Implicit Nonlinear Analysis using MSC Nastran and Patran

NAS400 Workbook

January 2015

NA*V2013.1*Z*Z*Z*SM*NAS400-WBK

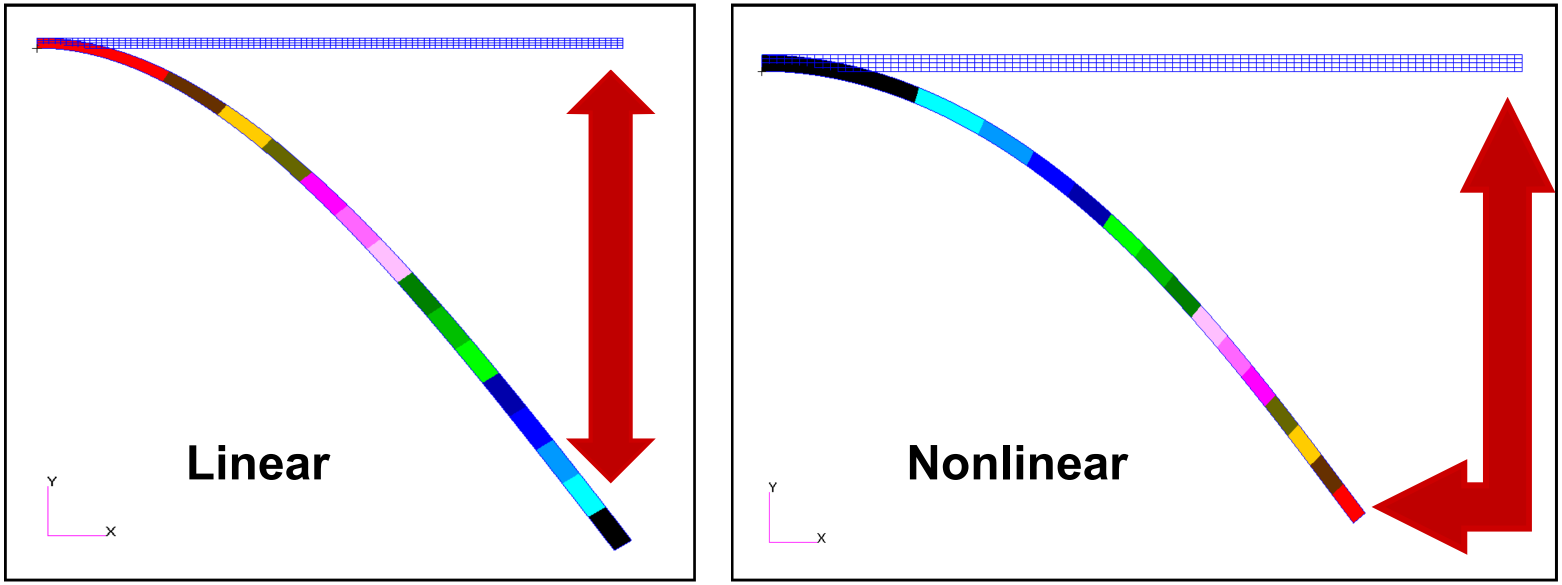
Legal Information

MSC.Software Corporation reserves the right to make changes in specifications and other information contained in this document without prior notice. The concepts, methods, and examples presented in this text are for illustrative and educational purposes only, and are not intended to be exhaustive or to apply to any particular engineering problem or design. MSC.Software Corporation assumes no liability or responsibility to any person or company for direct or indirect damages resulting from the use of any information contained herein.

CONTENTS

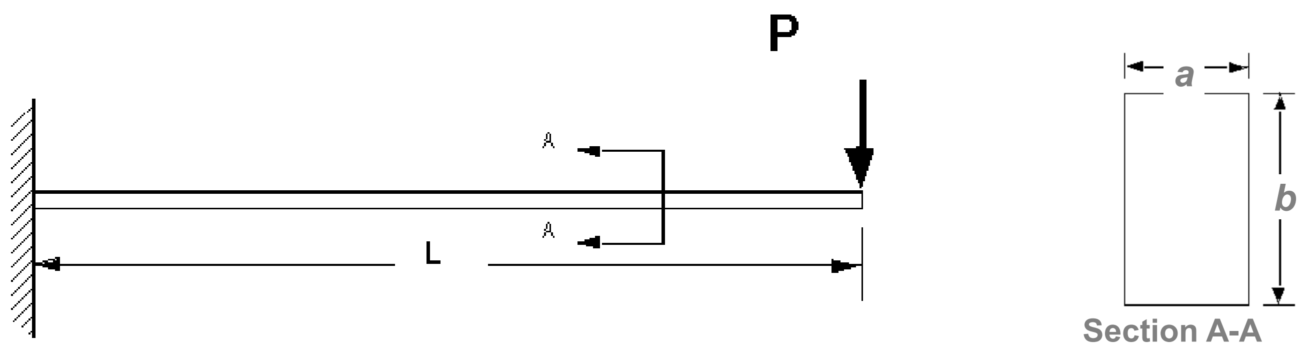
Workshop

WORKSHOP 1
LINEAR AND NONLINEAR ANALYSIS OF A CANTILEVER BEAM



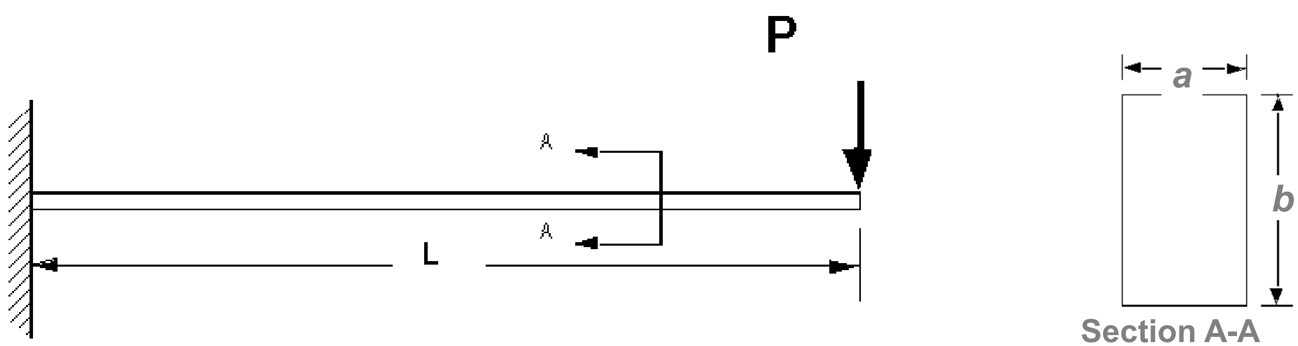
- **Workshop Objectives**
  - Small vs. large displacement analysis.
  - Linear elastic theory.
- **Software Version**
  - Patran 2013
  - MSC Nastran 2013.1
- **Files Required**
  - No supporting file is required.
- **Problem Description**
  - The simulation is to use 3D solid finite elements. The elements are uniformly spaced along the length of the beam (i.e. a mesh 100 elements wide and four elements deep).

- **Problem Description (Cont.)**
  - In this exercise, a cantilever beam is subjected to a static load. The beam is initially analyzed using small deformation theory. However, after reviewing the results, it becomes apparent that small deformation theory is not appropriate for this problem. Subsequently, a large deformation analysis is performed and its results are compared to the small deformation analysis.



(Data in next page)

• Problem Description (Cont.)



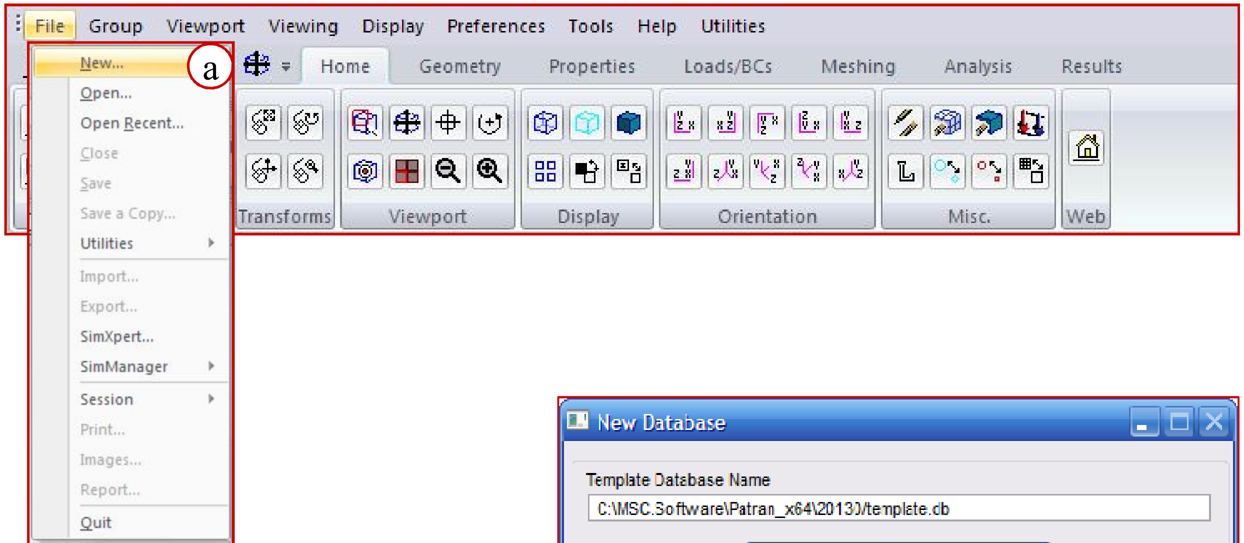
Length, L	100.0 in	2.54 m
a	1.0 in	25.4 mm
b	2.0 in	50.8 mm
Young's Modulus	30.0 x 10^6 lb/in^3	207 GPa
Poisson's Ratio	0.3	0.3
P	6000 lb	27200 N

- **Suggested Steps**
  1. Create a new database called **tip_load**.
  2. Create simple cantilever beam solid using XYZ vector coordinates **<100, 2, 1>**
  3. Apply a mesh seed to the model.
     - Select number of elements as **100** along the curve oriented in the **X** direction.
     - Select number of elements as **4** along the curve oriented in the **Y** direction.
  4. Mesh the solid using an isomesh with hex8 topology and a global edge length of **1.0**.
  5. Create an isotropic material with an elastic modulus of **30e6**, Poisson's ratio of **0.30**, and a density of **.00074**.
  6. Create 3D solid physical property for entire model and apply isotropic material.
  7. Fix translations in all directions along the left surface to create cantilever.
  8. Apply load of **1500** in the **-Y** direction to points on right end.
  9. Create and run a linear static analysis.
  10. Attach the **.op2** when the analysis is completed.

- **Suggested Steps**

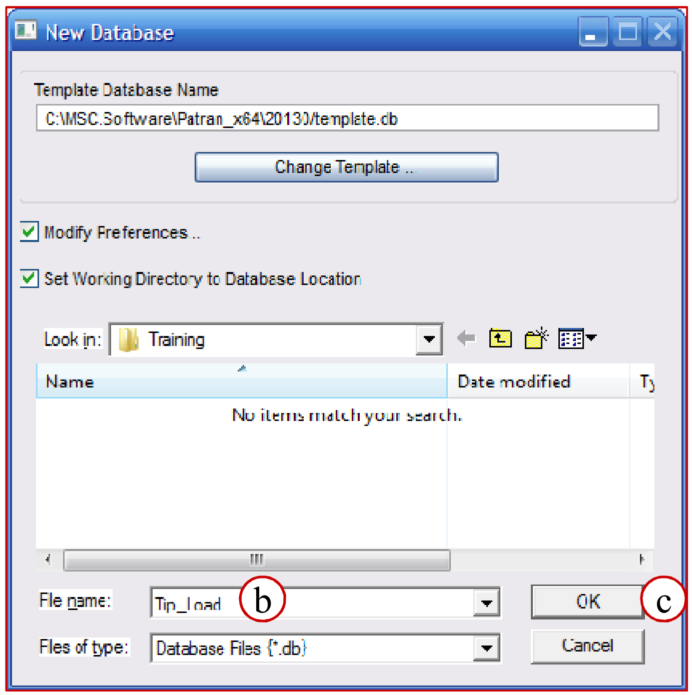
  11. Create a quickplot with fringe and deformation of displacement results.
  12. Create and run an implicit nonlinear analysis.
      - In Solution Parameters select **SOL400** Run.
      - Select .op2 as results output format.
      - Create a my_nonlinear subcase.
        - Change the increment type to Adaptive stepping
      - Run analysis with the my_nonlinear subcase.
  13. Attach **.op2** result file.
  14. Create a quickplot with fringe and deformation of the **y** component of the displacement results.
  15. Compare the linear and nonlinear results.
  16. Try to improve results.

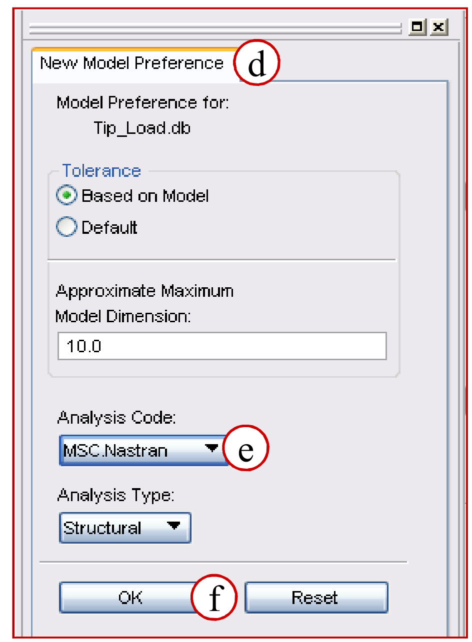
Step 1. Create New Database



To make this exercise as easy as possible to follow, the location on the form where the operation will be carried out will be pointed out by a red-circled letter that corresponds to the exercise step letter.

Where possible, the forms will be shown as they will appear after the instructions have been correctly followed.



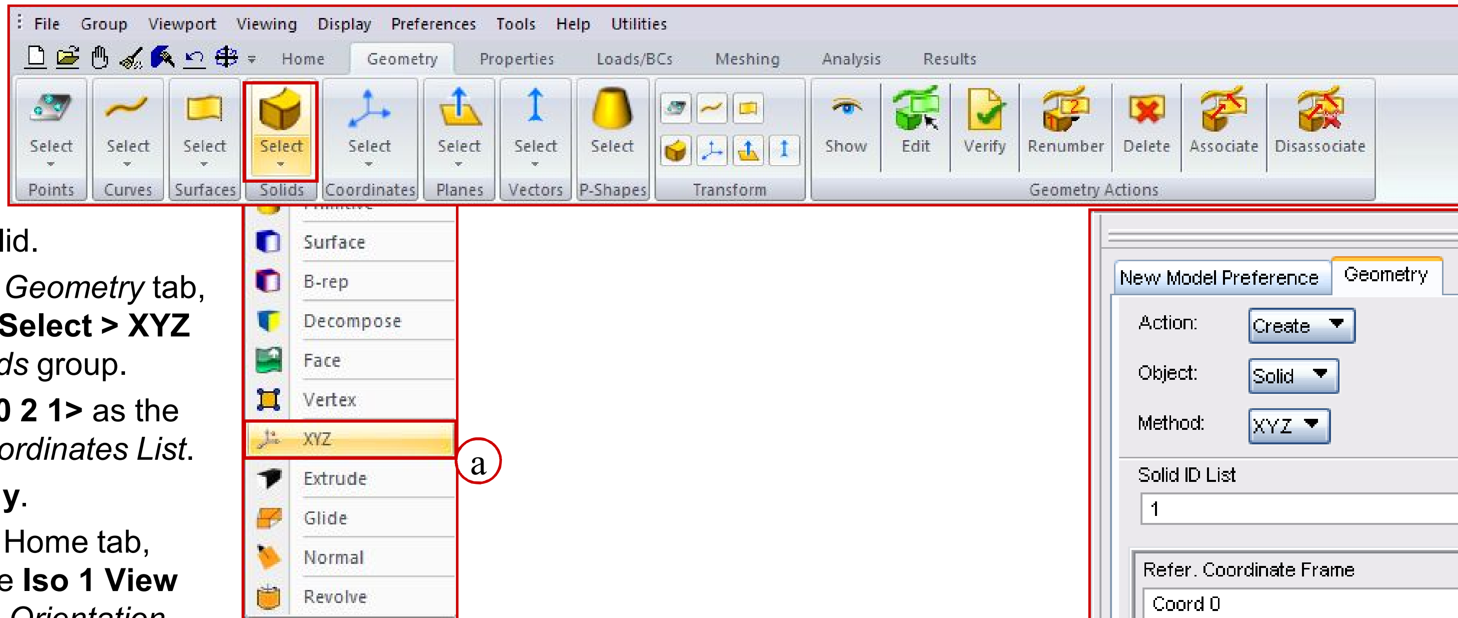


Create a new database. Name it Tip_Load.db.

a. Pull down *File* > **New**.
b. Enter **Tip_Load** as the *File name*.
c. Click **OK**.
d. Wait a few seconds until the *New Model Preference* opens.
e. Confirm **MSC Nastran** as the *Analysis Code*.
f. Click **OK**.

MSC Software

Step 2. Create Cantilever Beam



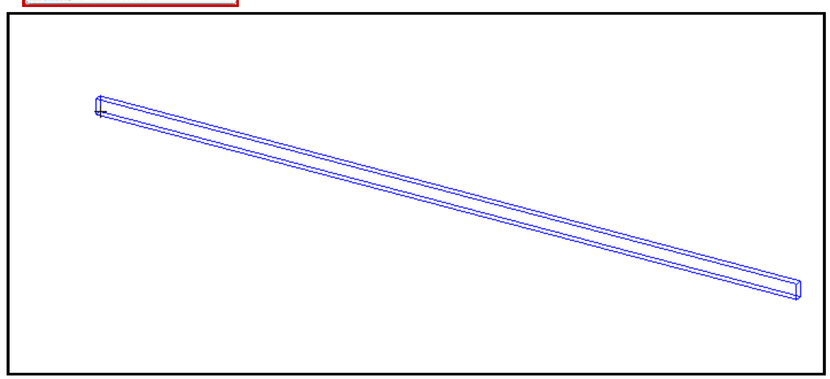
Create the solid.

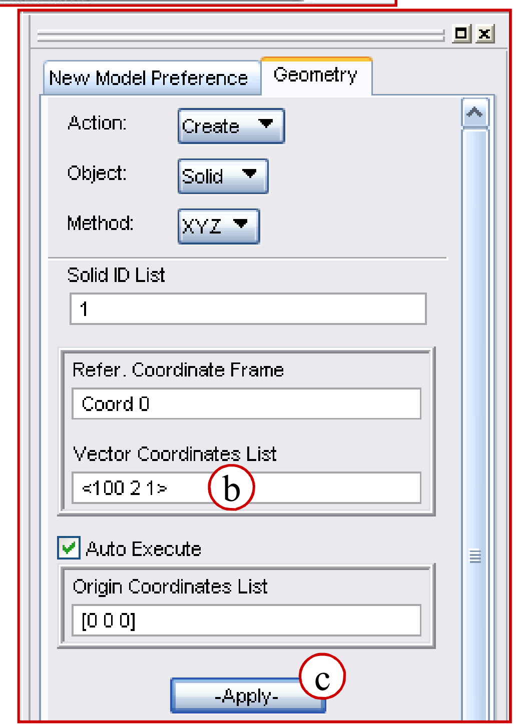
a. Under the *Geometry* tab, pull down **Select > XYZ** in the *Solids* group.

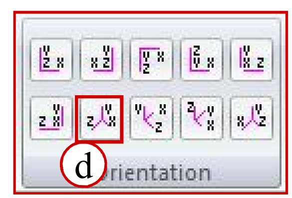
b. Enter **<100 2 1>** as the *Vector Coordinates List*.

c. Click **Apply**.

d. Under the Home tab, click on the **Iso 1 View** icon in the *Orientation* group.

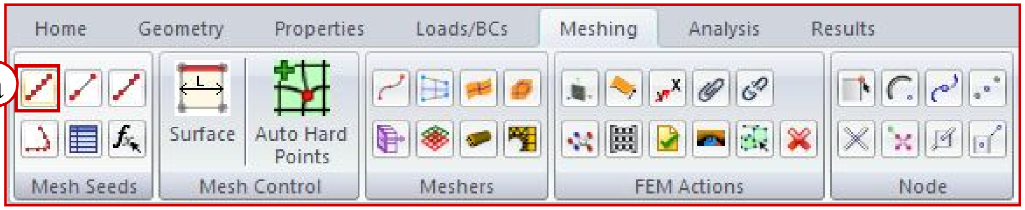


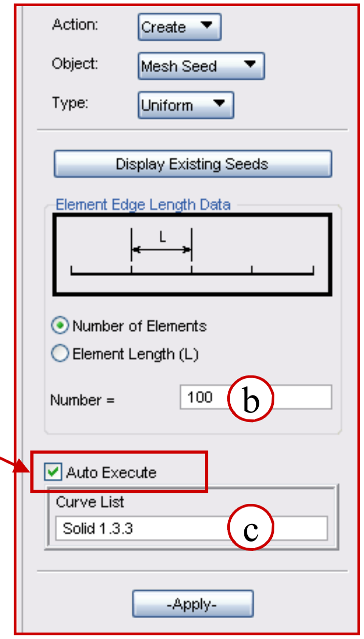


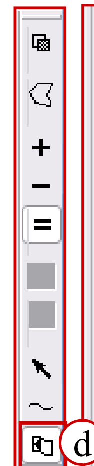


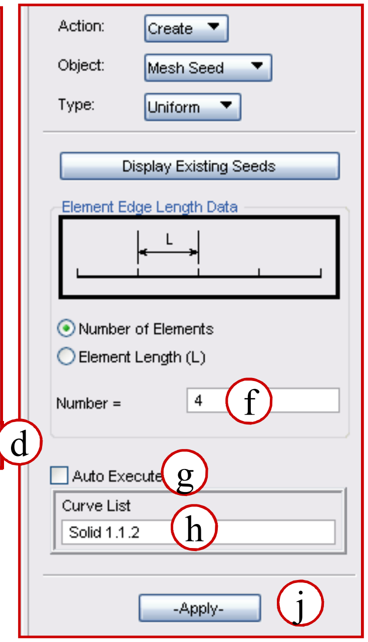
MSC Software

Step 3. Apply Mesh Seeds





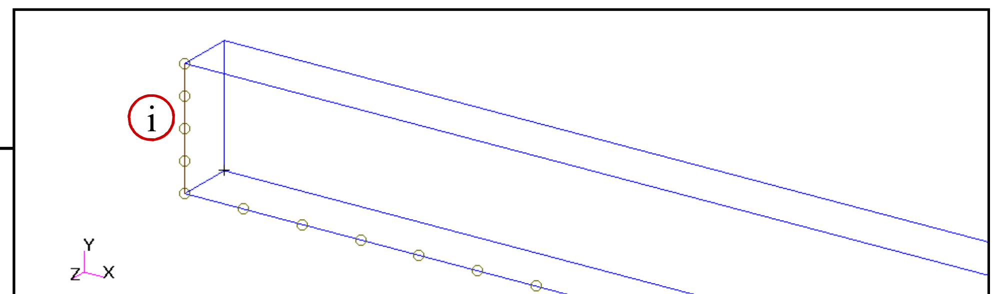


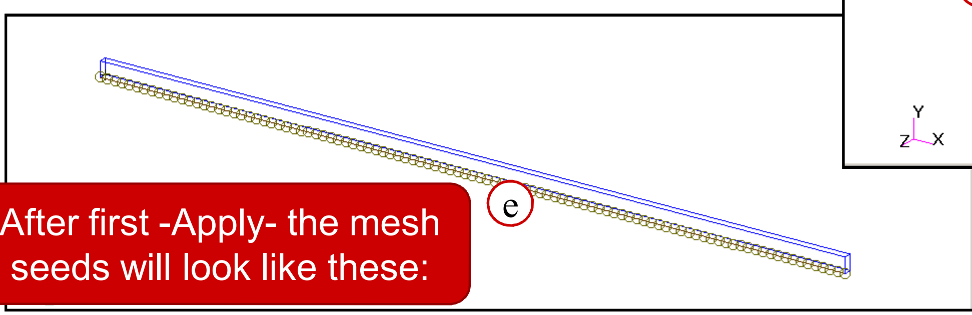


Create the finite element mesh seeds.

a. Under the *Meshing* tab, click **Uniform** in the *Mesh Seeds* group.
b. Enter **100** as the *Number*.
c. Click in the **Curve List** panel.
d. Select **Edge** from the *Picking Filters* tool bar.
e. Pick one of the bottom edges on the solid.
f. Enter **4** as the *Number*.
g. Uncheck *Auto Execute*.
h. Click in the **Curve List box**
i. Pick, on the left end of the solid, a vertical edge.
j. Click **Apply**.

See note on next page

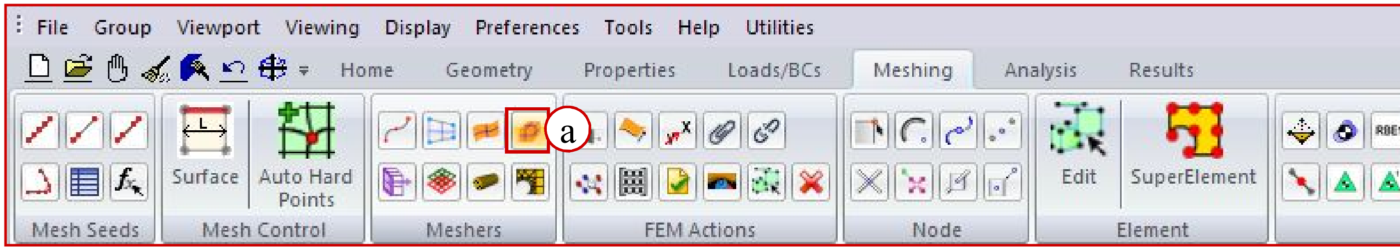


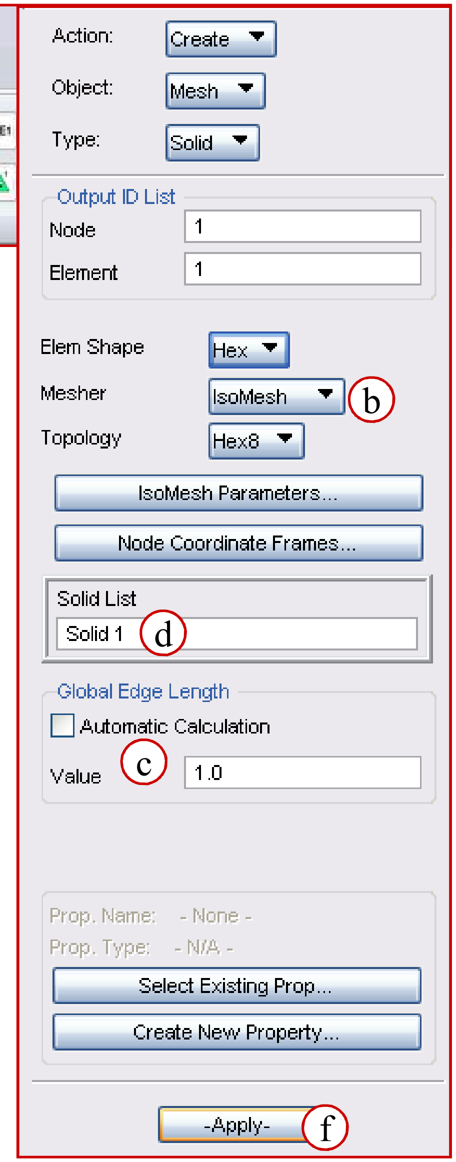


After first -Apply- the mesh seeds will look like these:

After second -Apply- the mesh seeds will look like these:

Step 4. Apply IsoMesh with Hex8 Topology



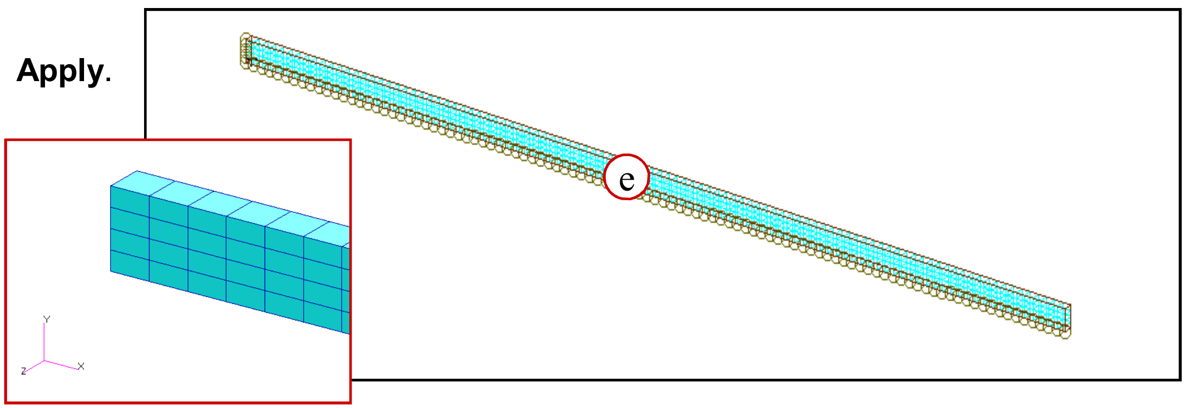


Create the finite element mesh.

a. Click **Solid** in the *Meshers* group.
b. Select **Hex** for the *Elem Shape,* **IsoMesh** for the *Mesher*, and **Hex8** for the *Topology*.
c. Enter **1.0** as the *Global Edge Length Value* and uncheck *Automatic Calculation***.**
d. Click in the **Solid List** box.
e. Select the **Solid** in the view port.
f. Click **Apply**.

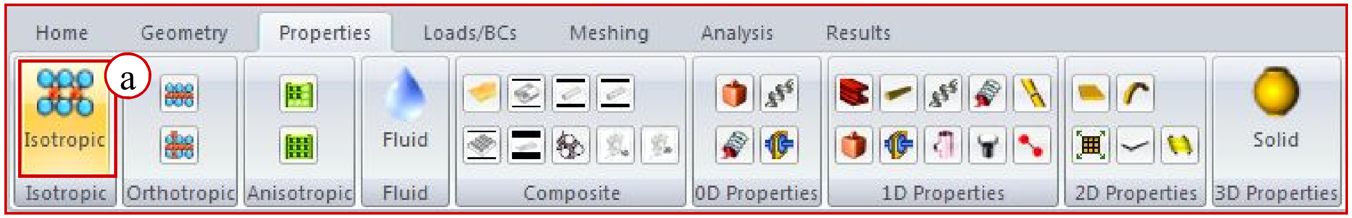
Auto-Execute Option

The Auto-Execute option will automatically apply the operation when the last field is filled in. In this case, the user can enter the number of elements for the mesh seed and then select the edge. Once the edge is selected, the operation will be applied. There is no need to click on the **-Apply-** button.



