CARDIFF UNIVERSITY
PRIFYSGOL CAERDYDD

ORCA – Online Research @ Cardiff

This is an Open Access document downloaded from ORCA, Cardiff University's institutional repository:https://orca.cardiff.ac.uk/id/eprint/70403/

This is the author's version of a work that was submitted to / accepted for publication.

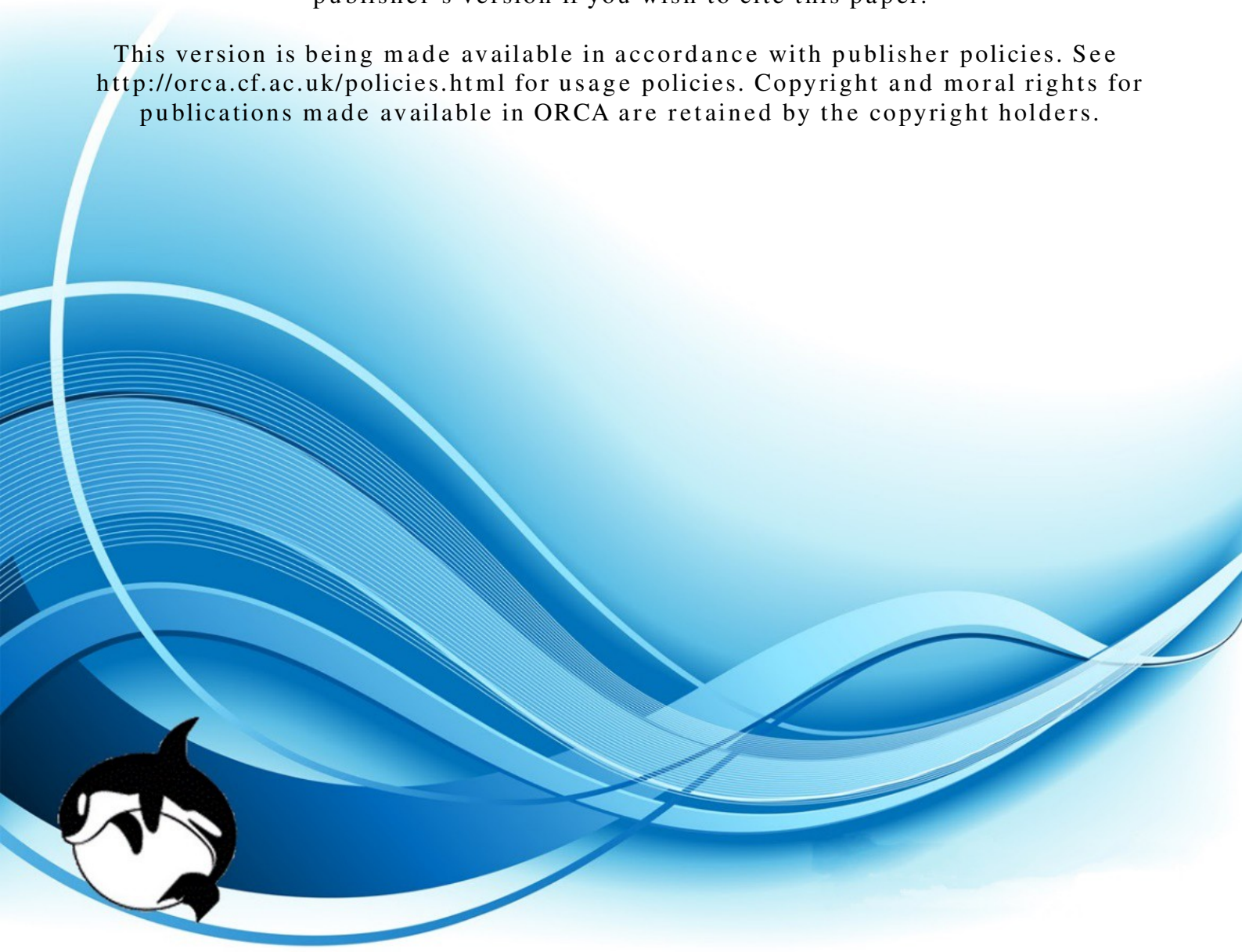
Citation for final published version:

Mishra, Aadarsh 2015. Finite element analysis of an aluminium cantilever beam. International Journal of Mechanical Engineering and Robotics Research 4 (1) , pp. 540-560.

Publishers page: http://www.ijmerr.com/v4n1/ijmerr_v4n1_60.pdf

Please note:
Changes made as a result of publishing processes such as copy-editing, formatting and page numbers may not be reflected in this version. For the definitive version of this publication, please refer to the published source. You are advised to consult the publisher's version if you wish to cite this paper.

This version is being made available in accordance with publisher policies. See http://orca.cf.ac.uk/policies.html for usage policies. Copyright and moral rights for publications made available in ORCA are retained by the copyright holders.



ISSN 2278-0149
Vol.4, No.1, January 2015

International Journal of Mechanical Engineering and Robotics Research

IJMERR

www.ijmerr.com

International Journal of Mechanical Engineering and Robotics Research

India

Email : editorijmerr@gmail.com or editor@ijmerr.com

International Journal of Mechanical Engineering and Robotics Research

ISSN 2278 – 0149 www.ijmerr.com
Vol. 4, No. 1, January 2015
© 2015 IJMERR. All Rights Reserved

Research Paper

FINITE ELEMENT ANALYSIS OF AN ALUMINIUM CANTILEVER BEAM

Aadarsh Mishra[1]*

*Corresponding Author: ***Aadarsh Mishra,*** ✉ *mishraA13@cardiff.ac.uk*

MSC Nastran is a multidisciplinary structural analysis application that is used by engineers in the static and dynamic analysis across the linear as well as nonlinear domains. In the given research paper an analysis has been done on an end loaded Aluminium cantilever beam and the finite elemental results have been compared with that of the beam theory.

Keywords: Aluminium Cantilever beam, Finite elemental analysis, MSC Nastran

INTRODUCTION

