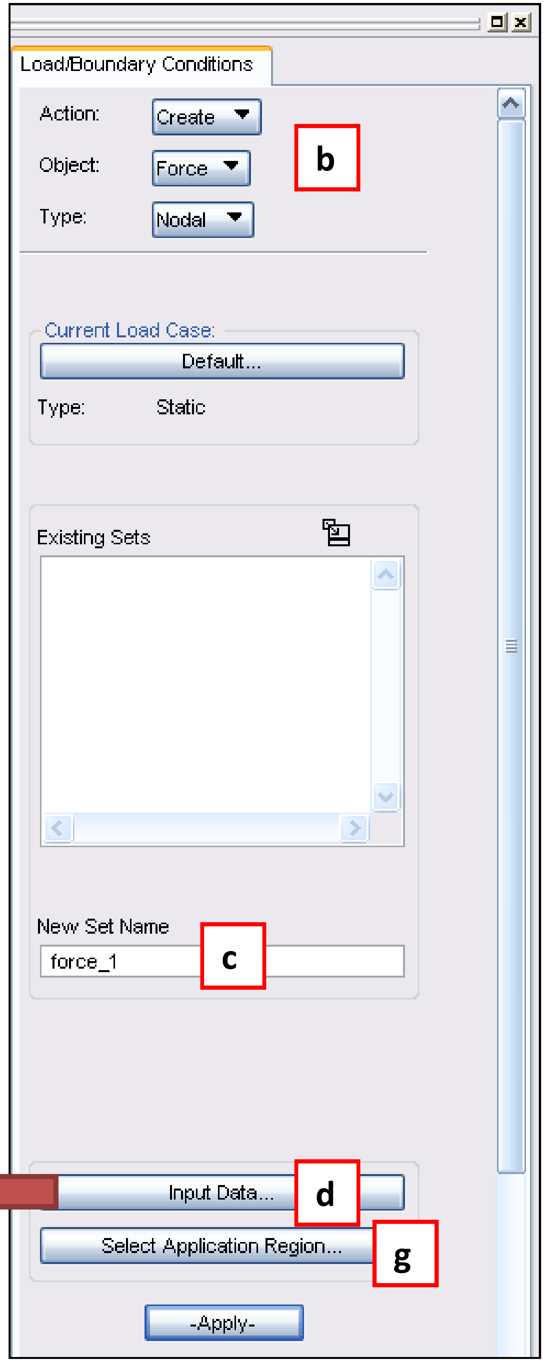
OPEN SECTION THIN-WALLED BEAM

LOAD APPLICATION

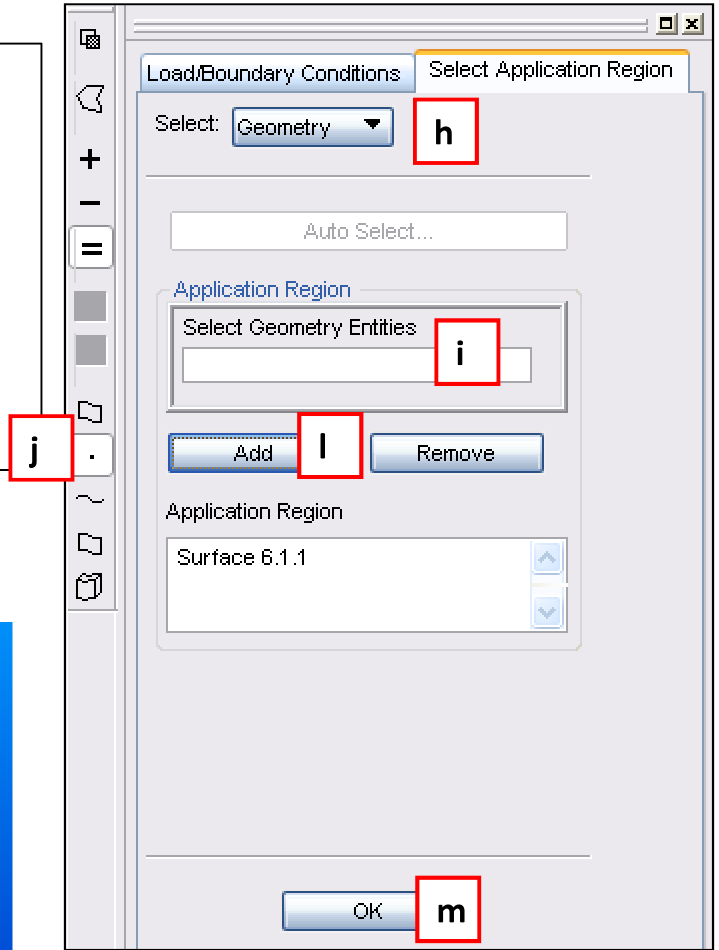
Apply load for **the 1st load case:**

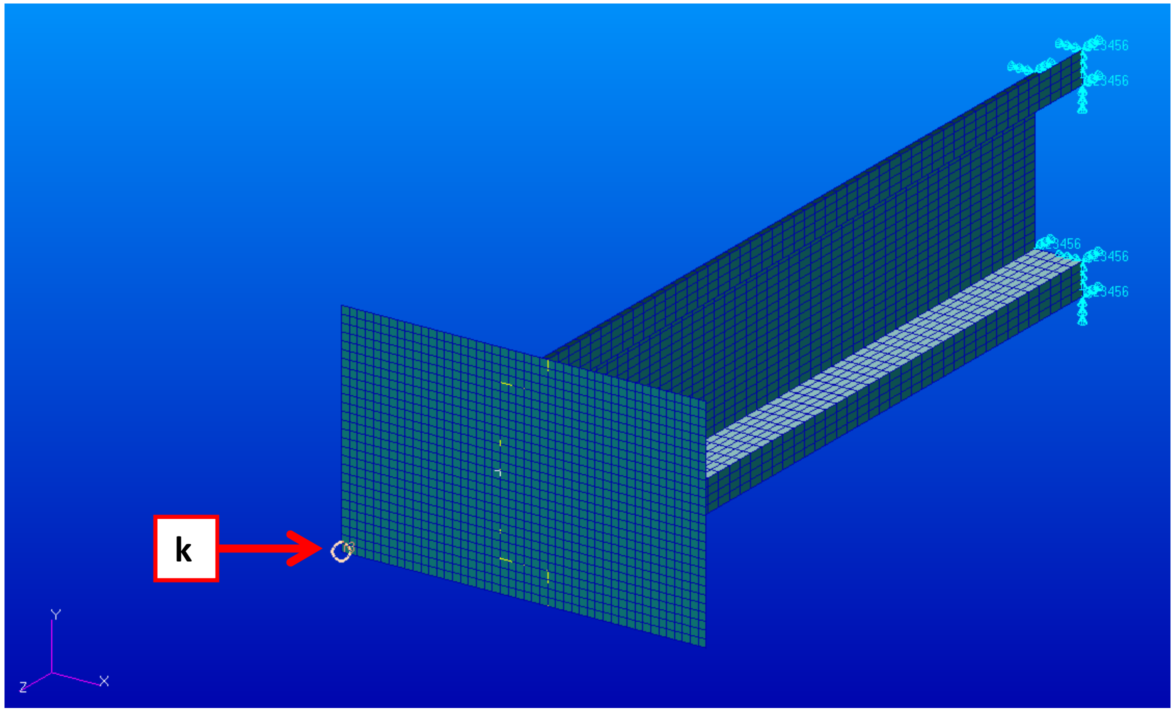
a. Click on the **Loads/BCs** tab
b. Loads/BCs: **Create/Force/Nodal**
c. Enter **force_1** as the New Set Name
d. Click **Input Data…**
e. Enter **<0,5000,0>** for the Force
f. Click **OK**
g. Click **Select Application Region…**





h. Select **Geomtery**
i. Click on the **Select Geometry Entities** panel
j. Select the **Point or Vertex** icon
k. Select the **bottom left** corner of the additional surface
l. Click **Add**
m. Click **OK**
n. Click **Apply** in *Load/Boundary Conditions* right menu





Apply load for **the 2nd load case:**

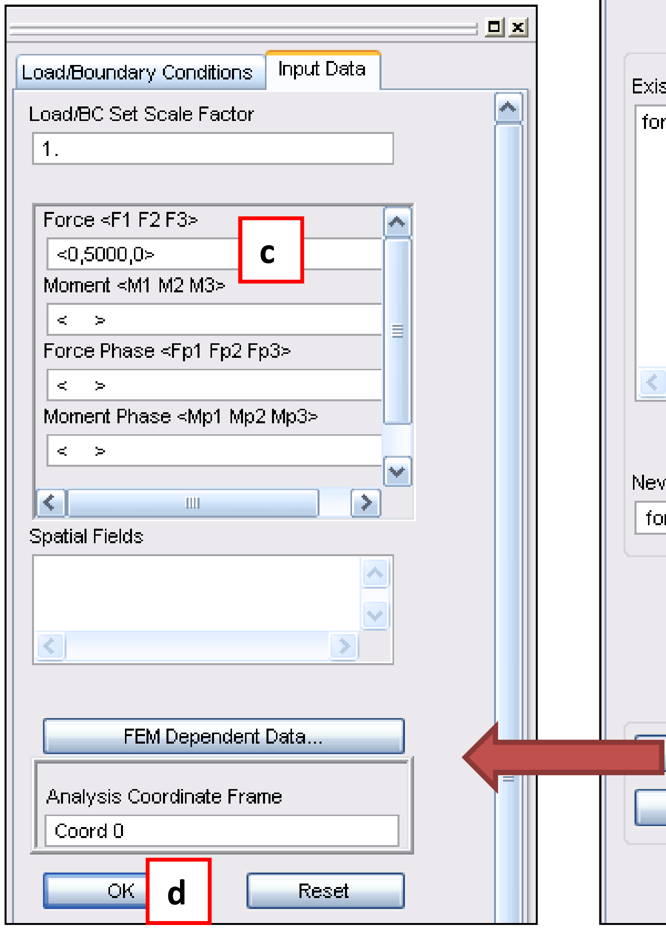
a. Enter **force_2** as the New Set Name