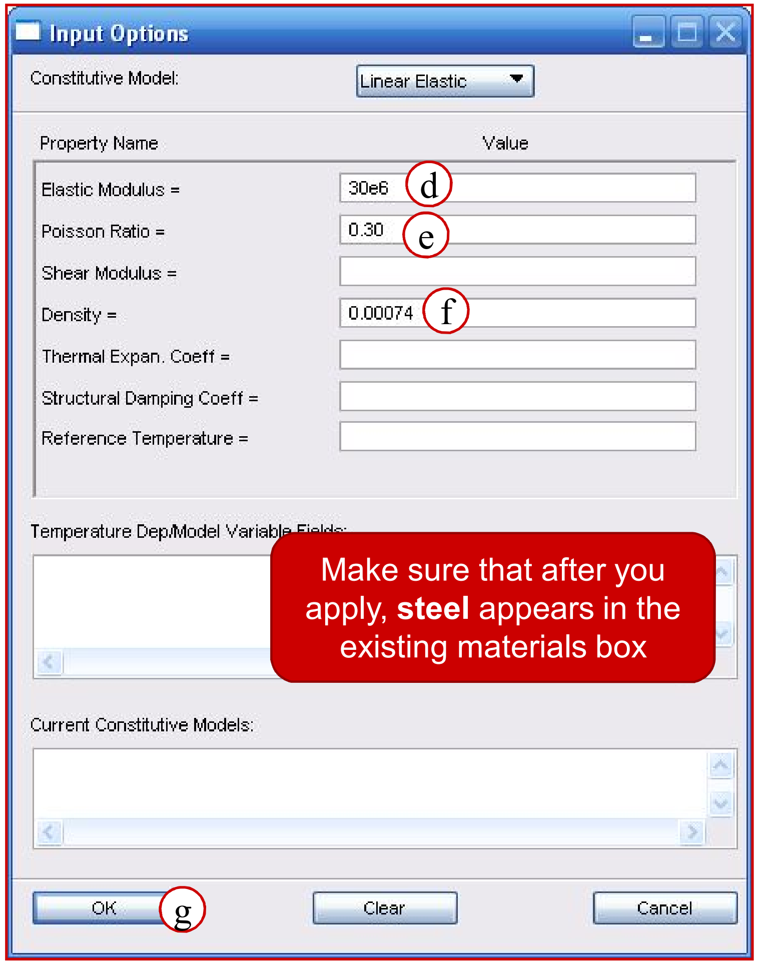
MSC Software

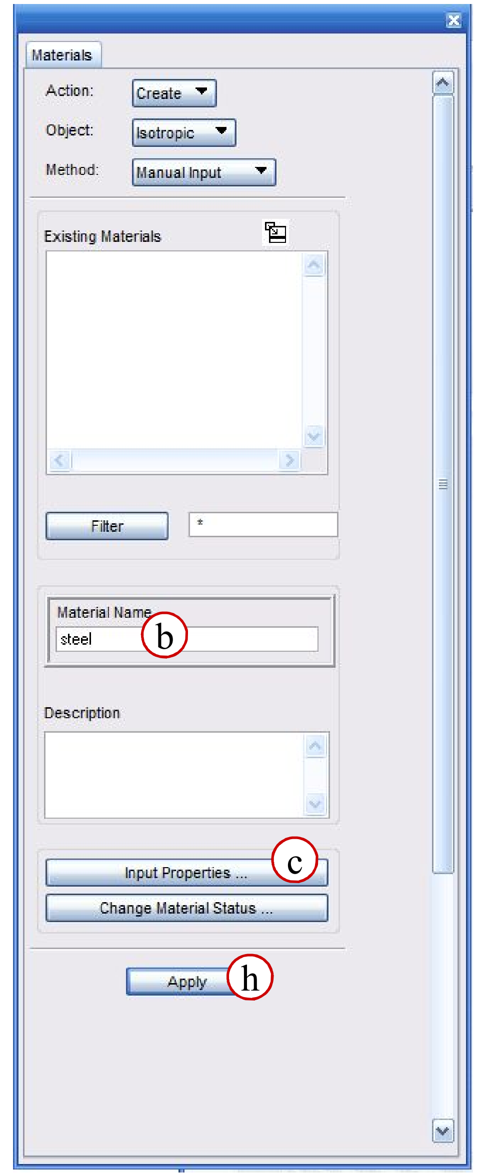
Step 5. Specify Material Properties



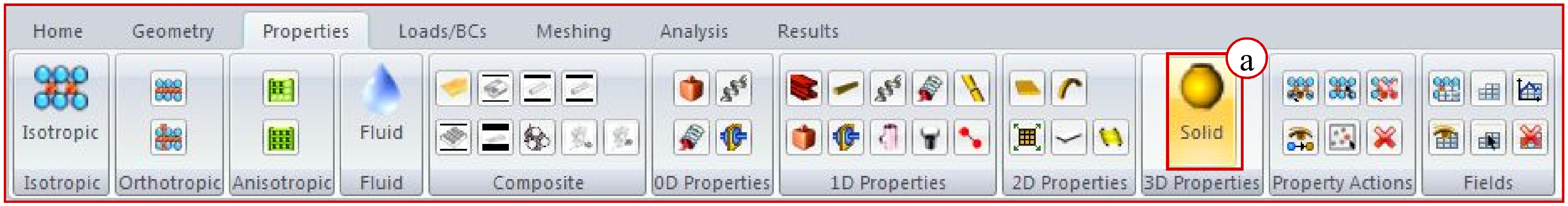
Create the material property: steel

a. Under the *Properties* tab, click **Isotropic** in the *Isotropic* group.
b. Enter **steel** as the *Material Name.*
c. Click **Input Properties**.
d. Enter **30e6** as the *Elastic Modulus.*
e. Enter **0.30** as the *Poisson Ratio.*
f. Enter **0.00074** as the *Density*.
g. Click **OK.**
h. Click **Apply**.



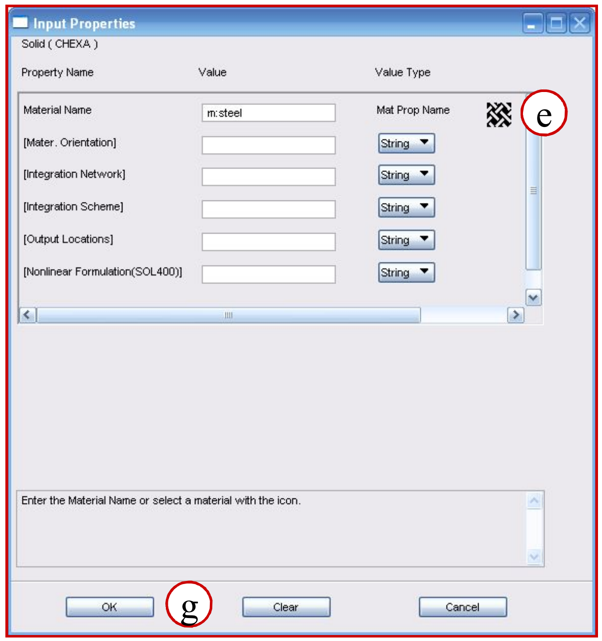


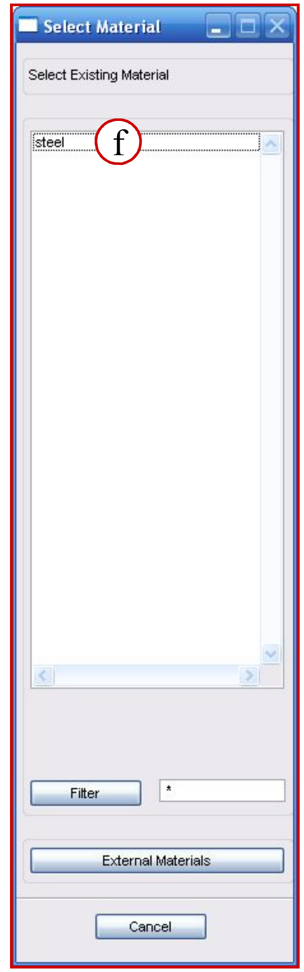
Step 6. Specify Solid Properties

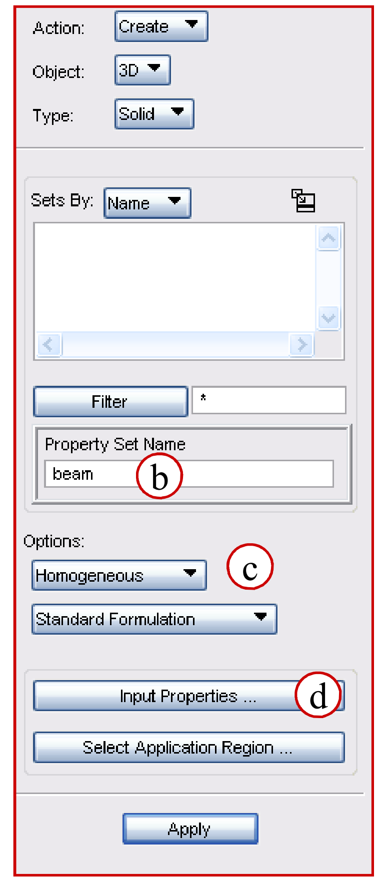


Create the properties of the beam.

a. Click **Solid** in the *3D Properties* group.
b. Enter **beam** as the *Property Set Name.*
c. Confirm **Homogeneous** / **Standard Formulation** for *Options*.
d. Click on **Input Properties**.
e. Click on the **Mat Prop Name** icon.
f. Choose **steel** from the *Select Existing Material* list.
g. Click **OK** .



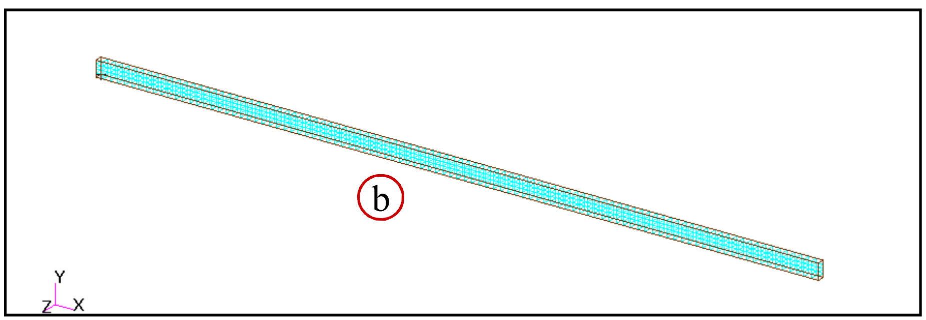


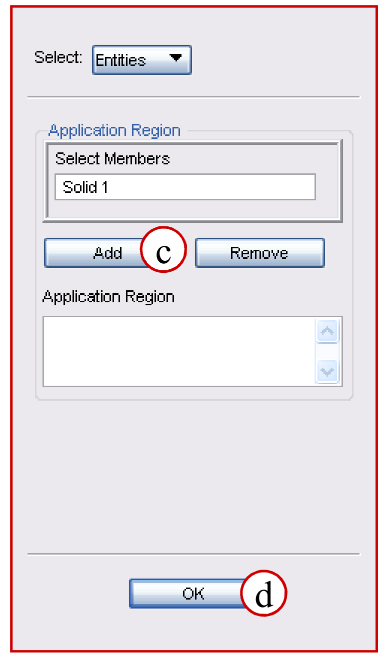


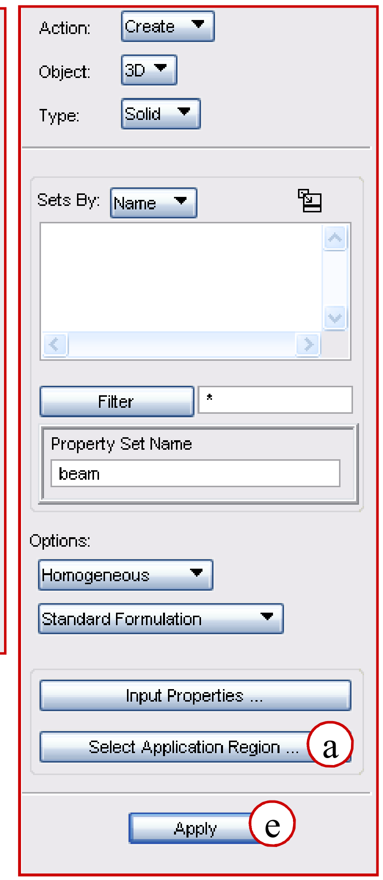
MSC Software

Step 6. Specify Solid Properties (Cont.)

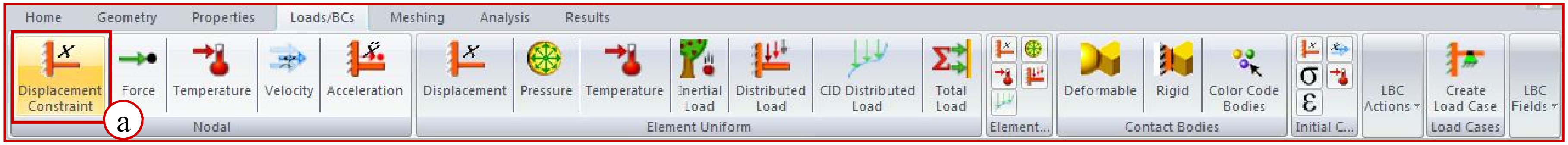
a. Click on **Select Application Region**.
b. Select the Solid from the viewport.
c. Click **Add**.
d. Click **Ok**.
e. Click **Apply**.





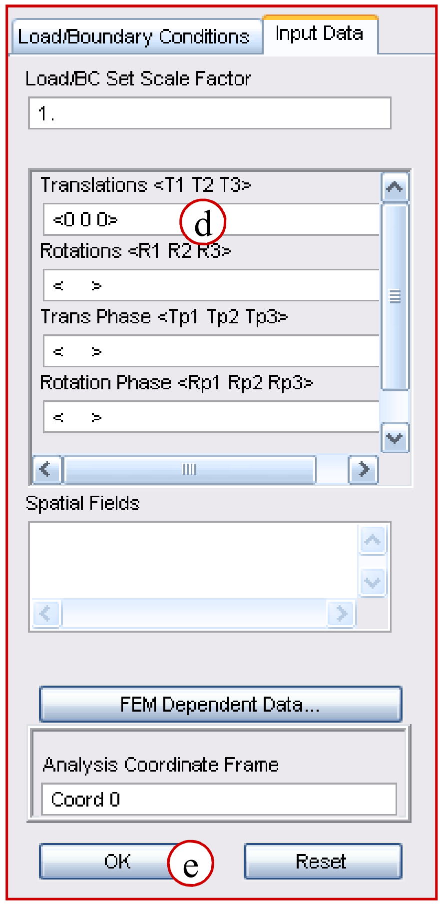


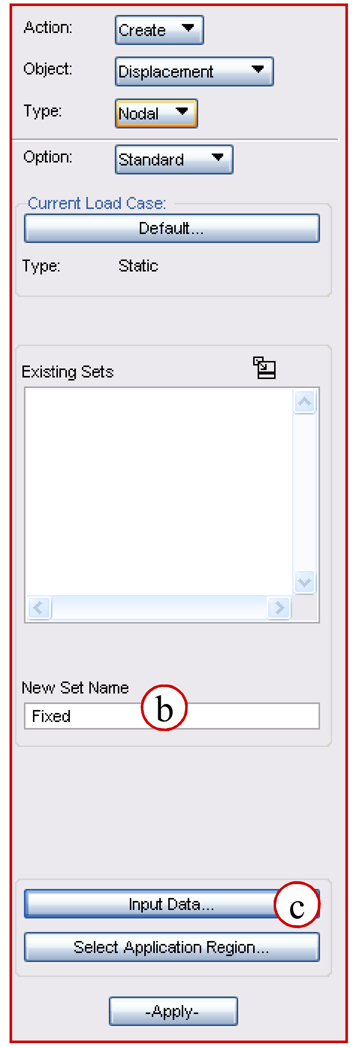
Step 7. Create Displacement Constraint



Fix the left side by creating a displacement boundary.

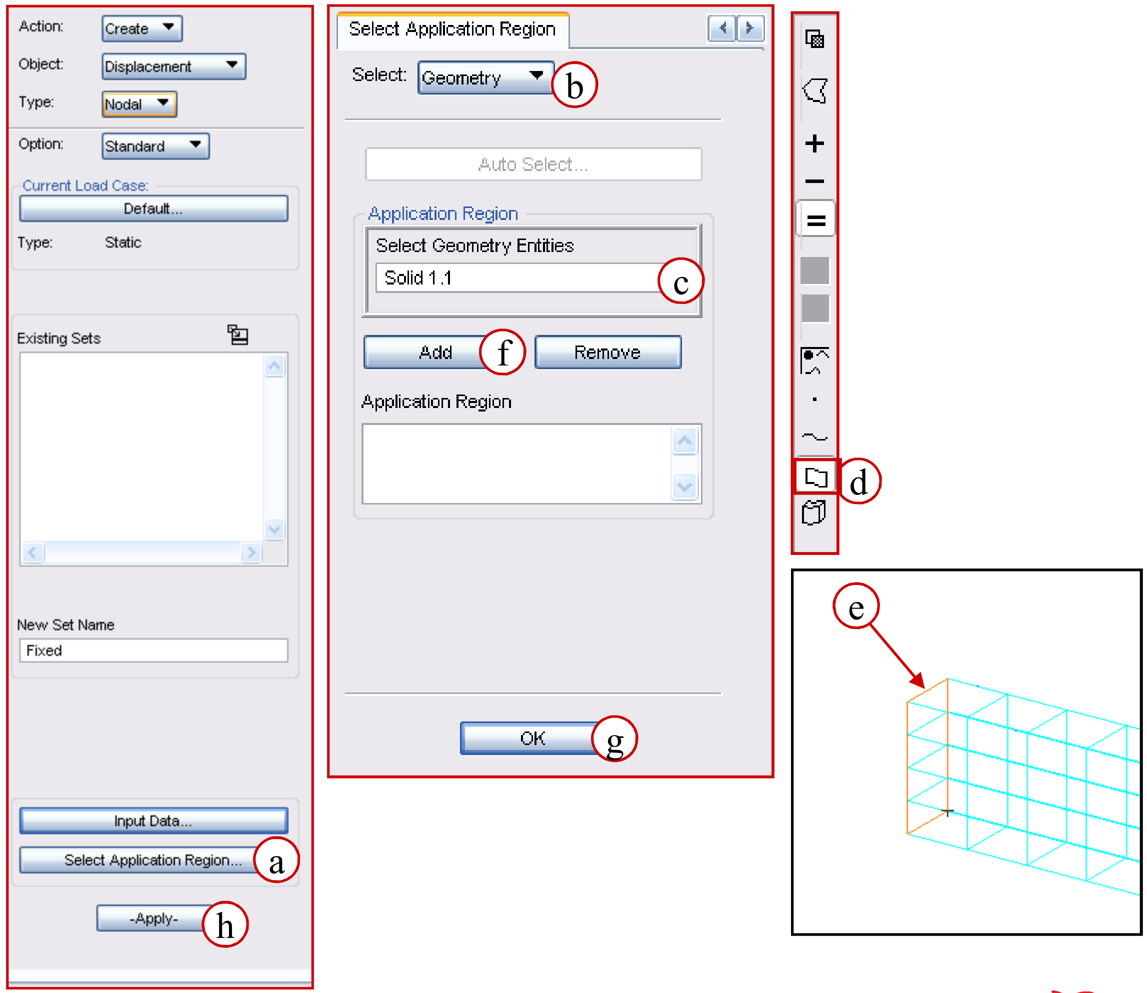
a. Click **Displacement Constraint** in the *Nodal* group.
b. Enter **Fixed** as the *New Set Name.*
c. Click on **Input Data**.
d. Enter **<0 0 0>** for the *Translations*.
e. Click **OK**.



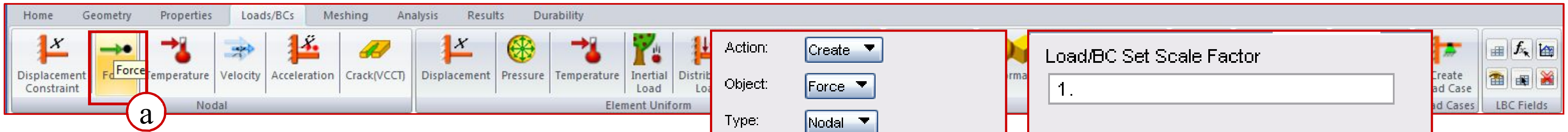


Step 7. Create Displacement Constraint (Cont.)

a. Click on **Select Application Region**.
b. Verify **Geometry** is chosen for *Select*.
c. Click in **Select Geometry Entities** box.
d. Select **Surface or Face** in the *Picking Filters* tool bar.
e. Select the Left edge of the surface.
f. Click **Add**.
g. Click **OK**.
h. Click **Apply**.



Step 8. Create and Apply Force



Create the load to be placed on the right tip.

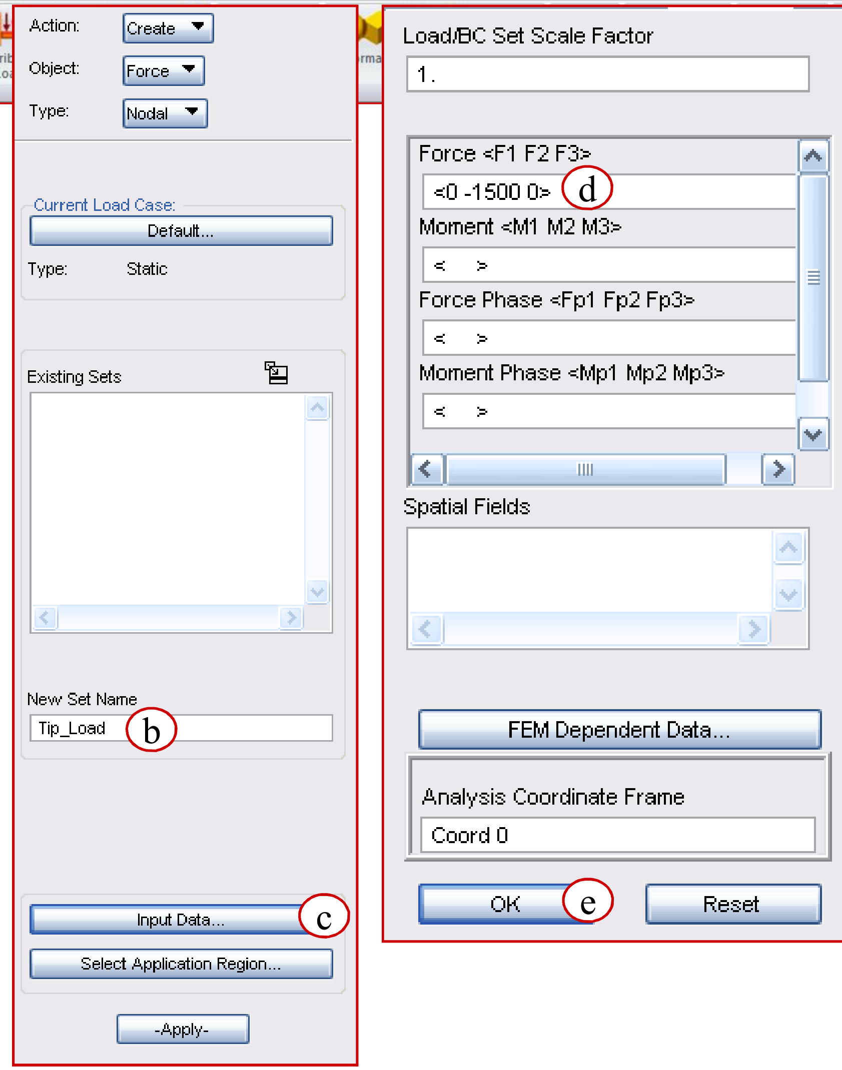
a. Click **Force** in the *Nodal* group.

b. Enter **Tip_Load** as the *New Set Name.*

c. Click on **Input Data**.

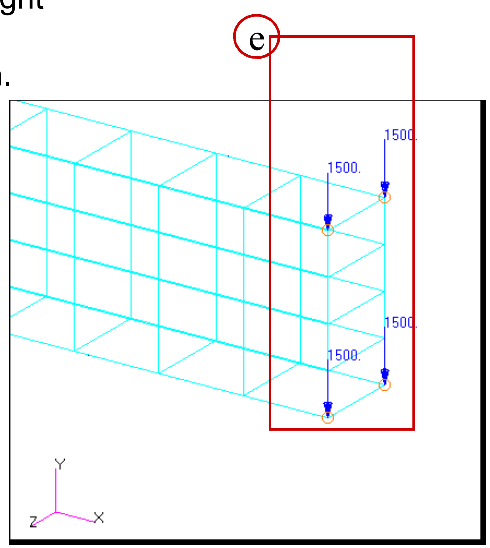
d. Enter **<0 -1500 0>** for the *Force*.

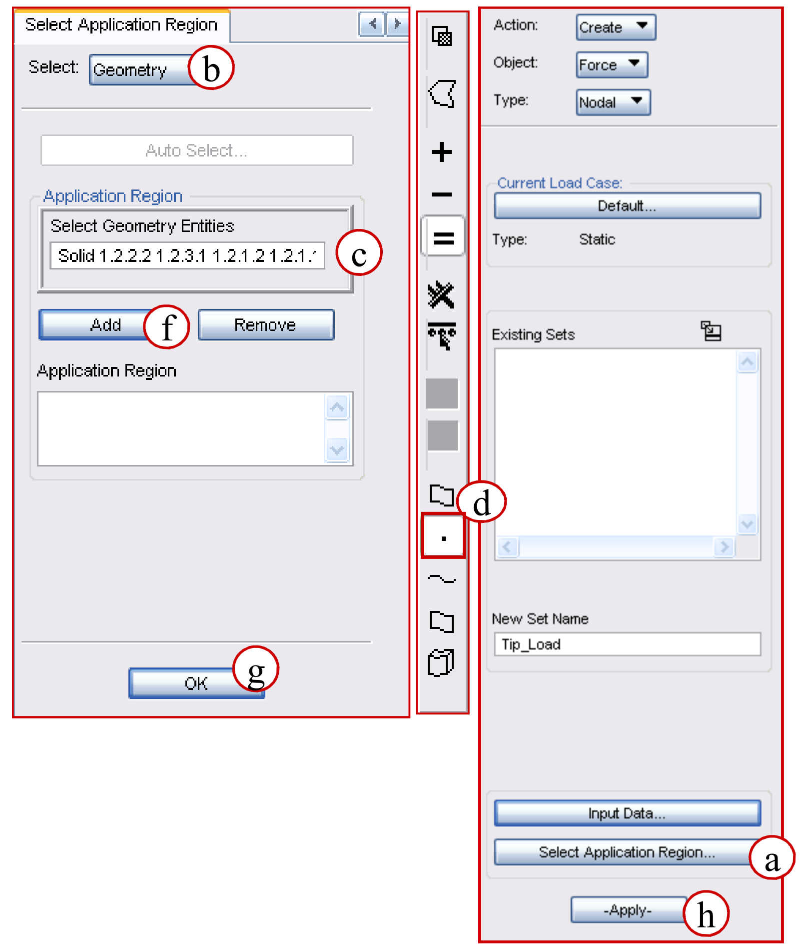
e. Click **OK.**



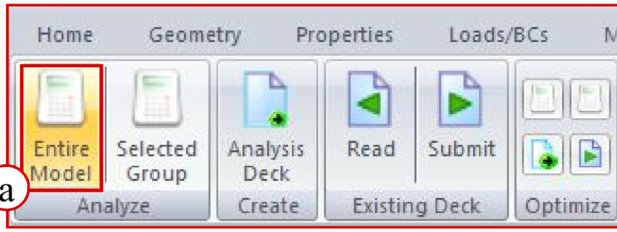
Step 8. Create and Apply Force (Cont.)

a. Click on **Select Application Region.**
b. Verify **Geometry** is chosen for *Select*.
c. Click in **Select Geometry Entities** box.
d. Select **Point or Vertex** in the *Picking Filters* tool bar.
e. Select the Vertices on the Right Edge by clicking each while holding down the shift button.
f. Click **Add**.
g. Click **OK**.
h. Click **Apply**.