Engineers have been using MSC Nastran to ensure that the structural systems have the required strength and stiffness to preclude failure (which includes excess stresses, resonance, buckling, and deformations) that may compromise the structural function and safety. MSC Nastran has an important role in improving the economy and passenger comfort of structural designs.

Nastran software is based on a very sophisticated numerical methods, in which the Finite Element Method is the most prominent. Nonlinear Finite Elemental problems are solved either with implicit or explicit numerical techniques.

Manufacturing industries are using MSC Nastran's unique multidisciplinary approach for the structural analysis at various points in the product development. MSC Nastran is used for the following ways:

Figure 1: Home Page of the Patran Software

- There is a provision of virtual prototype in the early design process, thereby saving

[1] Department of Mechanical Engineering, Cardiff School of Engineering, Cardiff University, United Kingdom.

costs which are traditionally associated with physical prototyping.

- There are certain remedy structural issues which occur during a product's service thereby reducing downtime and costs.
- For Optimising the performance of existing designs or develop unique product design differentiators which in turn lead to industry advantages over competitors.

EXPERIMENTAL PROCEDURE

In the given analysis, an end loaded cantilever beam of Aluminium has been used. The cross section of the beam is 400 mm * 40 mm.

The analysis of beam is started by first creating the corner points of the beam.

The geometry icon is selected and is used to build the structure. In the geometry icon, select the Points and option 'XYZ'. On the left hand side select Create, Point, XYZ. These are the default menu selections.

The coordinates of the first point which is to be created are entered. These are [x, y, z] and are in square brackets. After entering the coordinates of the first point, Click Apply to create the point. If 'Auto Execute' is selected, the point will be created as soon as the Enter key is pressed.

The point [0, 0, 0] will not be visible as the origin is always shown as a cross. Moving the cursor to the origin will indicate the point's position.

Figure 2: Figure Depicting the Origin Point and the Geometry Icon

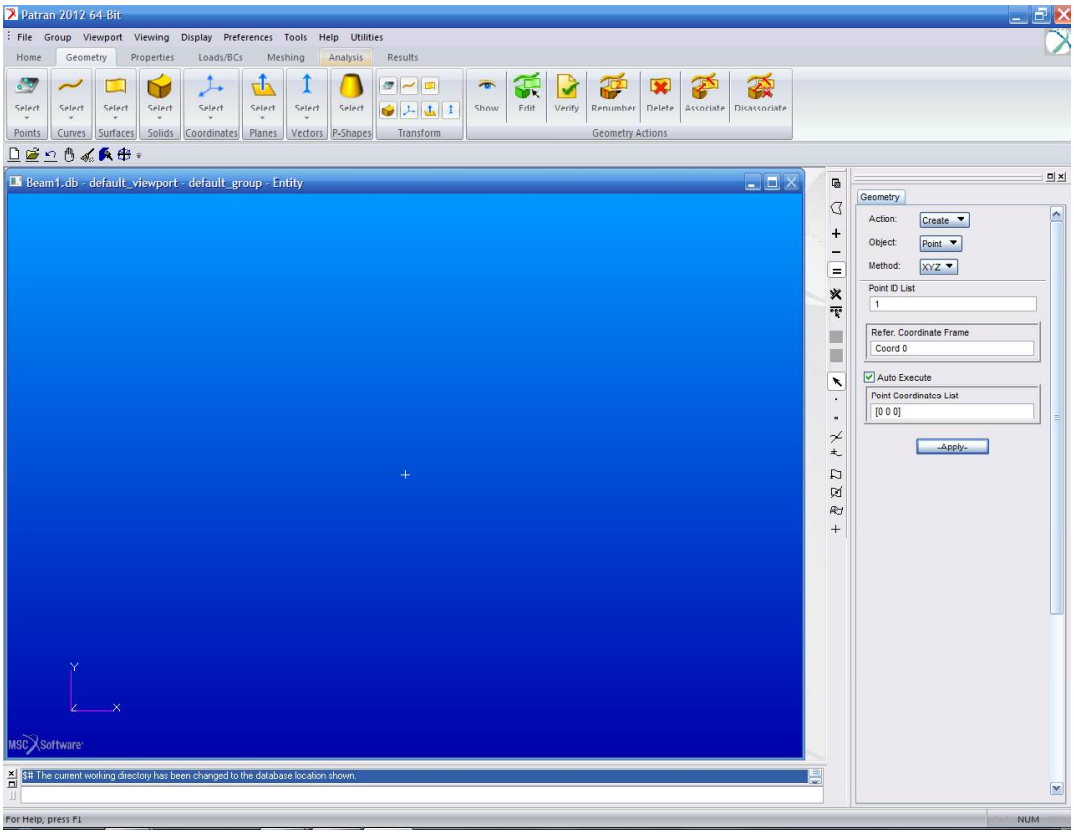
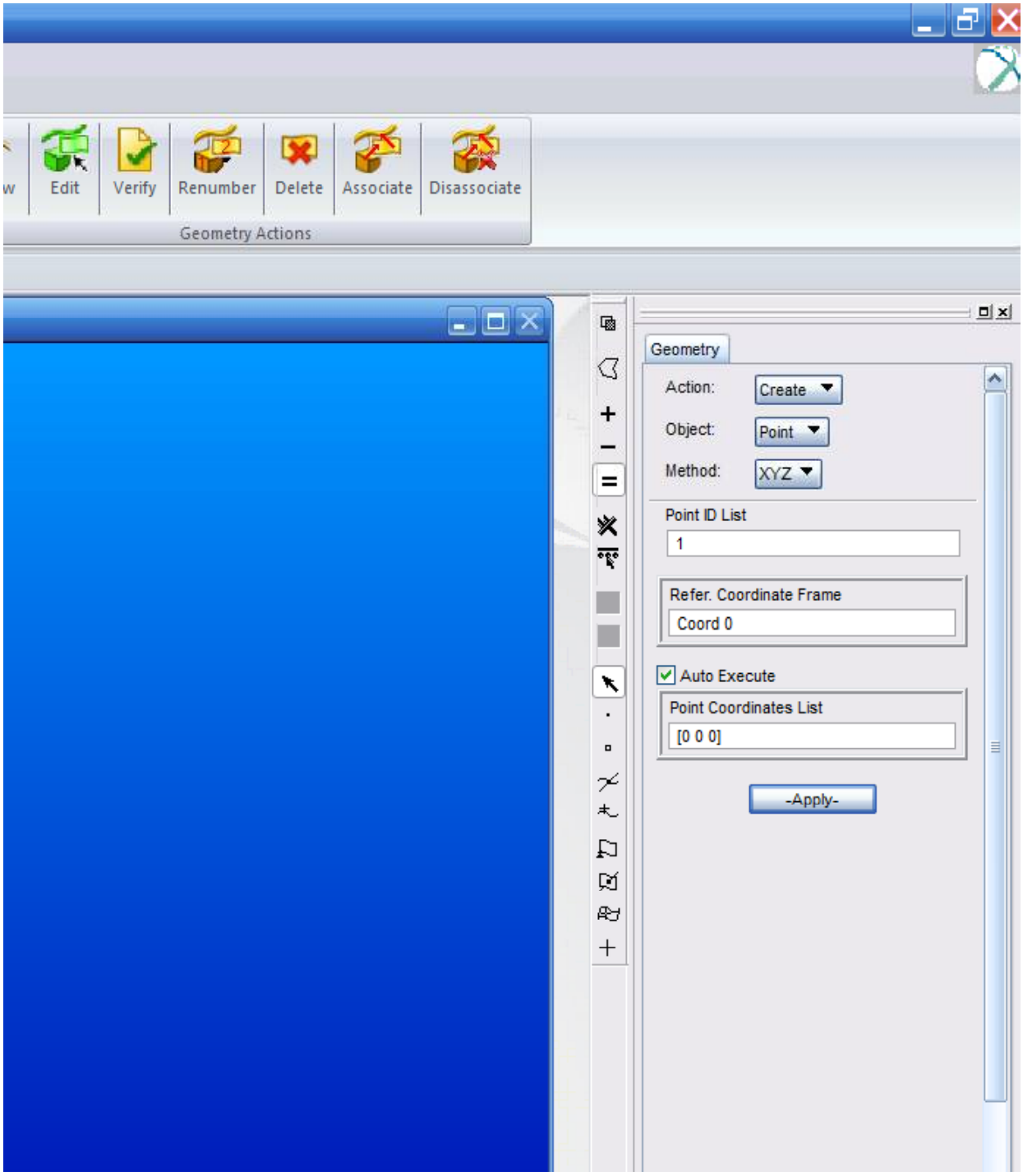
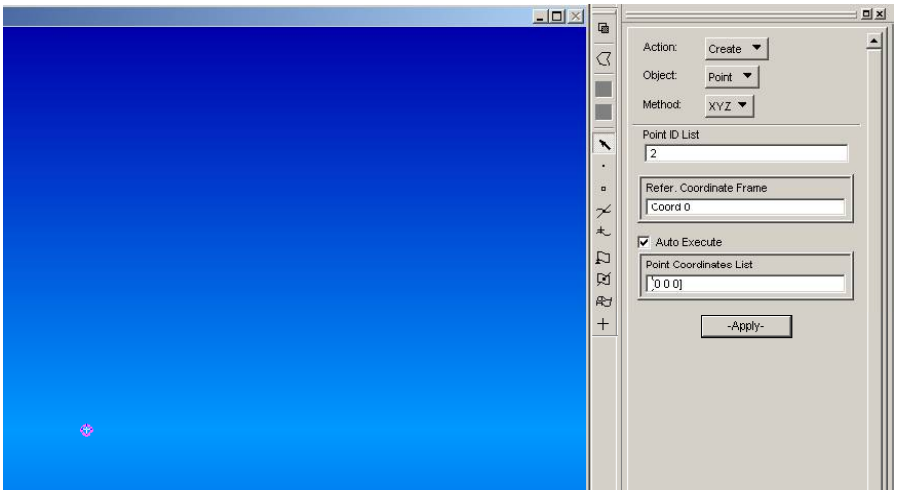