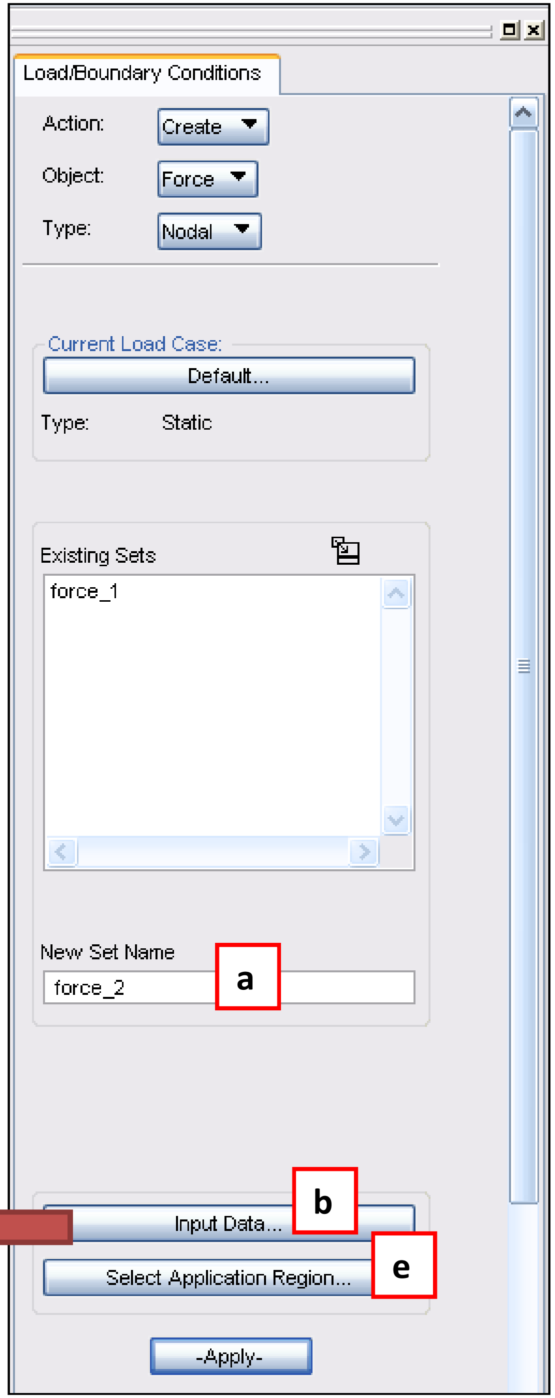
b. Click **Input Data…**

c. Enter **<0,5000,0>** for the Force

d. Click **OK**

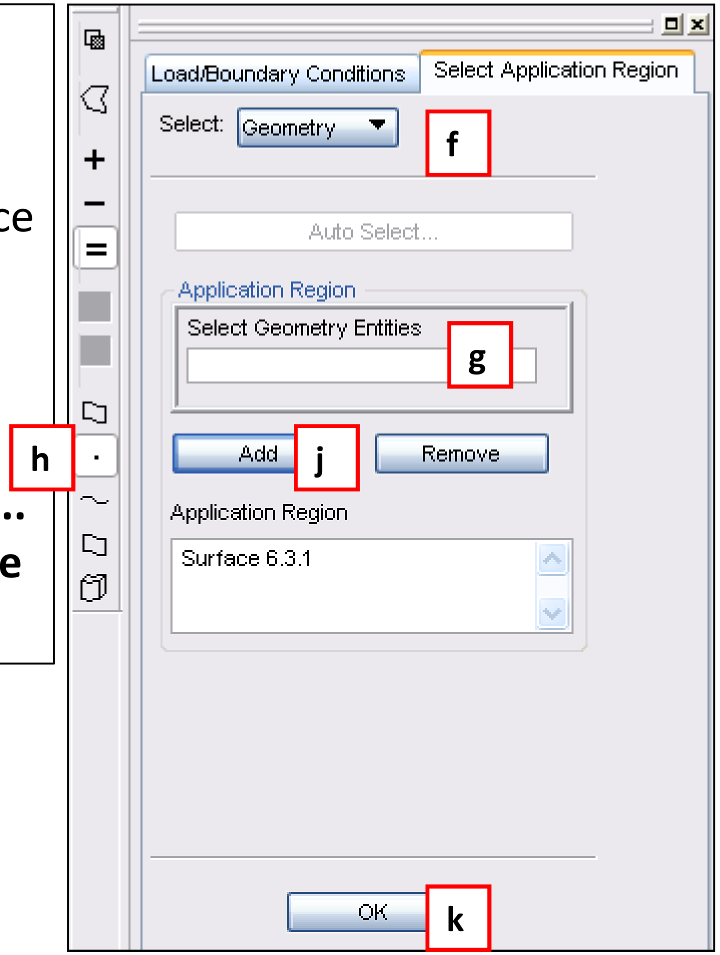
e. Click **Select Application Region…**

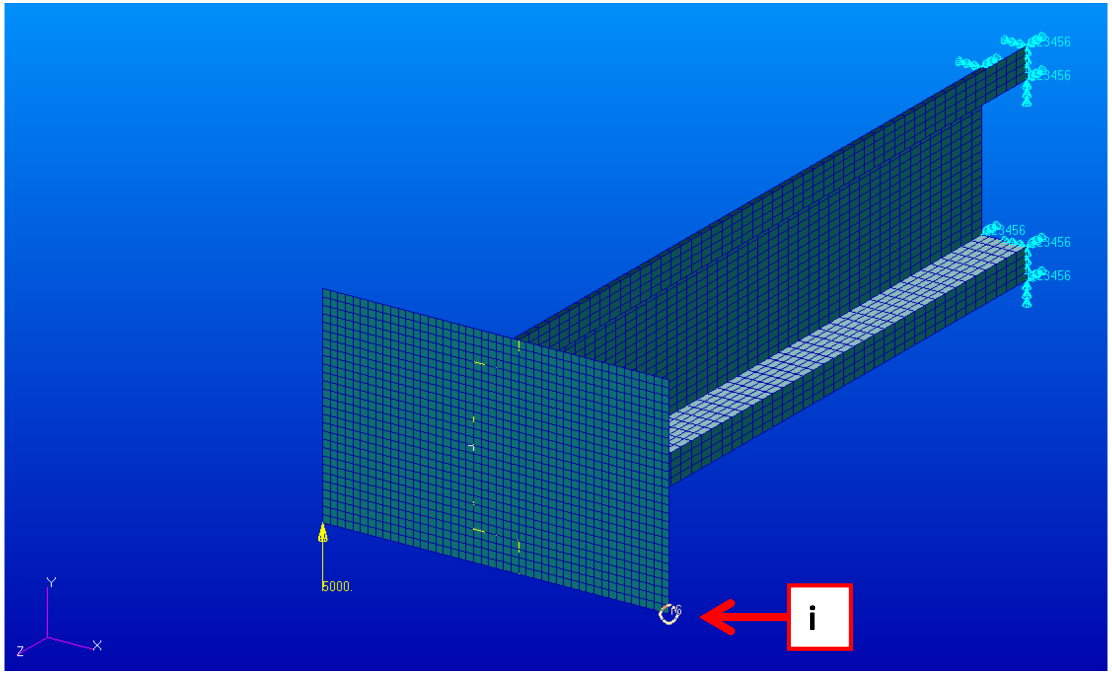




f. Select **Geomtery**
g. Click on the **Select Geometry Entities** panel
h. Select the **Point or Vertex** icon
i. Select the **bottom right** corner of the additional surface
j. Click **Add**
k. Click **OK**
l. Click **Apply** in *Load/Boundary Conditions* right menu

m. Top menu: **Display** and click on **Load/BC/Elem. Props...**
n. In *Loads/BCs* change color from yellow to red for **Force**
o. Click **Cancel**



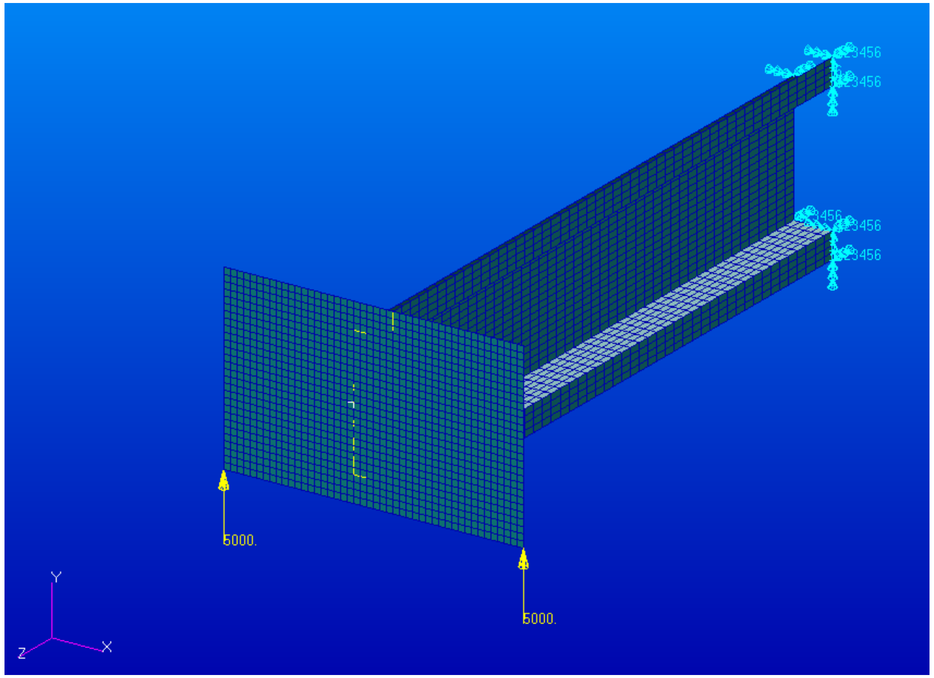


The following figure shows the FE model of the beam with applied loads and boundary conditions.

<u>Note:</u> geometry of the model is displayed because load and fixing have been applied to the geometrical entities.

Save the figure of the FE model of the beam with applied loads and boundary conditions (<u>remember about **white** background</u>):

a. **File/Images…**
b. Choose Image Format: **JPEG**
c. Click **Apply**

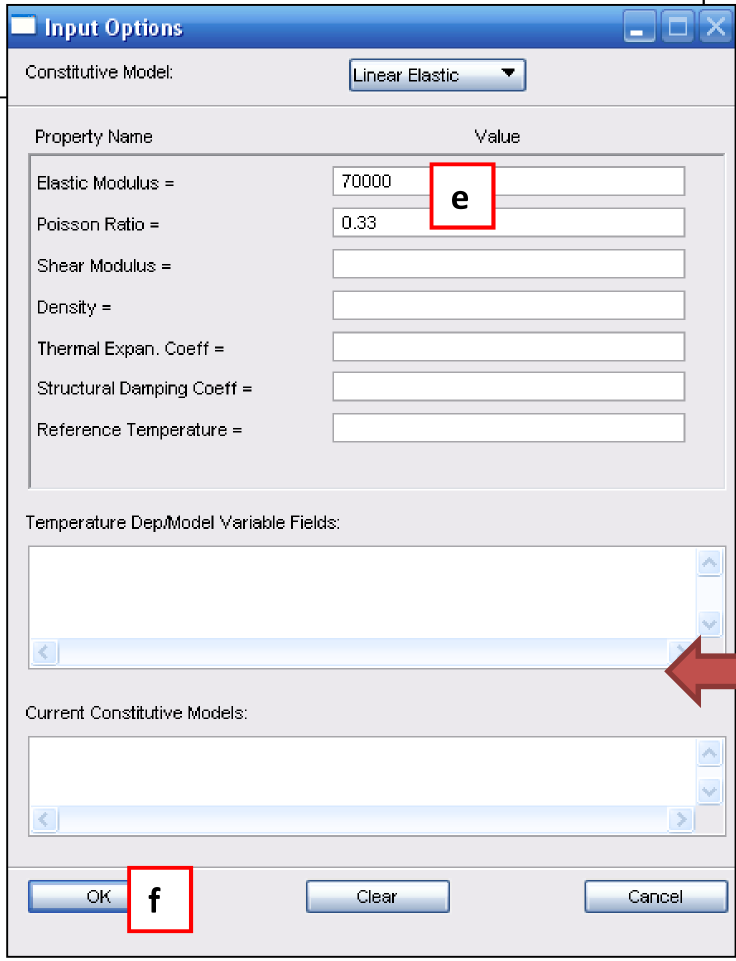


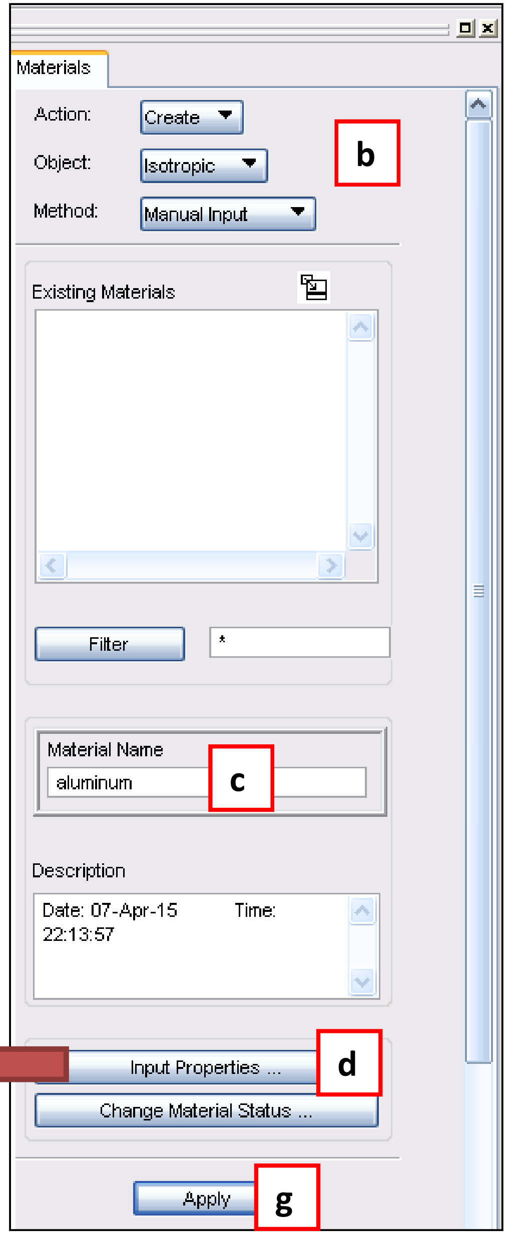
OPEN SECTION THIN-WALLED BEAM

MATERIAL PROPERTIES DEFINITION

Define material properties
(isotropic; linear elastic; aluminum, E = 70000 MPa; ν = 0.33):

a. Click on the **Properties** tab
b. Create/Isotropic/Manual Input
c. Enter **aluminum** as the Material Name
d. Click **Input Properties…**
e. Enter **70000** as Elastic Modulus and **0.33** as Poisson Ratio
f. Click **OK**
g. Click **Apply**



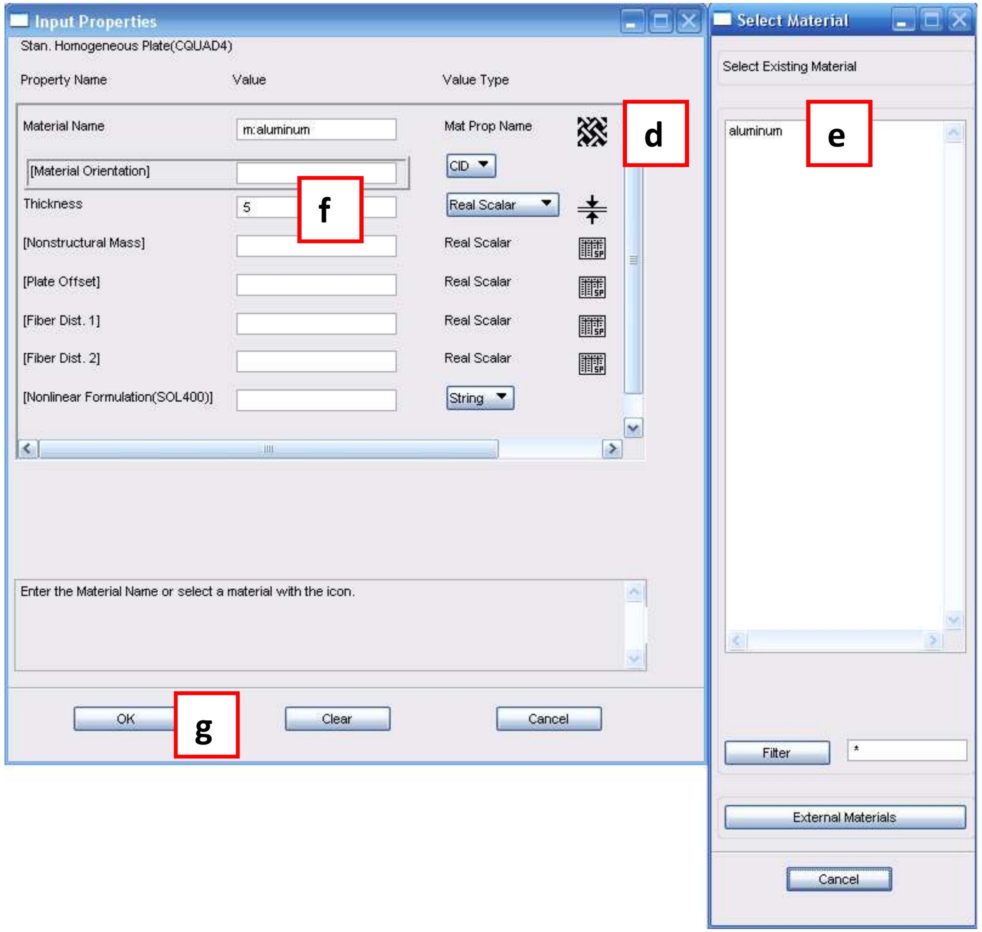


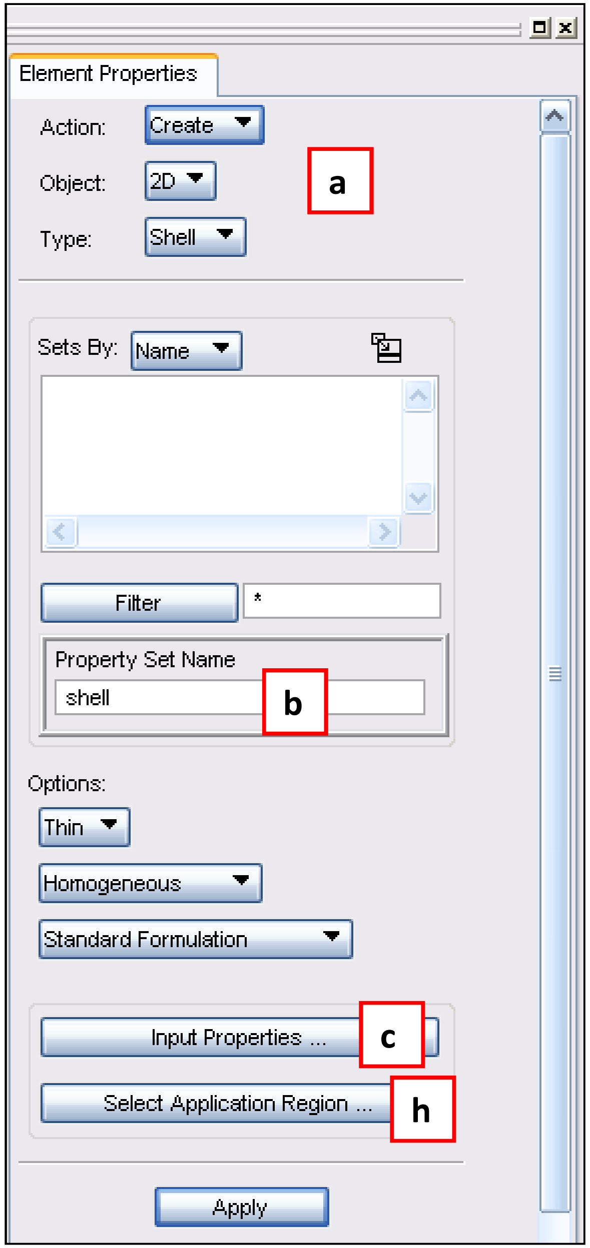
OPEN SECTION THIN-WALLED BEAM

MATERIAL PROPERTIES ASSIGNMENT

Assign the properties to the model:

a. Properties: **Create/2D/Shell**
b. Enter **shell** as the Property Set Name
c. Click **Input Properties…**
d. Click on the **Mat Prop Name** icon
e. Select **aluminum**
f. Enter **5** as the Thickness
g. Click **OK**
h. Click **Select Application Region…**

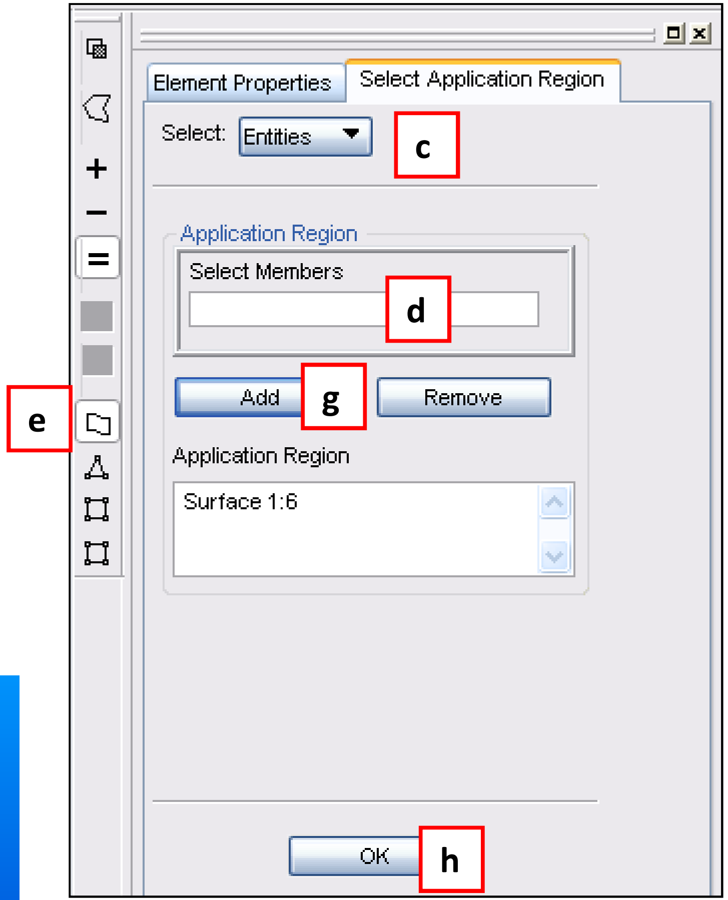


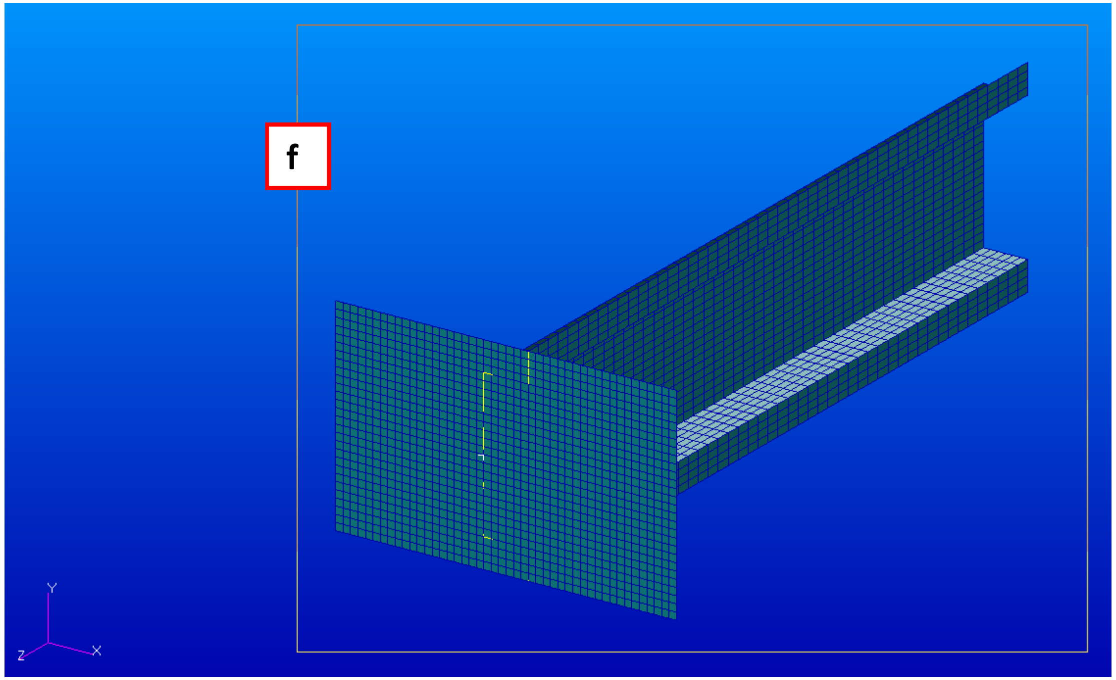


In **Home tab**:

a. Click on the **Iso 1 view** icon
b. Click on the **Fit view** icon

c. Select **Entities**
d. Click on the **Select Members** panel
e. Select **Surface or face** icon
f. Select all surfaces
g. Click **Add**
h. Click **OK**

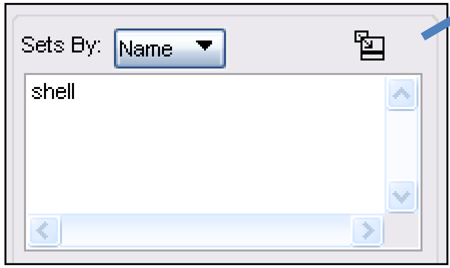


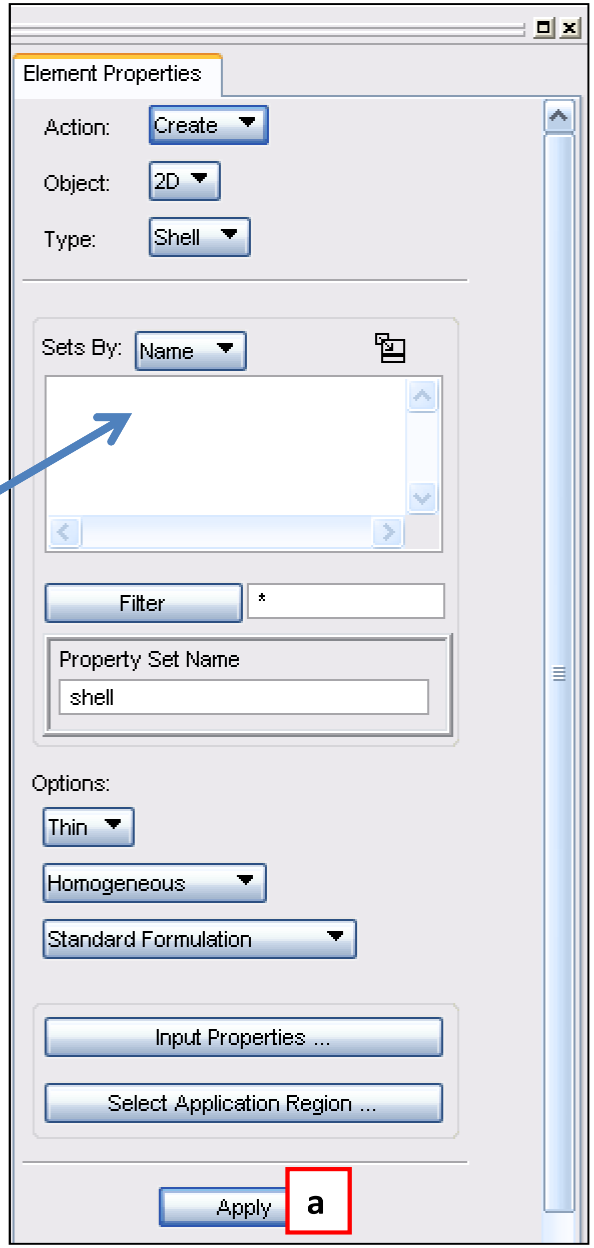


a. Click **Apply**

In this way the material properties have been assigned to the beam’s model.

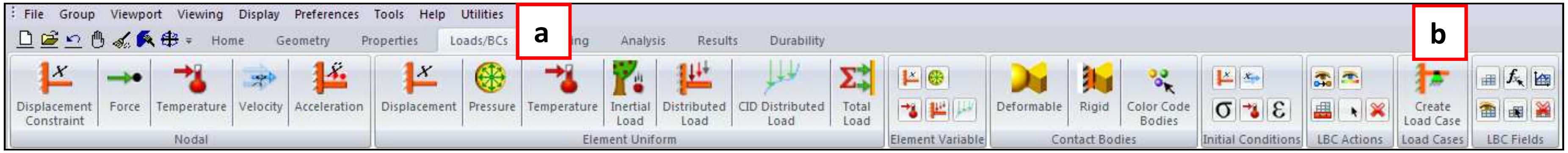
The property set name „*shell*” has been created.
The following window will appear after clicking Appy.



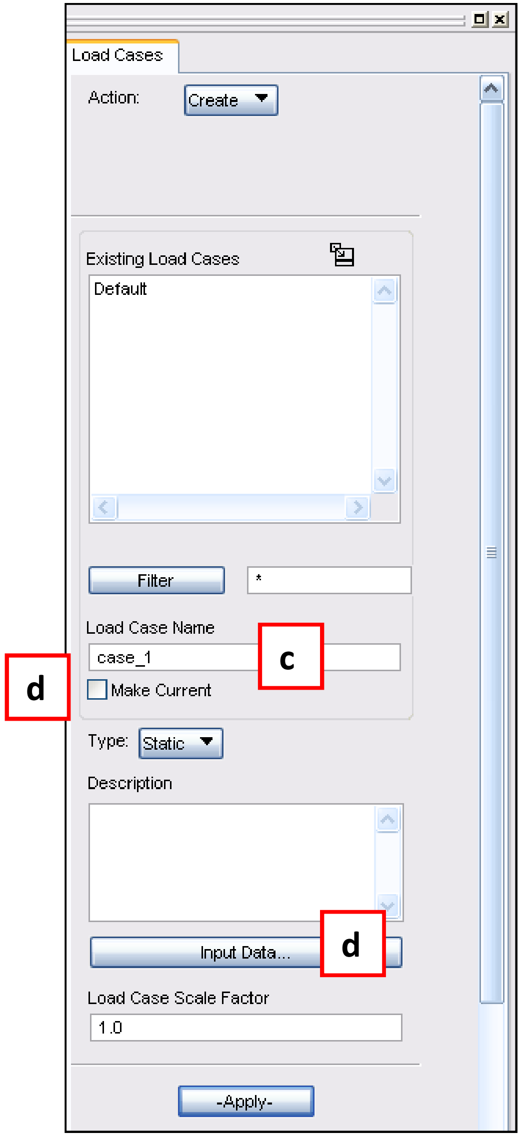


OPEN SECTION THIN-WALLED BEAM

LOAD SUBCASES CREATION



a. Click on the **Loads/BCs** tab
b. Click on the **Create Load Case** in Load Cases
c. Enter **case_1** as the Load Case Name
d. Uncheck **Make Current**
e. Click **Input Data…**