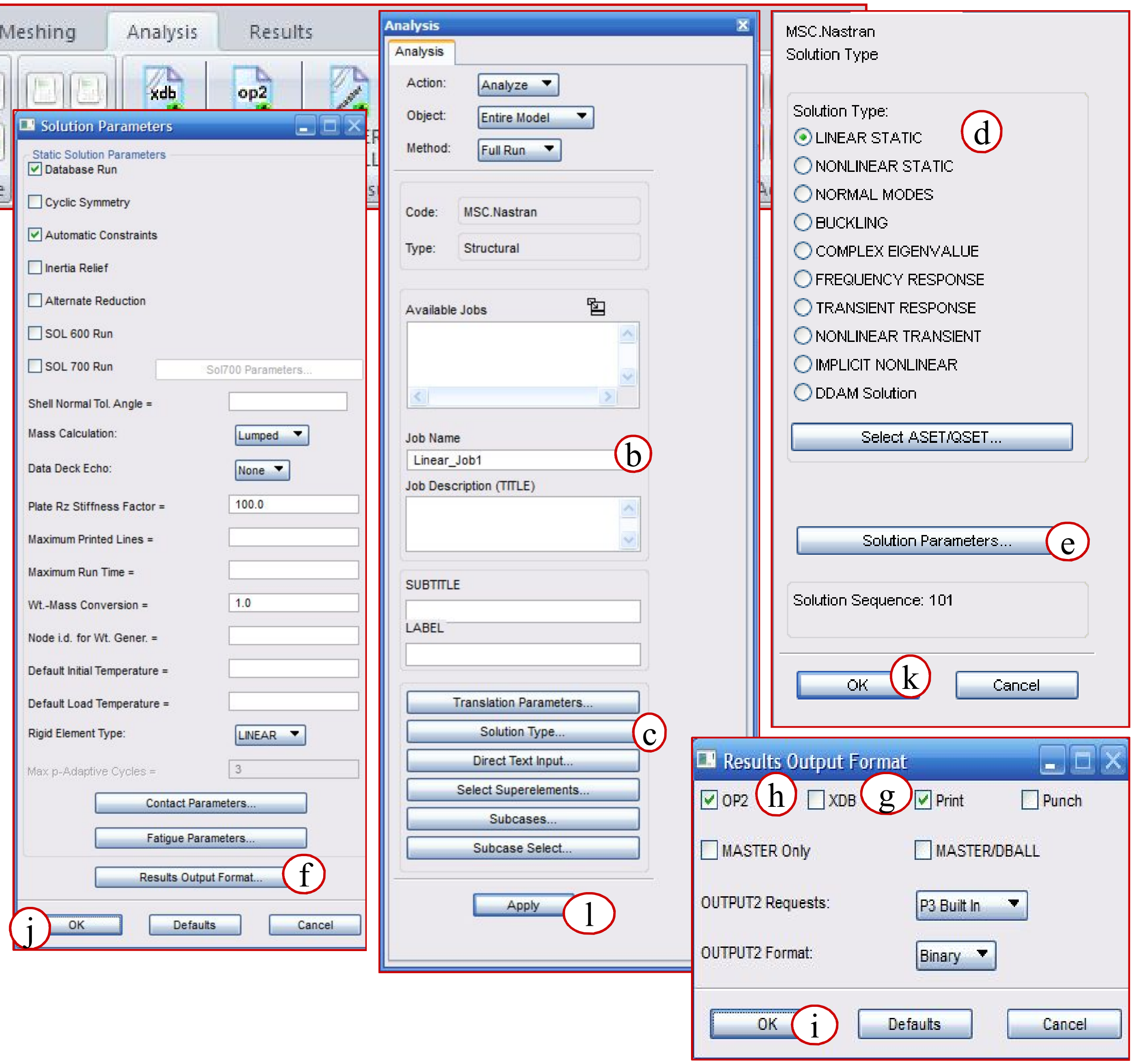


Step 9. Analysis: Create Linear Static Job



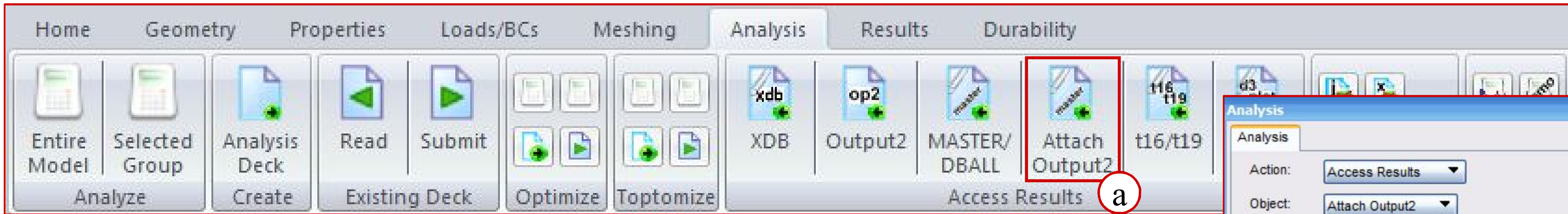
Run a linear analysis.

a. Under the *Analysis* tab, click **Entire Model** in the *Analyze* group.
b. Enter **Linear_Job1** as the *Job Name.*
c. Click on **Solution Type.**
d. Verify **Linear Static** as the *Solution Type.*
e. Click **Solution Parameters**
f. Click **Results Output Format**
g. Uncheck *XDB*
h. Check *OP2*
i. Click **OK.**
j. Click **OK.**
k. Click **OK.**
l. Click **Apply.**



MSC Software

Step 10. Analysis: Access Results



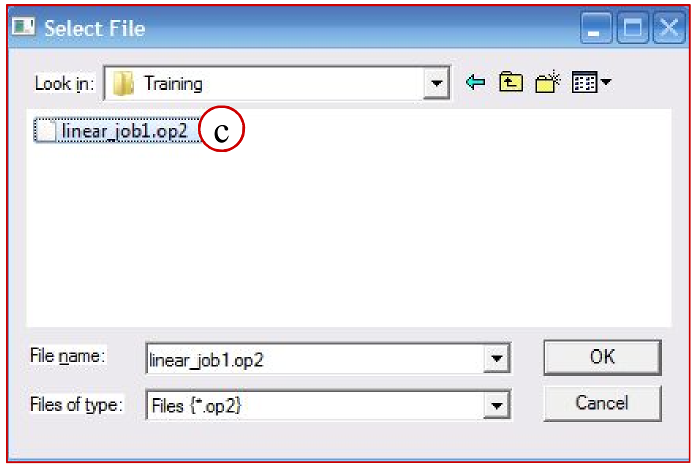
Attach the results file, when the analysis job is completed.

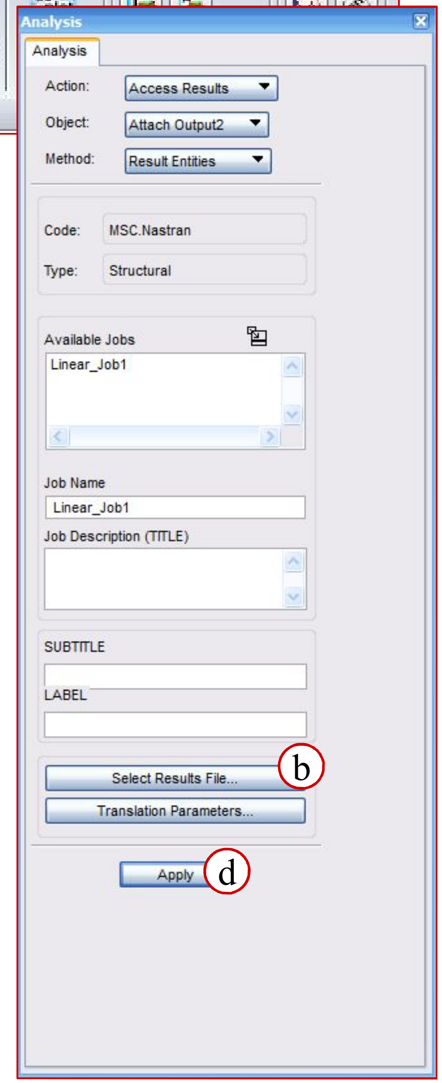
a. Under the *Analysis* tab, click **Attach Output2** in the *Access Results* group.

b. Click **Select Results File**

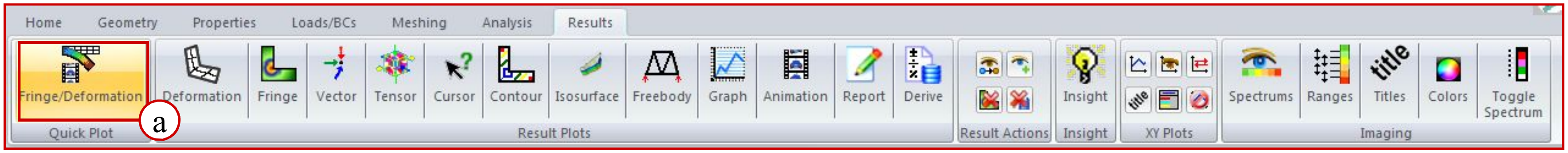
c. Select **linear_job1.op2**.

d. Click on **Apply**.



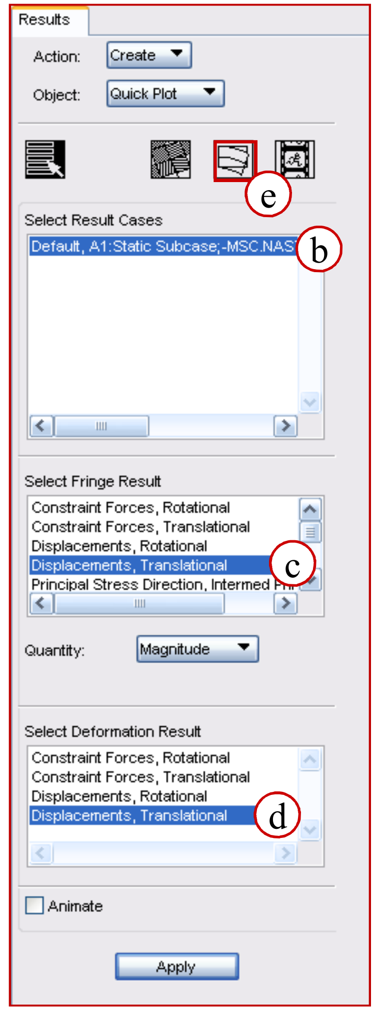


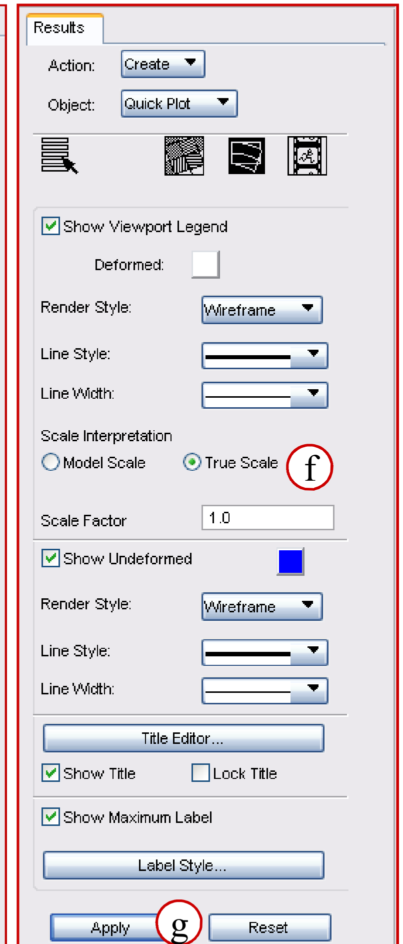
Step 11. Create Fringe Deformation Plot



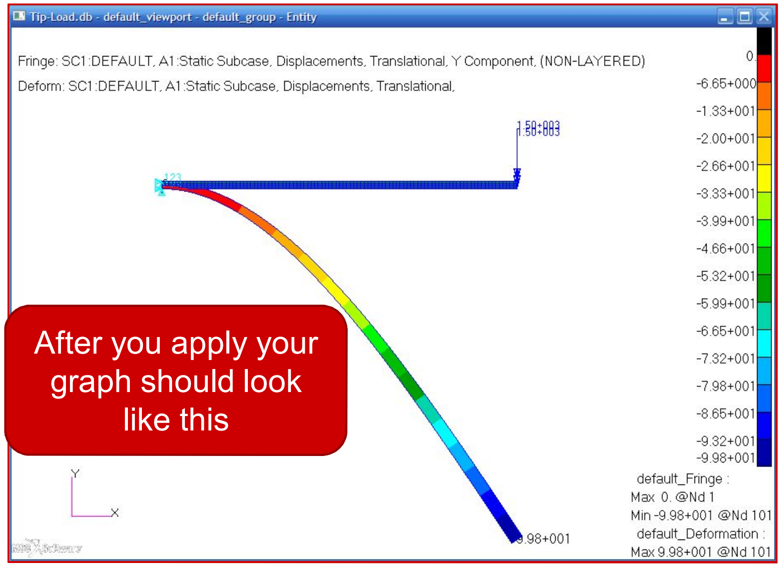
Post-Process the linear job results.

a. Under the *Results* tab, click **Fringe/Deformation** in the *Quick Plot* group.
b. Select **Default A1: Static Subcase** from the *Select Results Cases.*
c. Select **Displacements, Translational** from the *Select Fringe Result*.
d. Select **Displacements, Translational** from the *Select Deformation Result*.
e. Click on **Deform Attributes** icon.
f. Select **True Scale**.
g. Click **Apply**.
h. Under the *Home tab*, click on the **Front View** icon
i. Click **Fit View** to get a view of the entire beam.





Step 11. Create Fringe Deformation Plot (Cont.)



After you apply your graph should look like this

Linear beam theory predicts the maximum beam deflection in the Y-direction and stress to be:

$$U_{\max} = \frac{\left(PL^3\right)}{3EI} = \frac{4PL^3}{E \times ab^3}$$

$$U_{\max} = \frac{6{,}000 \times (100)^3}{30 \times 10^6 \times (1) \times (2)^3} = 100$$

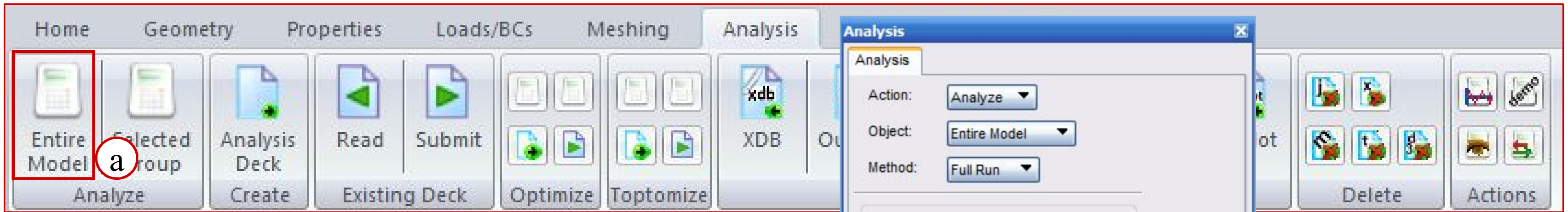
$$\sigma_{\max} = \frac{M_{\max} \times b}{I} = \frac{6PL}{a(b)^2}$$

$$\sigma_{\max} = \frac{6 \times 6{,}000 \times 100}{1 \times (2)^2} = 900{,}000$$

The maximum Y deflection of the beam can be taken directly off of the displayed spectrum/range. The largest value should correspond to a magnitude of 99.8, which is in fair agreement with our hand calculation of 100. You may still improve this by remeshing using a finer mesh. (You will be asked to do this after you have run a nonlinear analysis of present mesh.)
Linear beam theory assumes plane section remain plane and the deflection is small relative to length of the beam. As can be clearly seen by this analysis, the deflection is very large and this analysis is in violation of the underlying assumptions used for linear beam theory.

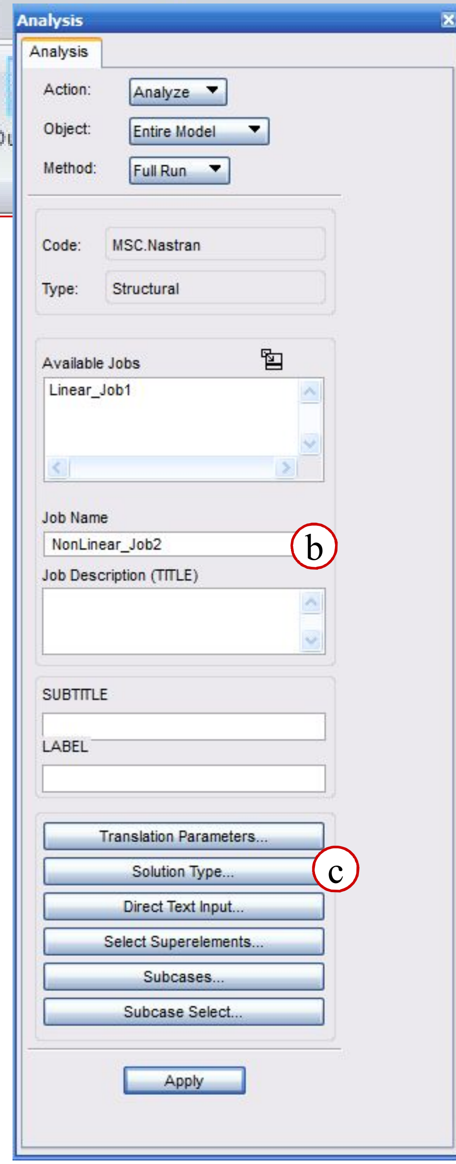
These results match the linear hand calculations and also show that the small deformation assumption is not valid and therefore, a non-linear, large deformation analysis needs to be performed. In large deformation analysis, the bending and axial stiffness are coupled. Thus, as the cantilever beam deflects, a portion of the applied load puts the beam in tension which tends to stiffen the beam in bending (i.e. “geometric stiffness”). Thus, one would expect to see a much <u>smaller</u> deformation in the <u>large deformation</u> analysis as compared to the <u>small deformation</u> analysis. To set up a large deformation analysis, one needs to change the analysis set-up and re-submit the job to MSC Nastran.

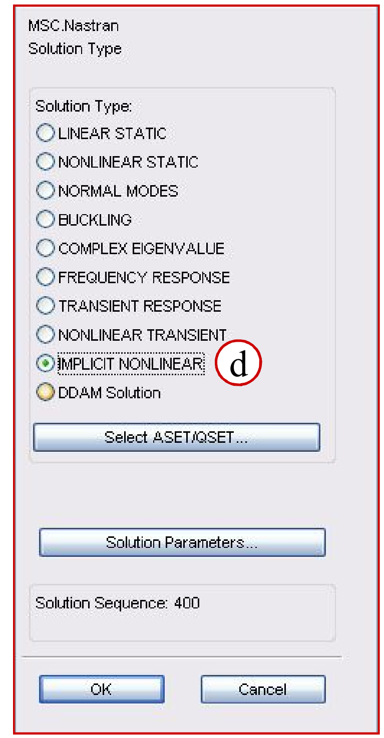
Step 12. Analysis: Create Implicit Nonlinear Job



Run Nonlinear analysis.

a. Under the *Analysis tab*, click **Entire Model** in the *Analyze* group.
b. Enter **Nonlinear_Job2** as the *Job Name*.
c. Click **Solution Type…**
d. Select **IMPLICIT NONLINEAR**.



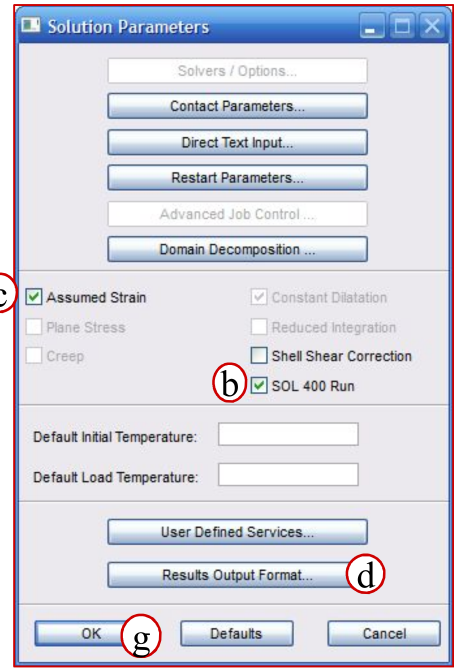


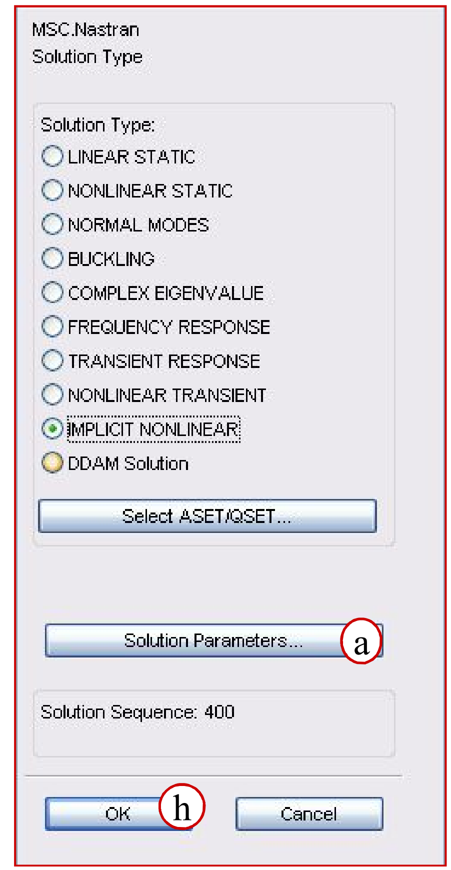
Step 12. Analysis: Create Implicit Nonlinear Job (Cont.)

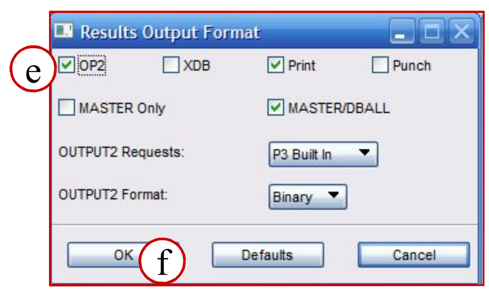
Run Nonlinear analysis.

a. Click **Solution Parameters**
b. Verify *SOL 400 Run* is checked
c. Verify *Assumed Strain* is checked
d. Click on **Results Output Format**.
e. Check *op2.*
f. Click **OK**.
g. Click **OK**.
h. Click **OK**.

The assumed strain formulation improves the bending behavior for plane stress, plane strain and solid elements (CQUAD and CHEXA)

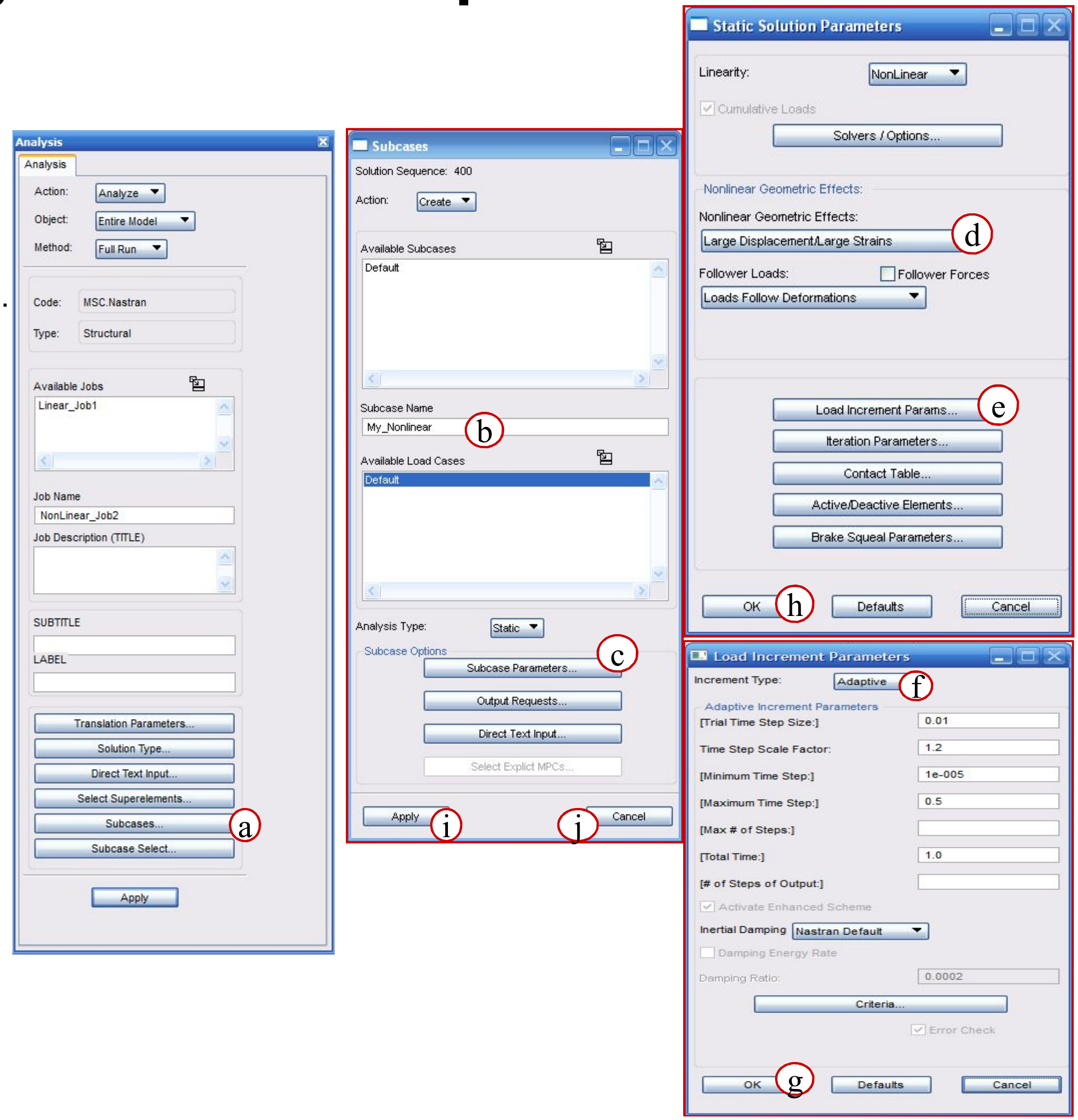






Step 12. Analysis: Create Implicit Nonlinear Job (Cont.)

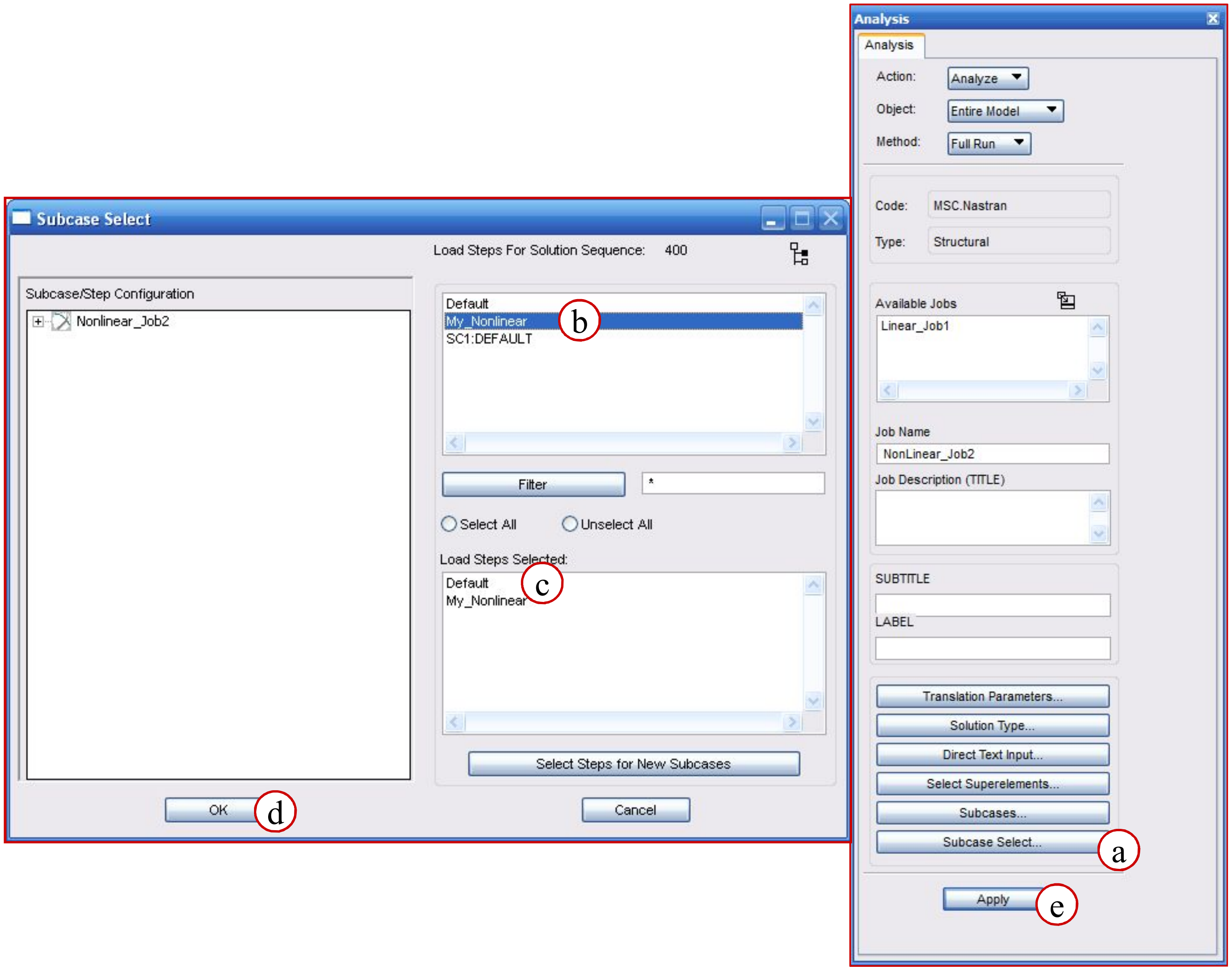
a. Click on **Subcases**.
b. Enter **My_Nonlinear** as the *Subcase Name.*
c. Click on **Subcase Parameters**.
d. Confirm that *Nonlinear Geometric Effects* is set to **Large Displacements/Large Strains**.
e. Click on **Load Increment Parameters**.
f. Pull down *Increment Type* to **Adaptive**.
g. Click **OK**.
h. Click **OK**.
i. Click **Apply**
j. Click **Cancel**



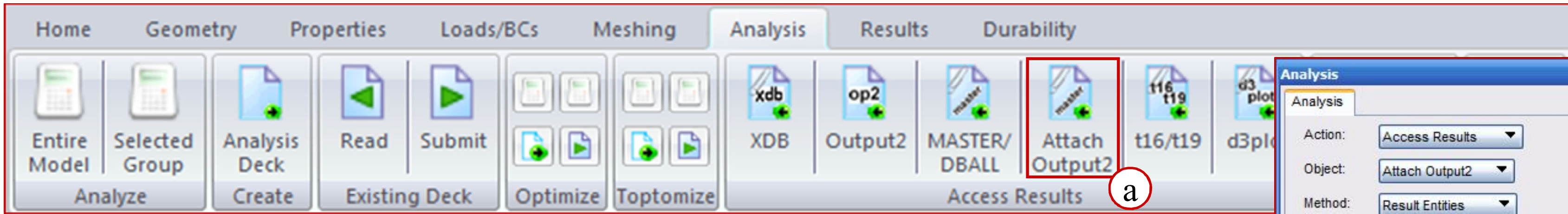
MSC Software

Step 12. Analysis: Create Implicit Nonlinear Job (Cont.)

a. Click on **Subcase Select**.
b. Deselect the **Default** subcase from *Load Steps Selected* (by clicking on it)
c. Select **My_Nonlinear** from the Existing Subcases.
d. Click **OK**.
e. Click **Apply**.