Figure 3: Diagram Depicting the Process of Creation of Points of the Cantilever Beam



The co-ordinates of the cantilever beam are [0, 0, 0], [3, 0, 0], [3, 0.4, 0] and [0, 0.4, 0] and these points are created either in a clockwise or an anticlockwise order. These points will appear on the screen as single light blue pixels.

Figure 4: Diagram Depicting the Point of Origin



The display of points can also be changed by using the slider to adjust point size to suit screen.

After creating the points, the surfaces corresponding to the beam's shape in the x-y plane are created. Click 'Select Surface' in the

Figure 5

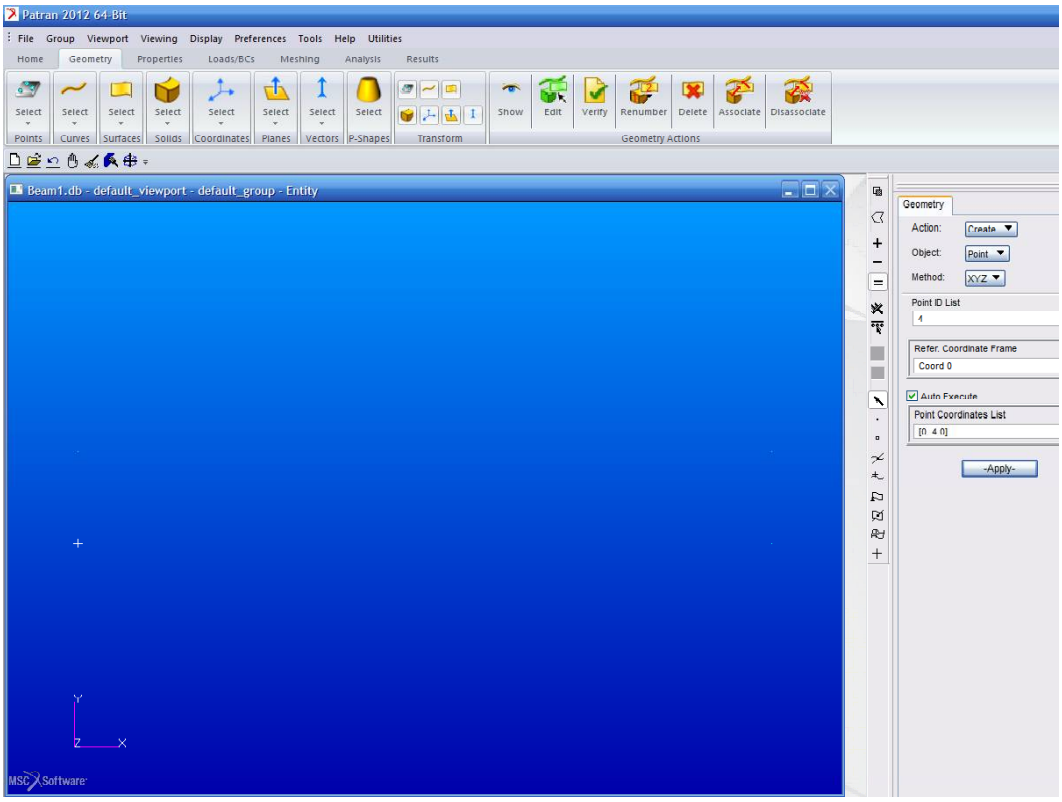


Figure 6

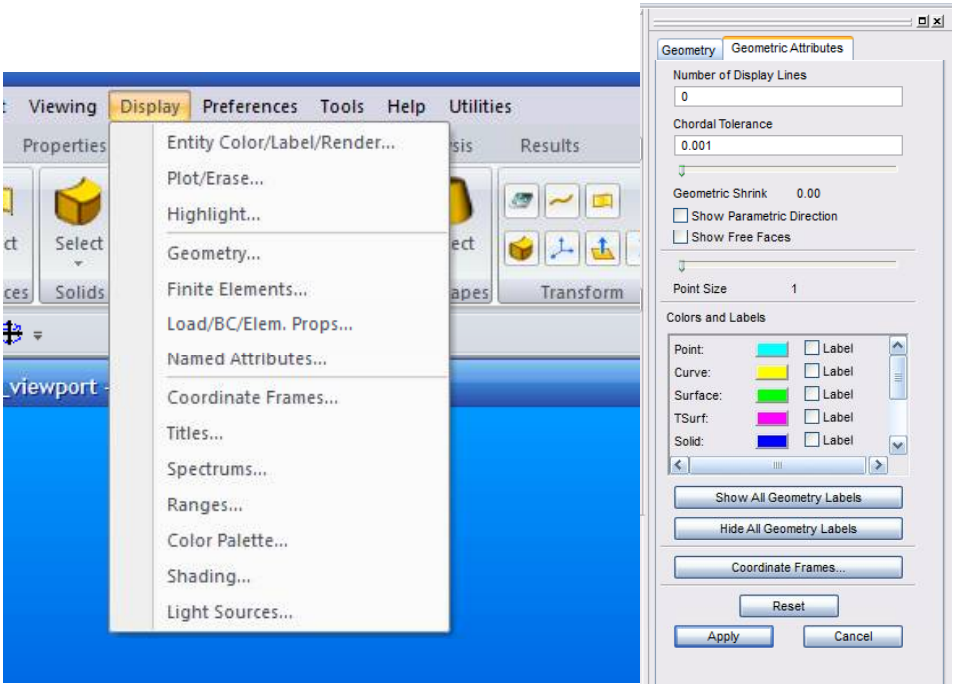


Figure 7

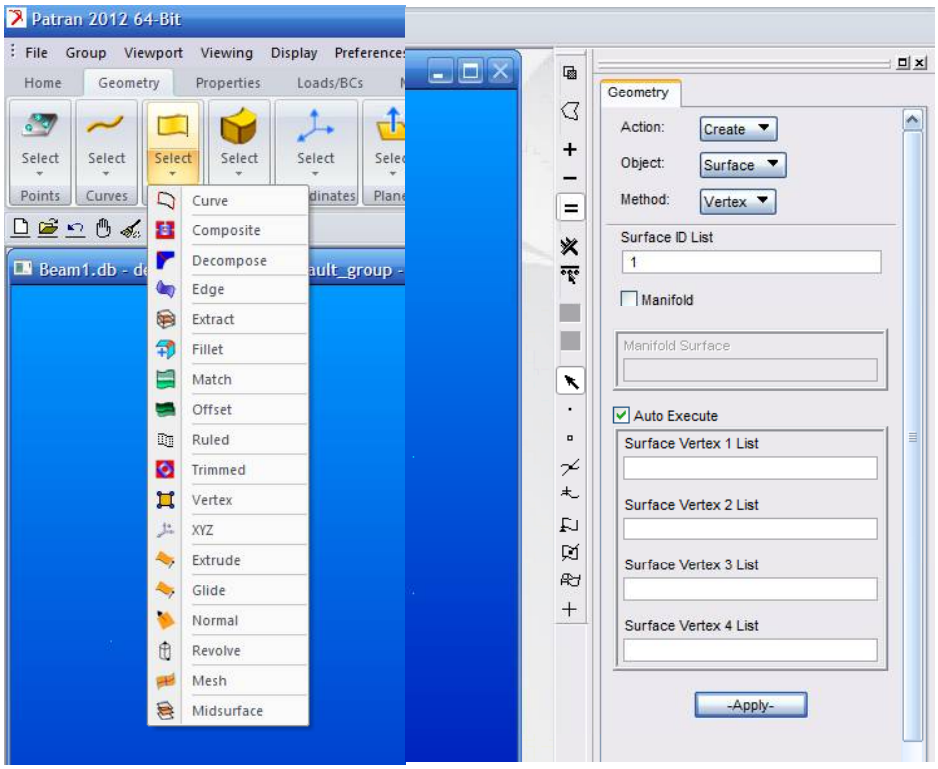


Figure 8

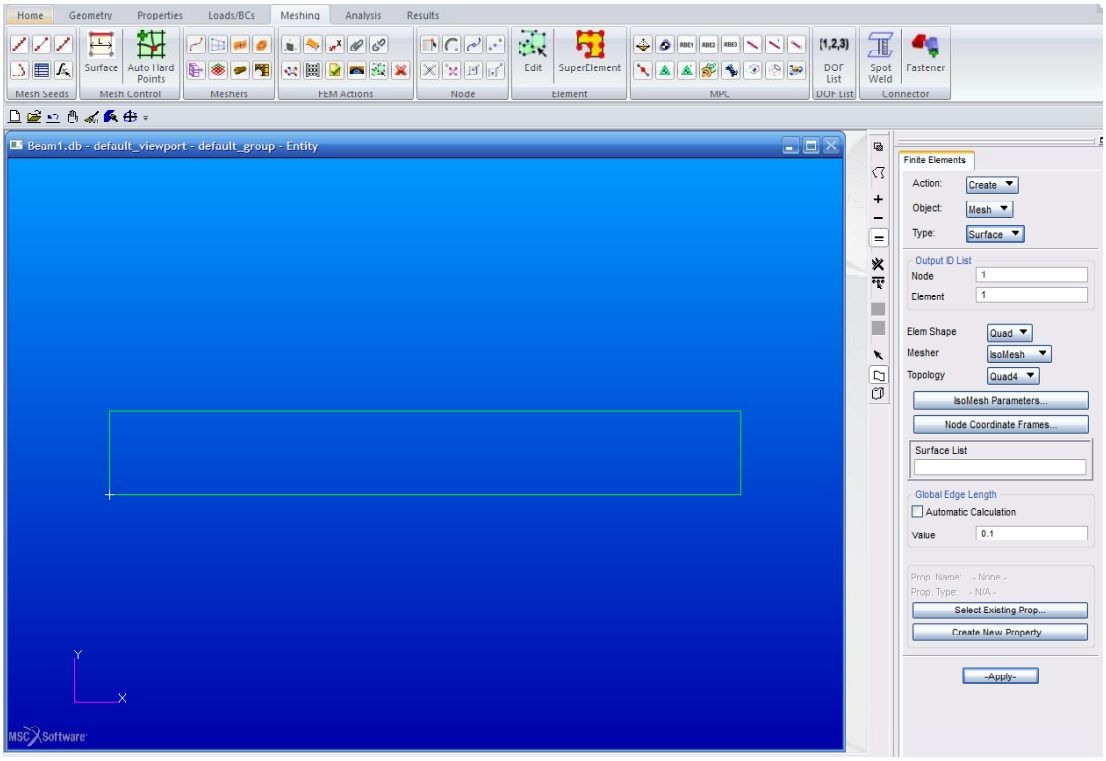
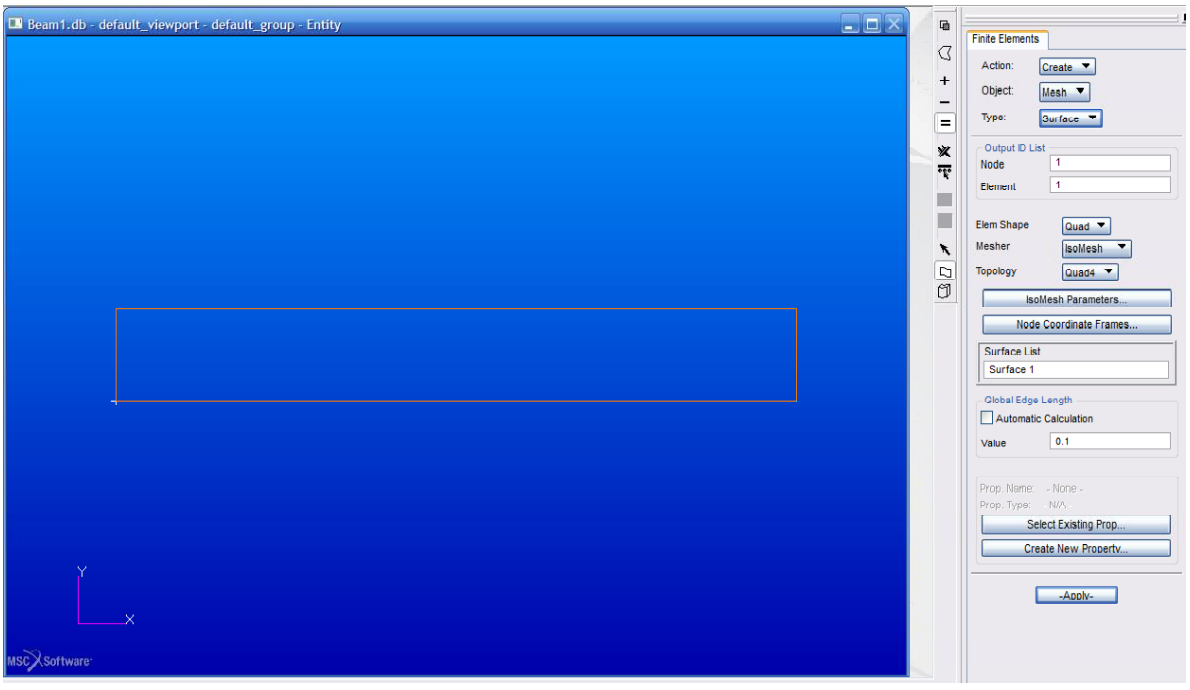


Figure 9



Geometry Menu and Select 'Vertex' from the options available. There will be a geometry icon in which Create, Surface, Vertex will be appearing. By placing the cursor in Vertex boxes,select the corner points.

The surface is then divided into Quadrilateral Finite Elements by selecting the Meshing menu and then the surface mesher, Options selected are Action: Create; Object: Mesh; Type: Surface. The type of element shape are selected as: Quadratic, IsoMesh and Quad8. Finally, required length of element is selected which will be the approximate length of each element side.

Now putting the cursor in Surface List box, select the surface by clicking on it, this will change its colour. The name of surface appears in the 'Surface List' box. The analysis is then proceeded by clicking 'Apply'.

The finite element mesh created is depicted by the figure given below:

After the element meshing, specify the required properties for the elements by selecting the properties menu. In the properties menu, there are options like: Isotropic Materials and 2D properties.