f. Click **once** on the **Displ_constraint**

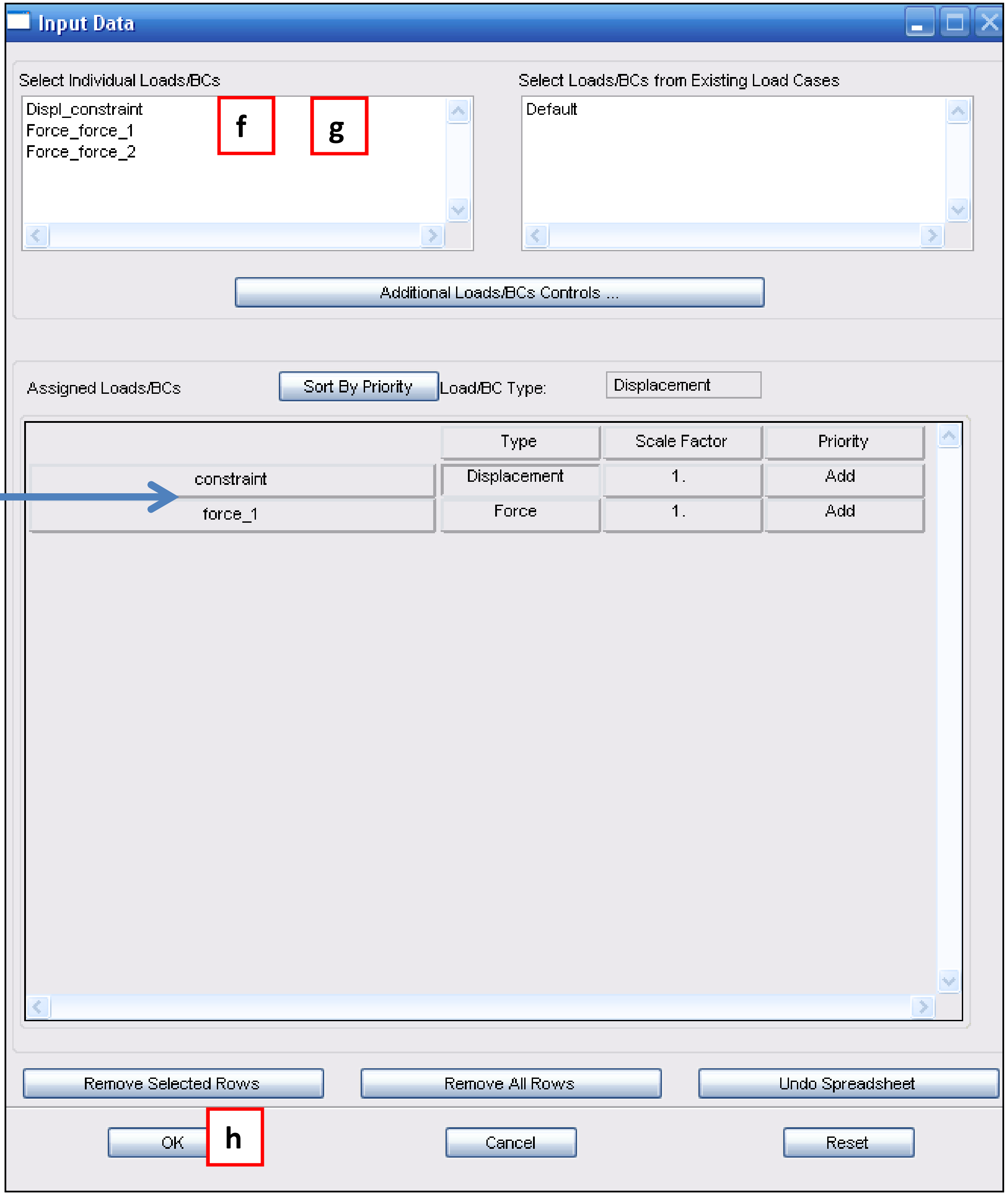
g. Click **once** on the **Force_force_1**

You will see the assigned loads and boundary conditions to the load subcase no. 1.

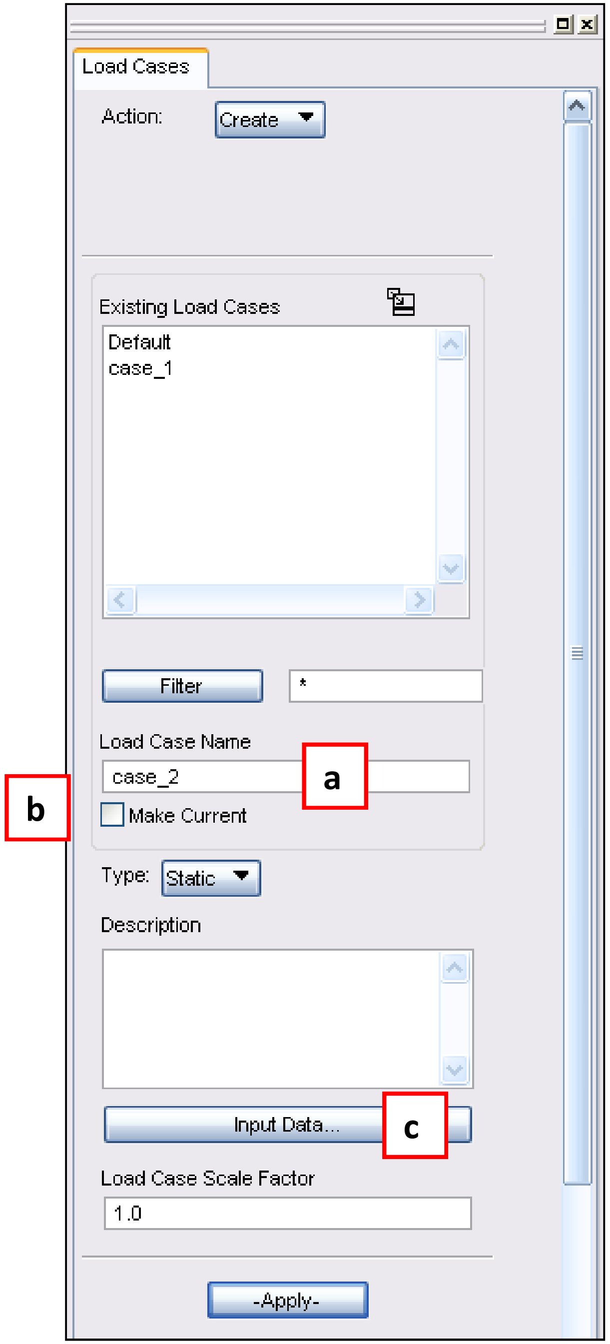
<u>**Note:**</u> You can delete the row *in Assigned Loads/BCs* by selecting the appropriate row (clicking on it) and clicking on **Remove Selected Rows**.

h. Click **OK**

i. Click **Apply**



a. Enter **case_2** as the Load Case Name
b. Uncheck **Make Current**
c. Click **Input Data…**



d. Select **force_1** row

e. Click **Remove Selected Rows**

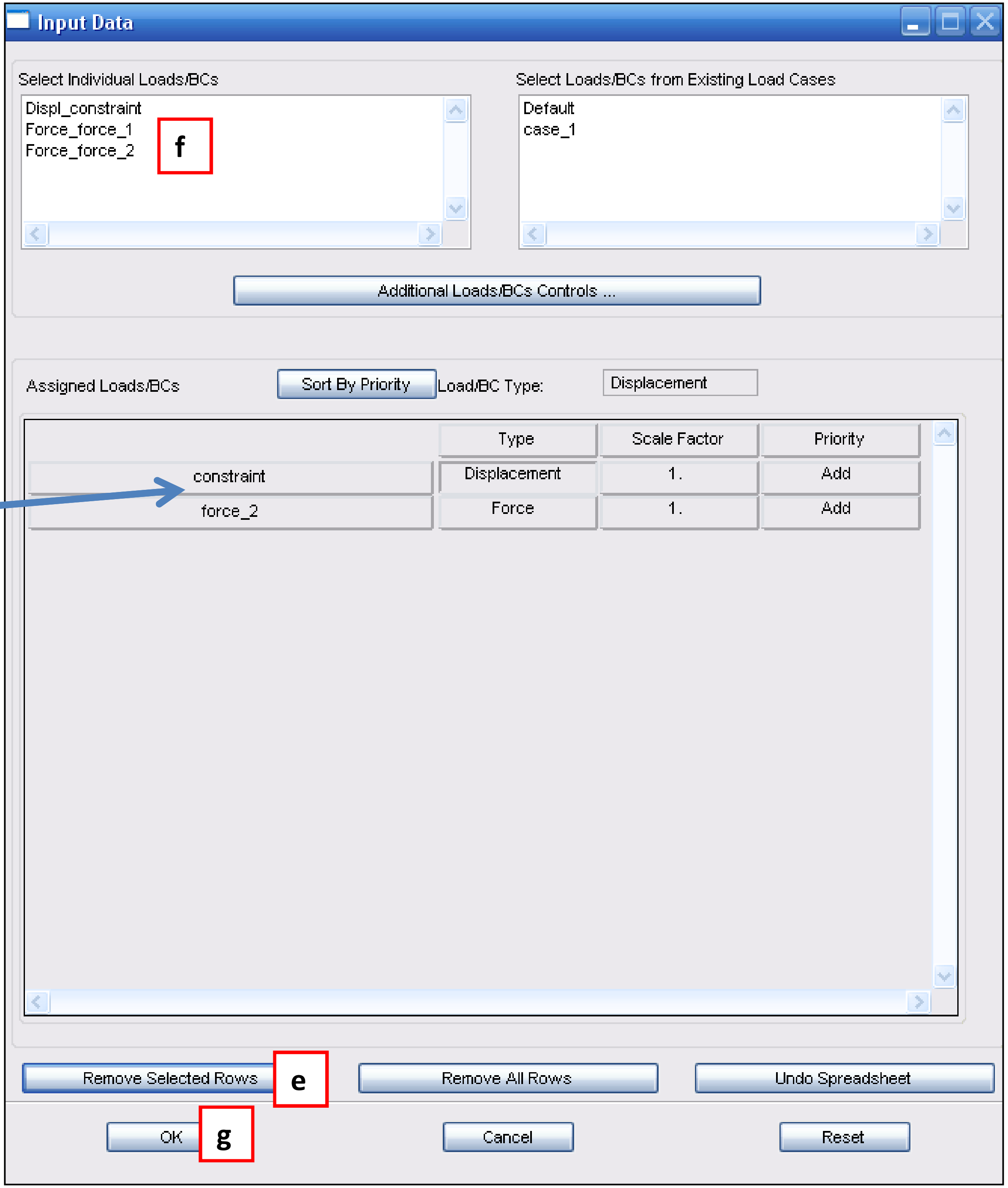
f. Click **once** on the **Force_force_2**

You will see the assigned loads and boundary conditions to the load subcase no. 2.

Note: You can delete the row *in Assigned Loads/BCs* by selecting the appropriate row (clicking on it) and clicking on **Remove Selected Rows**.

g. Click **OK**

h. Click **Apply**

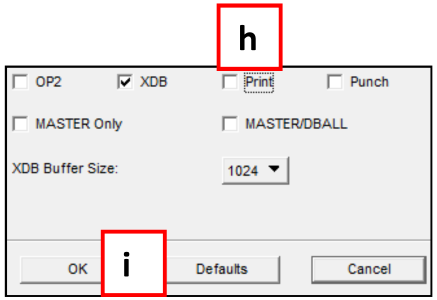


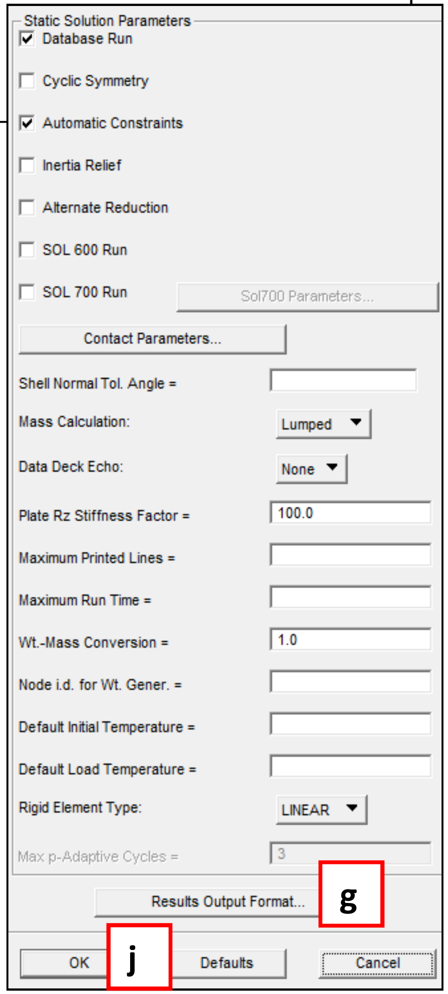
OPEN SECTION THIN-WALLED BEAM

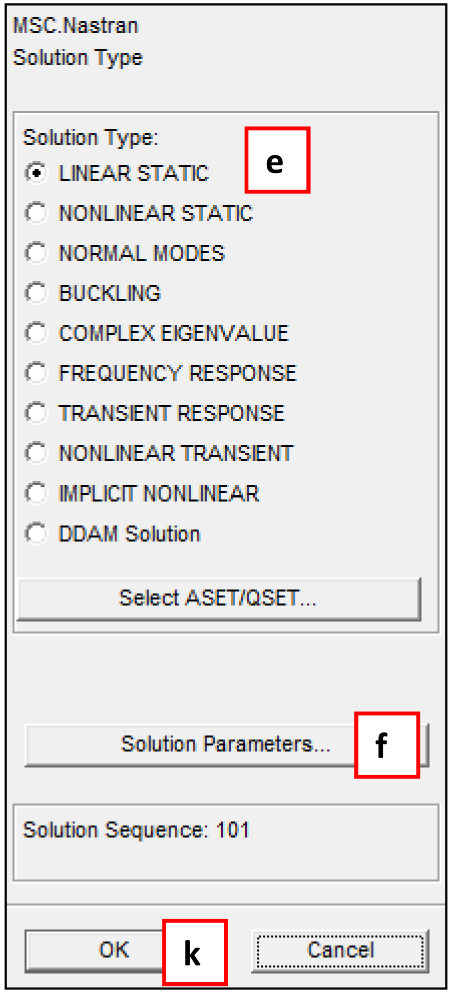
ANALYSIS WITH TWO SUBCASES

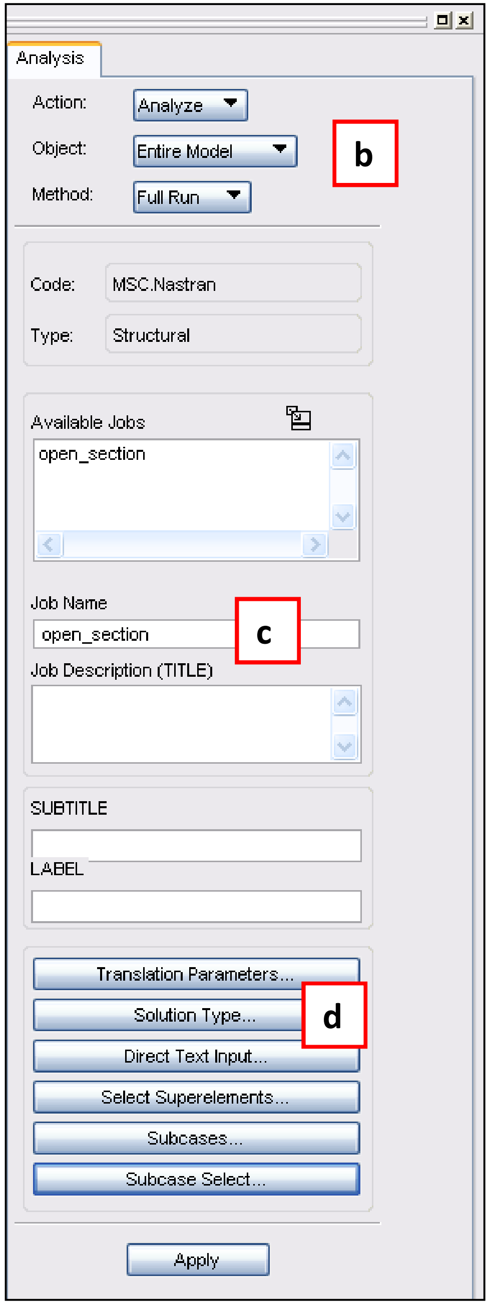
Run a linear static analysis:

a. Click on the **Analysis** tab
b. Choose **Analyze/Entire Model/Full Run** from right menu
c. Enter **open_section** as the Job Name
d. Click **Solution Type…**
e. Select **LINEAR STATIC** as the Solution Type
f. Click **Solution Parameters…**
g. Click **Results Output Format…**
h. Uncheck **Print**
i. Click **OK**
j. Click **OK**
k. Click **OK**