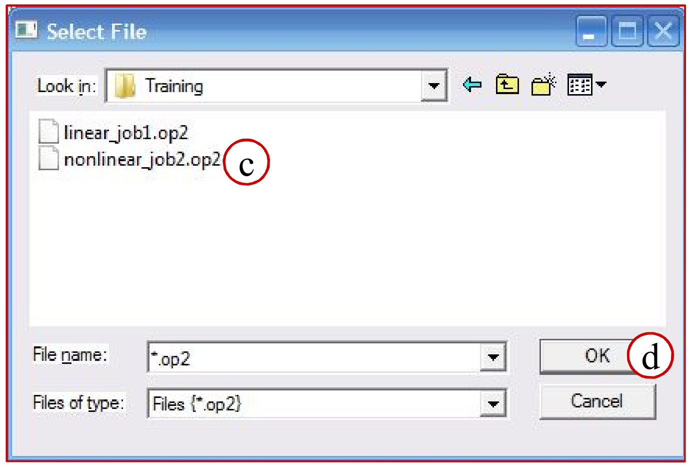


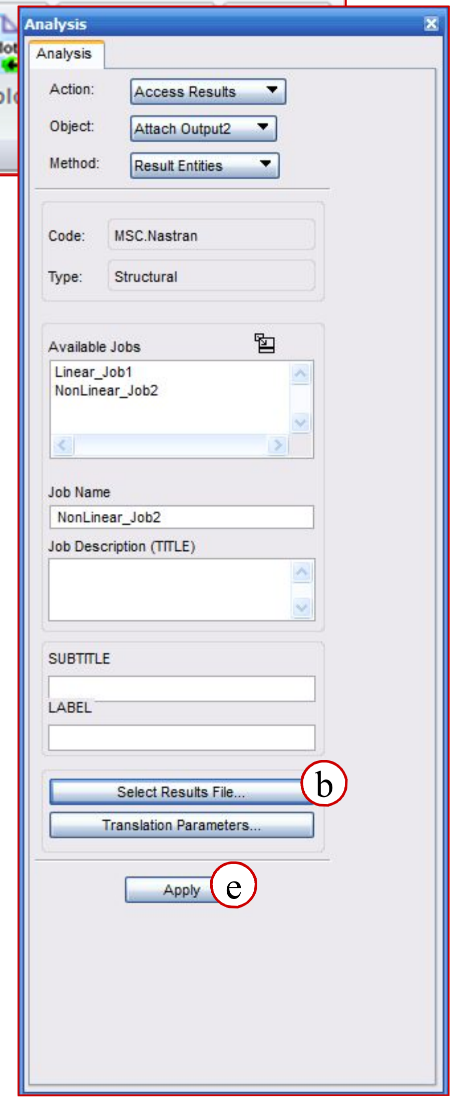
Step 13. Analysis: Access Results



Access the Results of the nonlinear analysis.

a. Under the *Analysis tab*, click **Attach Output2** in the *Access Results* group.
b. Click on **Select Results File.**
c. Select the file, ***nonlinear_job2.op2***.
d. Click **OK.**
e. Click on **Apply**.

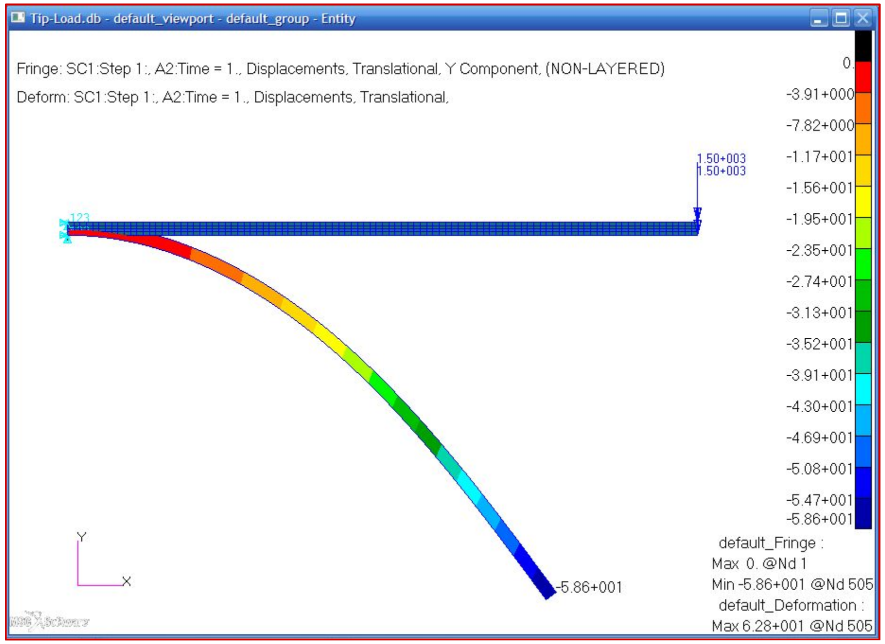


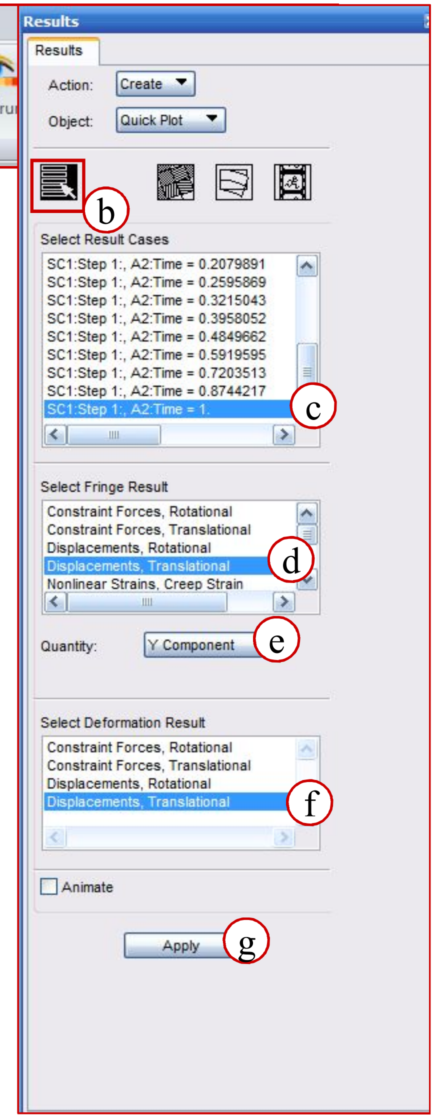


Step 14. Results: Create Displacement Plot

Post-Process the nonlinear results.

a. Under the *Results* tab click **Fringe/Deformation** in the *Quick Plot* group.
b. Click on the **Select Results** icon.
c. Select the final non-linear result case.
d. Select **Displacements, Translational** from the *Select Fringe Result*.
e. Select **Y Component** for the *Quantity*.
f. Select **Displacements, Translational**.
g. Click **Apply**.





Step 15. Compare Your Results

- Finally get the maximum Y deflection from the fringe spectrum/range. Enter that value into the first table below. Also compare the quickplots obtained with both jobs.

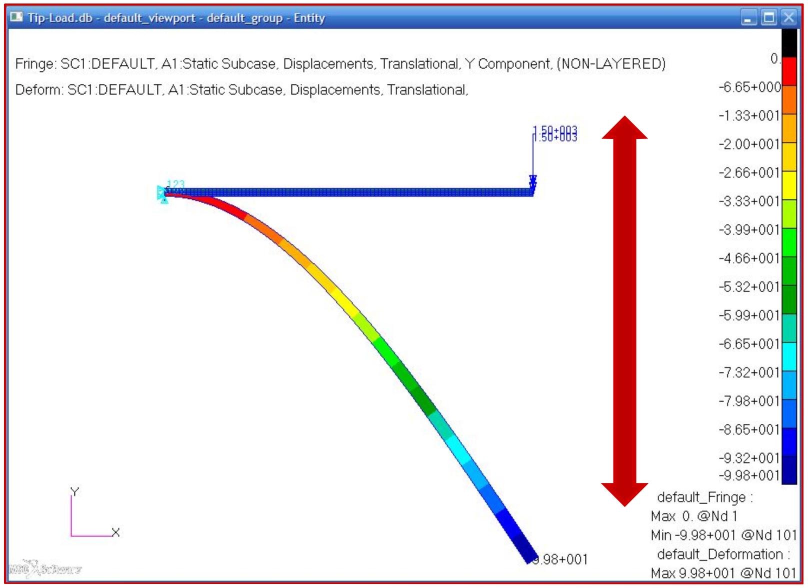
	Small Deflection	Large Deflection
MSC Nastran		

	Small Deflection	Large Deflection
MSC Nastran	-99.80	-60.70
Theory	-100.0	-58.59
Compare with these		

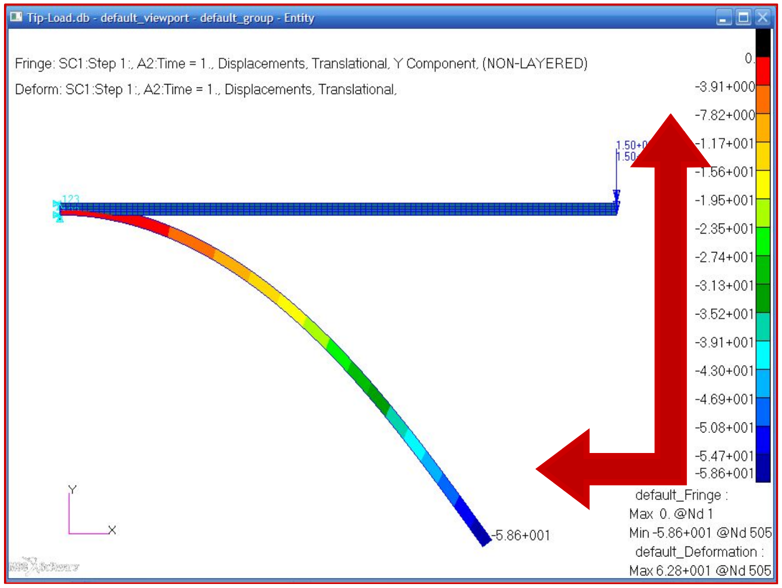
Step 15. Compare Your Results (Cont.)

As shown in the results obtained, inclusion of large deformation effects are very important in realistically modeling the physical behavior of the cantilever model.

Linear_job1



Nonlinear_job2



Notice the horizontal displacement in the nonlinear solution

Step 16. Try and Improve Your Results

- **If you have time, do the following:**
  1. Detach both result files (use Analysis, Delete Results Attachment).
  2. Delete the existing mesh.
  3. Create a new finer mesh to compare with the results of the previous analysis.
  4. Since you applied the element properties and LBC's to the geometry you do not need to update or change them.
  5. Run both linear and nonlinear jobs again.

Note: You might also want to try using the 'Large Strain' nonlinear formulation in the Input Properties Form… this might be more important than refining the mesh. We will cover this in the Nonlinear Elements Section.

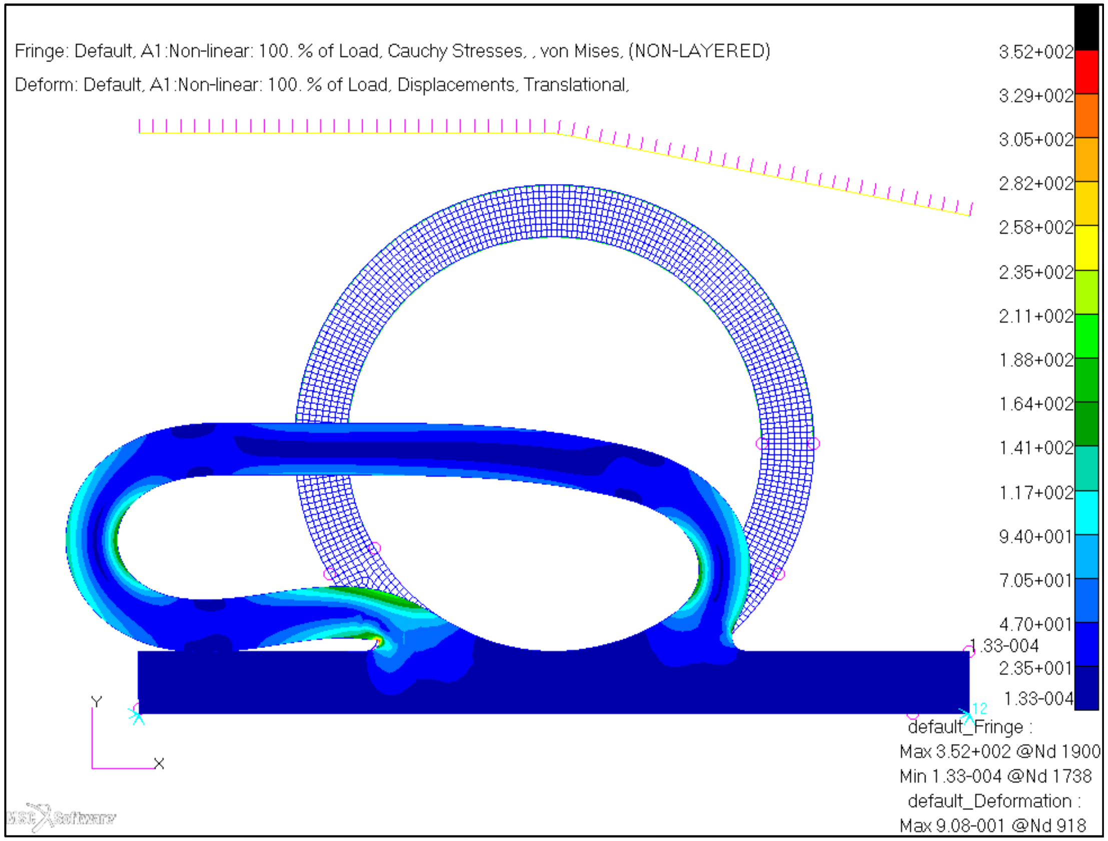
Step 16. Try and Improve Your Results (Cont.)

6. Attach the result files and make quick plots as before. Compare the Y-displacements with the ones obtained before and the theoretical values (previous page). See if the results improved with the finer mesh… it isn't required that they do so, but is often expected.

	Small Deflection	Large Deflection
MSC Nastran		
Enter your new results		

	Small Deflection	Large Deflection
MSC Nastran	-99.80	-60.70
Theory	-100.0	-58.59
Compare with these results		

WORKSHOP 2
RUBBER DOOR SEAL



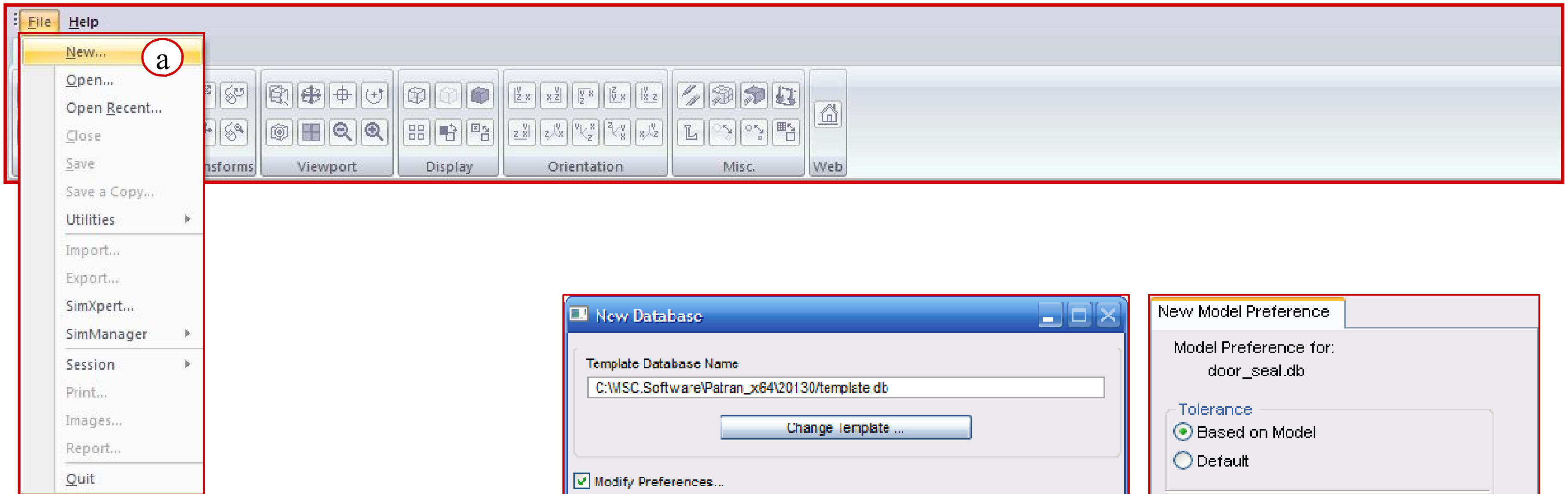
- **Workshop Objectives**
  - Large displacement/ large strain analysis.
  - Contact analysis using rigid-deformable contact.
  - Hyperelastic material model.
- **Software Version**
  - Patran 2013
  - MSC Nastran 2013.1
- **Required Files**
  - *rubber_seal.ses*
- **Problem Description**
  - In this Exercise we analyze a trunk door seal. The purpose of the analysis is to examine the stresses and deflections created during the closing of a door. The seal is made of a rubber material and therefore will be modeled using hyperelastic material properties. The trunk door is considered very stiff relative to the rubber seal and can be modeled as a rigid body.

- **Suggested Exercise Steps**
  1. Open a New Patran Database
  2. Run the **rubber_seal.ses**
  3. Mesh the top three surfaces using **Isomesh** and use **Paver** *Mesher* for the Bottom Surface
  4. Verify the Element for Cracks
  5. Remove the Duplicate Nodes by *Equivalence*
  6. Re-Verify the Element for Cracks
  7. Align the Element Normals by Selecting any Element with a Positive Z-Direction Normal
  8. Fix the Bottom of the Seal by Applying a Displacement Constraint
  9. Define the **Deformable** Contact Body for all Four Surfaces
  10. Define the **Rigid** Contact Body for the Curves Representing the Door
  11. Preview the **Rigid Body Motion**
  12. Create **Hyperelastic** Material Properties
  13. Apply **2D Solid** Properties
  14. Set up **SOL400** Analysis
      - Request *OP2* Output
      - Request Element Strains Output

- **Suggested Exercise Steps (Cont.)**

15. Attach the *OP2* File Attachment
16. Plot Results
    - Plot **Logarithmic Strains** and **Displacement Translation** Results
    - Plot **Cauchy Stress** and **Displacement Translational** Results
17. Quit Patran

Step 1: Open a New Database



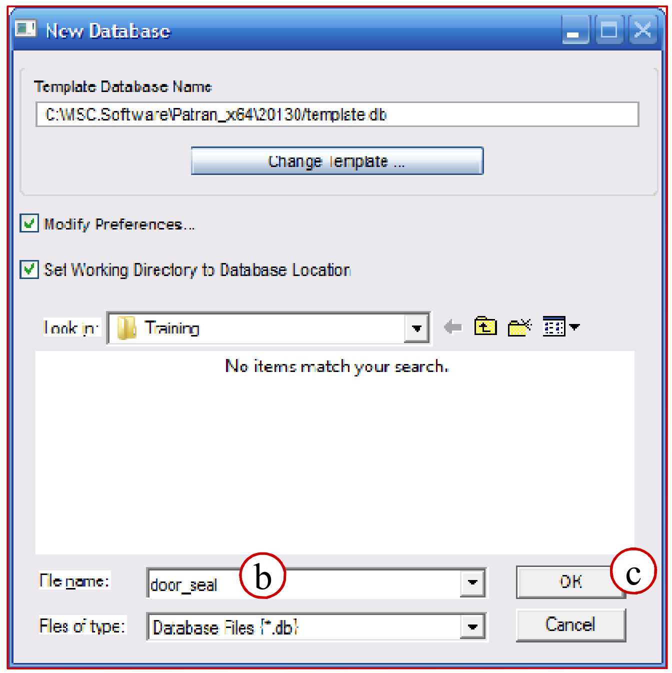
Open a new database named **door_seal.db**:

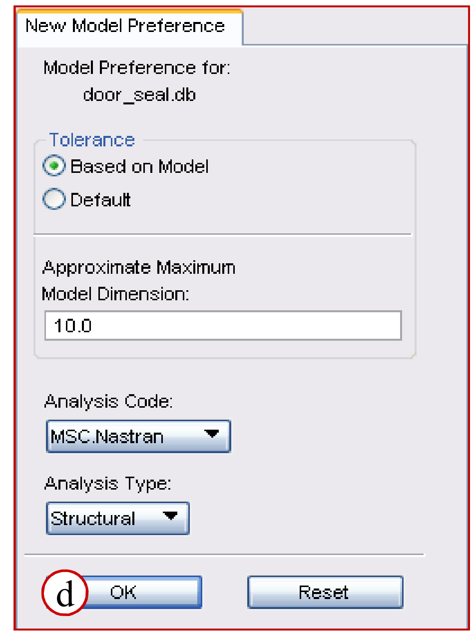
a. Pull down *File* > **New**.

b. Type **door_seal** as *File name*.

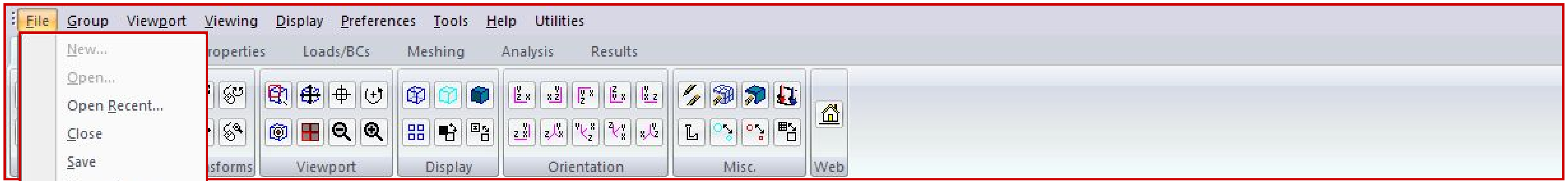
c. Click **OK**.

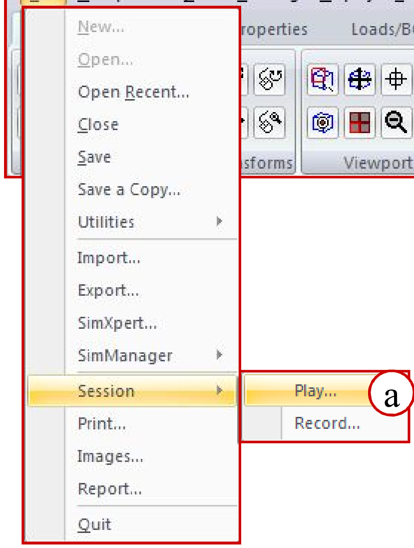
d. Click **OK** to select **MSC.Nastran** as the *Analysis Code*.





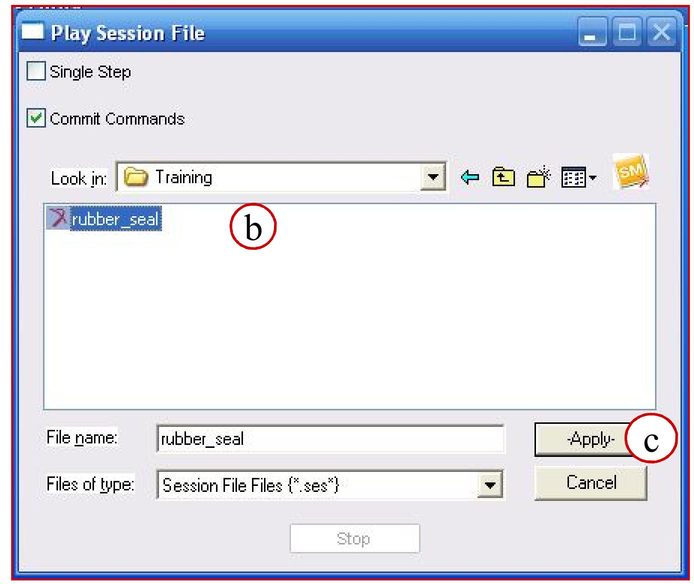
Step 2. Run the Provided Session File

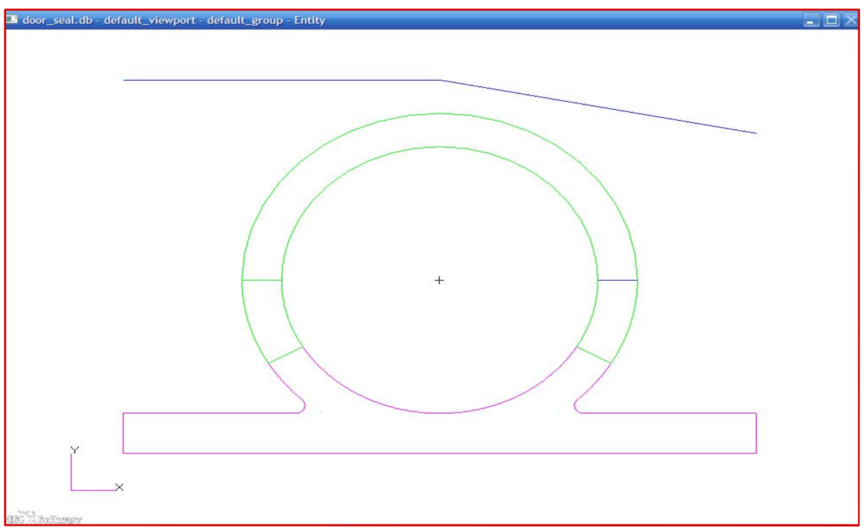




Run the provided session file.

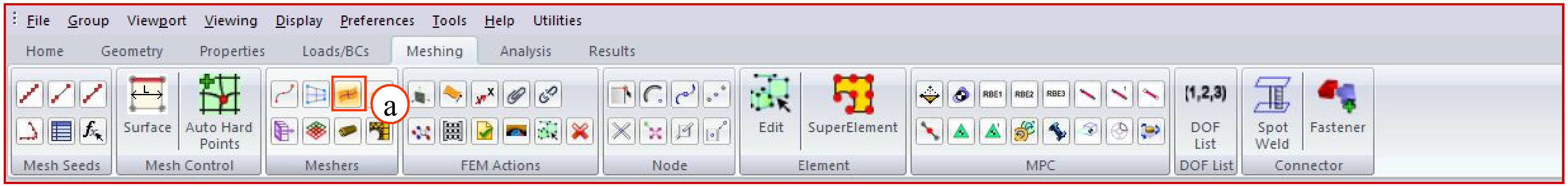
a. Pull down *File* and select *Session* and click **Play**.
b. Select **rubber_seal.ses** as the *File name.*
c. Click **Apply**. (This action will run the session file. Please do not interrupt it.)





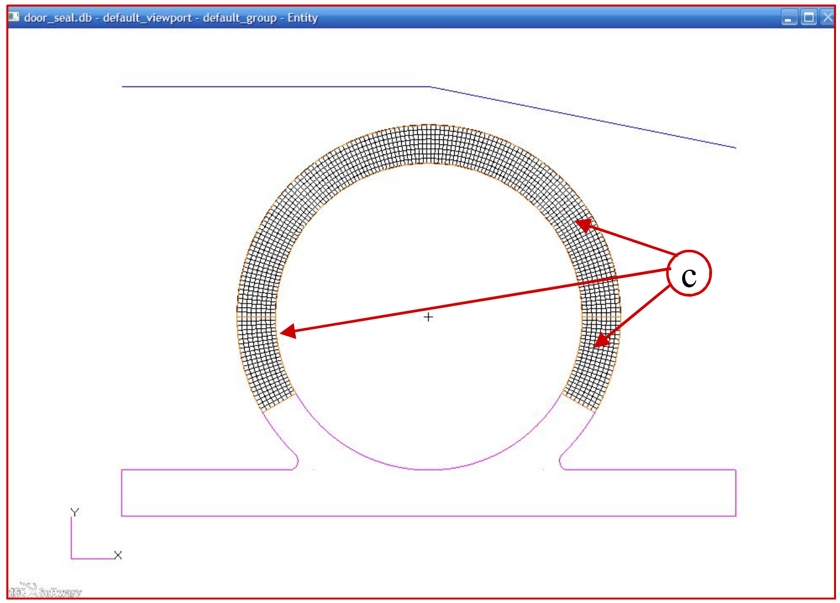
When the session file is done the viewport will contain all the geometry for the rubber seal and trunk rigid body.

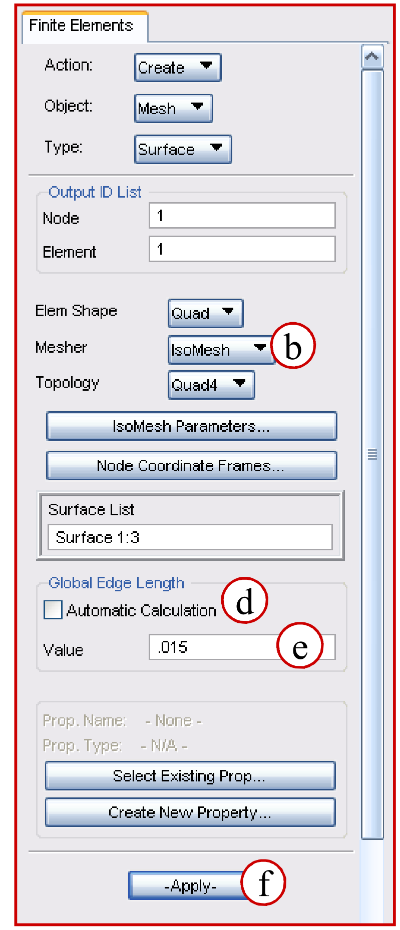
Step 3. Create the Mesh



Mesh the seal.

a. Under the *Meshing* tab, click **Surface** in the *Meshers* group.
b. Select **IsoMesh** as the *Mesher*.
c. Pick all three (Green) surfaces (Surface 1:3).To select multiple surfaces, hold the SHIFT key down while picking.
d. Uncheck *Automatic Calculation.*
e. Enter **.015** as the *Value*.
f. Click **Apply**.

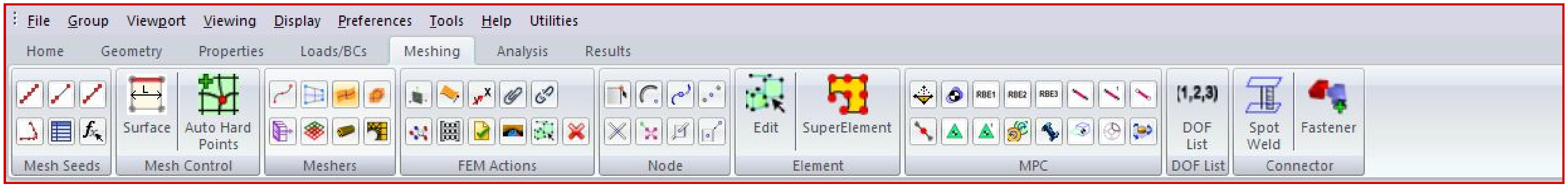




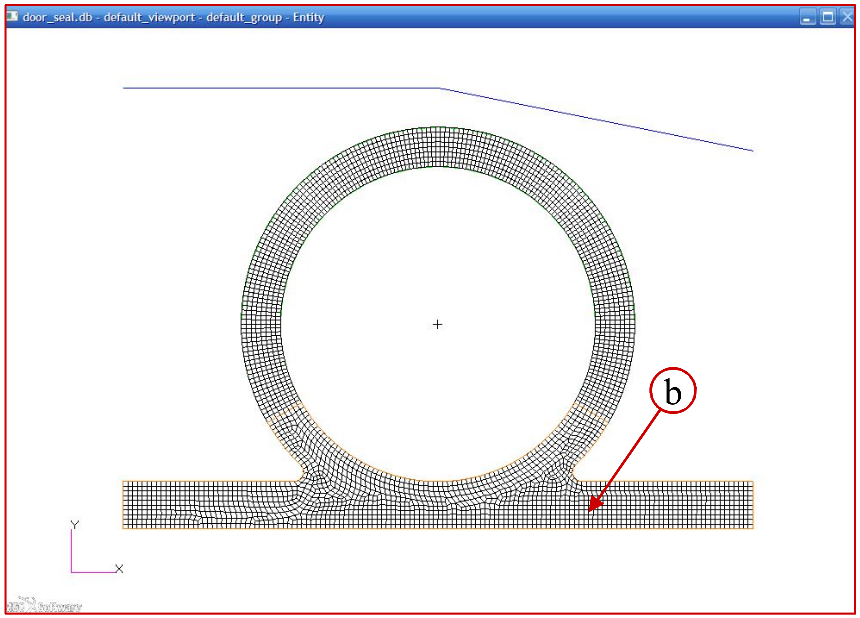
We use IsoMesh to mesh the top three surfaces since they are simple (Green) surfaces.