Figure 10

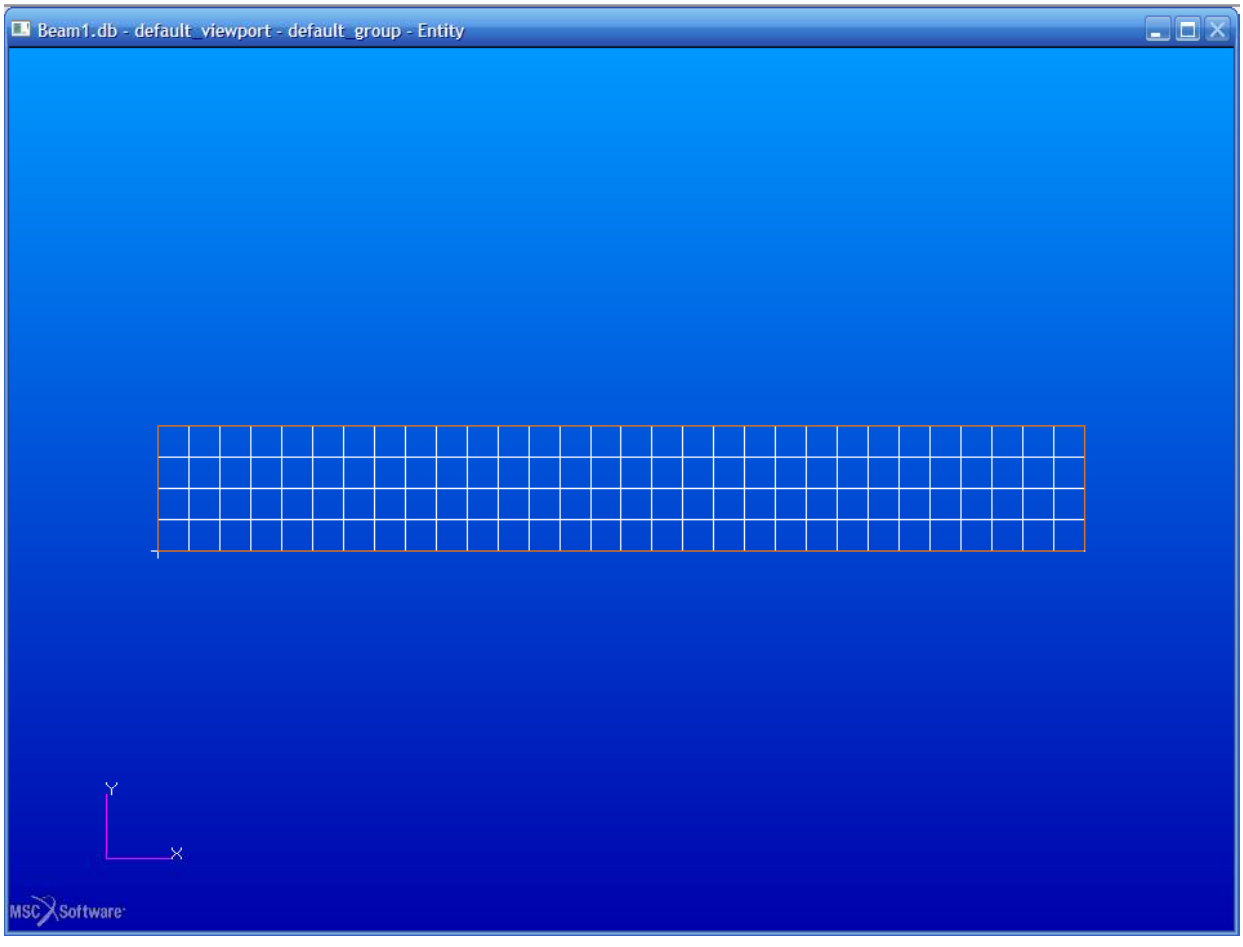


Figure 11

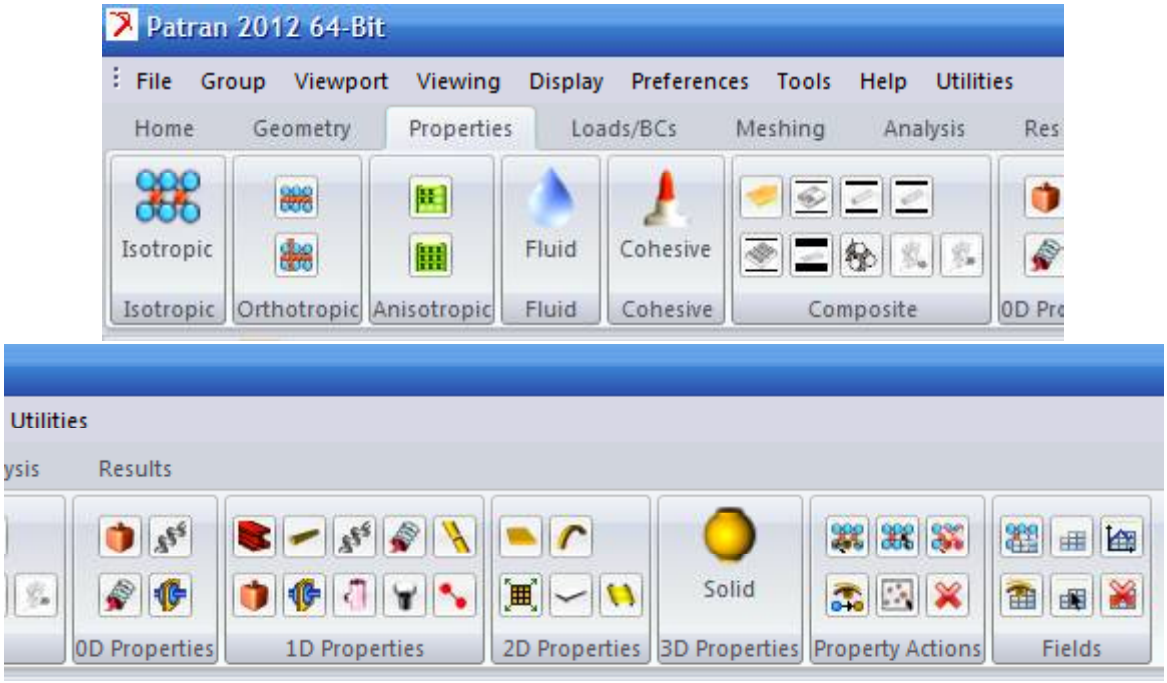


Figure 12

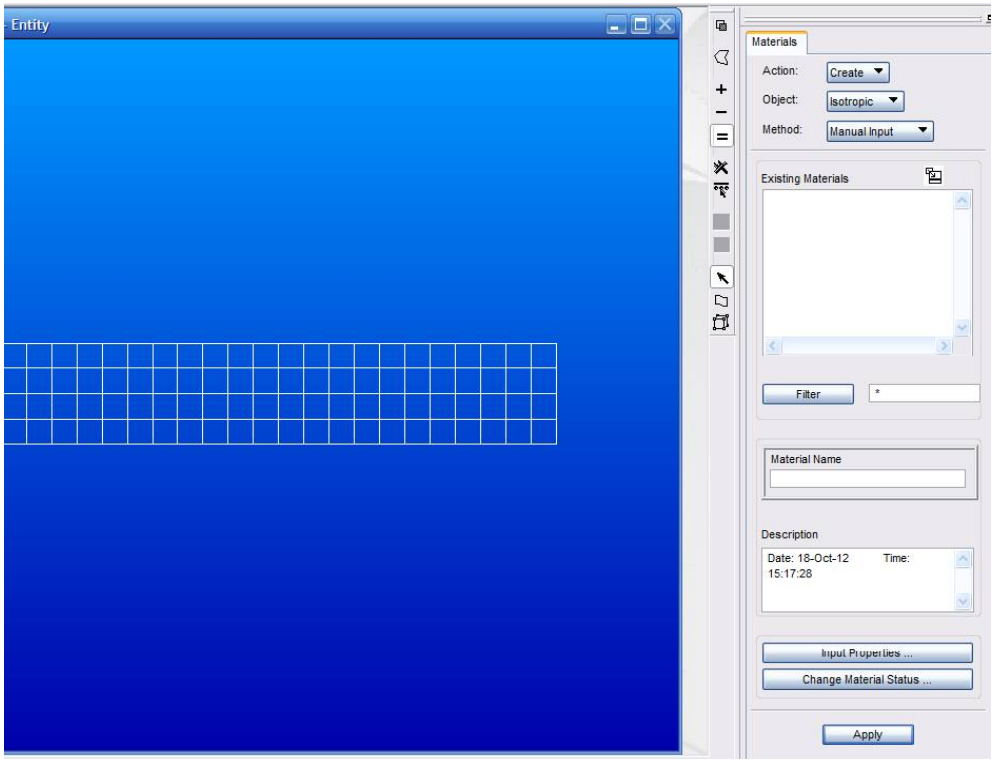


Figure 13

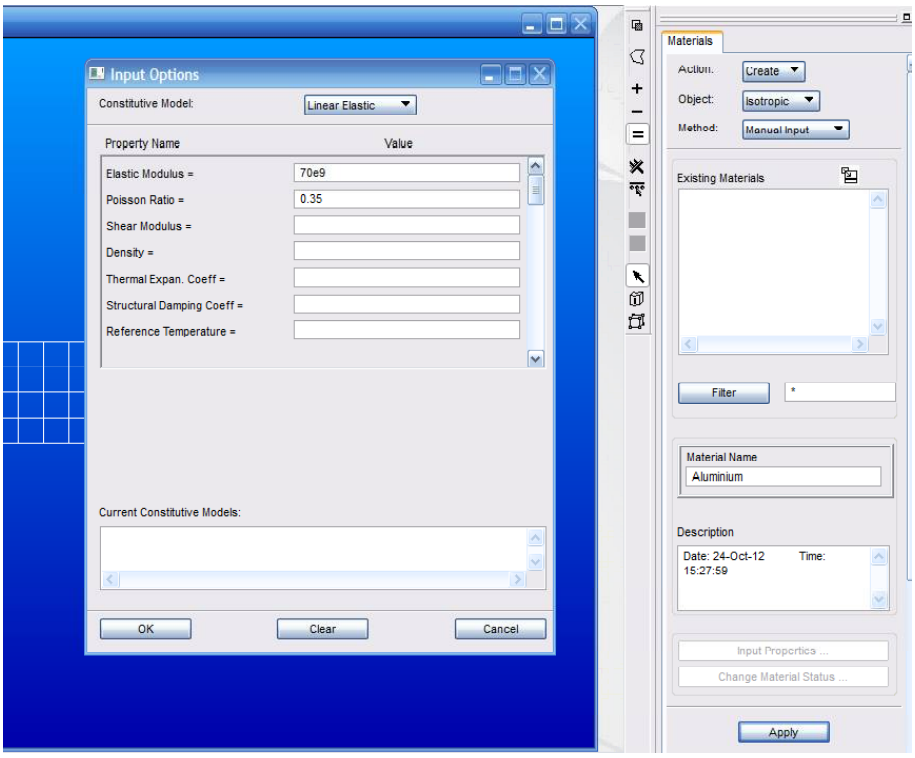
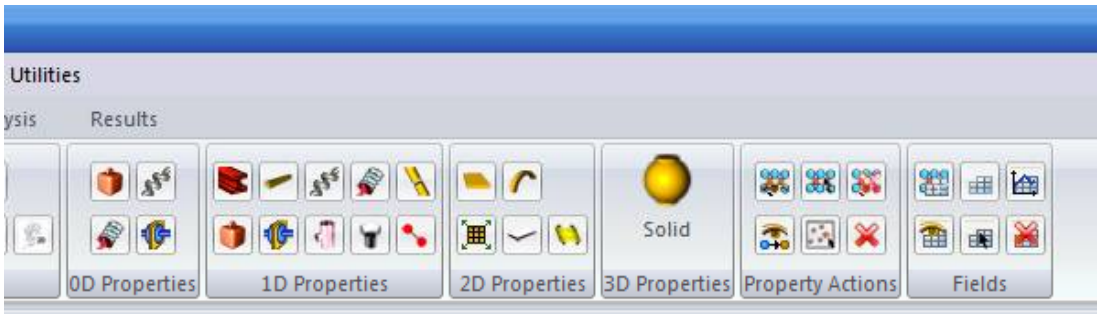


Figure 14



Since the given Cantilever beam which is being analysed is an Isotropic Material, Select Isotropic Material from the menu. Select Create, Isotropic, Manual Inputand then specify the Material Name. Then click on the Input properties.

A pop-up menu page opens in which the properties of materials have to be specified. The properties entered should be in S.I units. However, any two of Elastic Modulus, Poisson Ratio, and Shear Modulus are required. Click 'Ok' to return to Materials menu, and then Click Apply to create the material.

Similarly, the properties of the Finite Elements are also specified. Select the Shell option within the 2D Properties.

- This option will select the Crate, 2D, Shell menu.
- A shell element is a plane element that can deflect in and perpendicular to its plane. It is used to model a plate that has a thickness that is considerably smaller than the dimensions of the element.
- A shell element may also be curved (a curved shell element) e.g., part of the wall of a tube.

Further, there will be a Pop-up menu page for specifying 2D Shell properties.

- Specify Property Set Name
- Click on the Materials Property Name icon to list the materials available.
- Click on Input Properties

Now, a new popup Select Material menu appears showing available materials. Click on the required material. It will then be entered as the Material Name entry and the popup will disappear.

The shell element thickness has to be specified in S.I Units. Finally, Click Ok.

The last step is to choose the Elements that are to be allocated this property. Then click on the 'Select applicant region'.

- Put cursor in Select Members box
- Set surface pick option. The Pick options available vary according to the choice to be made.
- Select the surface on the drawing. Its outline will change colour and its name will appear in the Select Members box.