Run a linear static analysis:

m. Click **Subcase Select…**

n. Click **Unselect All**

o. Select **case_1**

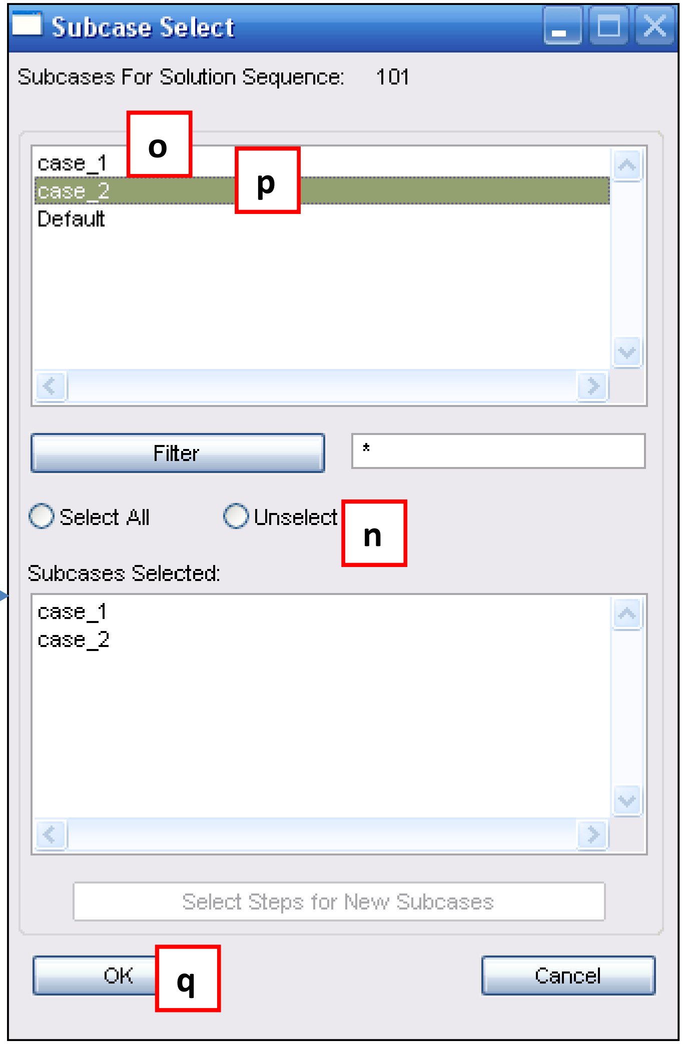
p. Select **case_2**

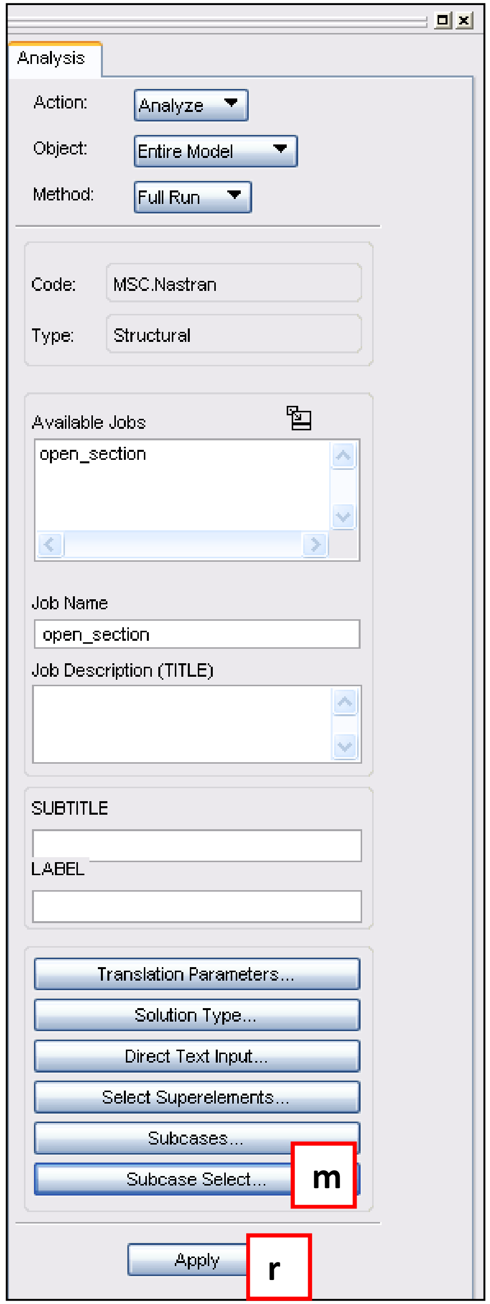
Then, "case_1" and "case_2" will appear under "*Subcases Selected:*".

It means that two load cases will be solved during one run.

q. Click **OK**

r. Click **Apply**



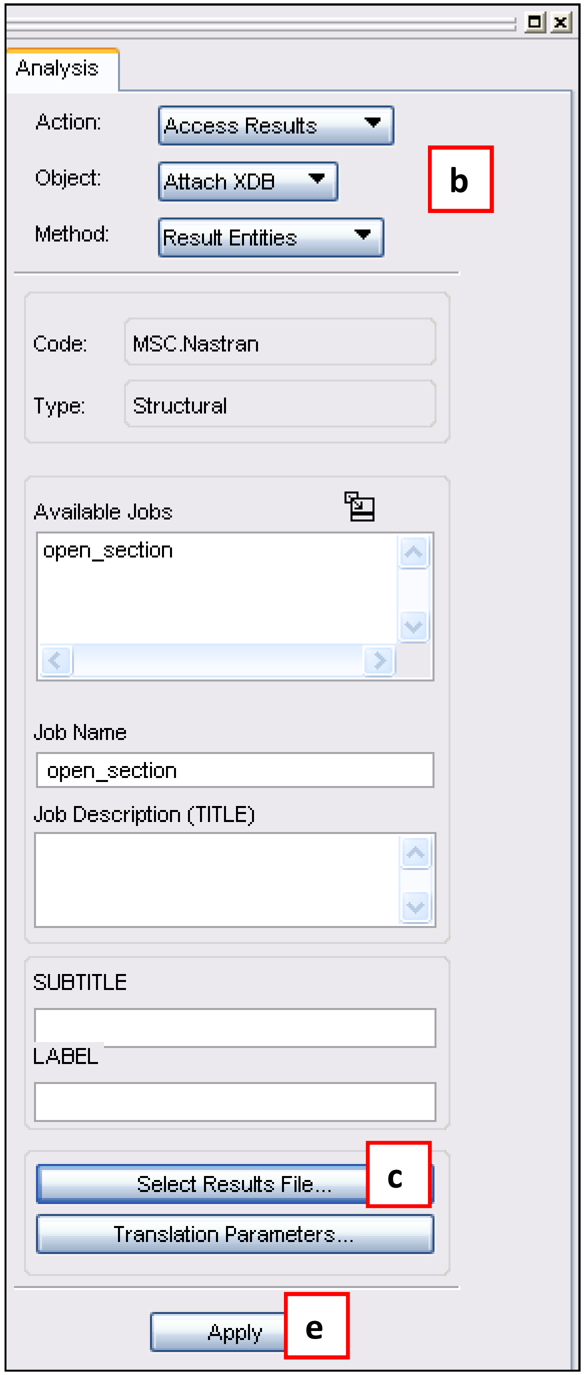


OPEN SECTION THIN-WALLED BEAM

ATTACHING THE RESULTS

Attach the results file, when the analysis job is completed:

a. Click on **Analysis** tab
b. Access Results/Attach XDB/Result Entities
c. Click **Select Results File…**
d. Select **open_section.xdb** file and click **OK**
e. Click **Apply**



OPEN SECTION THIN-WALLED BEAM

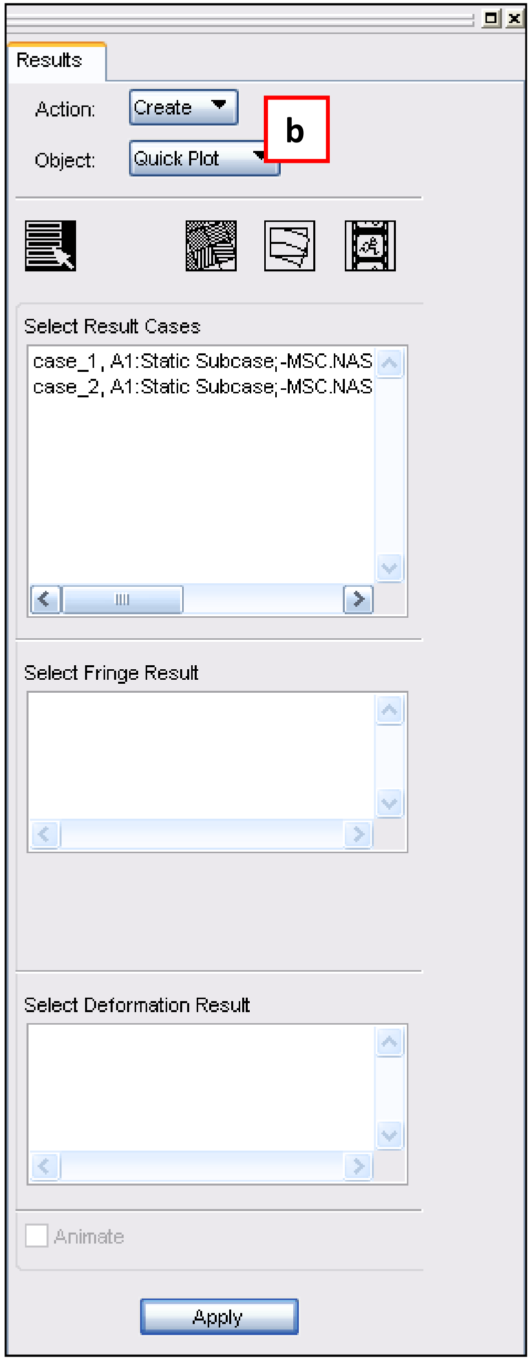
POSTPROCESSING OF THE RESULTS

Postprocess the results:

a. Click on the **Results** tab

b. Results: **Create/Quick Plot**

It can be noticed that there are 2 cases avialable for postprocessing of the results.



The **required** plots with the results for **BOTH** cases (**case 1** and **case 2**):

1) Fringe Result: **Displacements, Translational**
   Quantity: **Magnitude**
   Deformation Result: Displacements, Translational

2) Fringe Result: **Displacements, Translational**
   Quantity: **Y**
   Deformation Result: Displacements, Translational

3) Fringe Result: **Stress Tensor**
   Position… **AT Z1 and Z2** with Maximum option
   Quantity: **von Mises**
   Deformation Result: Displacements, Translational

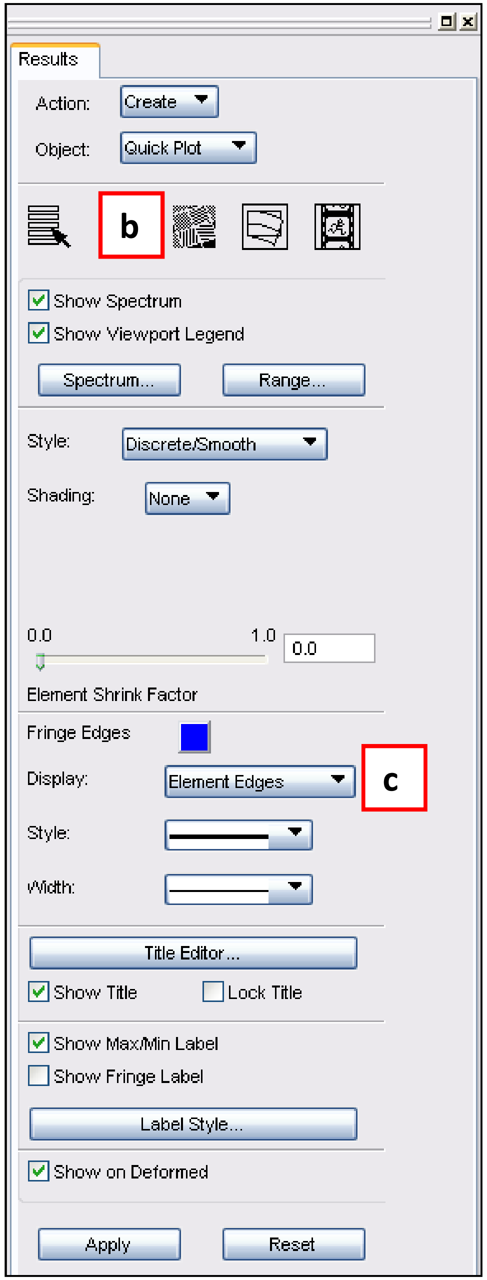
Total number of plots with the results for **BOTH** cases = 2 (cases) x 3 (plots) = **6**