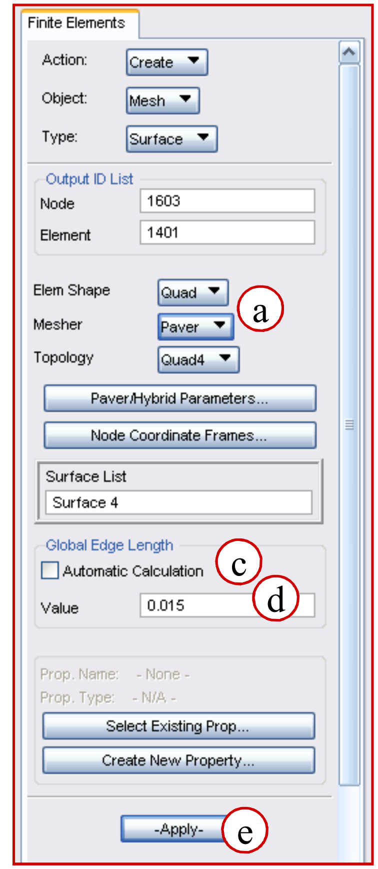
Continue working in Elements form on the next page.

Step 3. Create the Mesh (Cont.)



a. Select **Paver** as the *Mesher*.
b. Select the bottom (Magenta) surface (Surface 4).
c. Uncheck *Automatic Calculation.*
d. Enter **0.015** as the *Value*.
e. Click **Apply**.

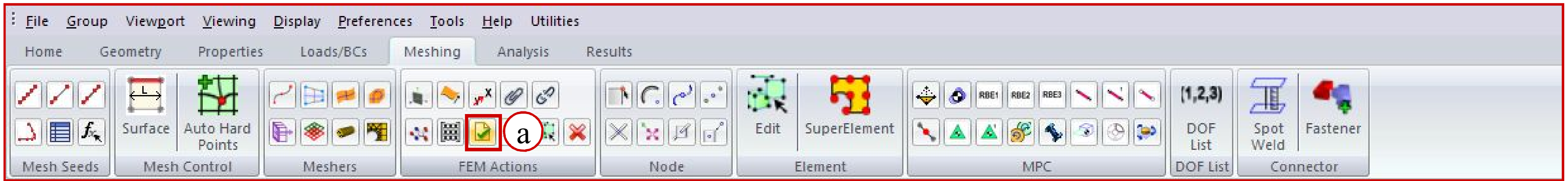




We use Paver to mesh the bottom part since it is a complex (Magenta) surface.

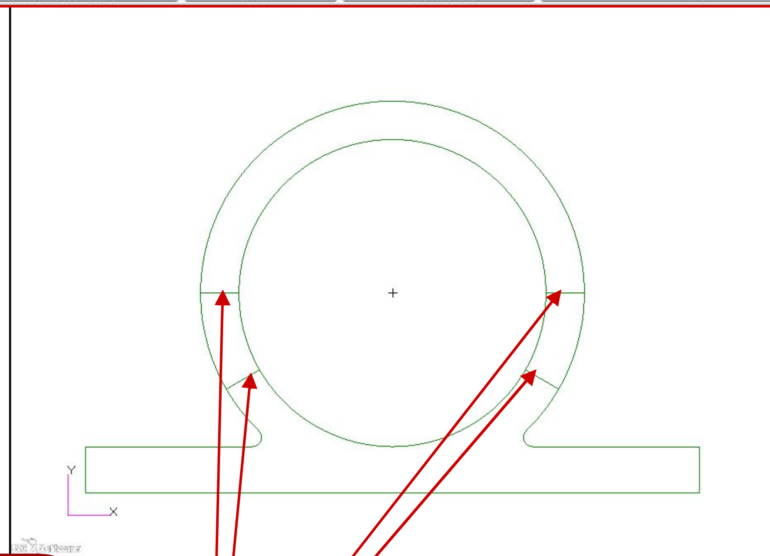
MSC Software

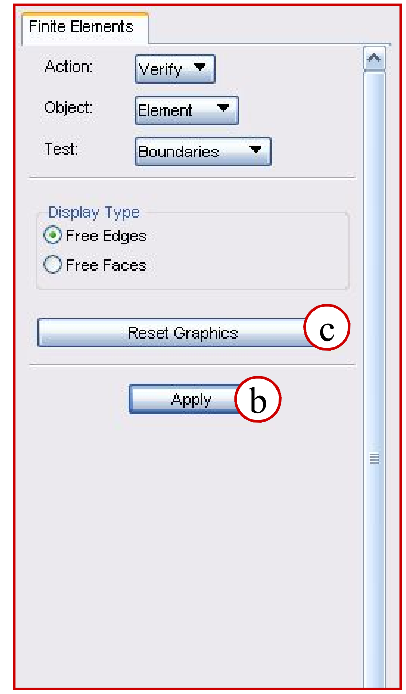
Step 4. Verify the Element for Cracks



Verify the Element Boundaries.

a. Click **Verify** in the *FEM Actions* group
b. Click **Apply**.
c. When done, click **Reset Graphics**.

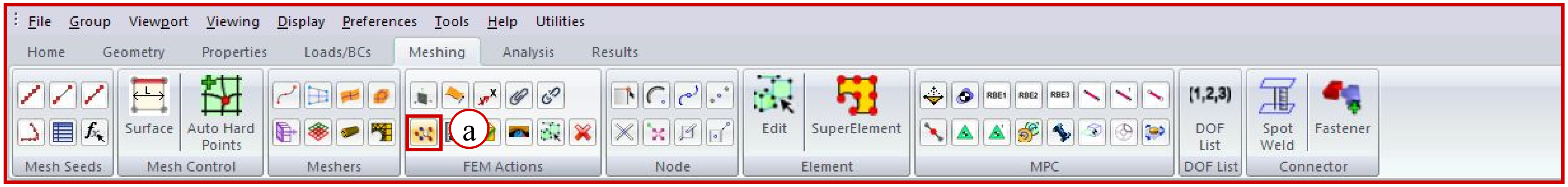




This option will show where there are free element edges. Free element edges are essentially ‘cracks’ in the model where duplicate nodes exist. Patran will not automatically remove duplicate nodes when meshing congruent surfaces like the ones in this model. Step 5 will show the **Equivalence** option to remove duplicate nodes.

These lines show ‘cracks’ in the FE Model. They need to be removed before the Analysis can be performed.

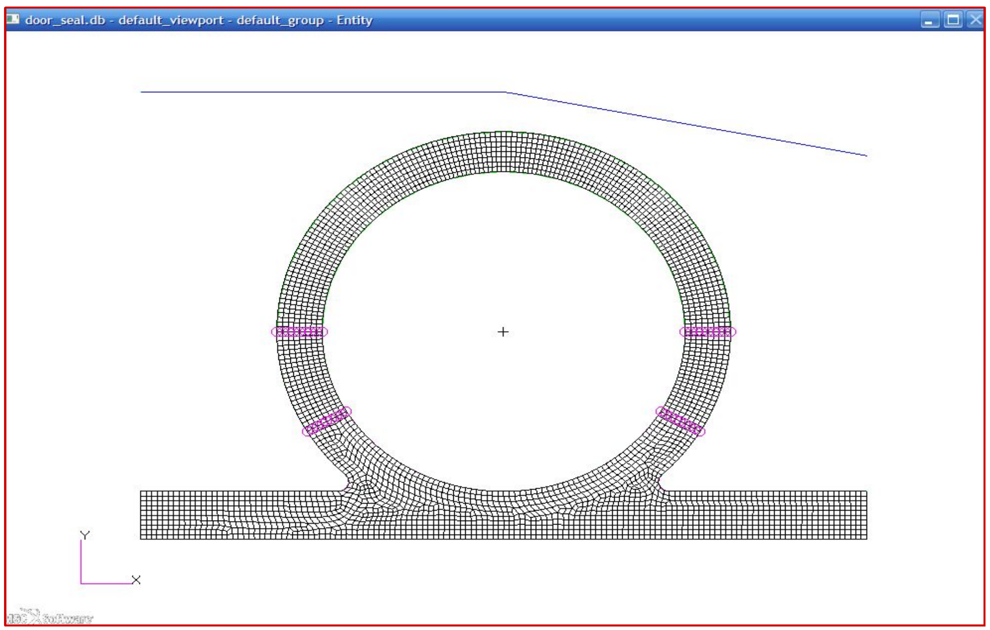
Step 5. Remove the Duplicate Nodes

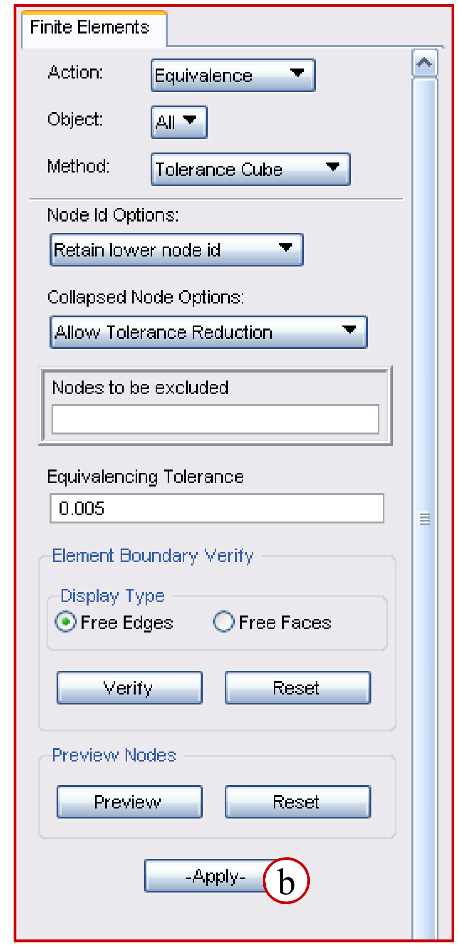


Equivalence the model.

a. Click **Equivalence** in the *FEM Actions* group.

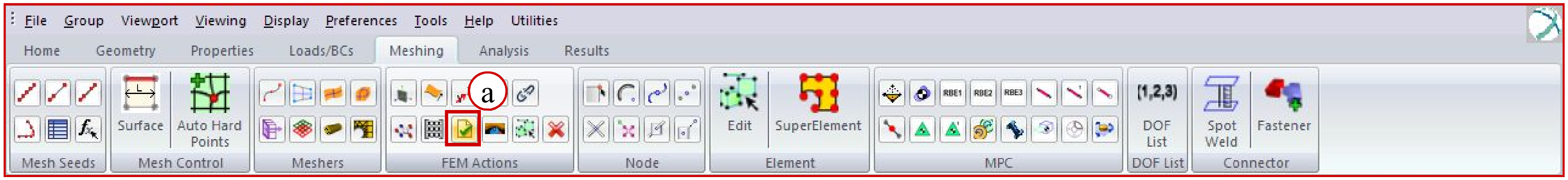
b. Click **Apply**.





Equivalence any duplicate nodes created during meshing. This process will delete all the overlapped nodes (where the pink circles are), and have only one node left.

Step 6. Re-Verify the Element for Cracks

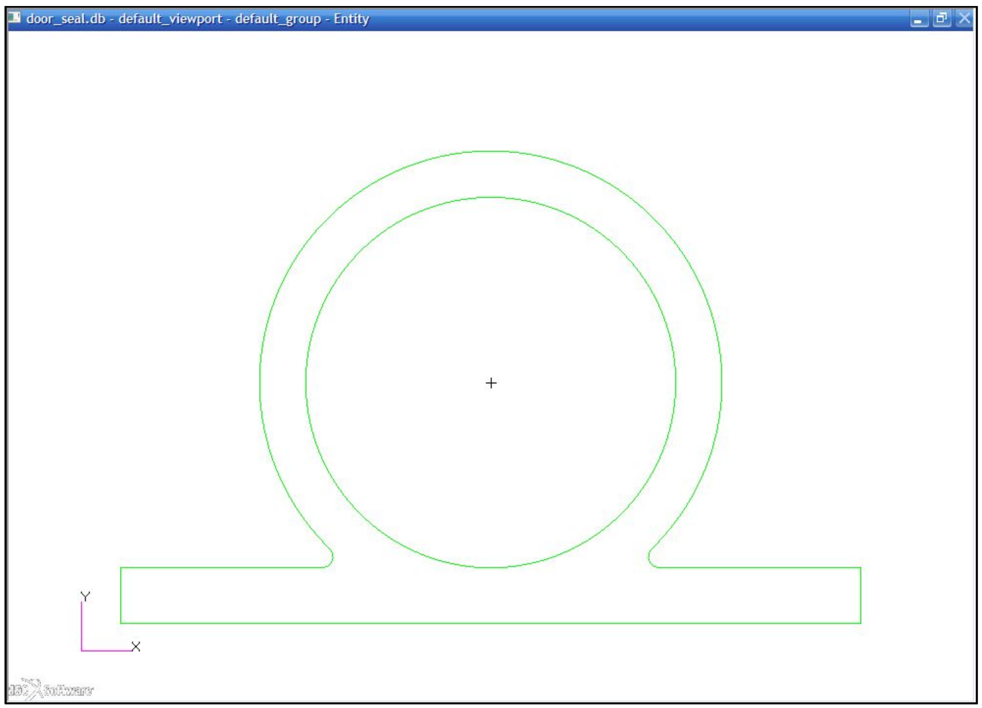


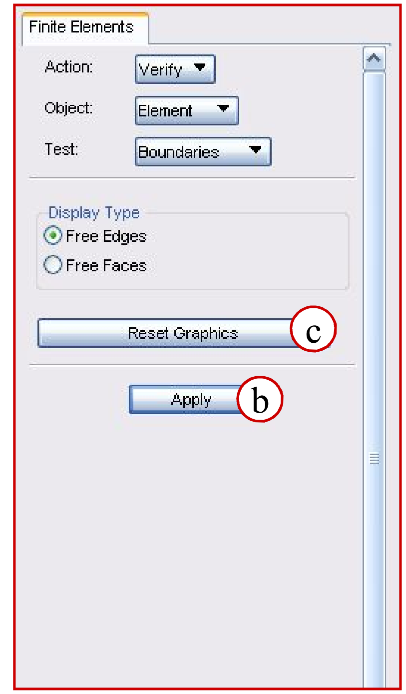
Re-Verify the Element Boundaries.

a. Click **Verify** in the *FEM Actions* group**.**

b. Click **Apply**.

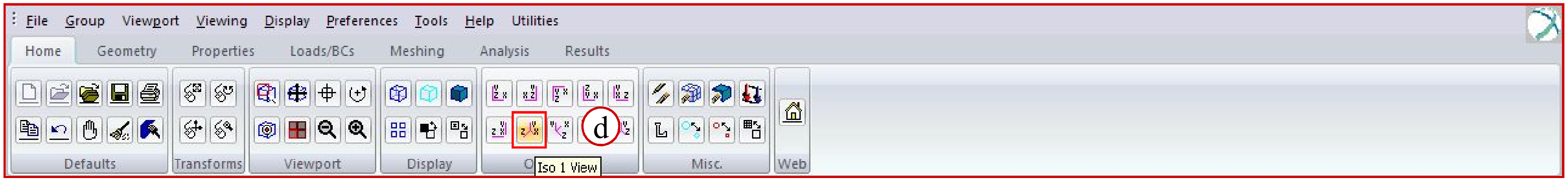
c. Click **Reset Graphics**.

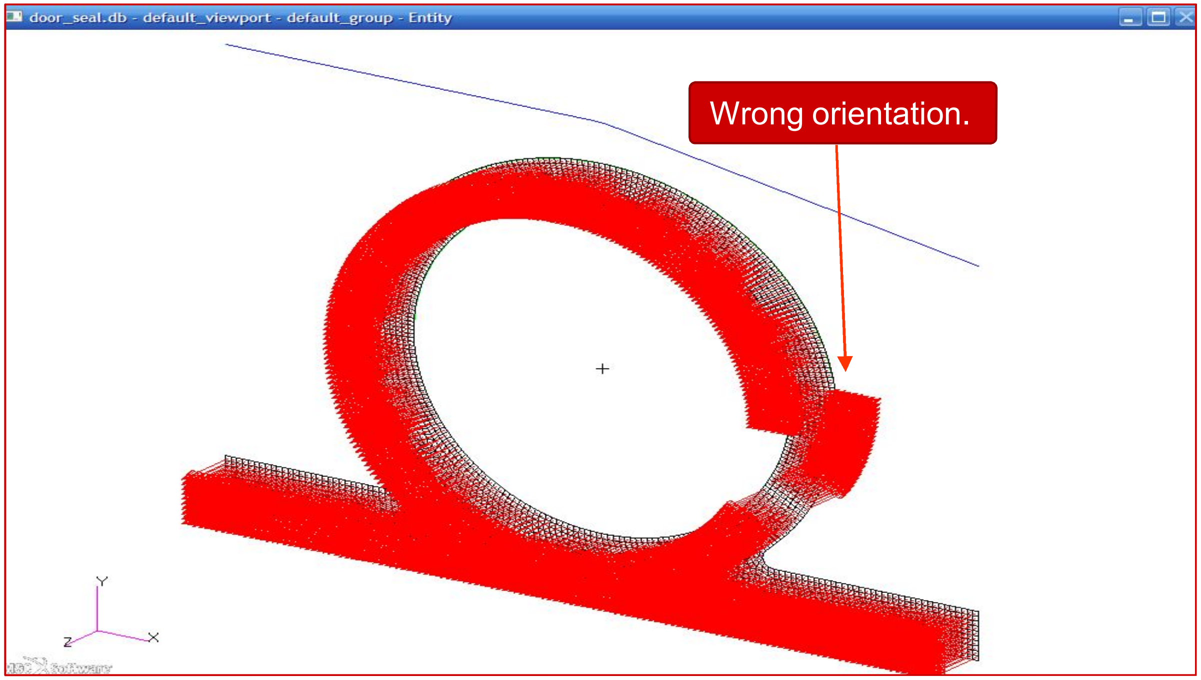


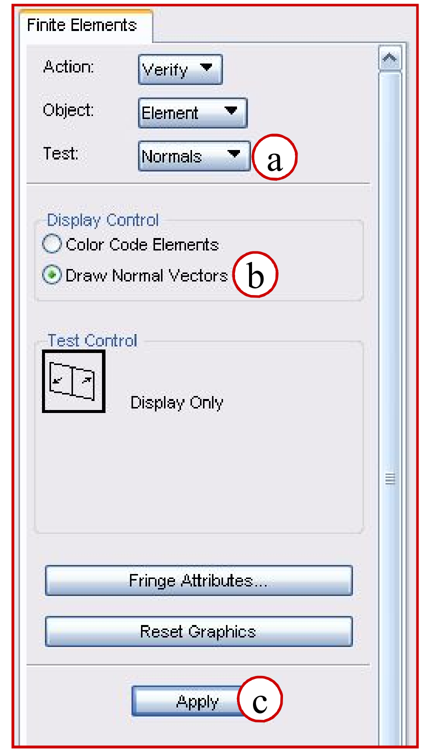


After the Equivalence Operation, the ‘cracks’ are now gone.

Step 7. Align the Element Normals







Verify the element normals.

a. Select **Normals** for *Test*.

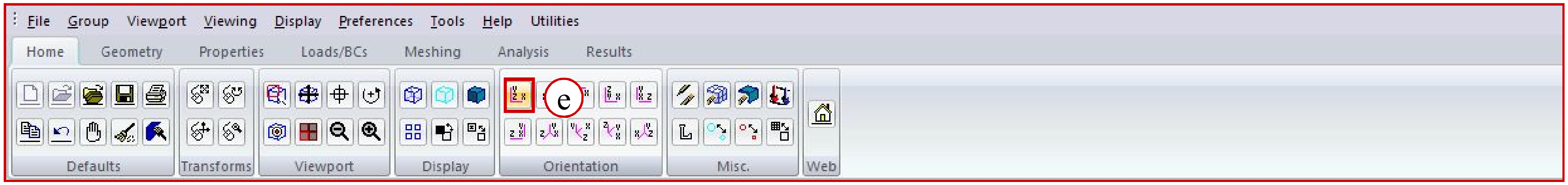
b. Select **Draw Normal Vectors**.

c. Click **Apply**.

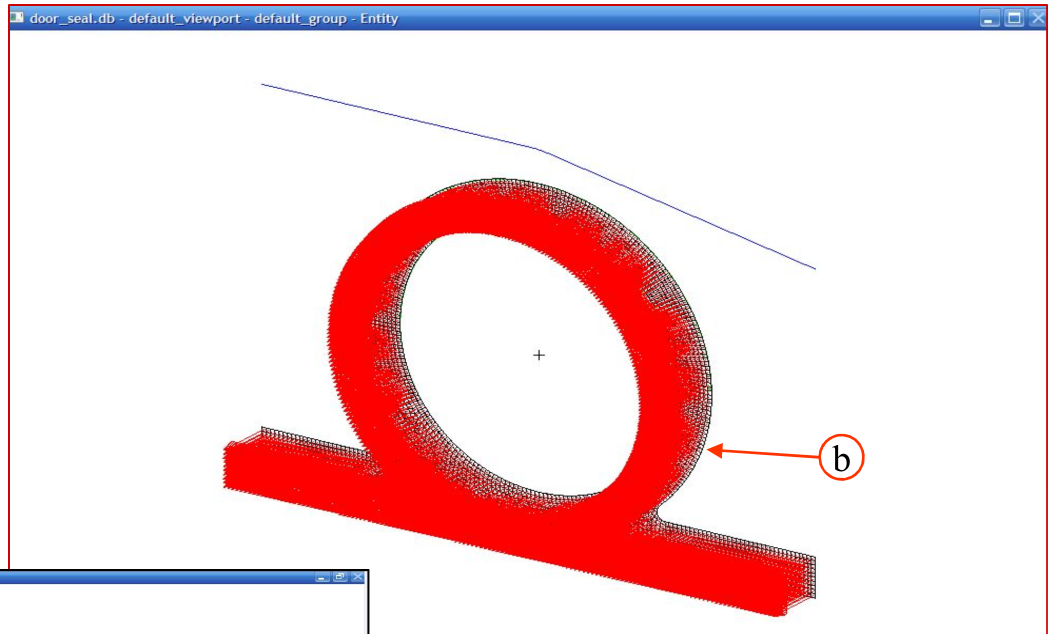
d. Under *Home* tab, Click **Iso1 View** in the *Orientation* group.

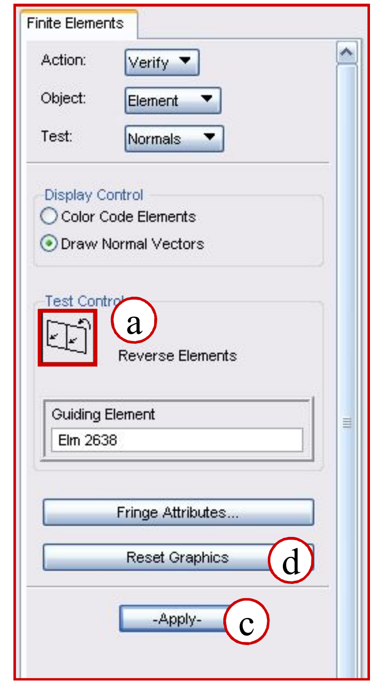
Since this is a 2-D solid model, all element normals must point in the positive Z-direction. Verify the element normals, and correct those whose normals point in the wrong direction.

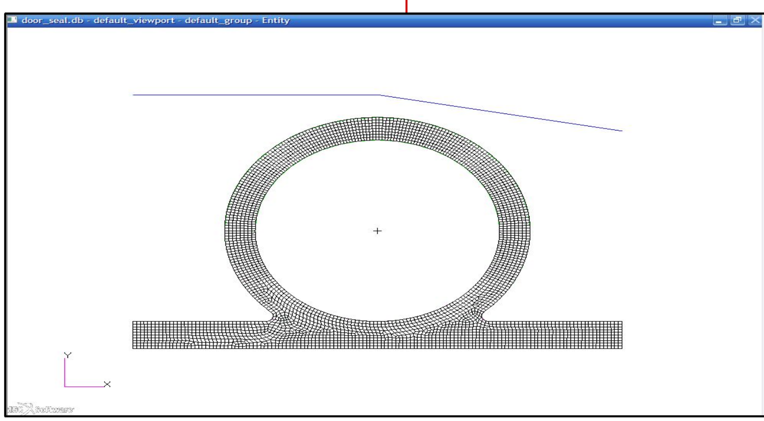
Step 7. Align the Element Normals (Cont.)



a. Click on **Reverse Elements**.
b. Pick any element pointing in the positive Z-direction as the guiding element.
c. Click **Apply**.
d. Click **Reset Graphics**.
e. Click the **Front View** Icon.



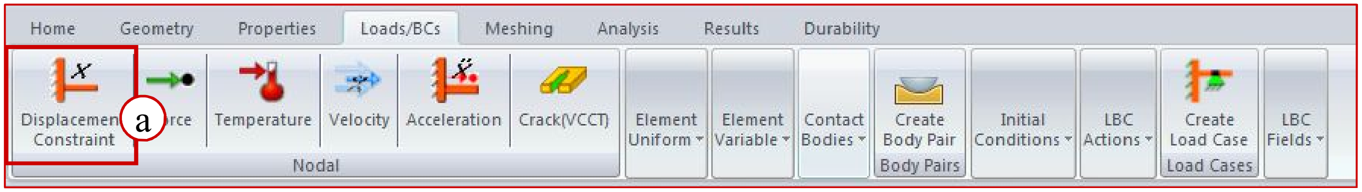




After all operations, your model should look like the image to the left

MSC Software

Step 8. Fix the Bottom of the Seal



Create the Boundary Condition fixing the bottom of the rubber seal.

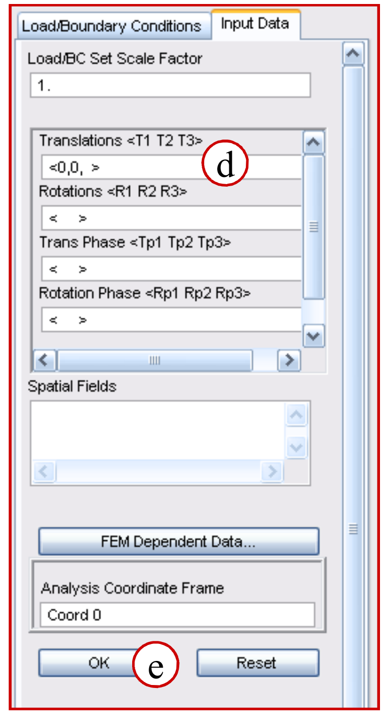
a. Under the *Loads/BCs* tab click **Displacement Constraint** in the *Nodal* group.

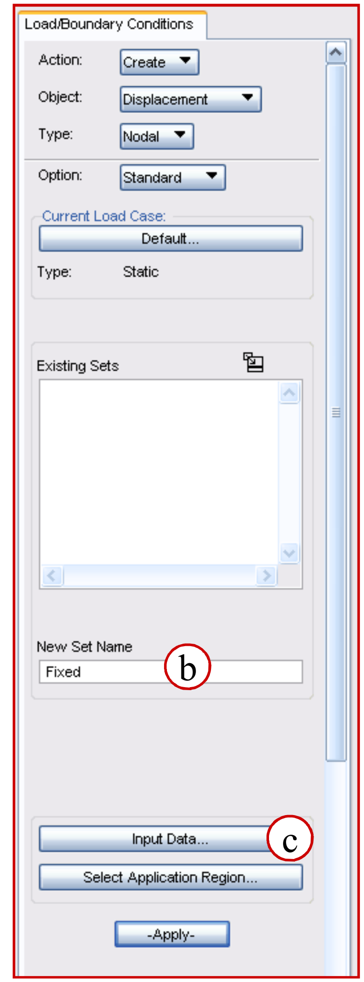
b. Enter **Fixed** as the *New Set Name.*

c. Click on **Input Data**.

d. Enter **<0,0,   >** for the *Translation*.

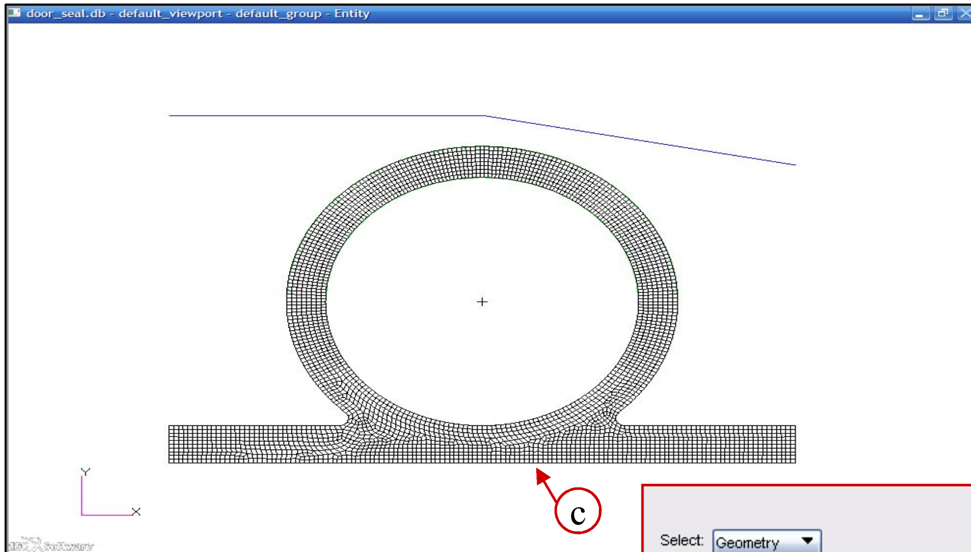
e. Click **OK**.