Figure 15

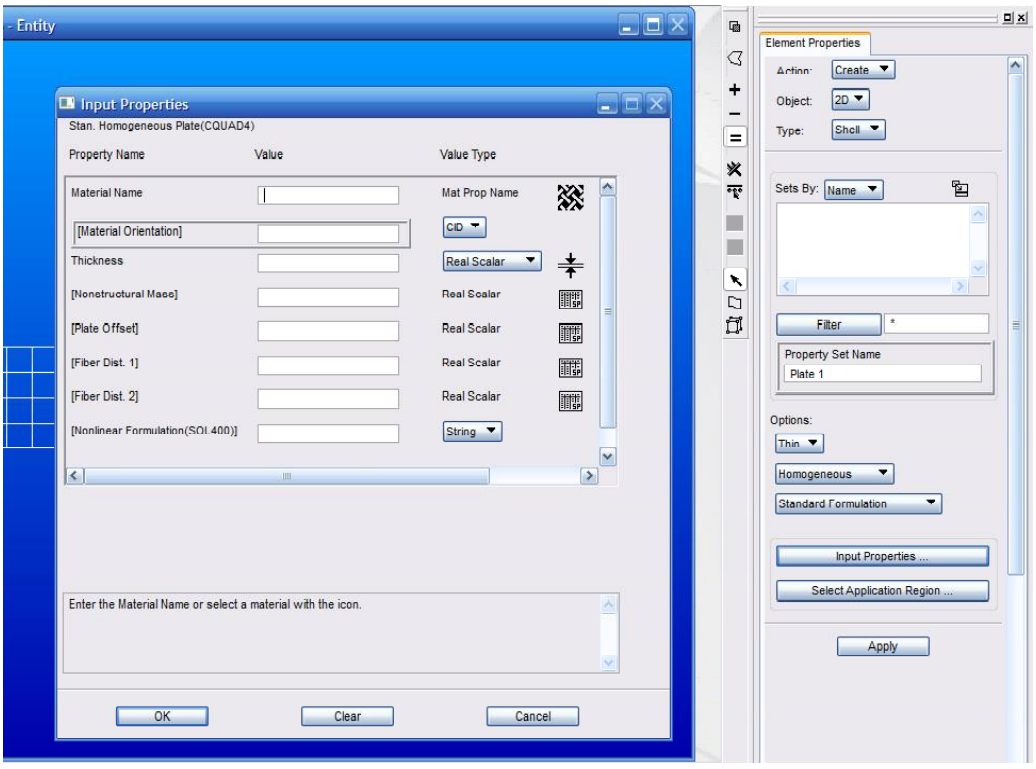


Figure 16

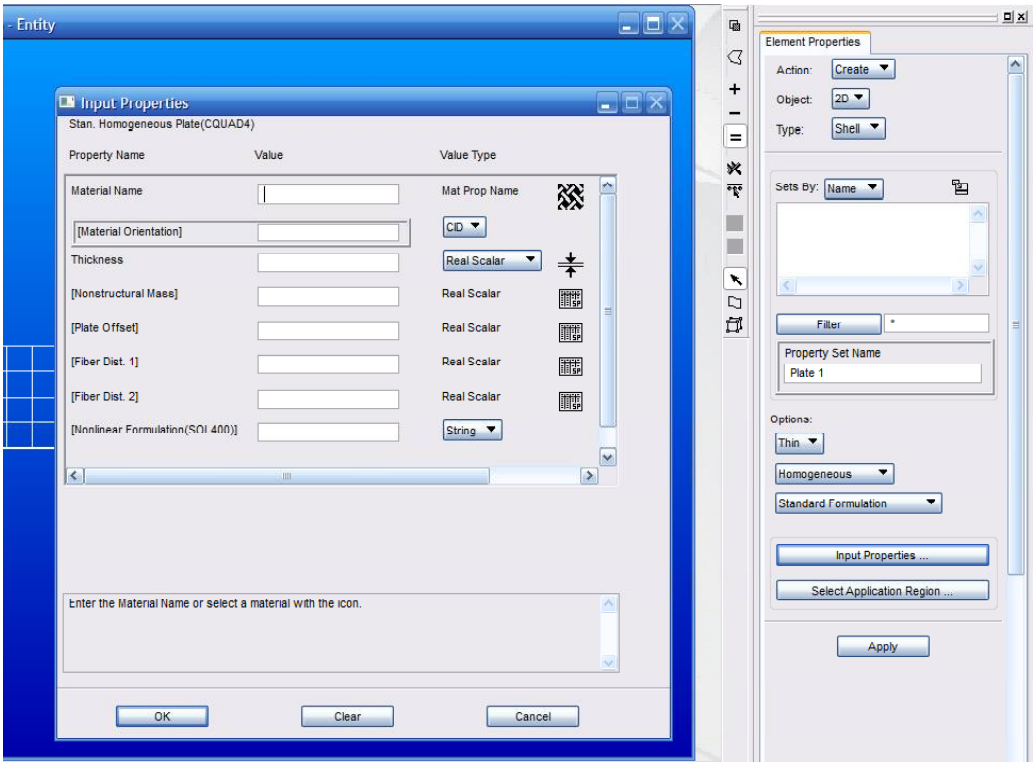


Figure 17

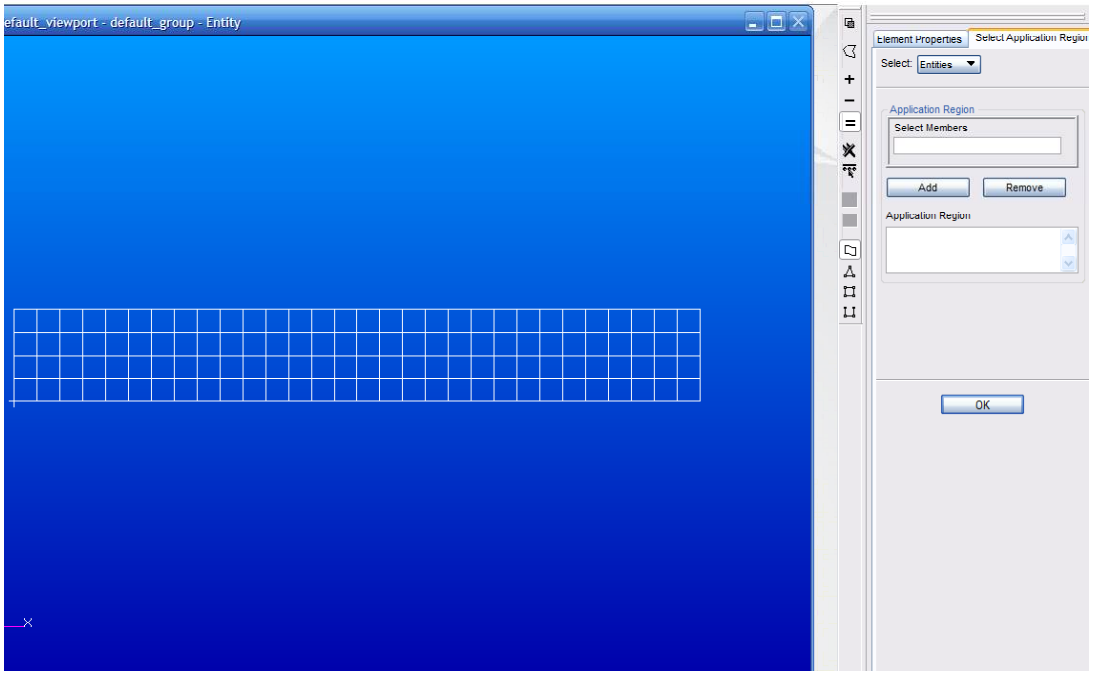


Figure 18

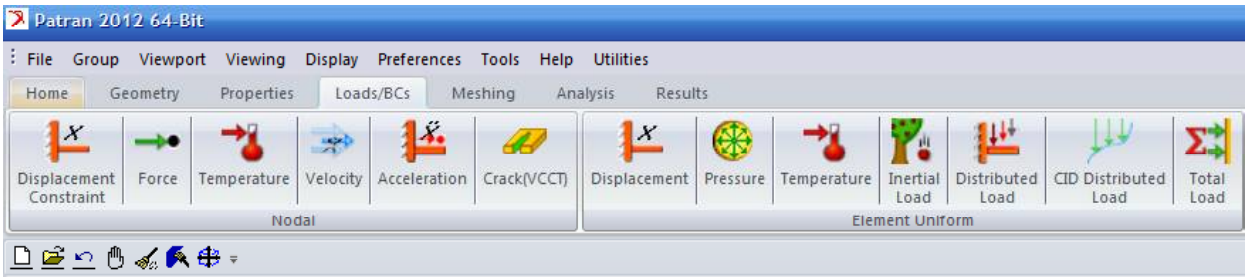
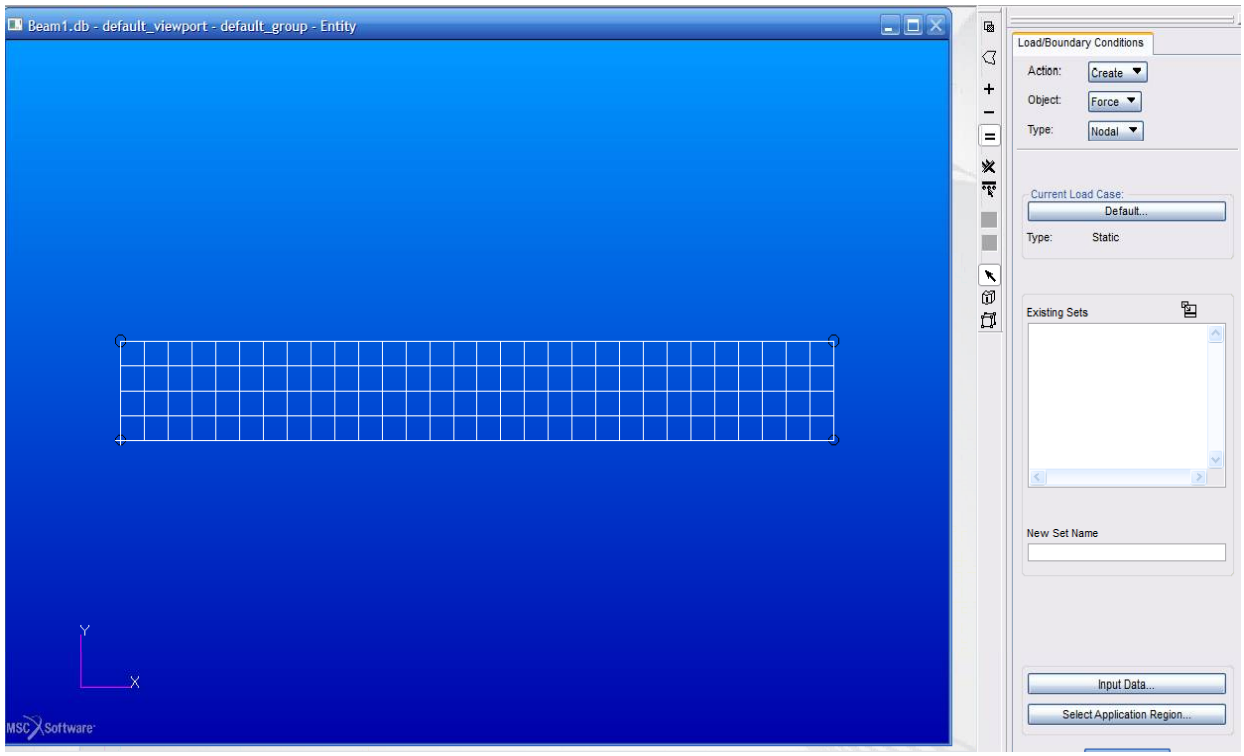


Figure 19



- Click Add which will transfer the selected member to the Application Region.
- When the Application region is complete, click 'ok'.

RESULTS AND DISCUSSION

After applying the element properties in the surface, specify the required Loads and Restraints.

- Select Create, Force, Nodal.
- Specify Name for Load.
- Select the Input data.
- Click the Select Application region.

By default Geometry items are selected, e.g., arcs, surfaces, points. This is called the Geometry Filter.

- Put cursor in Select Nodes box of the new window that appears.
- Select the 'Node' pick option. The Pick options available vary according to the choice to be made.
- Then click on the node to be loaded. The selected load is indicated.
- Node number will then appear in the Select Nodes box.
- Click Addto transfer Selected Nodes to the Application Region.
- Click Ok to indicate that specifying the Application Region is complete.
- The original window is then restored. Click Apply to signify that the loading specified is to be applied to the model
- When Apply is clicked the load applied is illustrated on the model in magnitude and direction.

Now specify required Restraints:

- Select Create, Displacement, Nodal
- Choose Name
- Click Input Data

Specify the translation restraint in the new window: Fixed = No Translation.