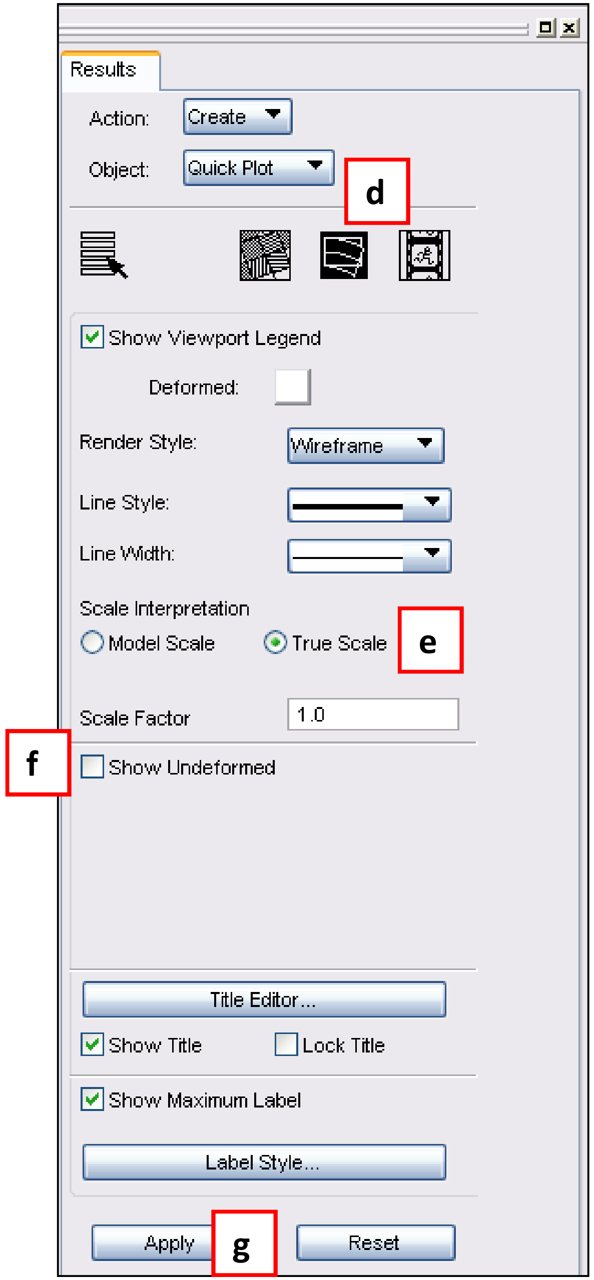
For each plot set the following options:

a. Click on **Reset Graphics** icon → in **Home** tab

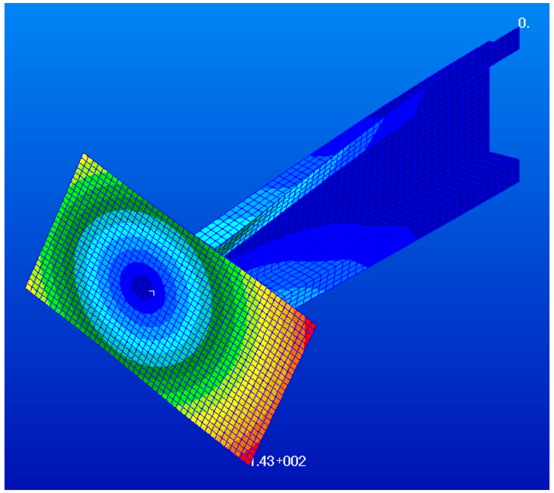
b. Click on the **Fringe Attributes** icon
c. Select **Element Edges** as Display

d. Click on the **Deform Attributes** icon
e. Select **True Scale**
f. Uncheck **Show Undeformed**
g. Click **Apply**

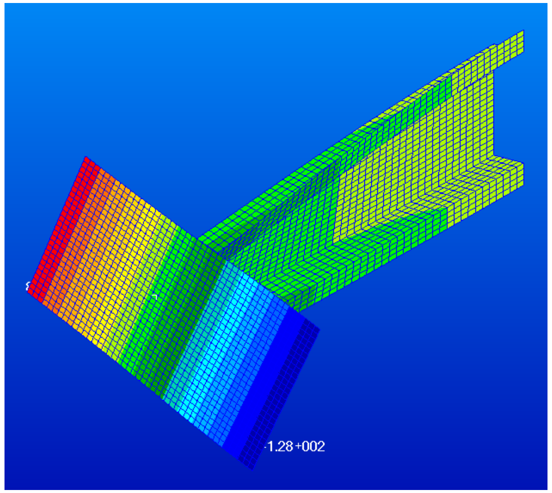




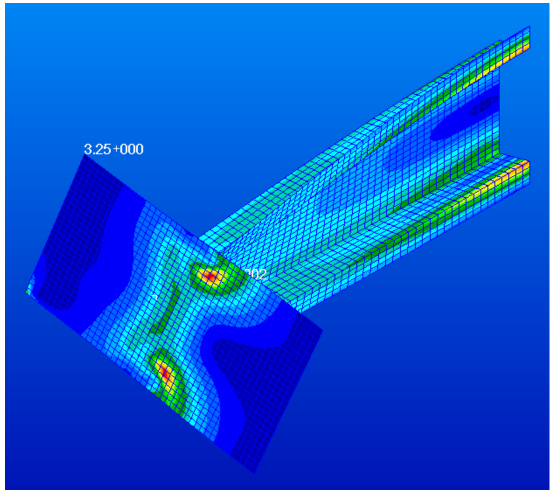
Verify the results with this reference



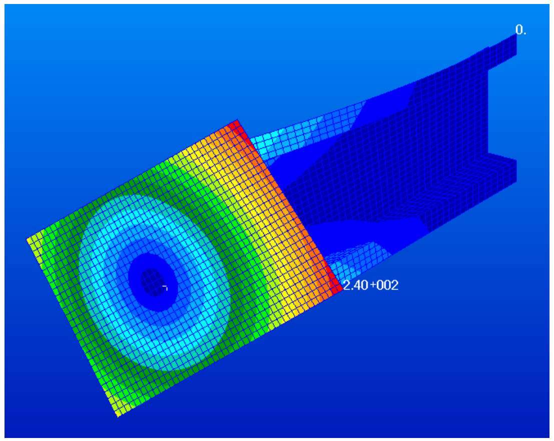
1st case, displacement magnitude



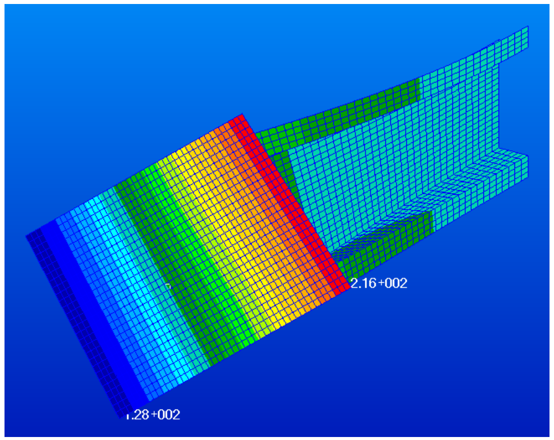
1st case, displacement Y



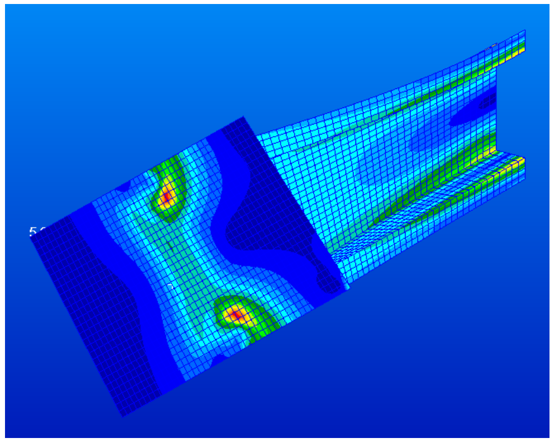
1st case, von Mises, Z1 and Z2



2nd case, displacement magnitude



2nd case, displacement Y



2nd case, von Mises, Z1 and Z2

OPEN SECTION THIN-WALLED BEAM

CALCULATION OF THE SHEAR CENTER „*X*" COORDINATE based on the data obtained from the analysis

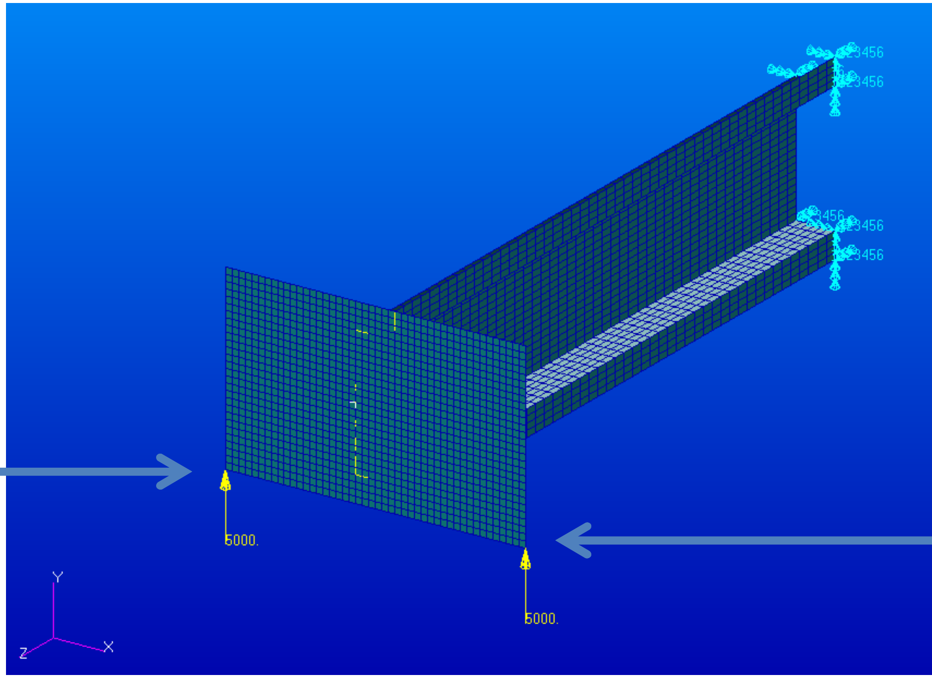
$$\boxed{\mathbf{X_{s.c.} = ?}}$$

Check the value of the displacement in the vertical direction Y of the two nodes located on the bottom edge of the additional surface:

Bottom **left** node

$f_{y\ \text{node 1 case 1}}$ = ?

$f_{y\ \text{node 1 case 2}}$ = ?

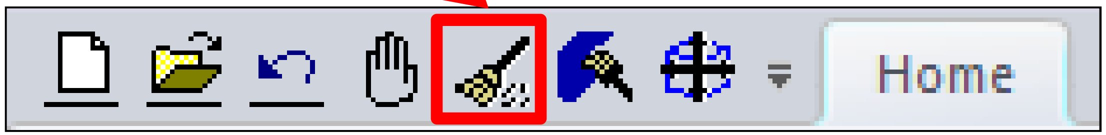


Bottom **right** node

$f_{y\ \text{node 2 case 1}}$ = ?

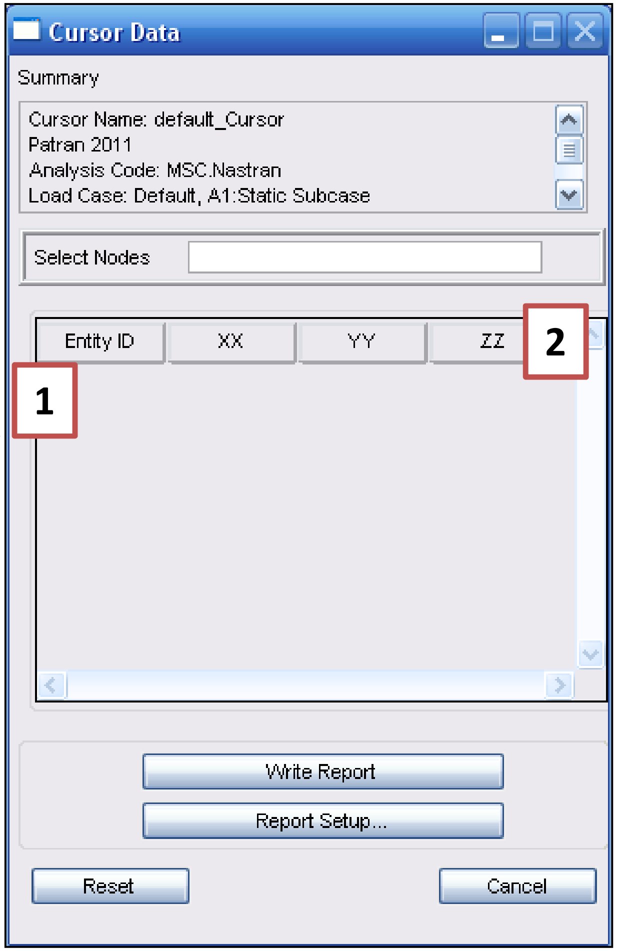
$f_{y\ \text{node 2 case 2}}$ = ?

Reset Graphics



a. **Results** tab: **Create/Cursor/Vector**
b. Select Result Cases: **case_1, A1:Static Subcase**
c. Select Cursor Vector: **Displacements, Translational**
d. Position...((NON-LAYERED))
e. Target Entity: **Nodes**
f. Click **Apply**

After clicking Apply the following window will appear.



After selection of the desired node you will see:

1 **Node ID**

2 **its 3 components of displacement (XX, YY, ZZ)**

g. Select the **bottom left** corner of the additional surface
h. Read and write down the **YY value** for the **bottom left** corner of the additional surface
i. Select the **bottom right** corner of the additional surface
j. Read and write down the **YY value** for the **bottom right** corner of the additional surface

k. Repeat a÷j steps for **case_2**

You should write down 4 different values of displacements (2 displacement values per one case).

Based on the **vertical displacements** of chosen two nodes (**YY**) from both analyses (Case 1 and Case 2) the following **table** can be done.