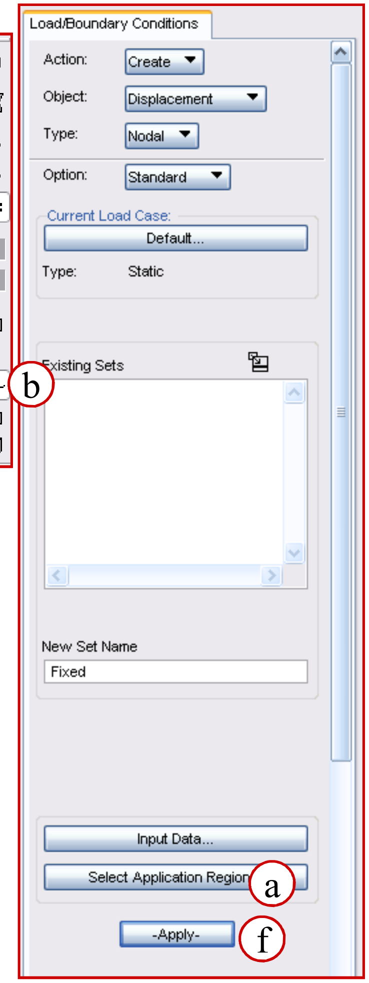


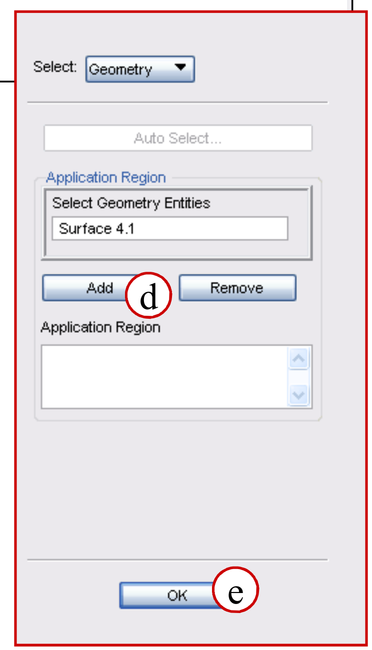


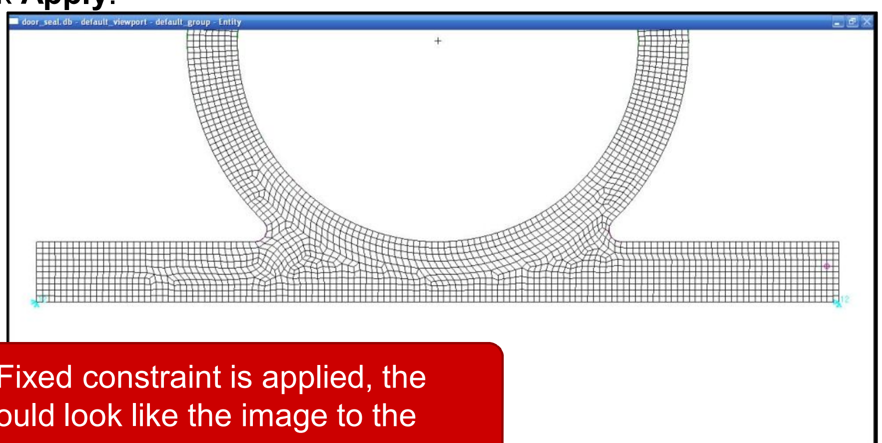
Step 8. Fix the Bottom of the Seal (Cont.)

a. Click on **Select Application Region**.
b. Select the **Curve or Edge** icon from the *Picking Filters* tool bar.
c. Select the bottom curve of the seal.
d. Click **Add**.
e. Click **OK**.
f. Click **Apply**.





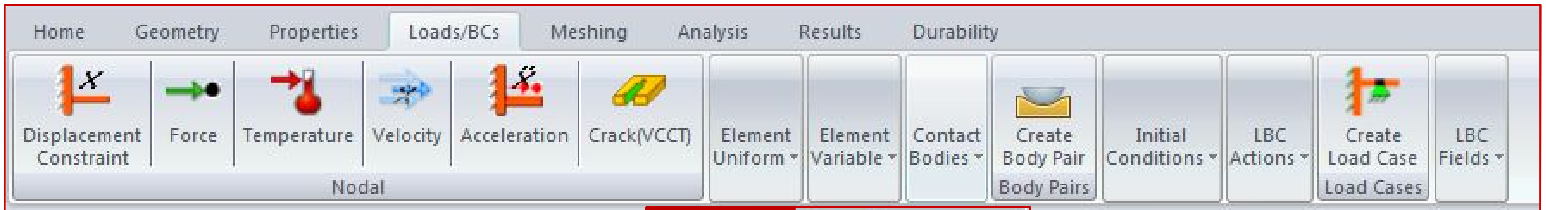




After the Fixed constraint is applied, the model should look like the image to the right.

MSC Software

Step 9. Define the Deformable Contact Body



Create the deformable contact body.

a. Click **Deformable** in the *Contact Bodies* group

b. Enter **Seal** as the *New Set Name.*

c. Select **2D** as the *Target Element Type*.

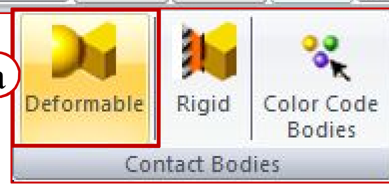
d. Click on **Select Application Region** form.

e. Select all four surfaces. To select multiple surfaces, hold the SHIFT key down while picking.

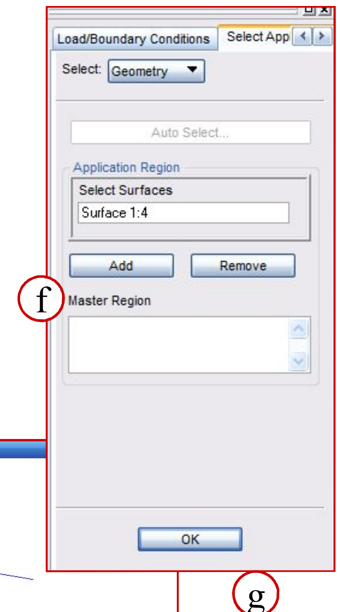
f. Click **Add**.

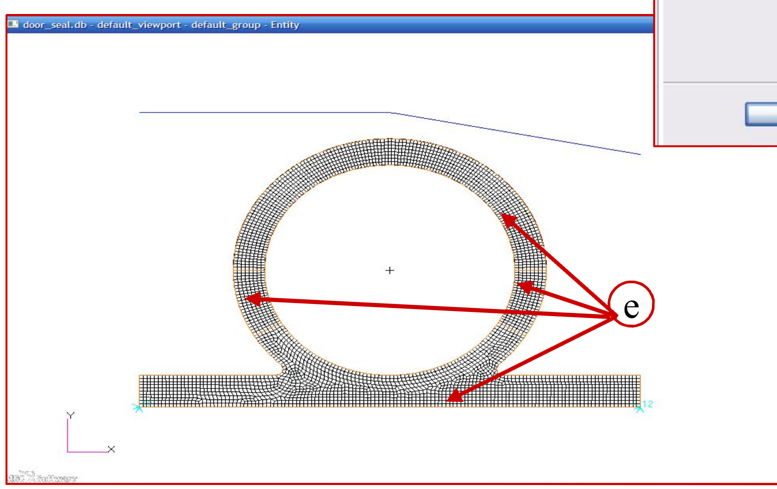
g. Click **OK**.

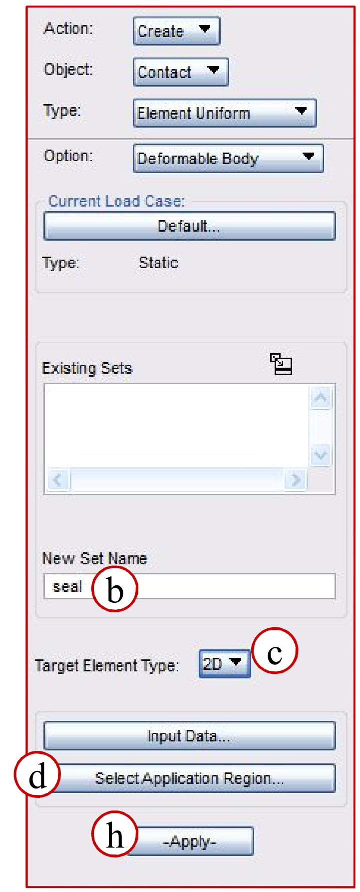
h. Click **Apply**.



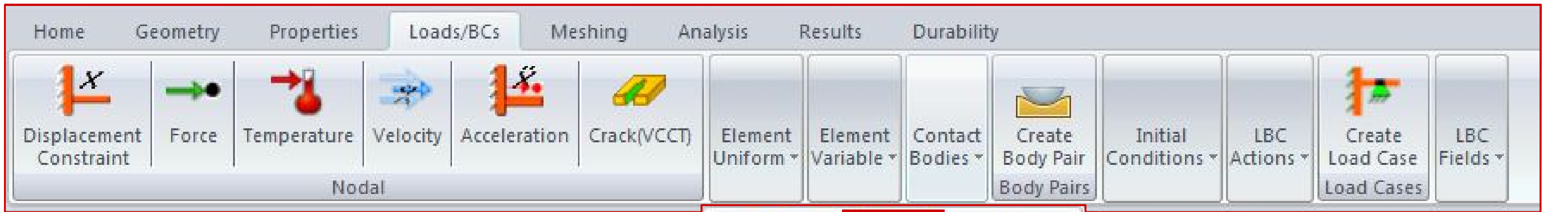
After you click Apply, you will see the contact markers (pink circles) in viewport.

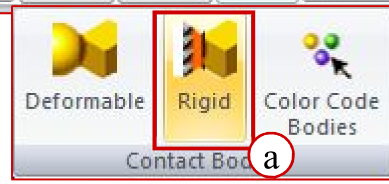






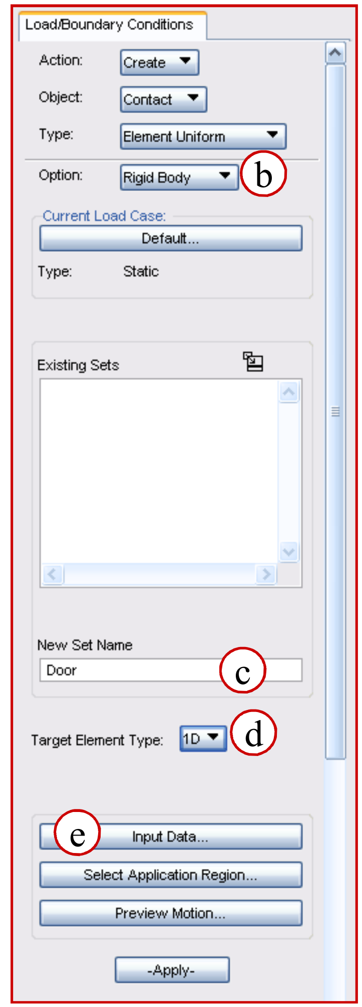
Step 10. Define the Rigid Contact Body

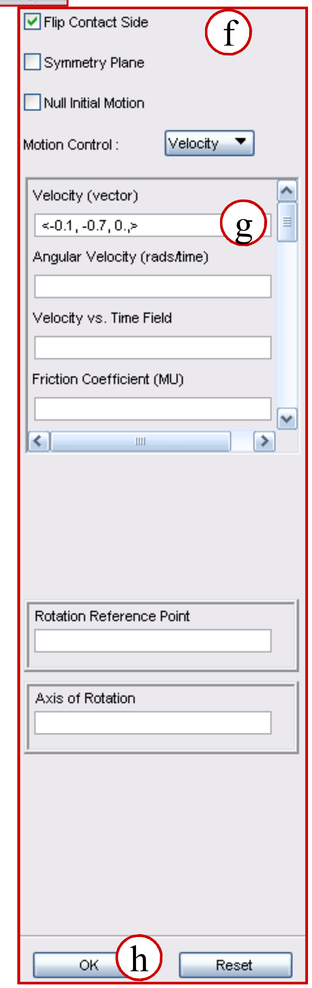




Create the rigid contact body.

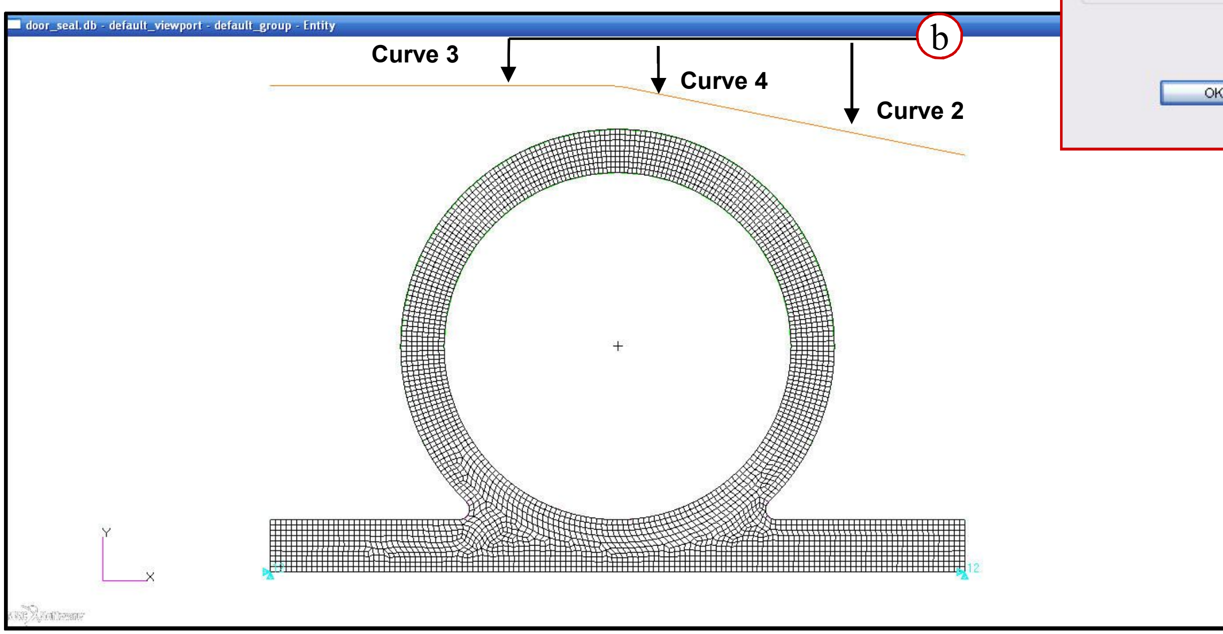
a. Under the *Loads/BCs* tab, click **Rigid** in the *Contact Bodies* group.
b. Verify **Rigid Body** is set for *Option*.
c. Enter **Door** as the *New Set Name.*
d. Select **1D** as the *Target Element Type*.
e. Click on **Input Data**.
f. Check **Flip Contact Side** (Flipping the Contact Side is needed, because of the orientations of the curves representing the door)
g. Enter **<-0.1, -0.7, 0.>** for the *Velocity*.
h. Click **OK**.

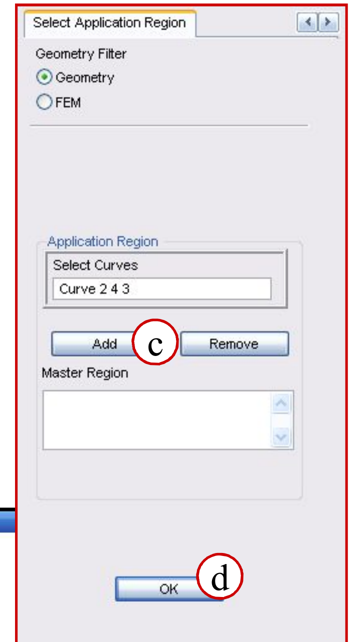


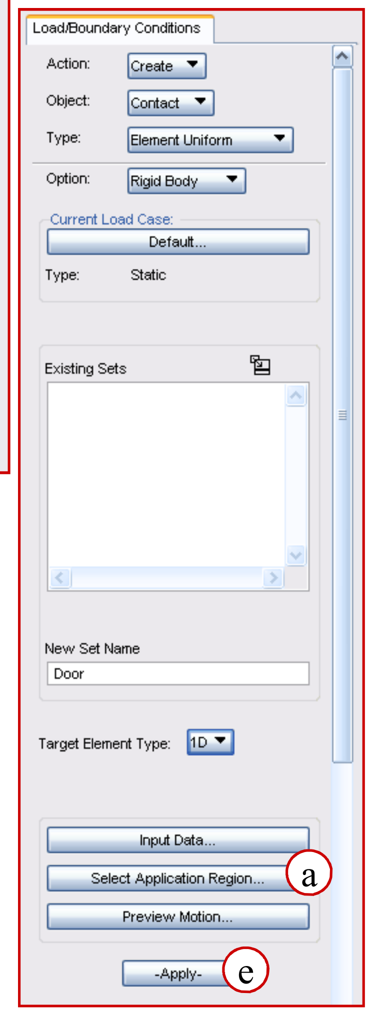


Step 10. Define the Rigid Contact Body (Cont.)

a. Click **Select Application Region**.
b. Select all three curves defining the rigid surface (Curve 2:4). To select multiple surfaces, hold the SHIFT key down while picking.
c. Click **Add**.
d. Click **OK**.
e. Click **Apply**.

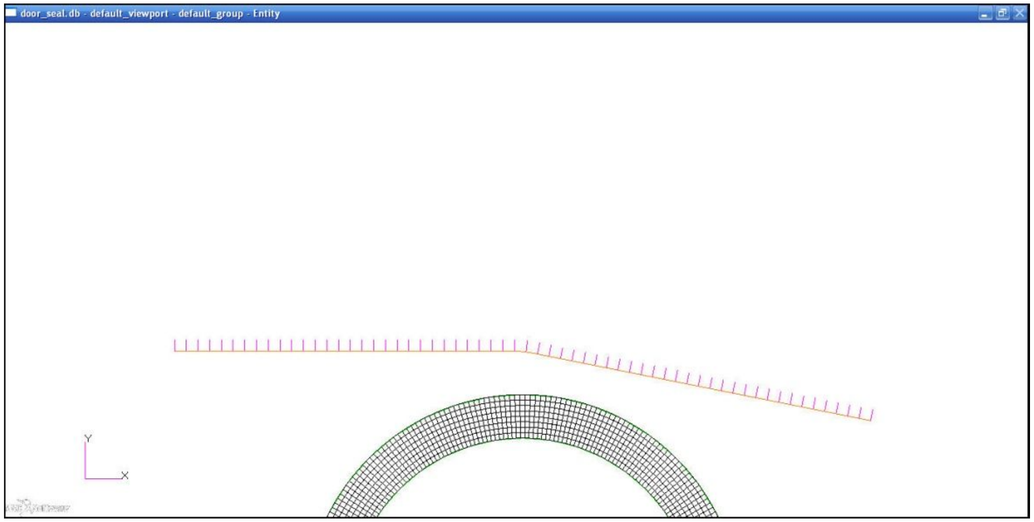


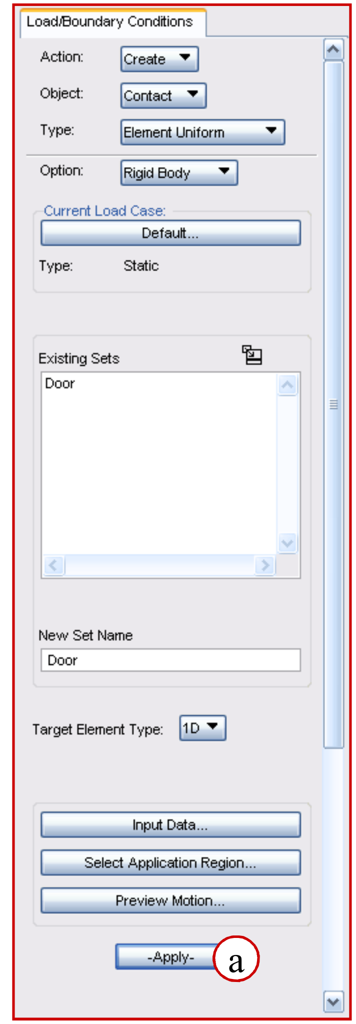




Step 10. Define the Rigid Contact Body (Cont.)

The rigid contact markers point toward the inside of the rigid body. Think of them as tick marks representing a wall. If the tick marks are reversed, Modify the Contact Body and flip the contact side on the Modify Data Form.





You should now see the contact (pink) markers in the viewport and the name of the object in the Existing Sets panel.

Step 11. Preview the Rigid Body Motion

Preview the Rigid Body Motion.

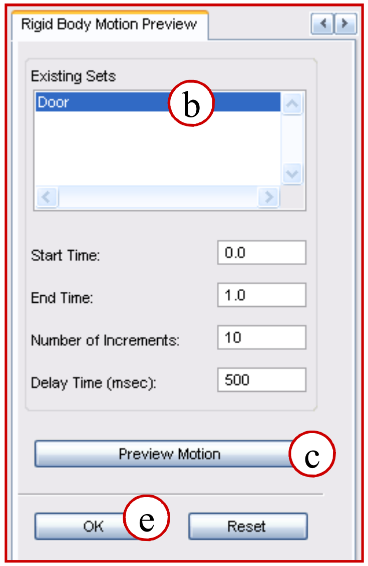
a. Click **Preview Motion**.

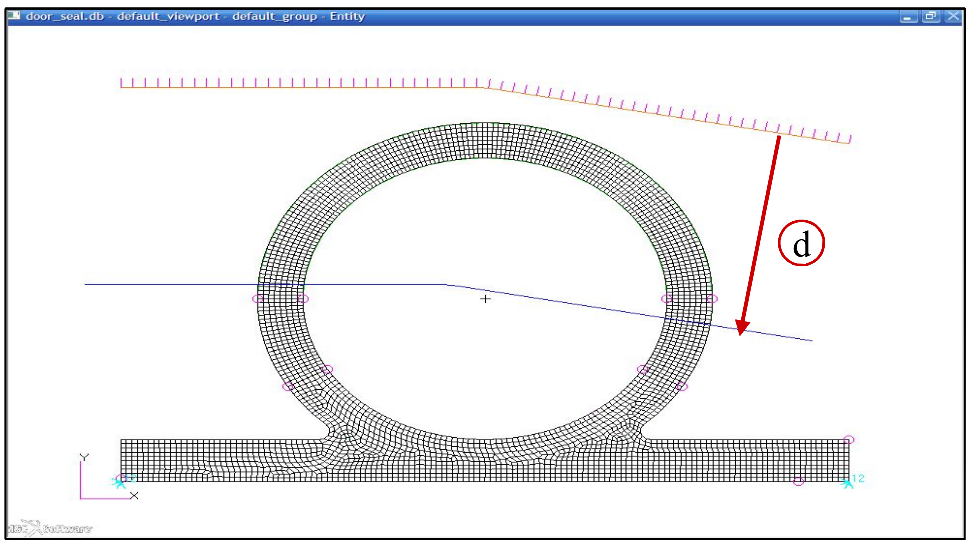
b. Select **Door**.

c. Click **Preview Motion**.

d. The Rigid Body should display the prescribed motion that was entered on the Input Data form.

e. Click **OK** when finished examining the motion.

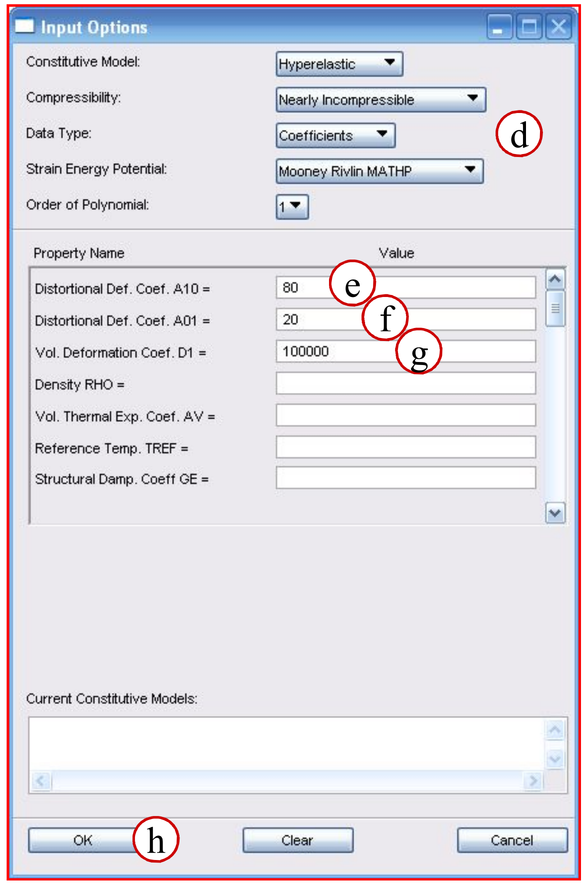


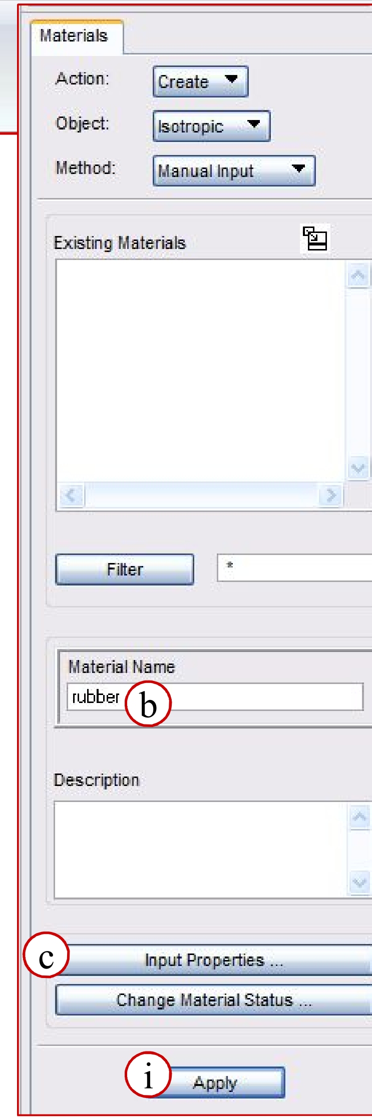


Step 12: Create Material

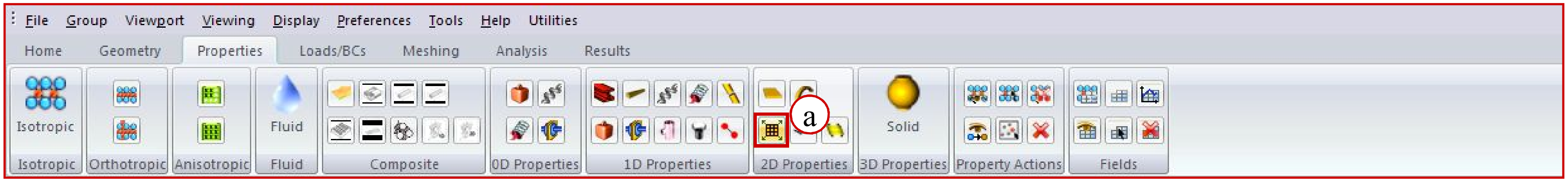
To create an isotropic Mooney-Rivlin material:

a. Under the *Properties* tab, click **Isotropic.**
b. Enter **rubber** for *Material Name.*
c. Click **Input Properties**.
d. *Hyperelastic > Nearly Incompressible > Coefficients > Mooney Rivlin MATHP > 1.*
e. Enter **80** for *Distortional Def. Coef. A10.*
f. Enter **20** for *Distortional Def. Coef. A01.*
g. Enter **100000** for *Vol. Deformation Coef. D1*.
h. Click **OK**.
i. Click **Apply**.





Step 13: Apply Properties



To apply properties:

a. Click **2D Solid** in the *2D Properties* group.

b. Enter **seal** for *Property Set Name.*

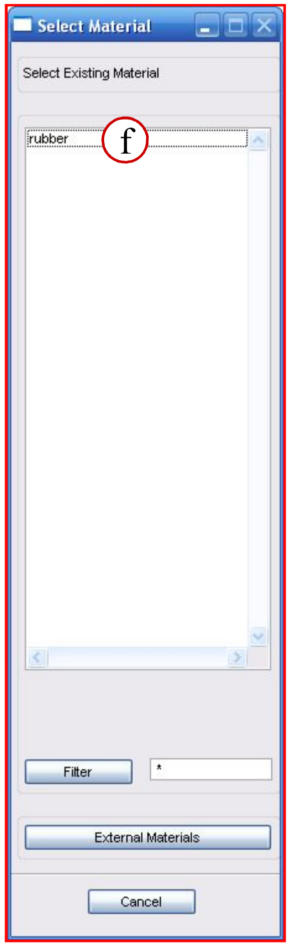
c. Change *Options* to **Hyperelastic Formulation**.

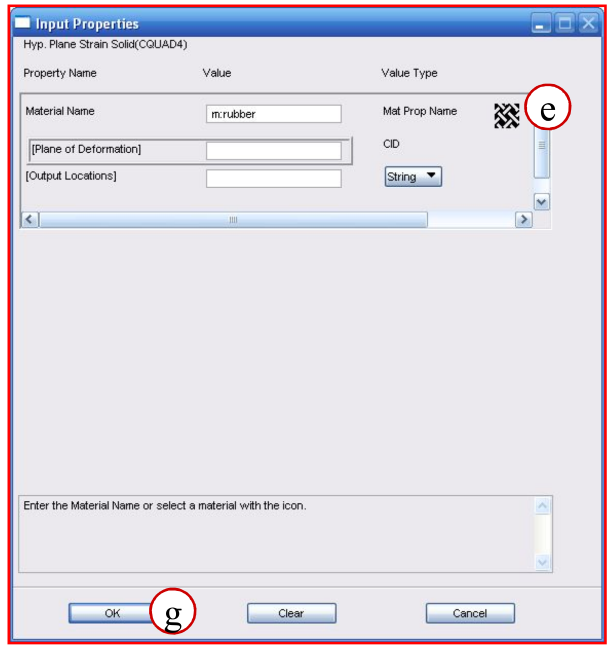
d. Click **Input Properties….**

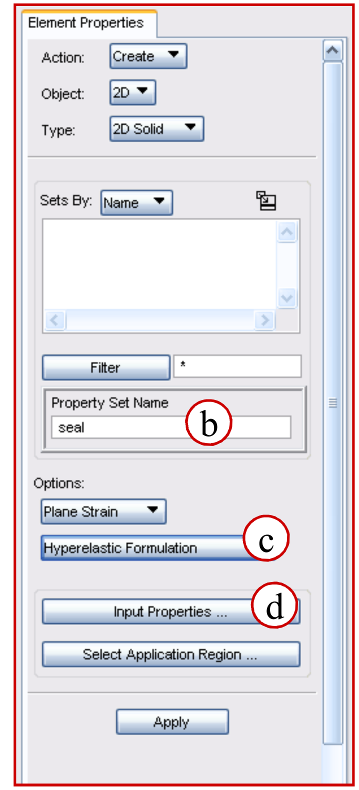
e. Click the **Mat Prop Name** Icon.

f. Select **rubber**.

g. Click **OK**.