Figure 20

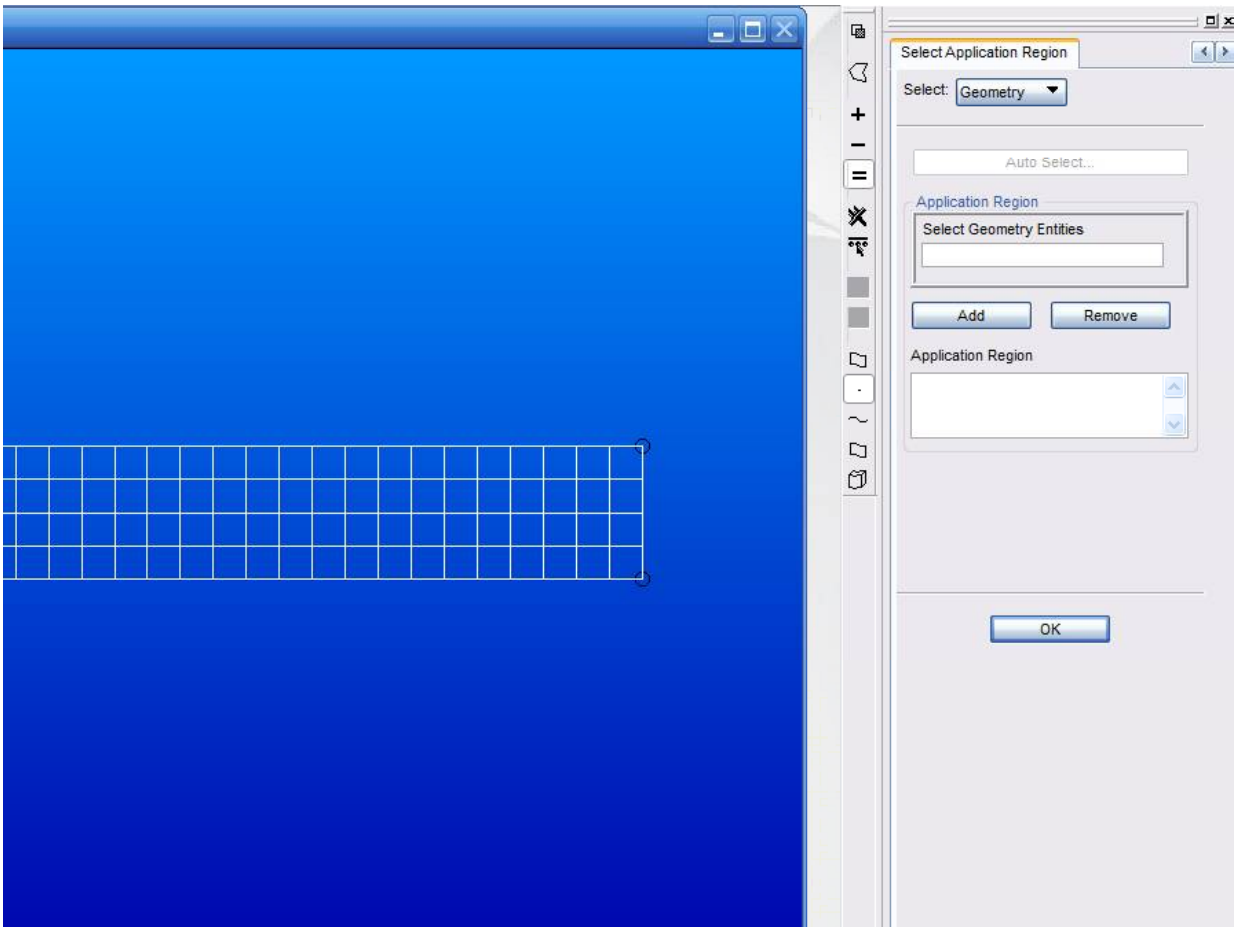
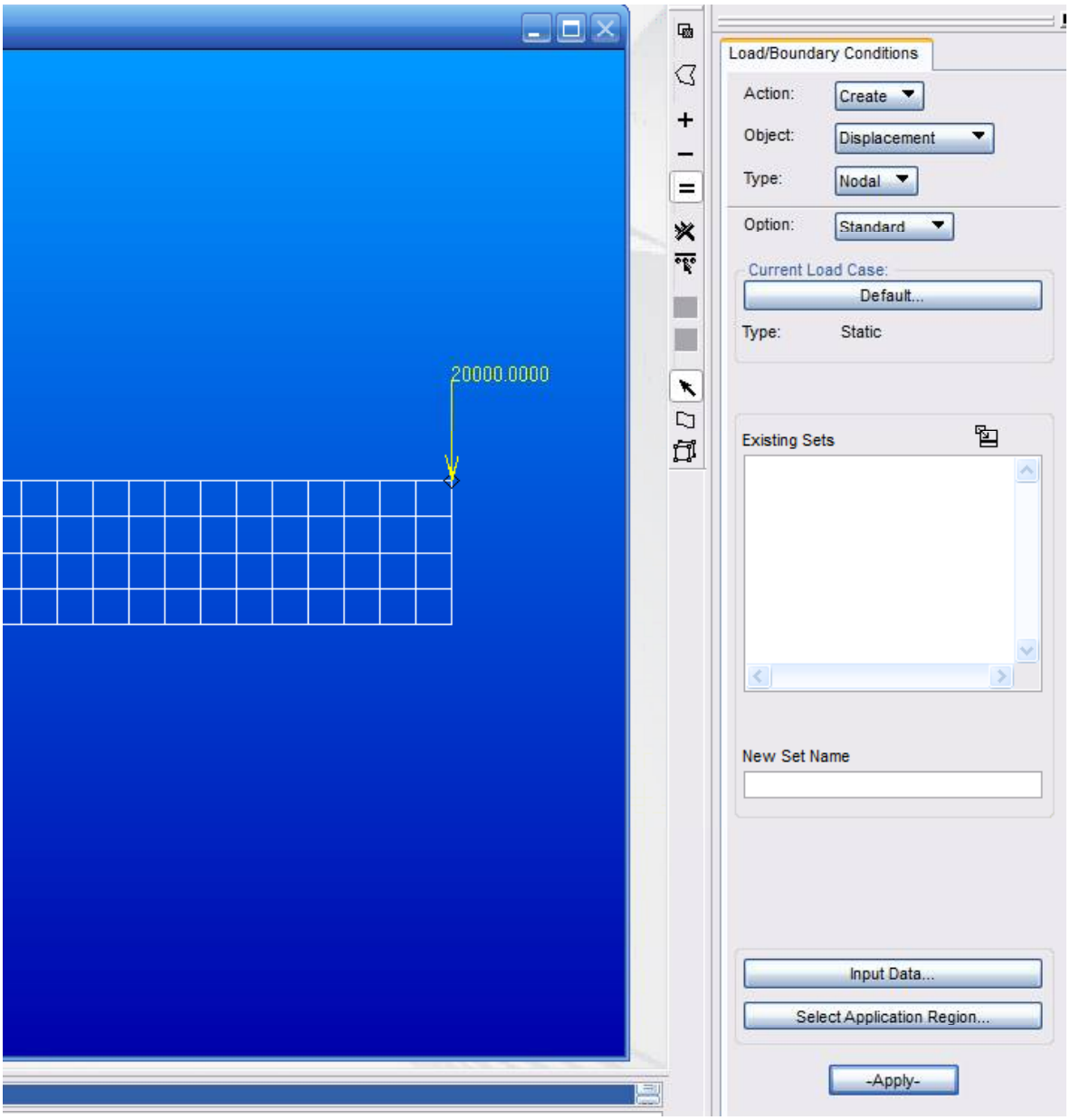


Figure 21



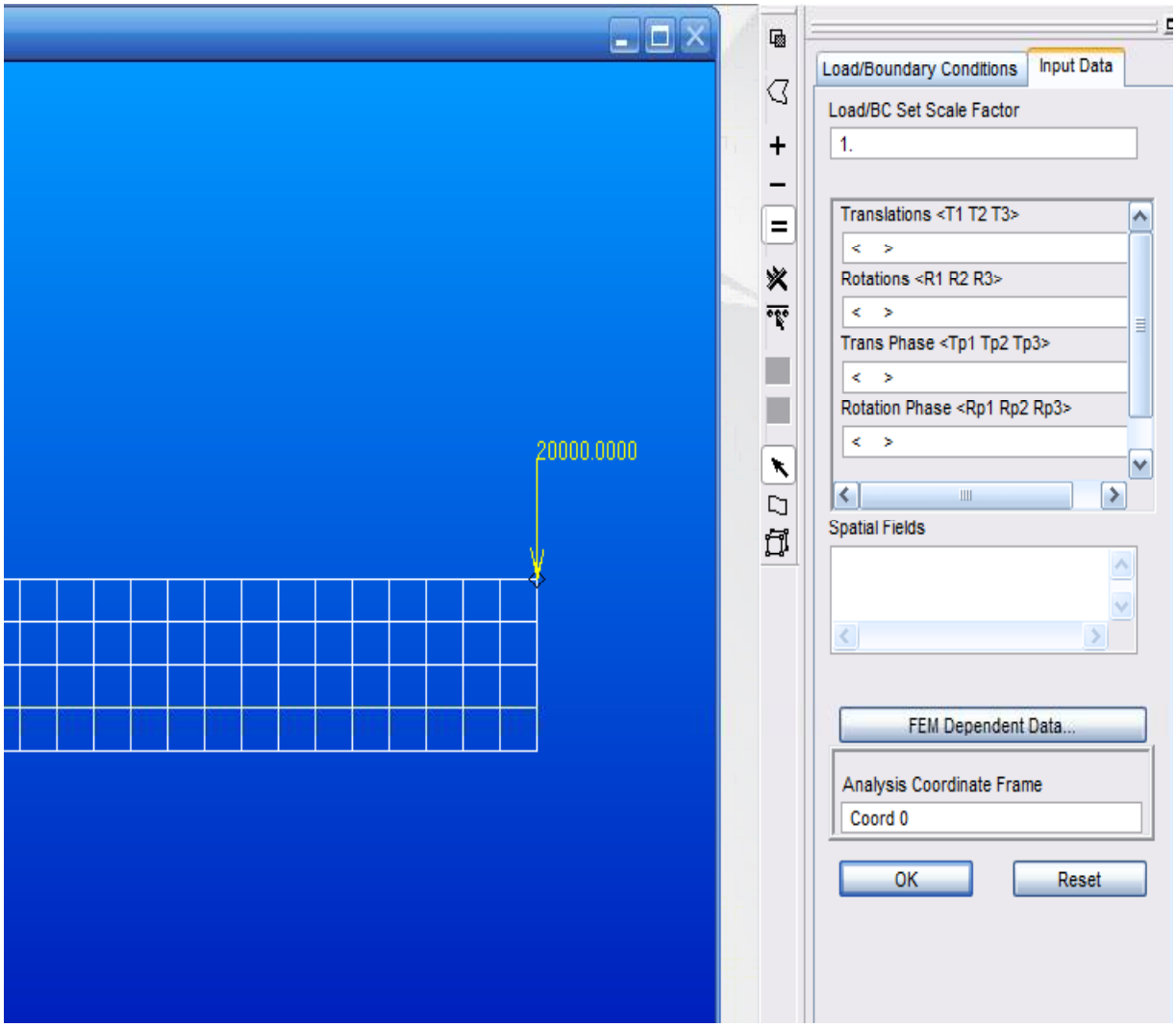
- Insert the translation: < 0, 0, 0 >

This imposes a ball joint restraint at each node selected.

- Insert the rotation: < 0,0,0>
- This prevents rotation about the x, y and z axes at each node selected.
- These two restraints effectively fix any nodes to which they are both applied.
- Click Okwhen complete.

As the cantilever beam is 'built in' at one end, so the nodes at the end of the beam need to be restrained. Hence click 'Select Application region'.

Figure 22



- Select Geometry so that the edge of the surface can be selected.
- Place cursor in Select Geometry Entities.
- Select the curve or edge pick option.
- Select the restrained edge with the cursor.

The selected curve will be shown in Select Geometry Entities.When the correct edge is chosen click Add to move the edge specification to the Application Region, and then click Ok.

The main Create, Displacement, Nodal window will then appear.Click Apply to implement the restraint.

A load of 20,000 N is made to act in the negative y direction. Displacement boundary condition is specified on curve for directions 1, 2, 3 and 4, 5, 6. This is applied to all finite element nodes on the curve.

- Direction 1 is the x axis, 2 is y axis, and 3 is z axis direction. These translation restraints are indicated by single arrows.