	YY [mm]			
	Case 1	**Case 2**	**delta [mm]**	**X coordinate [mm]**
Node 1	TBD	TBD	TBD	-200
Node 2	TBD	TBD	TBD	260

Tab. 1

These values are read from the program.

TBD – to be determined

"*delta*" is calculated in the following way:

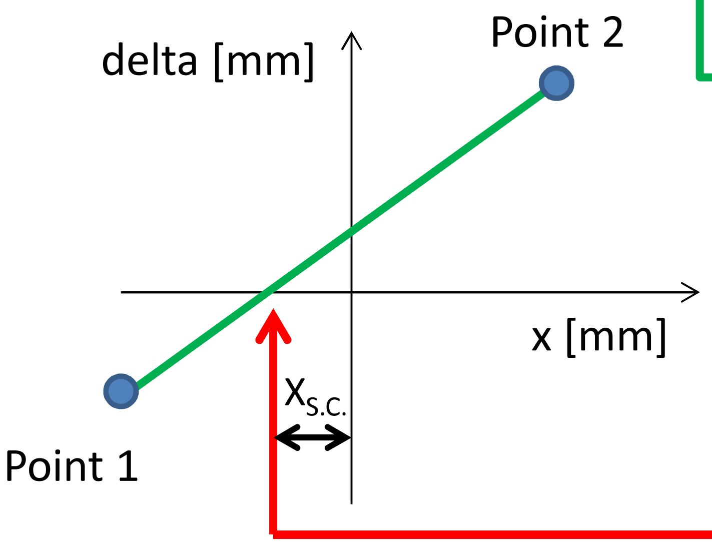
$$delta\ [mm] = YY_{case\ 2}\ [mm] - YY_{case\ 1}\ [mm]$$

Based on the vertical displacements of chosen two nodes (YY) from both analyses (Case 1 and Case 2) the following **graph** can be done.

	YY [mm]			
	Case 1	**Case 2**	**delta [mm]**	**X coordinate [mm]**
Node 1	TBD	TBD	TBD	-200
Node 2	TBD	TBD	TBD	260

Tab. 1

These values are the coordinates of two points: Point 1 and Point 2.



The „X" coordinate of the point of the intersection of the function and the horizontal axis is the searched „X" coordinate of the shear center $X_{S.C.}$

Hint: draw the trend line with option *„Display equation on chart"*

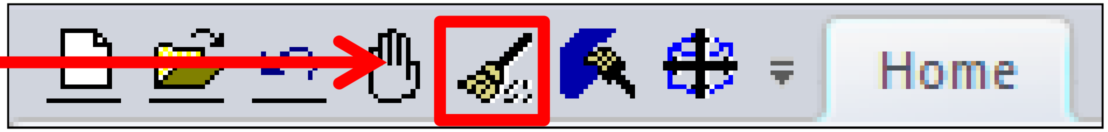
OPEN SECTION THIN-WALLED BEAM

APPLICATION OF LOAD THROUGH THE SHEAR CENTER

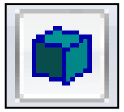
The task is to apply the force acting through the shear center.

The value of this force is equal to 5000 [N].

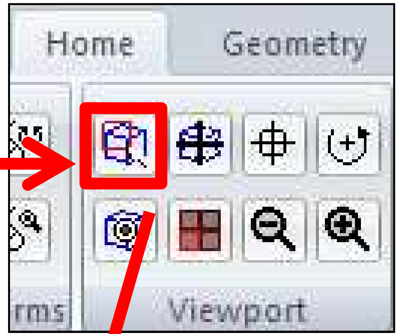
1 **Reset Graphics**



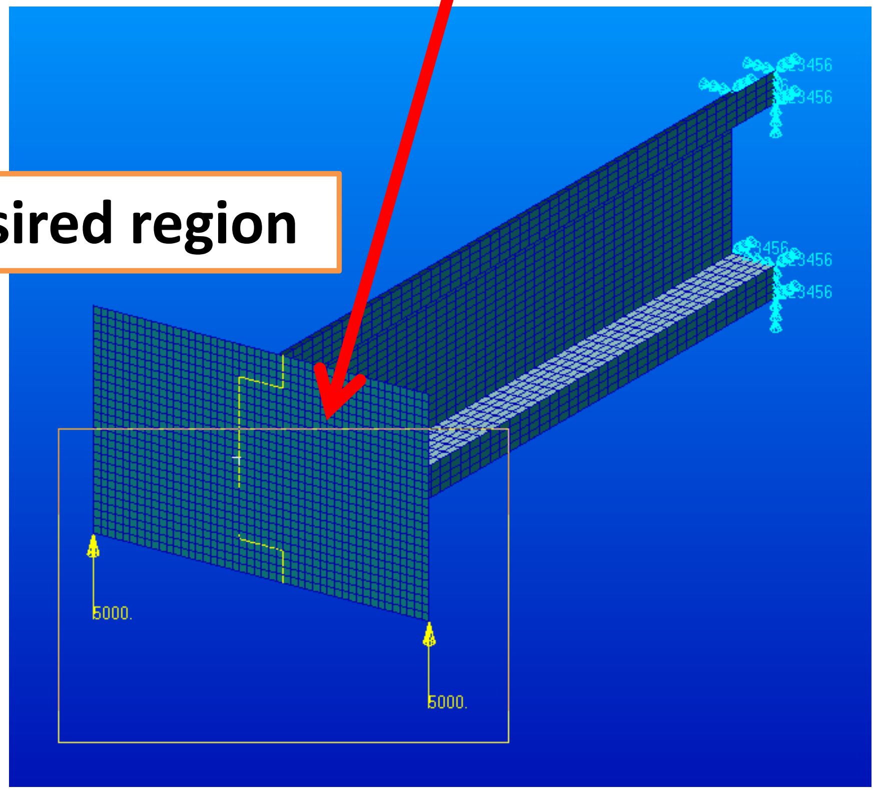
2 **Smooth shaded**



3 **View Corners**

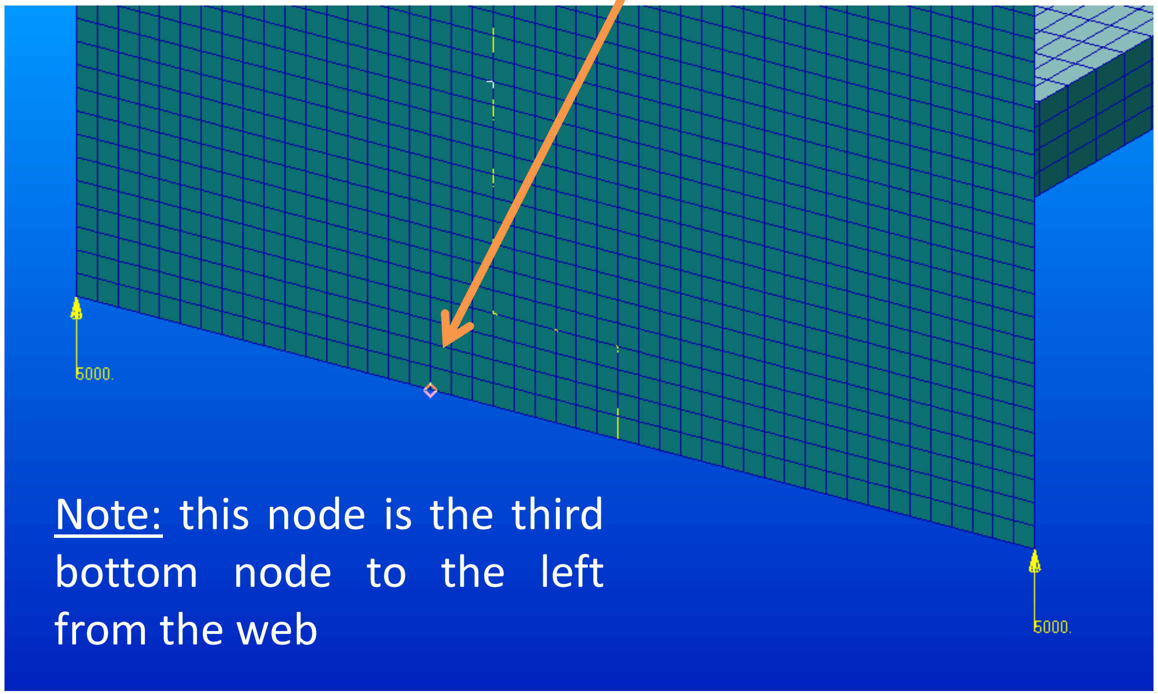


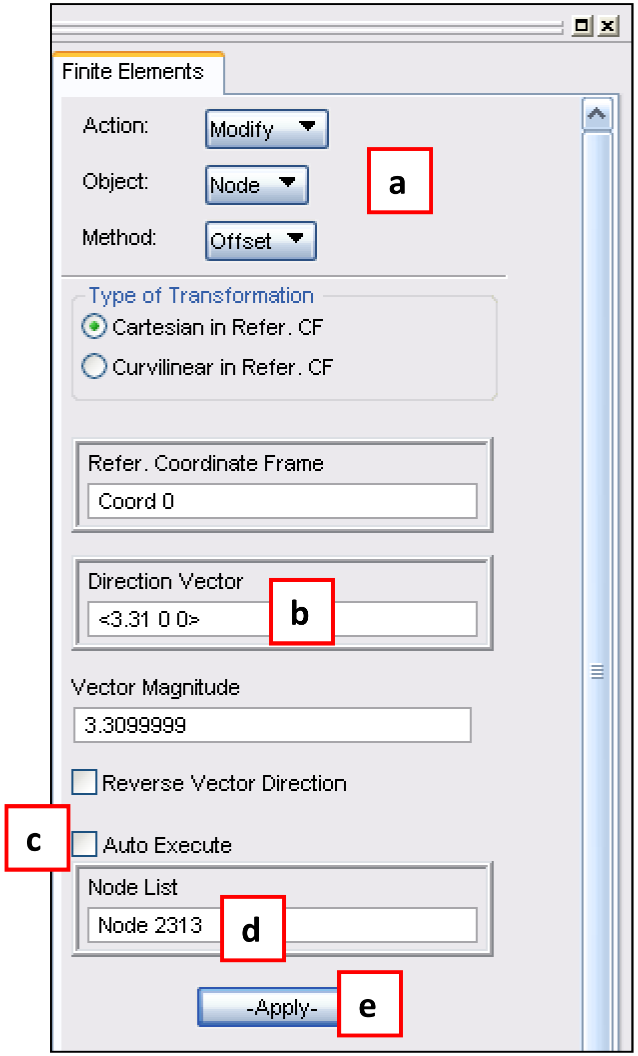
4 **Mark the desired region**



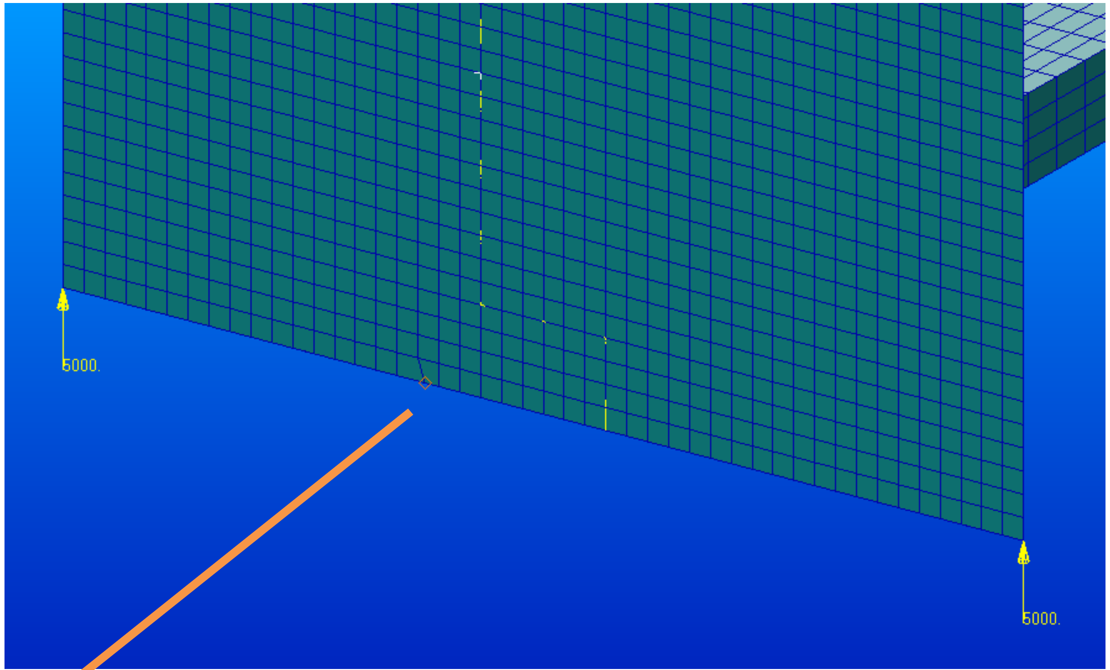
a. **Meshing** tab: **Modify/Node/Offset**
b. Direction Vector: **<3.31 0 0>**
c. Uncheck **Auto Execute**
d. Select **Node list** panel and select the node which has the following coordinates: **(-30, -140, 0)**
e. Click **Apply**

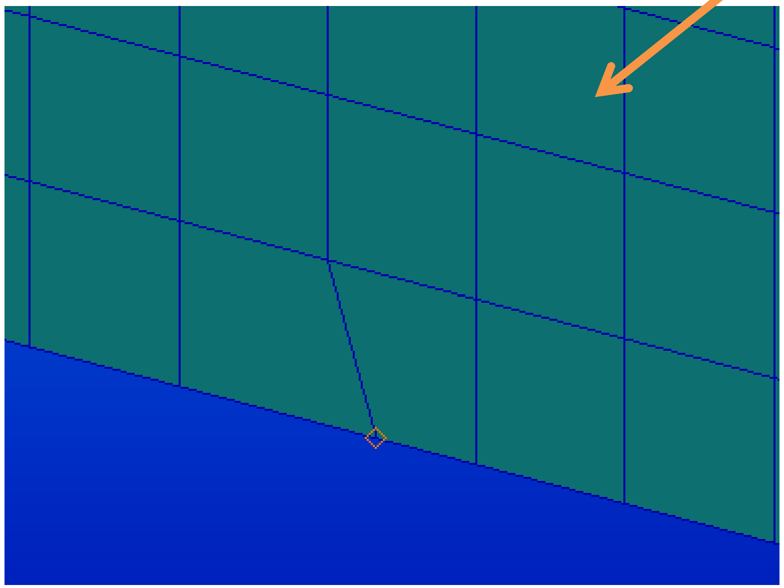
web





It can be seen that the location of this node has been changed along X axis.