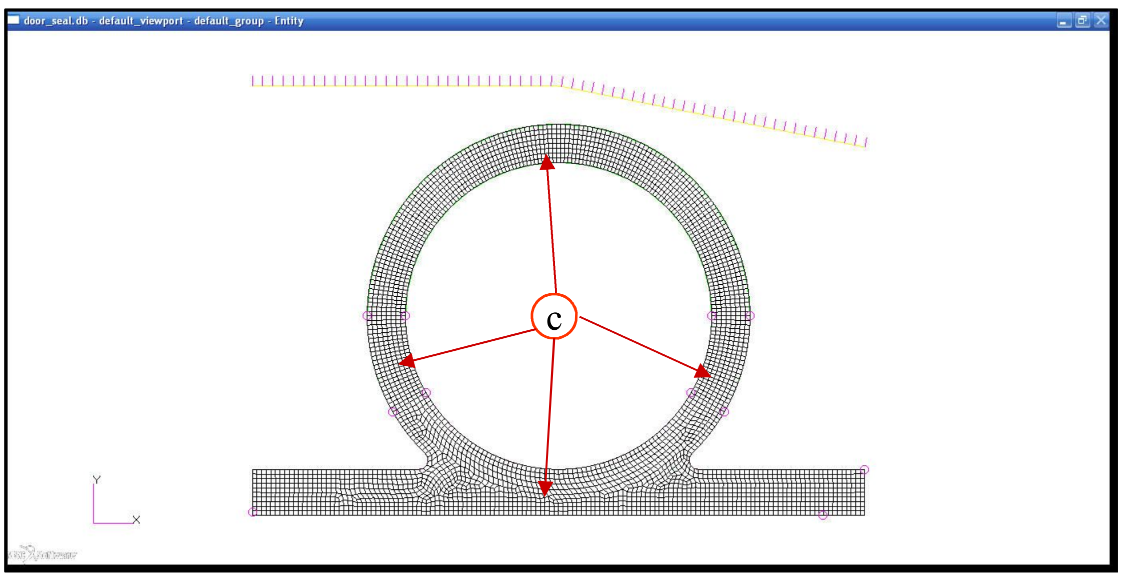


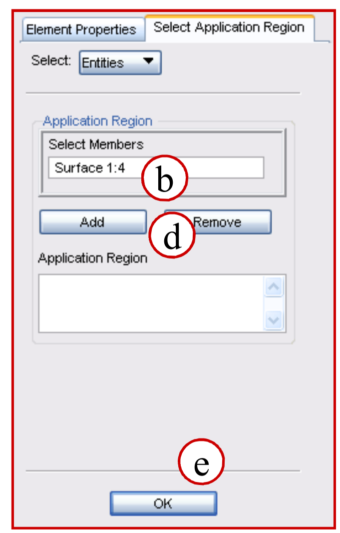


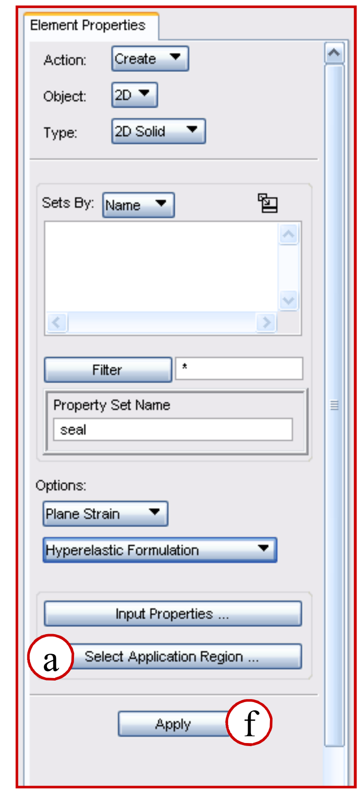
Step 13: Apply Properties (Cont.)

To apply properties:

a. Click **Select Application Region**.
b. Click in the **Select Members** text box.
c. Select all the surfaces.
d. Click **Add**.
e. Click **Ok**.
f. Click **Apply**.

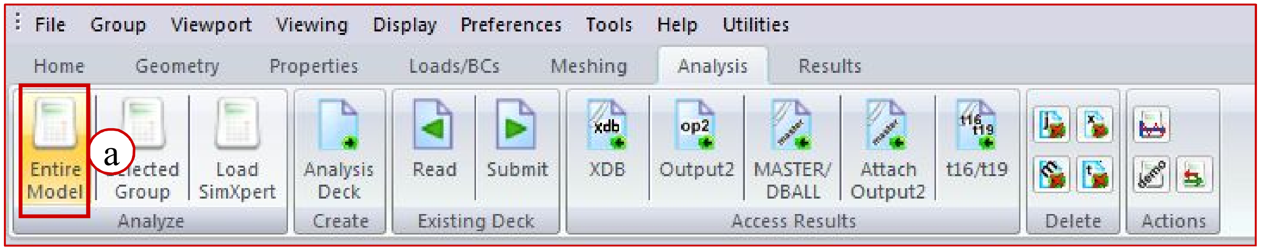






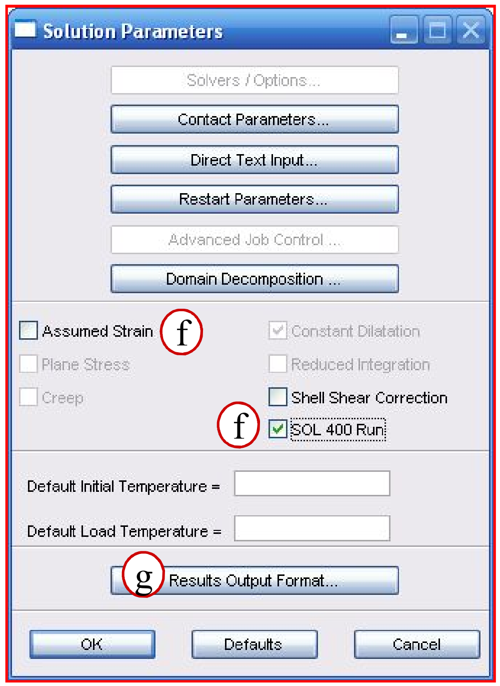
MSC Software

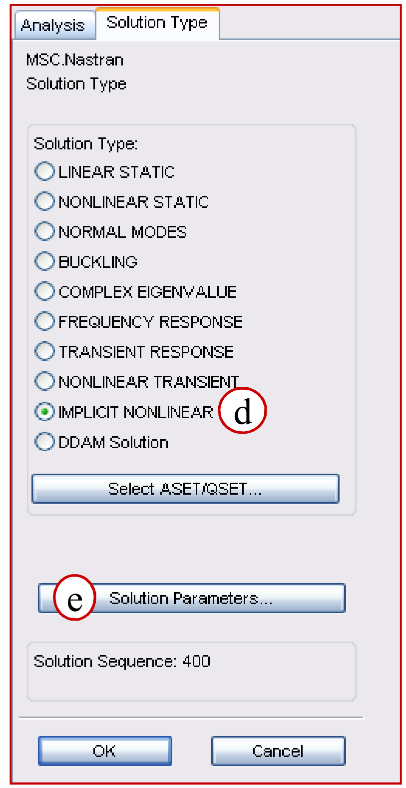
Step 14. Set up Analysis

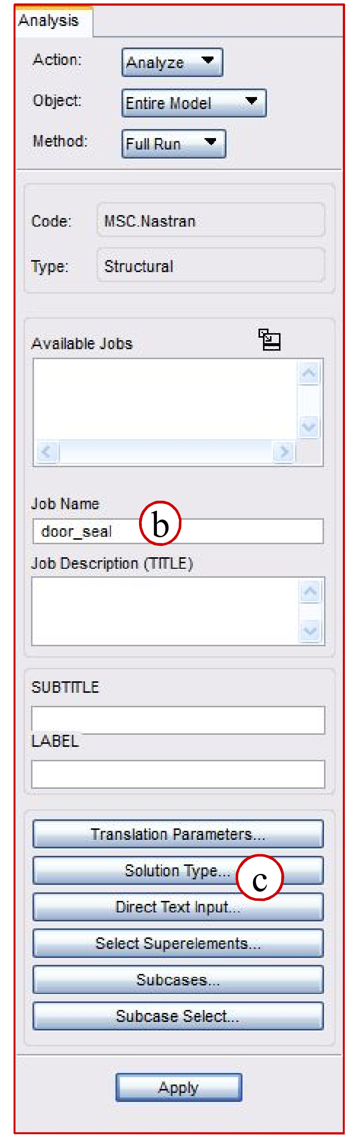


Set up the non-linear Analysis job.

a. Under the *Analysis* tab, click **Entire Model** in the *Analyze* group**.**
b. Enter **door_seal** as the *Job Name.*
c. Open **Solution Type**.
d. Select **Implicit Nonlinear** as *Solution Type*.
e. Click on **Solution Parameters…**
f. Confirm **SOL400 Run**, and uncheck **Assumed Strain**.
g. Click **Results Output Format**.

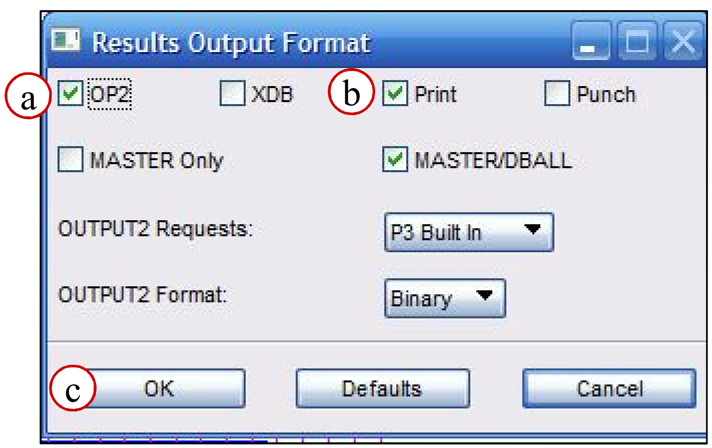


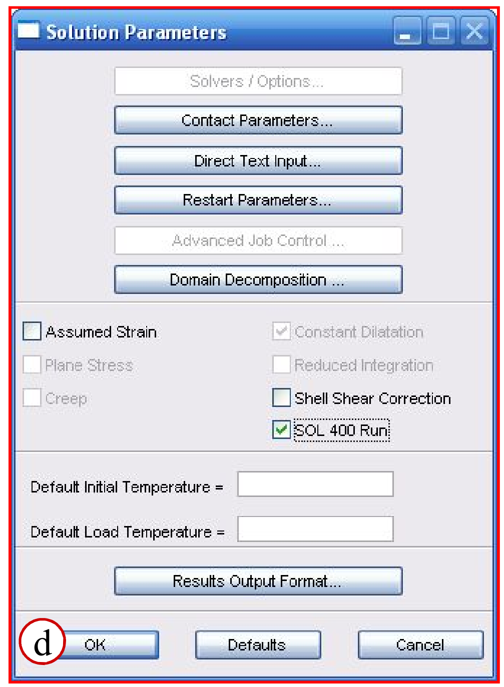


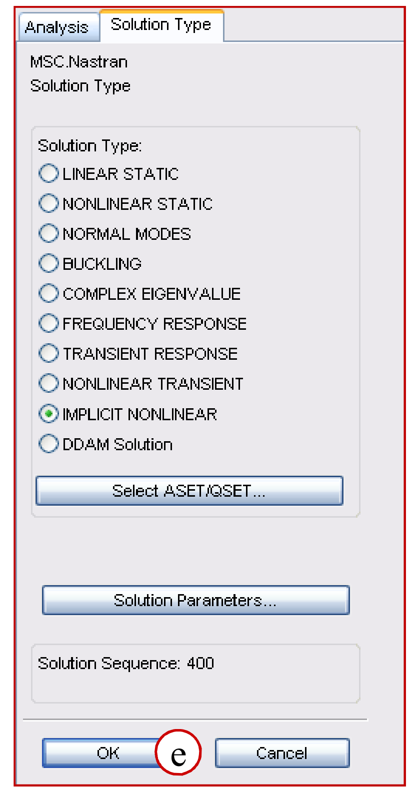


Step 14. Set up Analysis (Cont.)

a. Check *OP2*.
b. Check *Print*.
c. Click **OK**.
d. Click **OK**.
e. Click **OK**.

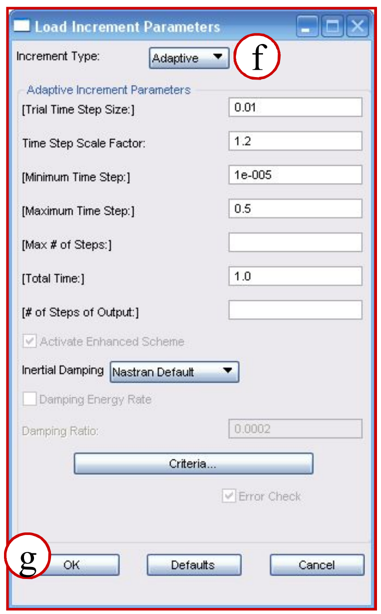


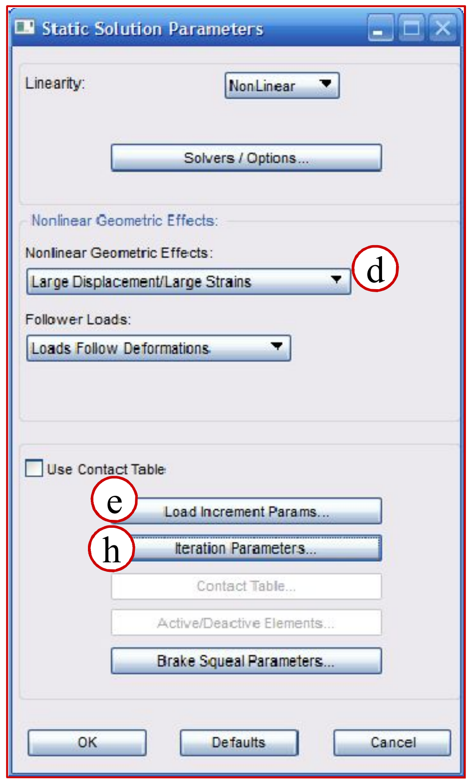


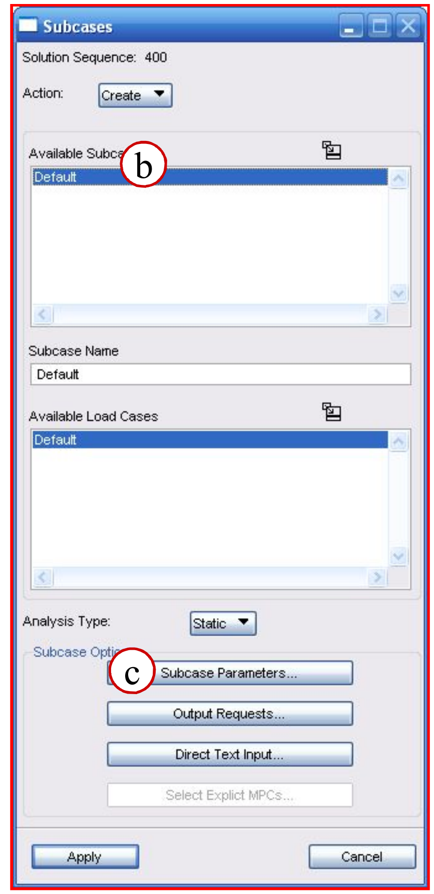


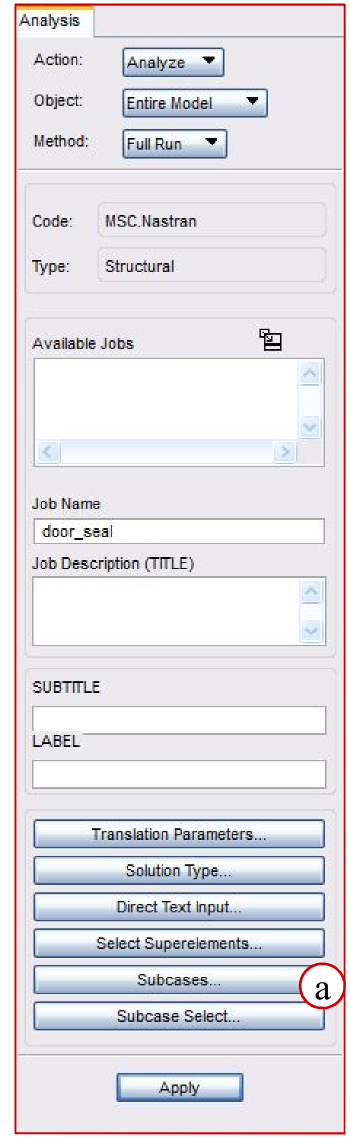
Step 14. Set up Analysis (Cont.)

a. Click on **Subcases**.
b. Click on **Default** under *Available Subcases.*
c. Click on **Subcase Parameters**.
d. Verify **Large Displacement/Large Strains** is chosen under *Nonlinear Geometric Effects*.
e. Click on **Load Increment Parameters**.
f. Pull down **Adaptive** for *Increment type.*
g. Click **OK**.
h. Click **Iteration Parameters**.



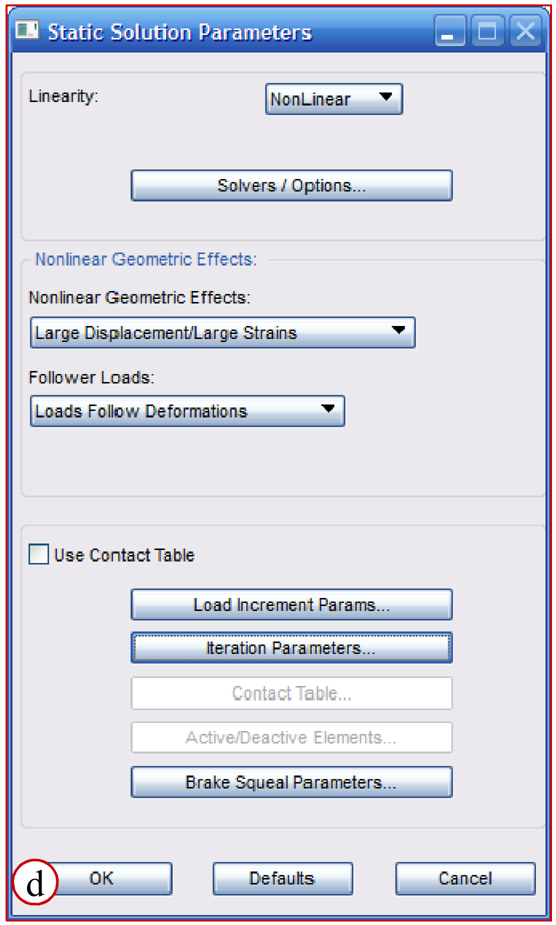


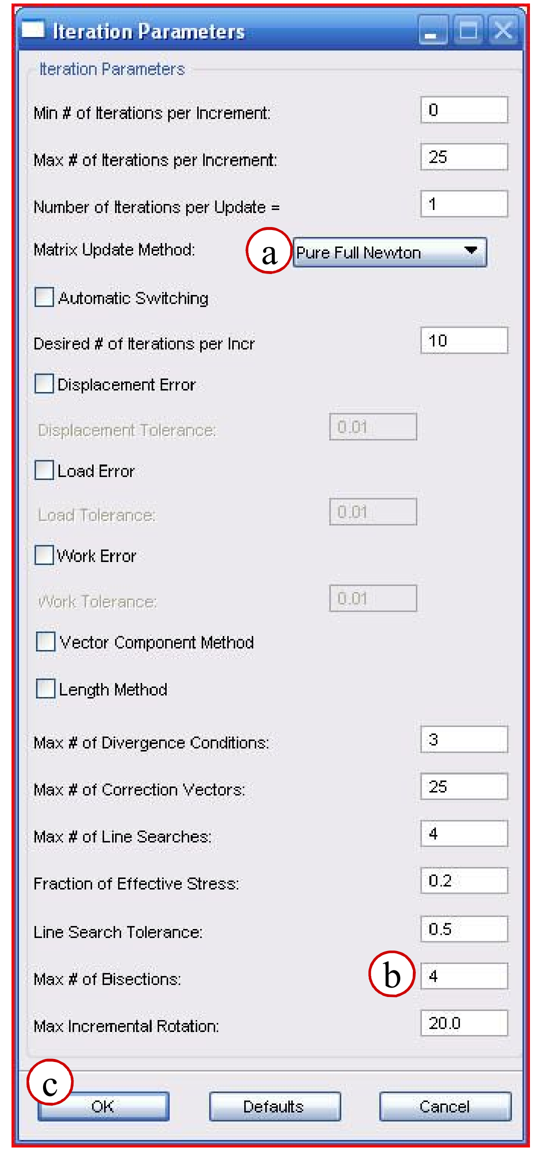




Step 14. Set up Analysis (Cont.)

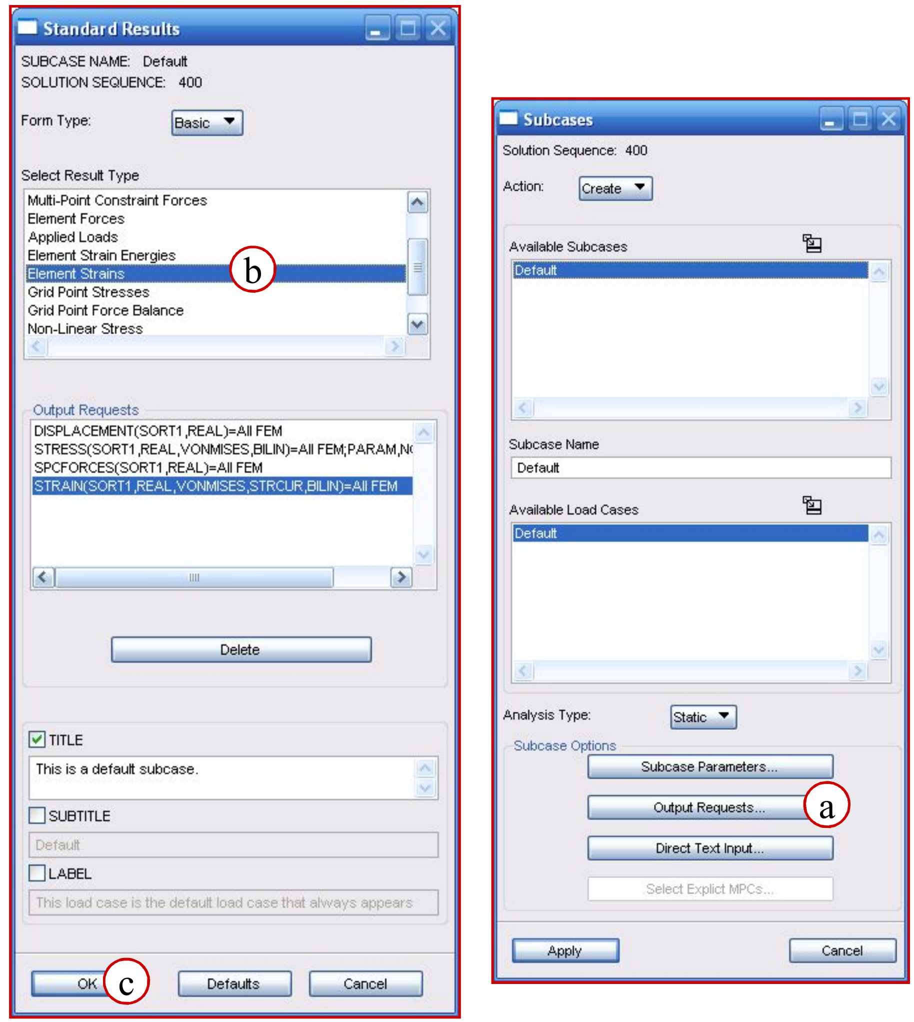
a. Select **Pure Full Newton** as *Matrix Update Method.*
b. Enter ***4*** as *Max # of Bisections*.
c. Click **OK**.
d. Click **OK**.





Step 14. Set up Analysis (Cont.)

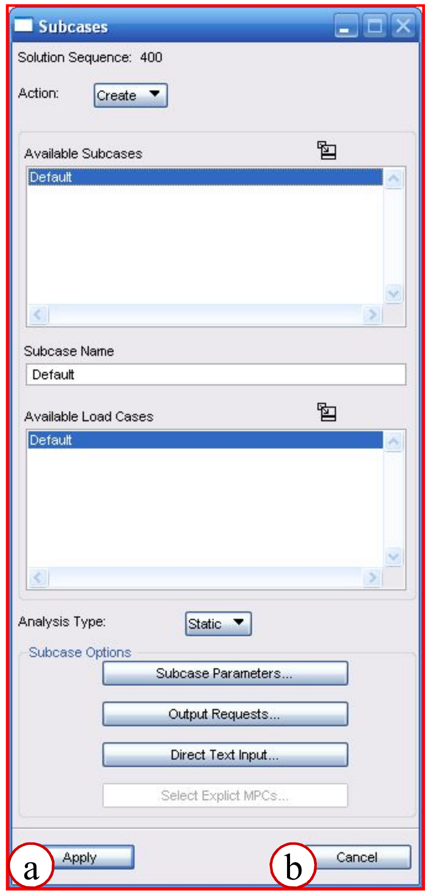
a. Click **Output Requests**.
b. Select **Element Strains**.
c. Click **OK**.

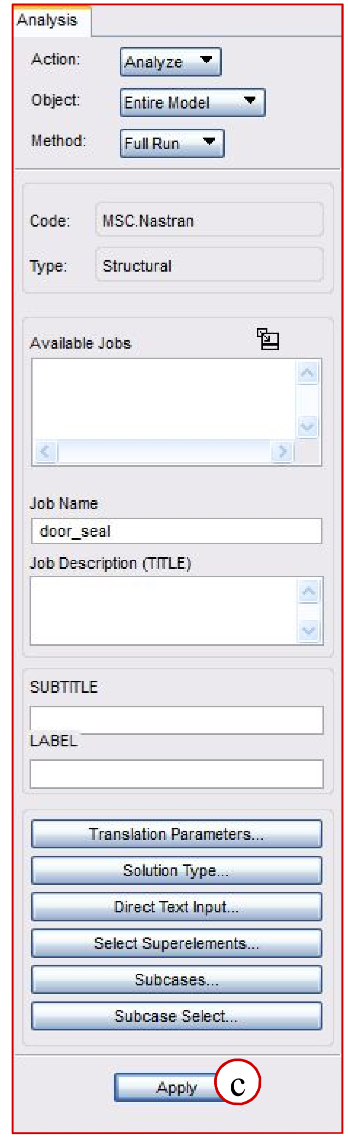


MSC Software

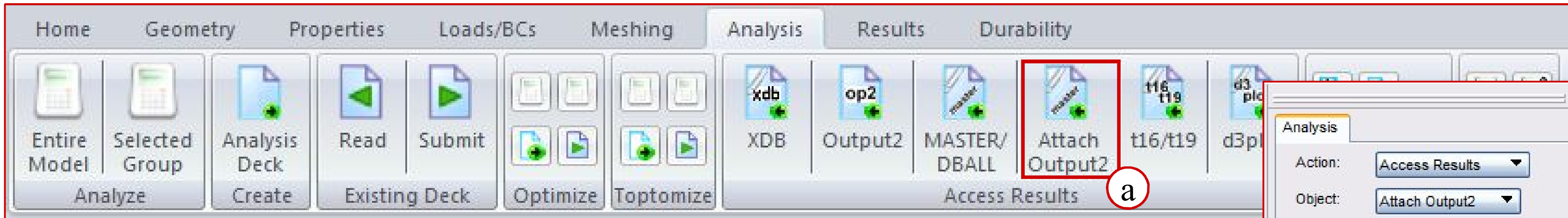
Step 14. Set up Analysis (Cont.)

a. Click **Apply**.
b. Click **Cancel**.
c. Click **Apply**.





Step 15: Access the Nonlinear Results



Read (Attach) results.

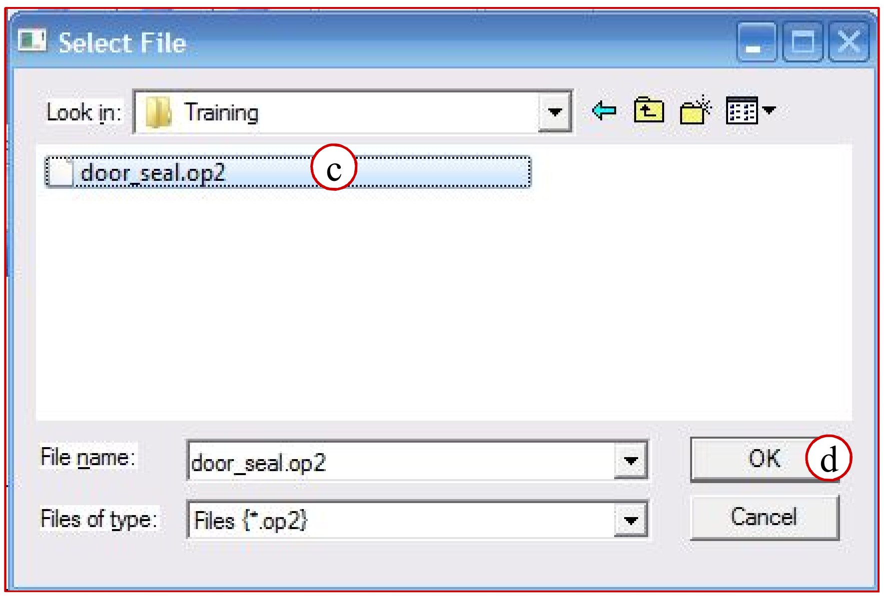
a. Under the *Analysis* tab, click **Attach Output2** in the *Access Results* group.