Figure 23

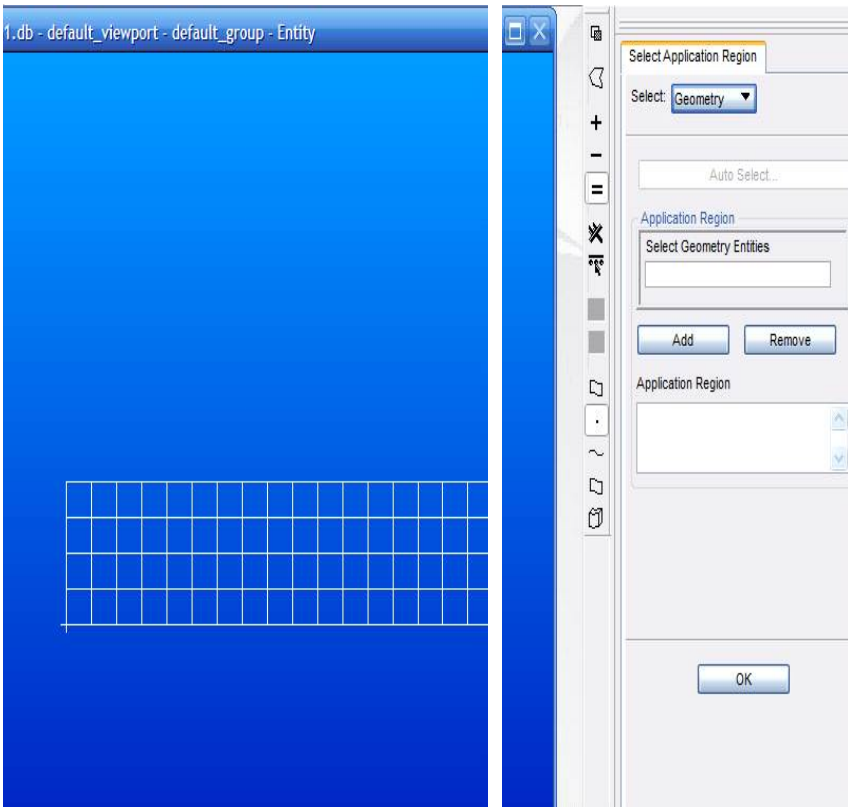


Figure 24

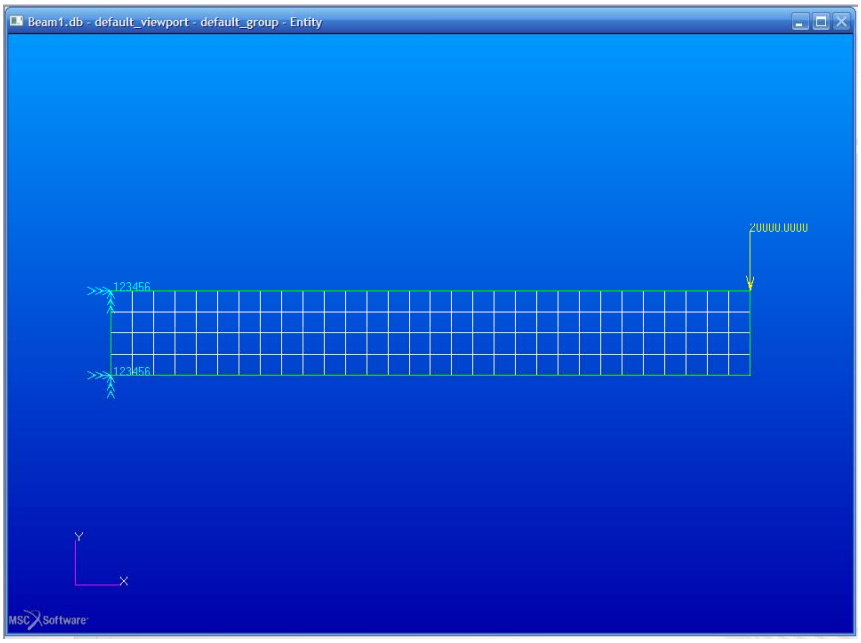


Figure 25

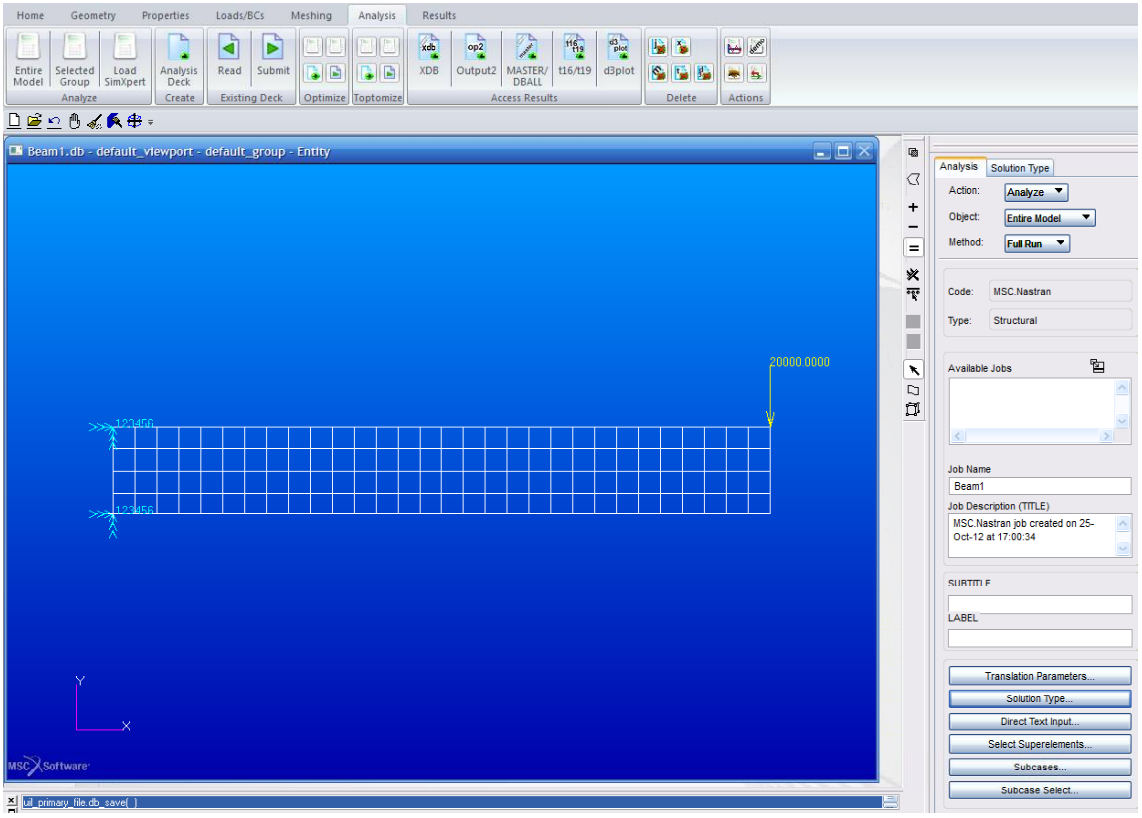


Figure 26

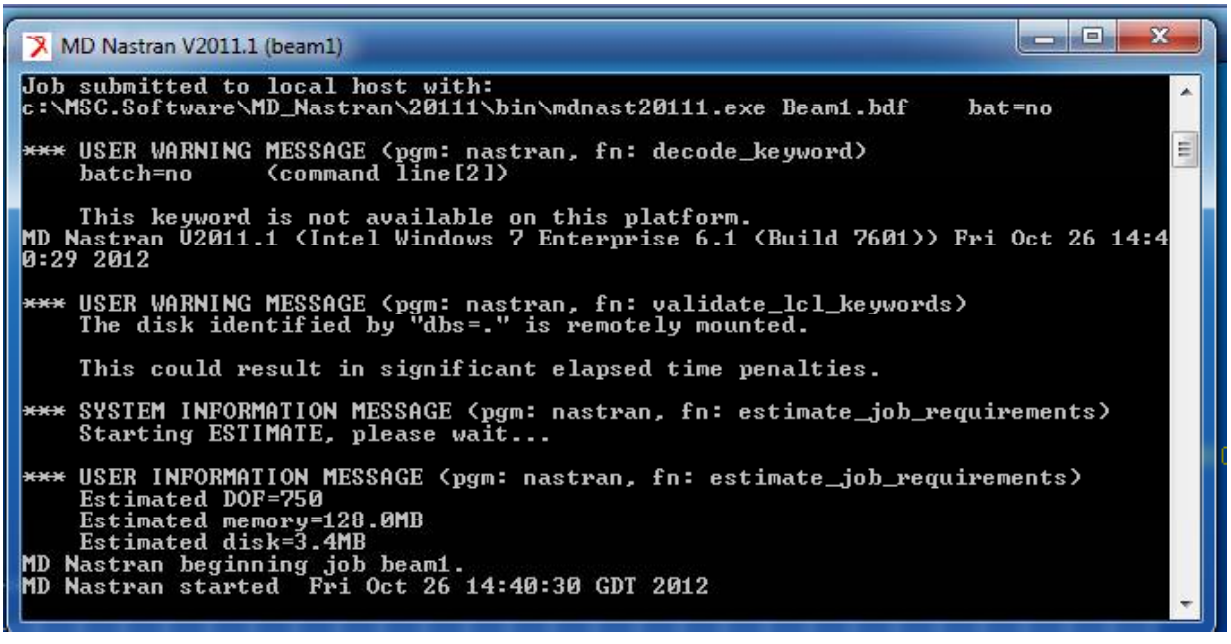


Figure 27

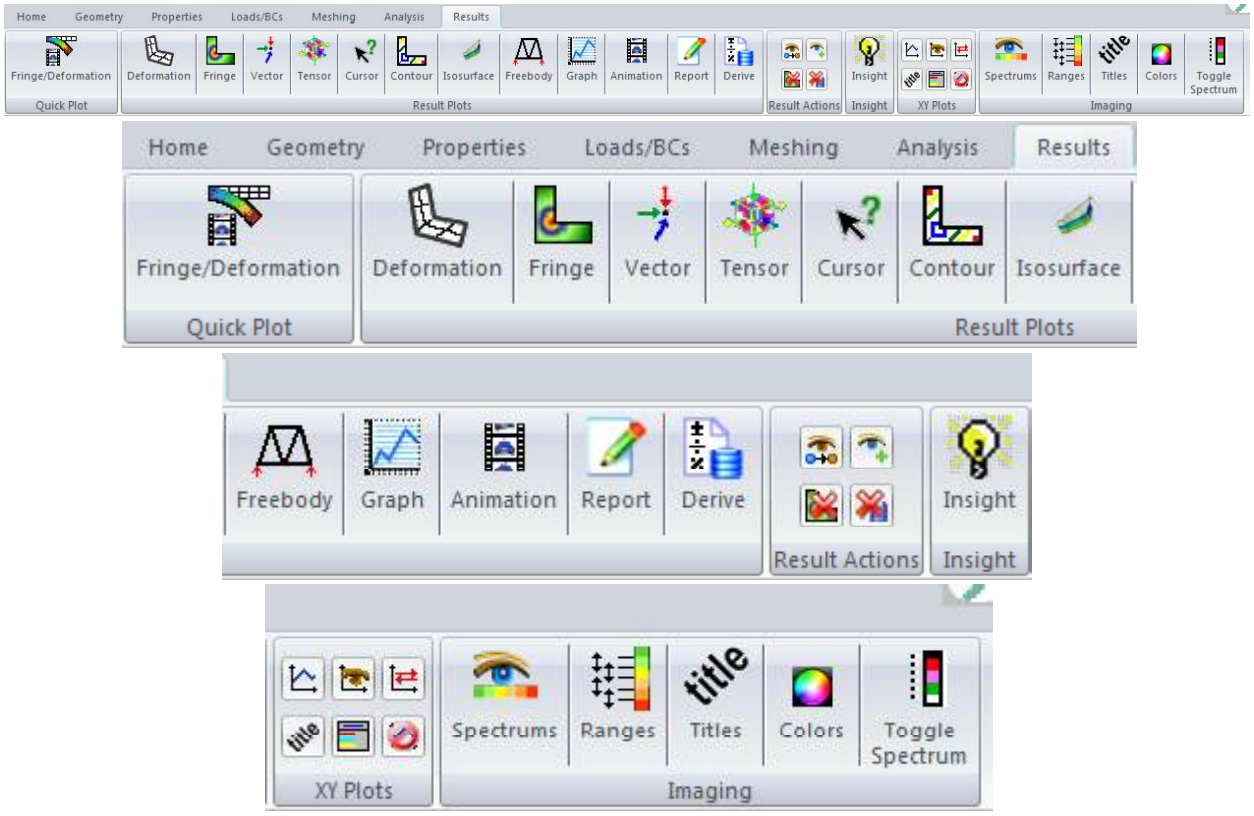


Figure 28

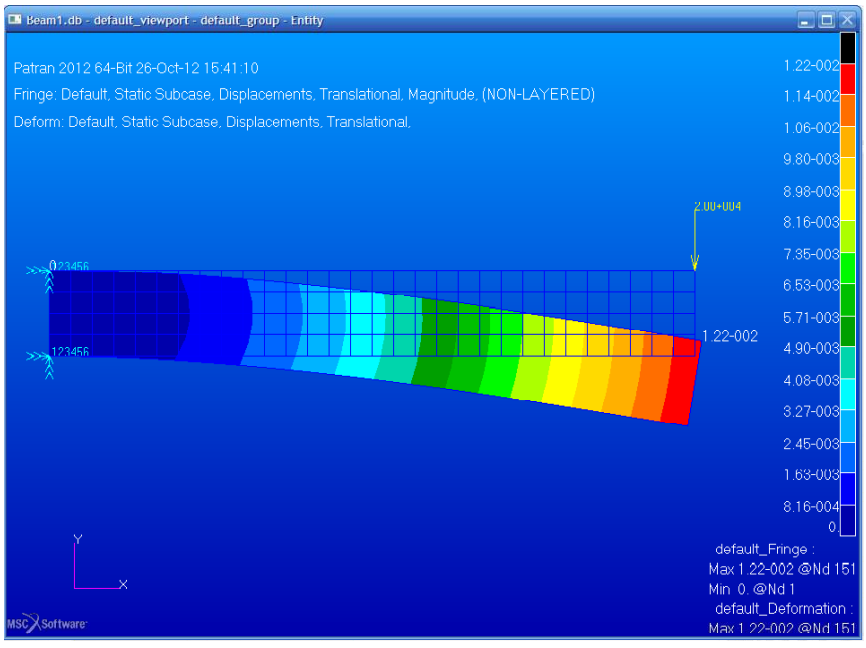


Figure 29

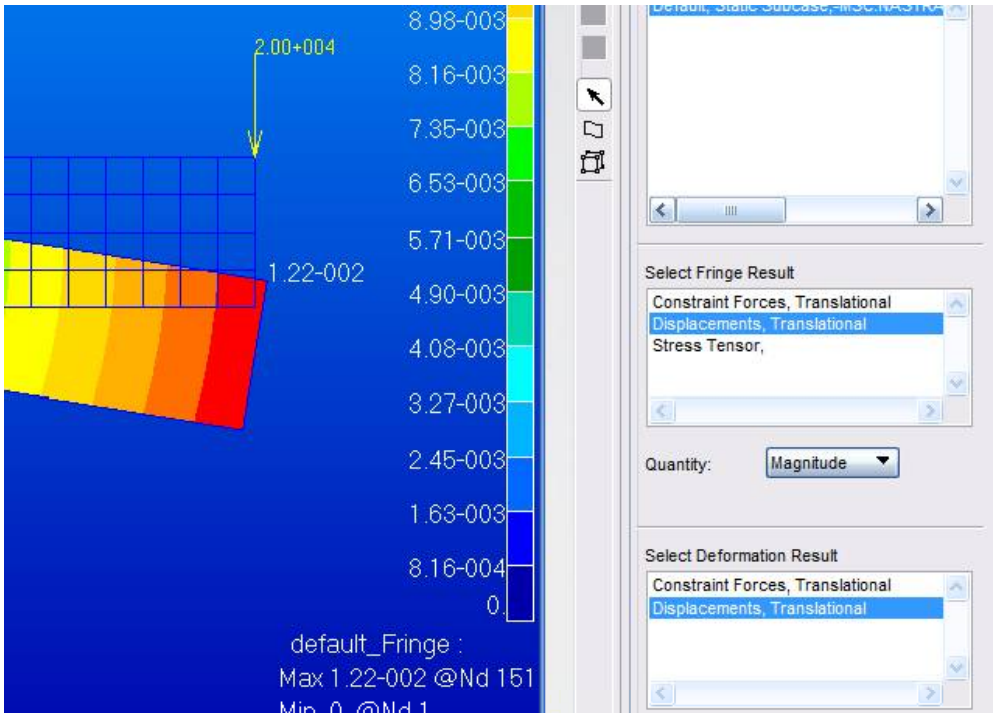


Figure 30

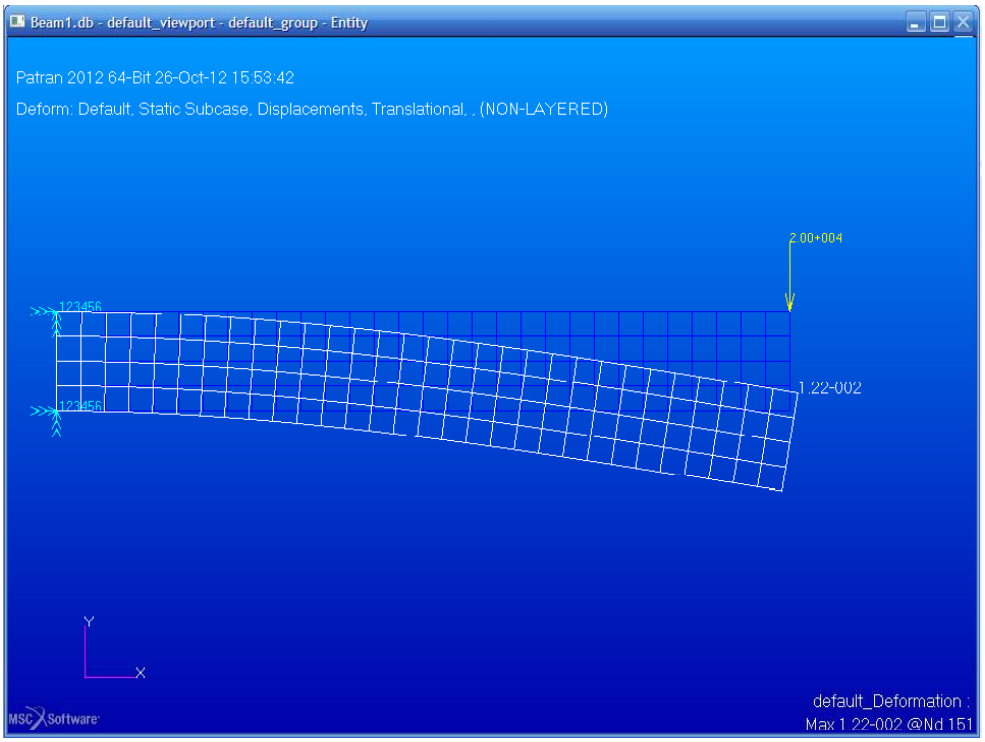
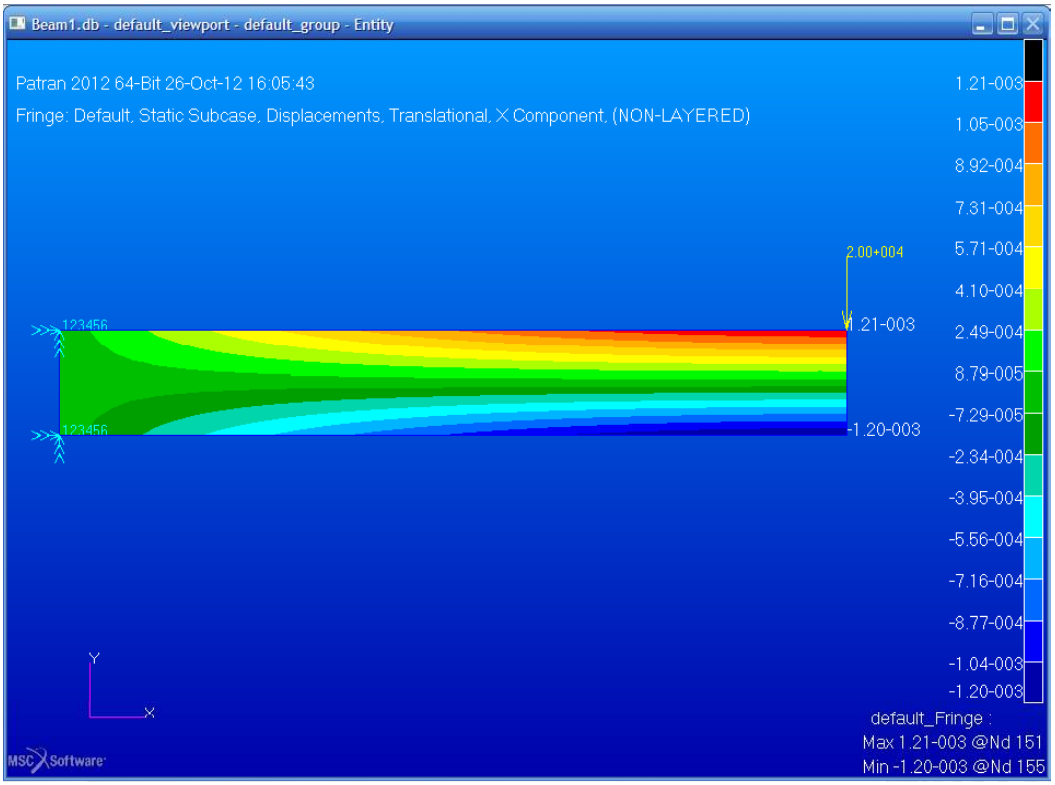


Figure 31



- Direction 4 is rotation about the x axis, 2 is rotation about the y axis, and 3 is rotation about the z axis. These rotation restraints are indicated by double arrows.

CONCLUSION

- To run the F.E. Analysis choose the Analysis menu.
- Then select the Analyze Entire Model option.
- Select Analyze, Entire Model, Full Run.
- Click on Solution Type to make the Output File selection for graphical results presentation.

To run the analysis click on Apply which is at the end of the menu list.A DOS window will appear on the screen giving NASTRAN job information.

Use the results menu to examine the results:

- Quick Plot gives a simple tool to plot deflected shape and fringe information.
- Results Plots for Deformation, Fringes, Contours, Vector fields, Tensor fields are sophisticated tools.
- Cursor gives values for particular results at specified positions.
- Select Quick Plot.
- Select the results case. (There will be more than one if the job has run more than once or if it contains more than one loadcase).
- Select the kind of information to show using coloured fringes, e.g., Displacements Translational.

- Select the kind of information to show as a deformation: Displacements Translational. Constraint Forces is not a meaningful plot.