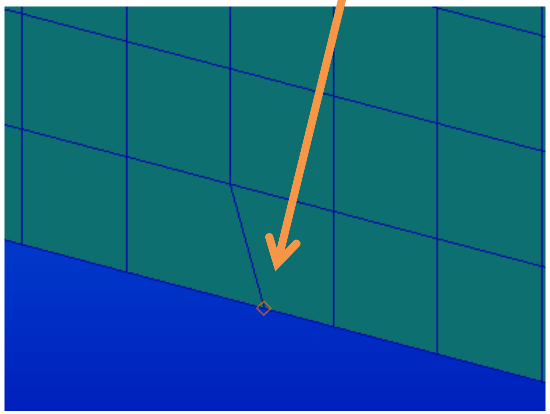


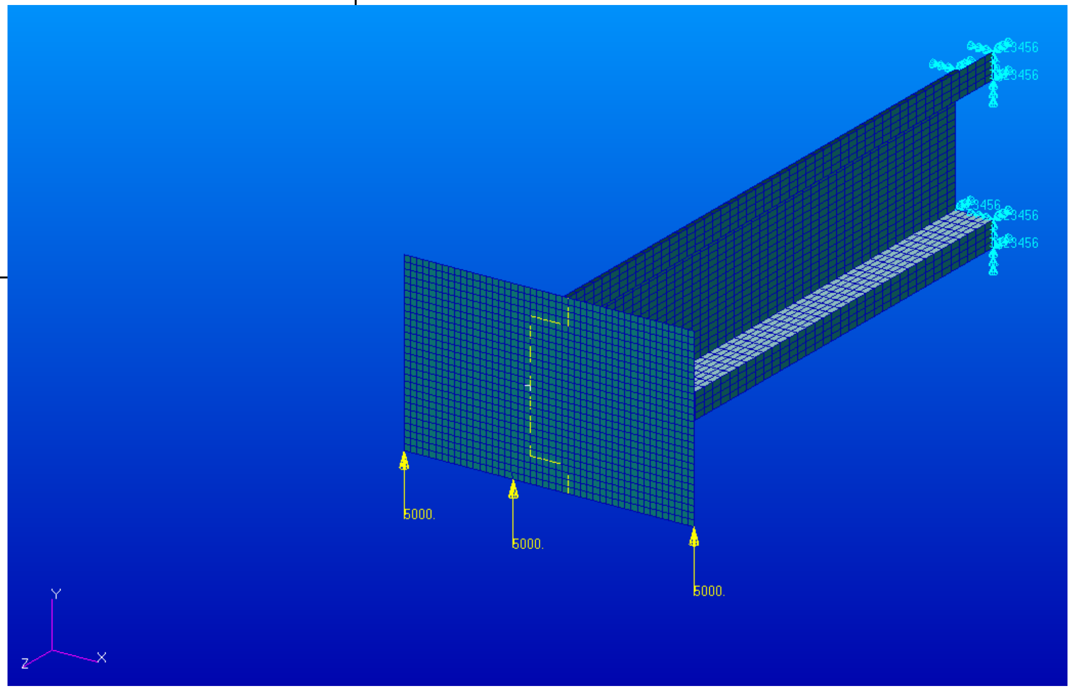


Apply load for **the 3rd load case:**

a. Click on the **Loads/BCs** tab
b. Loads/BCs: **Create/Force/Nodal**
c. Enter **force_3_SC** as the New Set Name
d. Click **Input Data…**
e. Enter **<0,5000,0>** for the Force
f. Click **OK**
g. Click **Select Application Region…**
h. Select **FEM**
i. Click on the **Select Nodes** panel
j. Select the **Node** icon
k. Select the **„shifted" node**
l. Click **Add**
m. Click **OK**
n. Click **Apply**

The figure below shows the third force acting through the shear center.



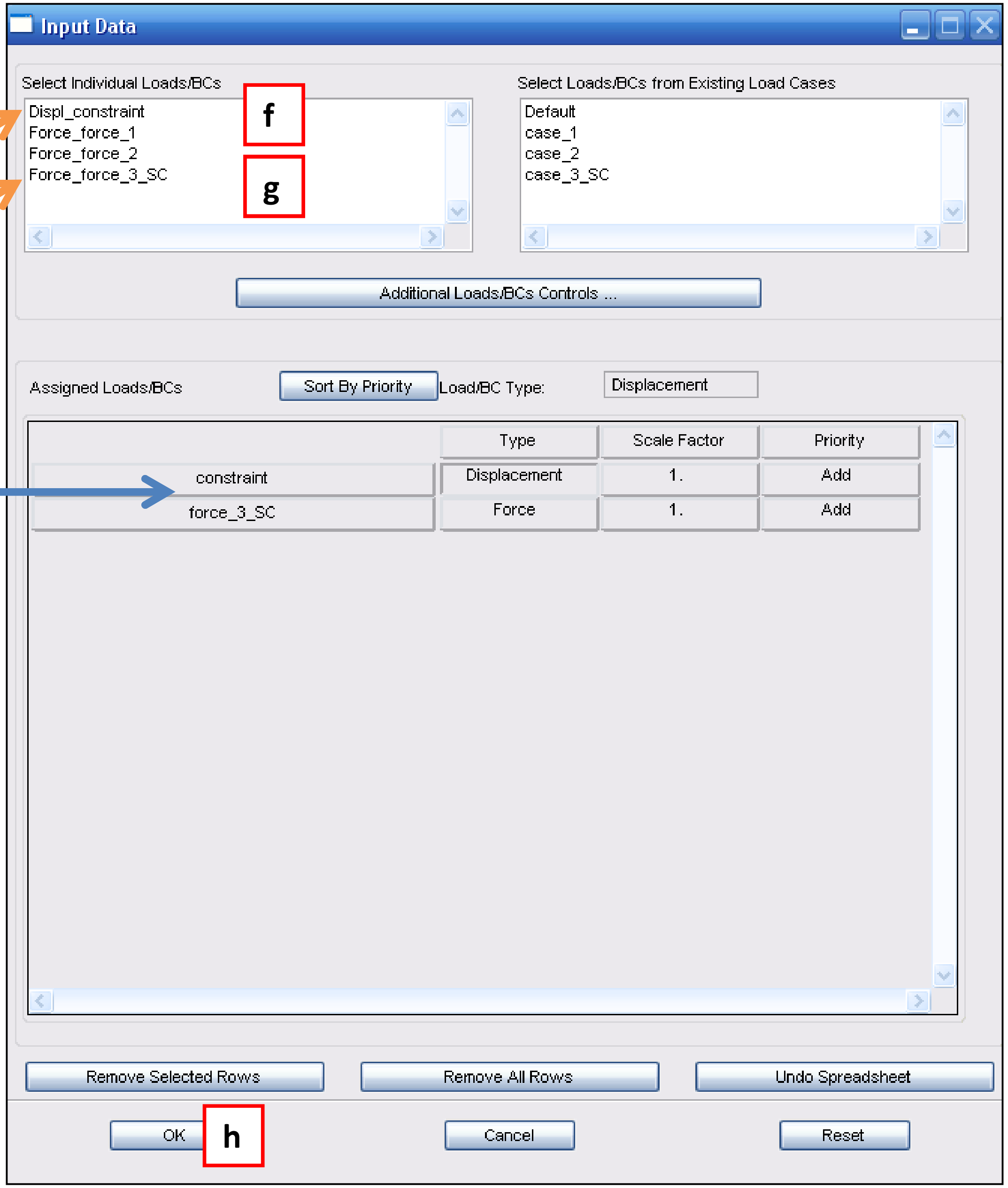


a. Click on the **Loads/BCs** tab
b. Click on the **Create Load Case** in Load Cases
c. Enter **case_3_SC** as the Load Case Name
d. Uncheck **Make Current**
e. Click **Input Data…**
f. Click **once** on the **Displ_constraint** in *Select Individual Loads/BCs*
g. Click **once** on the **Force_force_3_SC** in *Select Individual Loads/BCs*

You will see the assigned loads and boundary conditions to the load subcase no. 3.

<u>Note:</u> You can delete the row *in Assigned Loads/BCs* by selecting the appropriate row (clicking on it) and clicking on **Remove Selected Rows**.

h. Click **OK**
i. Click **Apply**



Run a linear static analysis:
a. Click on the **Analysis** tab
b. Choose **Analyze/Entire Model/Full Run** from right menu
c. Enter **open_section_SC** as the Job Name
d. Click **Solution Type…**
e. Select **LINEAR STATIC** as the Solution Type
f. Click **Solution Parameters…**
g. Click **Results Output Format…**
h. Uncheck **Print**
i. Click **OK**
j. Click **OK**
k. Click **OK**
l. Click **Subcase Select…**
m. Click **Unselect All**
n. Select **case_3_SC**
Then, ”case_3_SC” will appear under ”*Subcases Selected:*”.

o. Click **OK**
p. Click **Apply**

Attach the results file, when the analysis job is completed:
a. Click on **Analysis** tab
b. Access Results/Attach XDB/Result Entities
c. Click **Select Results File…**
d. Select **open_section_SC.xdb** file and click **OK**
e. Click **Apply**

Verify the results with this reference

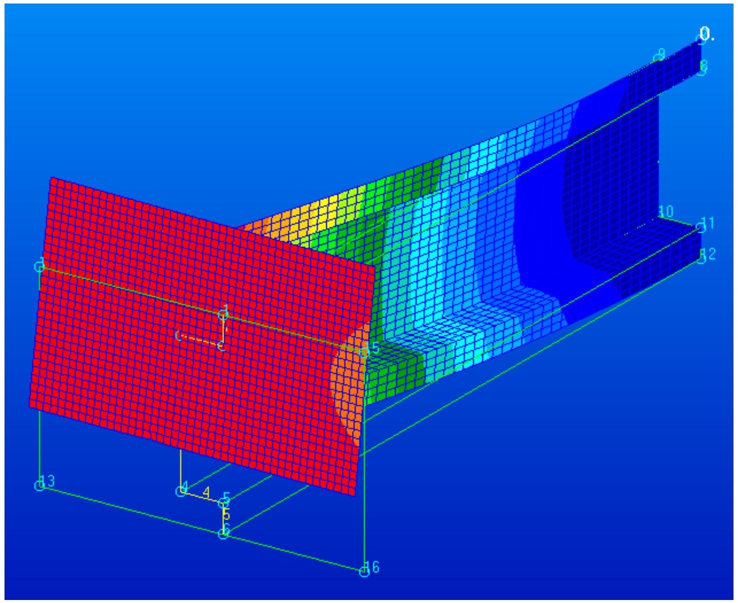
In **Deform Attributes** set Model Scale: **0.1**

<u>The **required** plots with the results for **3rd case**:</u>

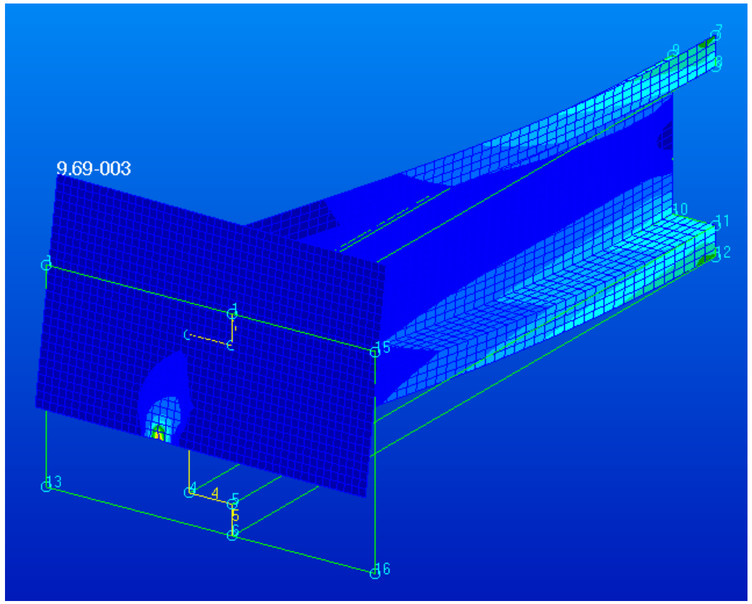
1) Fringe Result: **Displacements, Translational**
   Quantity: **Magnitude**
   Deformation Result: Displacements, Translational

2) Fringe Result: **Stress Tensor**
   Position… **AT Z1 and Z2** with Maximum option
   Quantity: **von Mises**
   Deformation Result: Displacements, Translational

Total number of plots with the results for **3rd case** = 1 (case) x 2 (plots) = **2**



3rd case, displacement magnitude



3rd case, von Mises, Z1 and Z2

Report should also contain:

a) **Figures**:

1) Geometrical model of the beam (1 figure)
2) FE model of the beam with load and boundary conditions (1 figure)
3) 8 plots with the results for 3 cases (8 figures = 2 x 3 + 2)

Total number of figures = 1 + 1 + 8 = **10**

A **white background** of **all** figures is **obligatory**.
A **date** on the plots with the results is **obligatory**.

b) **Table** with:

- the vertical displacements of chosen two nodes (YY) [mm]
- the calculated *delta* values [mm]
- the X coordinate [mm]

from both analyses (Case 1 and Case 2)

Note: the chosen nodes should be specified based on some description (not on the number) e.g. *Node 1 is located in the bottom left corner of the additional surface.*

Report should also contain:

c) **Graph** of function connecting coordinates of two points

d) **Equation** generated based on the trend line

e) **The X coordinate of the shear center** relative to the web (**$X_{S.C.}$**) calculated based on the data from the program (see: Table which should be filled with the appropriate values)

f) **Relative error** between $X_{S.C.}$ from the graph obtained based on the data from the program and $X_{S.C.\ theoretical}$ from analytical calculations (see: homework)

g) **Definition of the shear center**

i) **Conclusions**