When a Fringe figure is selected as well as a deflection the quantity plotted as a fringe is shown as a contour plot superimposed on the deflected shape. Here the displacement is being plotted $\sqrt{(u^2 + v^2 + w^2)}$, where (u, v, w) is the displacement vector.

$\sqrt{(u^2 + v^2 + w^2)} = 5.71$ mm

The interface between two colours has the stated contour value. Maximum and minimum values are given on the plot.

- Maximum deflection is given as 0.0122. As the data and properties have been specified in SI units this result is in SI units, i.e., in metres and is 0.0122 m or 12.2 mm.
- Beam theory result is a deflection of 12.05 mm on beam axis perpendicular to the axis. The FEA result for this is 12.16 mm which is a 0.9% difference.

In the diagram given below: X component of the deflection is being plotted as a fringe.

ACKNOWLEDGMENT

I would like to dedicate this Research work to my Father Late Ram Sewak Mishra and Mother Kanak Lata Mishra.

REFERENCES

1. http://www.mscsoftware.com/product/msc-nastran
2. Mishra A (2014a), "Analysis of Relation Between Friction and Wear", pp. 603-606.
3. Mishra A (2014b), "Microstructural Analysis of Wear Debris", pp. 416-421.
4. Mishra A (2014c), "The Generation of Mechanically Mixed Layers (MMLS) During Sliding Contact, pp. 578-582.
5. Mishra A (2014d), "Analysis of Application of Oxide Surface as Environmental Interface", pp. 548-552.
6. Mishra A (2014e), "Effect of Hardness on Sliding Behavior of Materials", pp. 566-569.
7. Mishra A (2014f), "Friction, Metallic Transfer and Wear Debris of Sliding Surface", pp. 574-577.
8. Mishra A (2014g), "Influence of Oxidation on the Wear of Alloys", pp. 584-587.
9. Mishra A (2014h), "Analysis of Formation of Oxide Surfaces", pp. 553-556.

International Journal of Mechanical Engineering and Robotics Research
Hyderabad, INDIA. Ph: +91-09441351700, 09059645577
E-mail: editorijmerr@gmail.com or editor@ijmerr.com
Website: www.ijmerr.com

ISSN 2278-0149
01
9 772278 014003