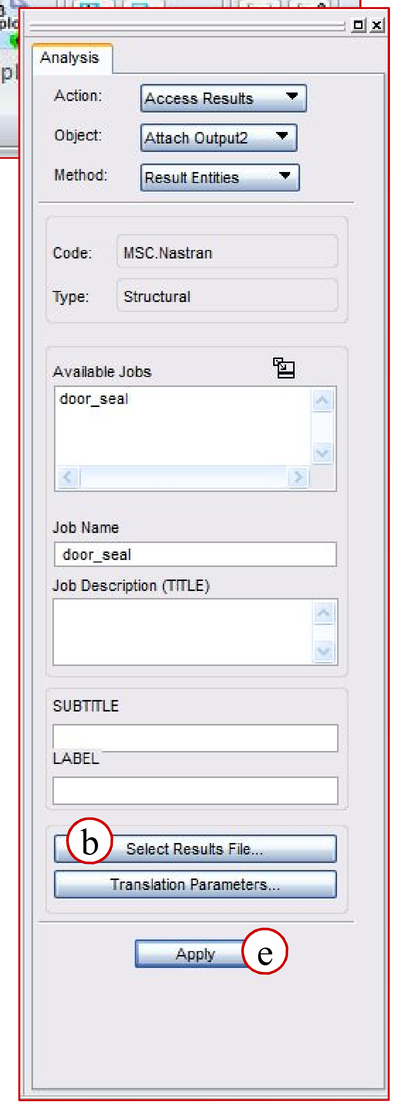
b. Click **Select Results File...**

c. Select ***door_seal.op2***.

d. Click **OK**.

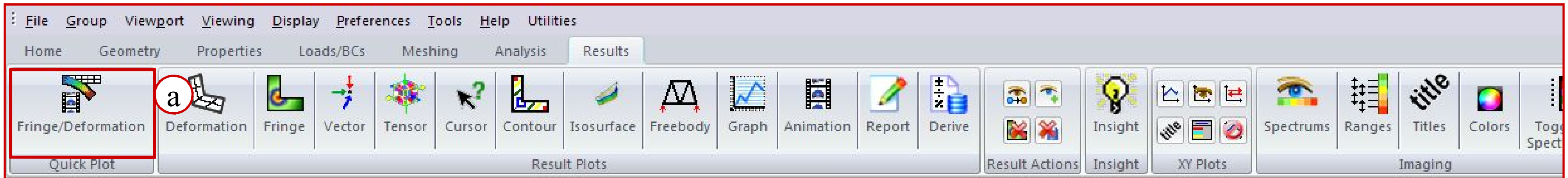
e. Click **Apply**.





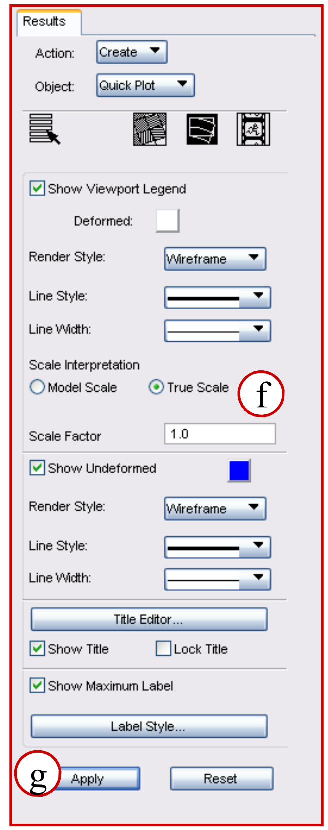
MSC Software

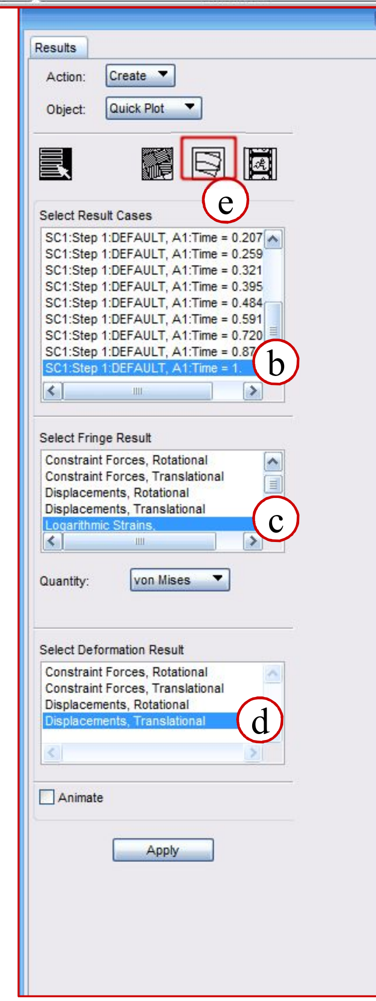
Step 16: View Results



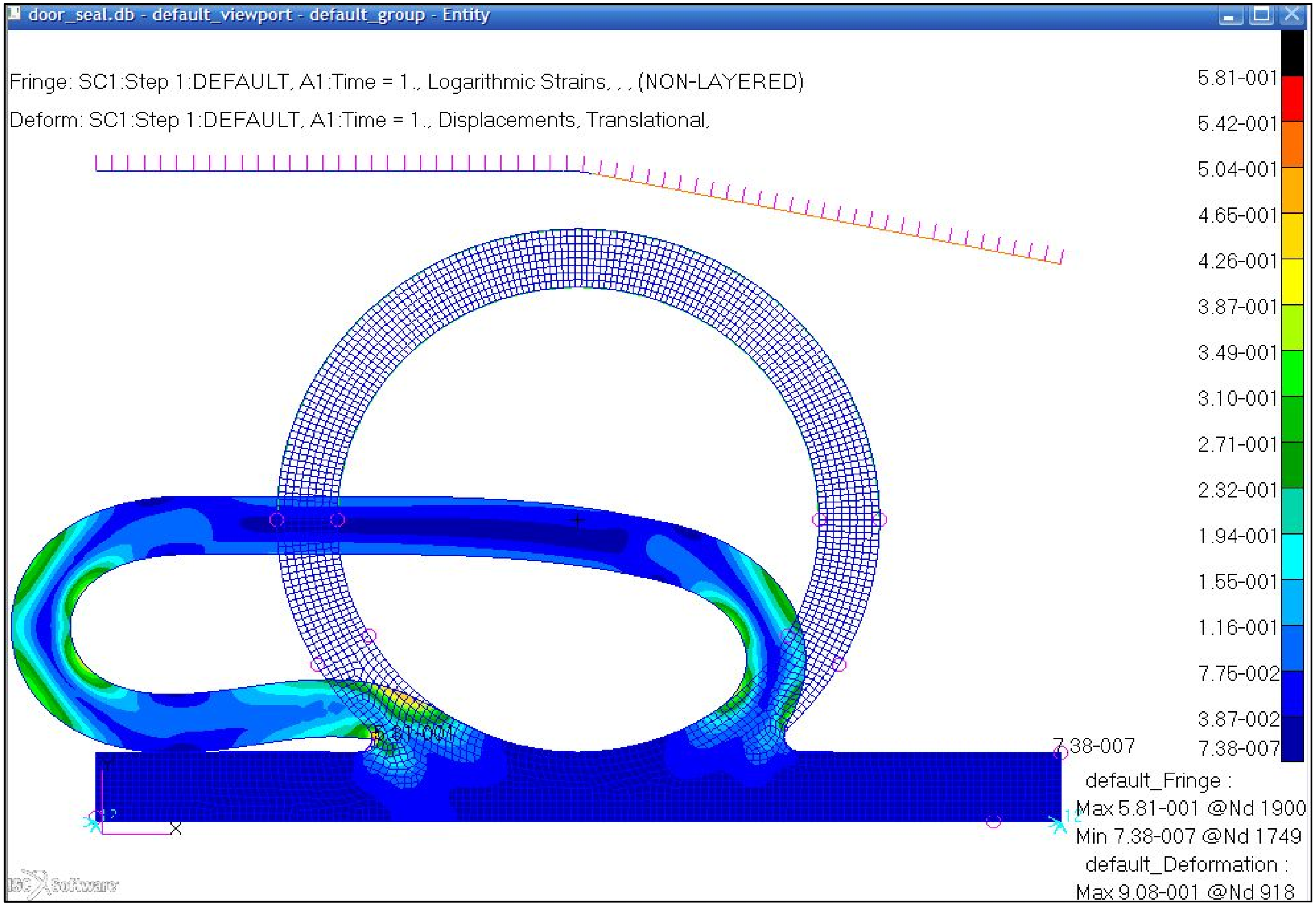
To create a fringe results plot:

a. Under the *Results* tab, click **Fringe/Deformation** in the *Quick Plot* group.
b. Select the last result case.
c. Select **Logarithmic Strains,** as the *Fringe Result.*
d. Select **Displacements, Translational**.
e. Click the **Deform Attributes** icon.
f. Change the *Scale Interpretation* to **True Scale**.
g. Click **Apply**.



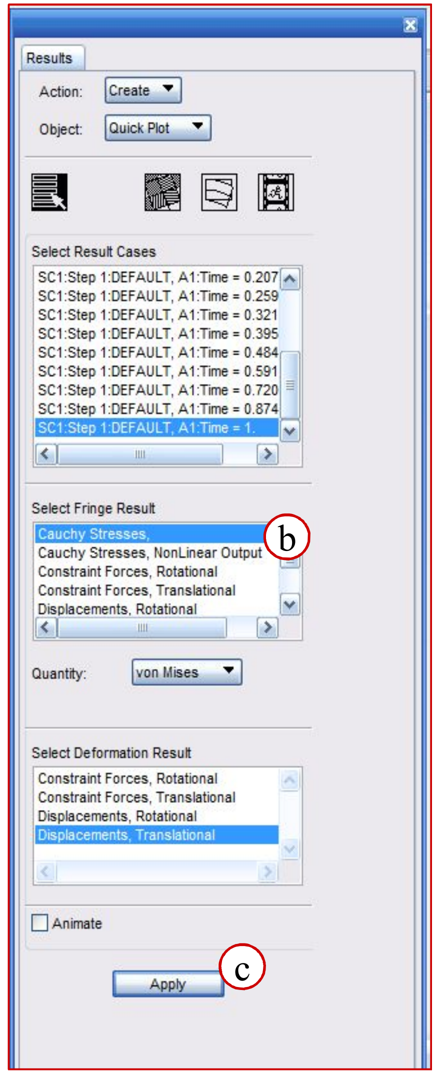


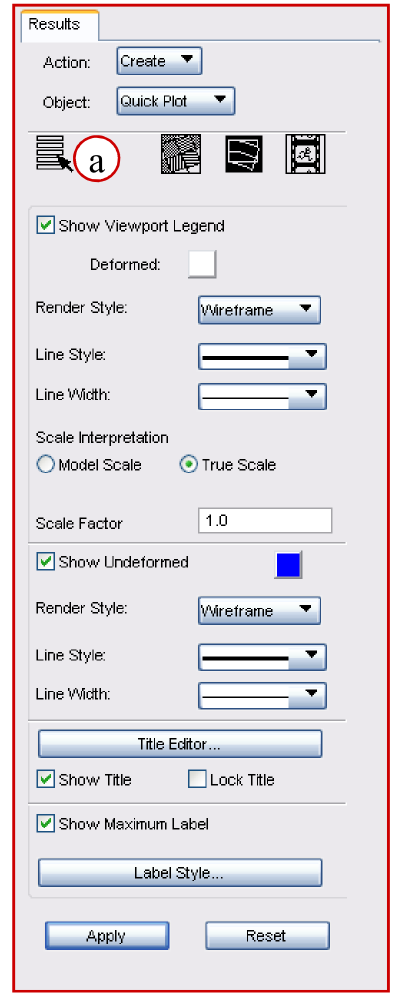
Step 16: View Results (Cont.)



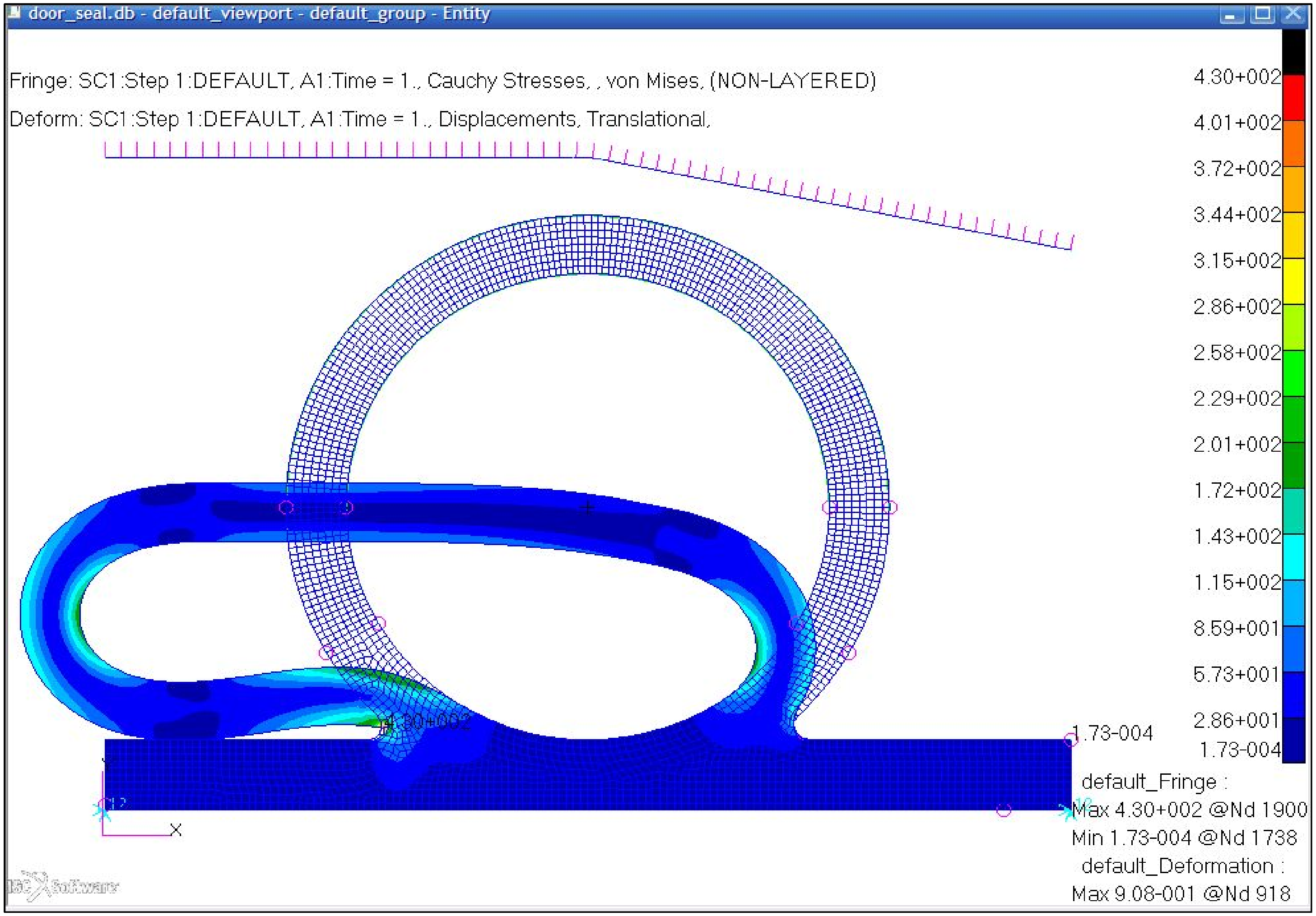
Step 16: View Results (Cont.)

a. Click the **Select Results** icon
b. Select **Cauchy Stresses** as the *Fringe Result*
c. Click **Apply**.

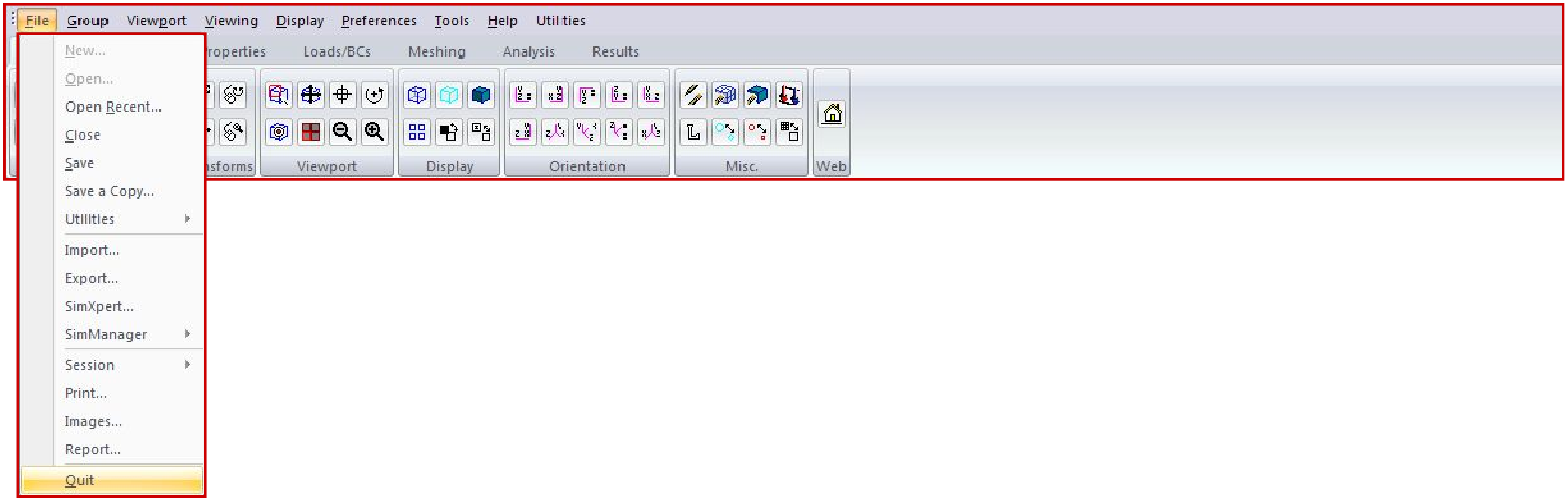




Step 16: View Results (Cont.)



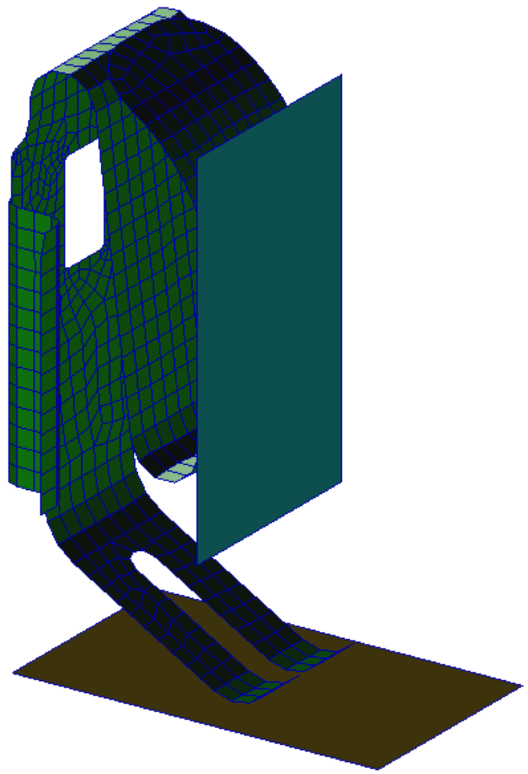
Step 17. Quit Patran



MSC Software

WORKSHOP 3

DEFORMABLE TO RIGID CONTACT

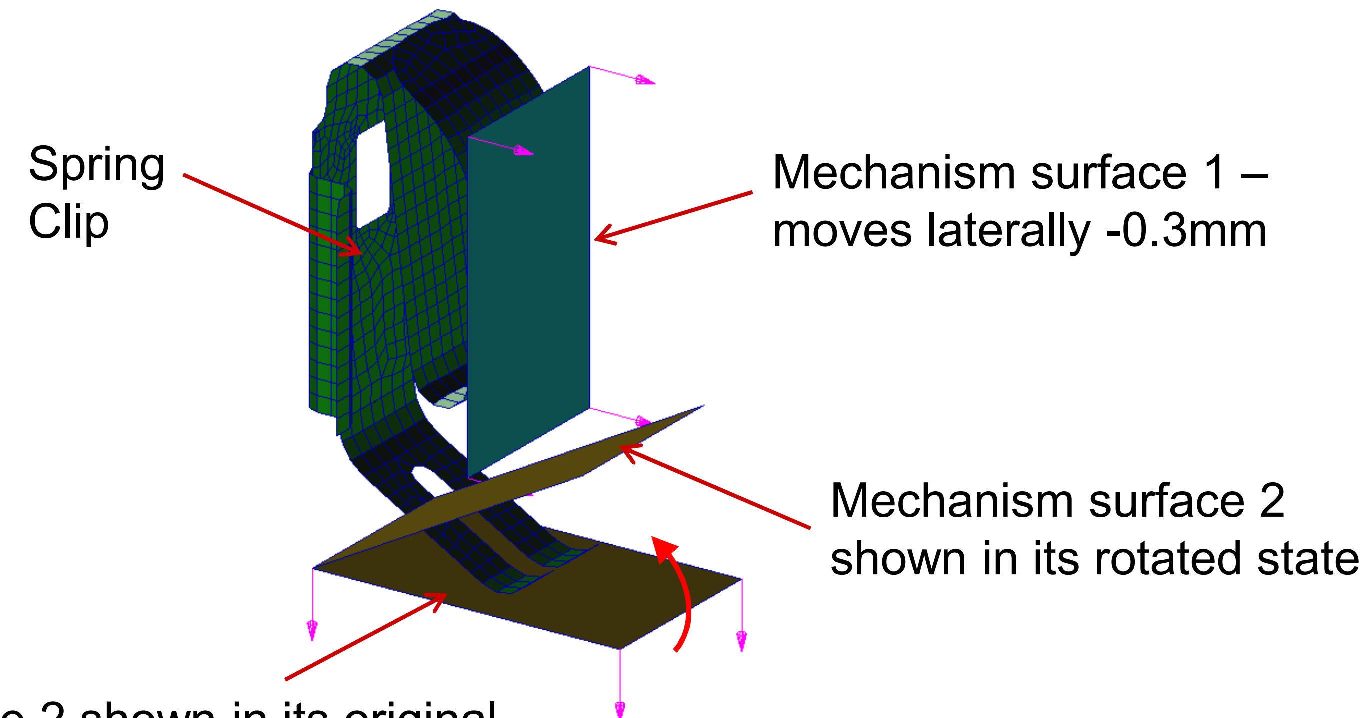


- **Workshop Objectives**
  - To perform a start to finish contact analysis with MSC Nastran that uses both deformable and rigid contact bodies
  - To gain experience in the definition of the motion of rigid contact bodies
  - To gain familiarity with the Patran contact analysis GUI

- **Software Version**
  - MSC Nastran 2013.1
  - Patran 2013

- **Required Files**
  - latch.db

- **Problem Statement**
  - This problem will simulate the closing of a latch mechanism and will include a spring clip that is compressed by moving surfaces of the closing door mechanism
  - Because of the high relative flexibility of the spring when compared to the door, the door surfaces will be represented with rigid contact bodies



Mechanism surface 2 shown in its original location – rotates about its left edge .5 radians.

- **Suggested Steps**
  1. Open the database **latch.db**
  2. Create a deformable contact body for the spring clip
  3. Create rigid contact bodies for the 2 rigid surfaces including prescribed motions
  4. Create contact pairs between the clip and the rigid bodies
  5. Fix the spring clip around the rectangular hole
  6. Set up and run the analysis.
     - Set Solution Type to SOL400 – implicit nonlinear
     - Request OP2 for Results Output Format.
     - Request contact results output
     - Run the analysis
  7. Review the MSC Nastran f06 file for errors.
  8. Correct the normal direction of contact
  9. Attach the OP2 results file
  10. Plot deformation and von Mises stress results
  11. Make a Marker Plot of Contact Status
  12. Plot Normal Contact Force as a Vector Plot
  13. Display a Fringe Plot of Contact Status
  14. Display a Fringe Plot of Normal Contact Stress

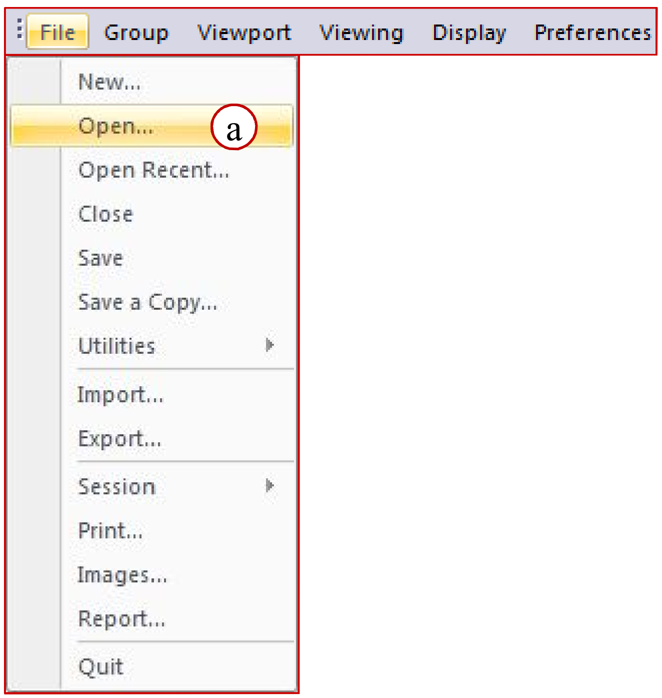
Step 1. Open the Existing Database

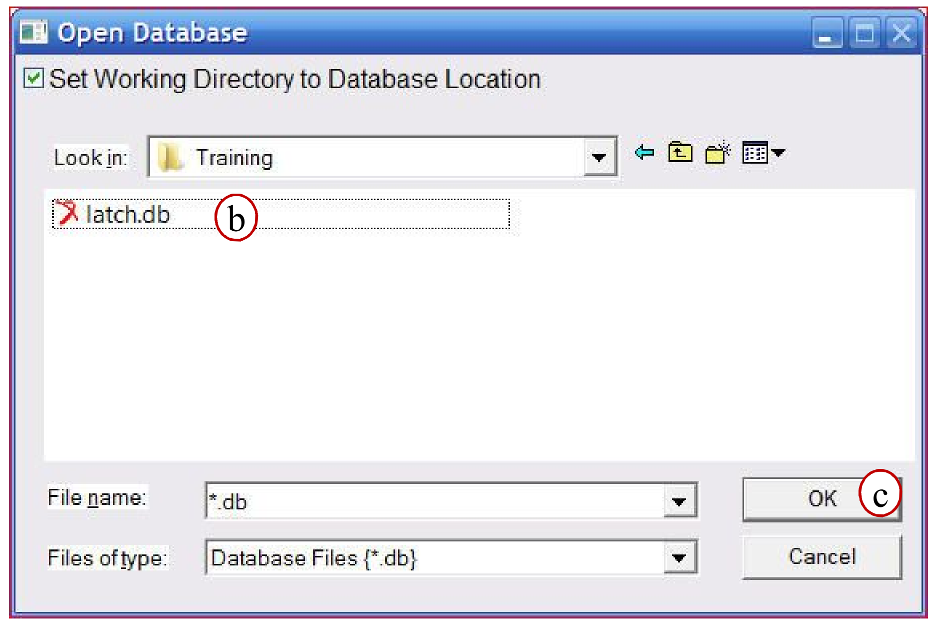
Open the database **latch.db**:

a. Pull down **File** > **Open**.

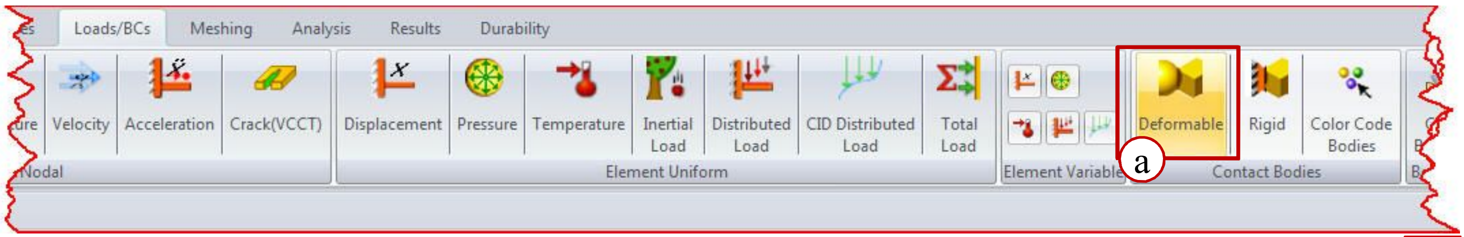
b. Select **latch.db** as the database file.

c. Click **OK**.



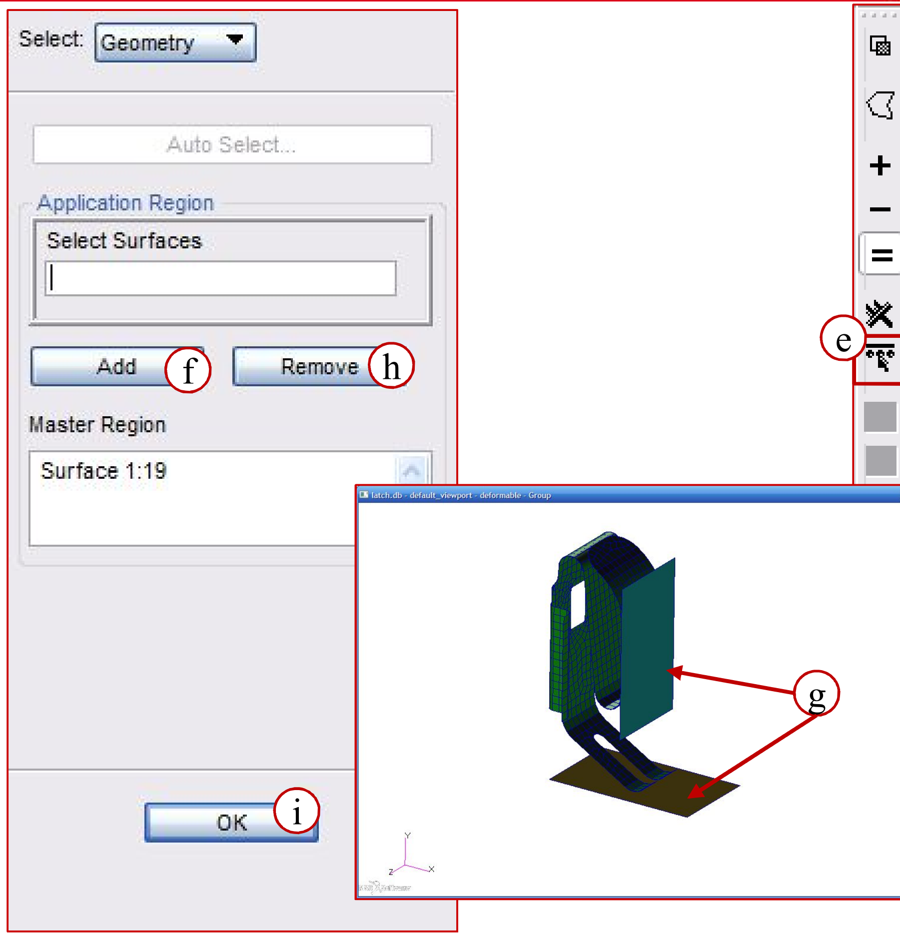


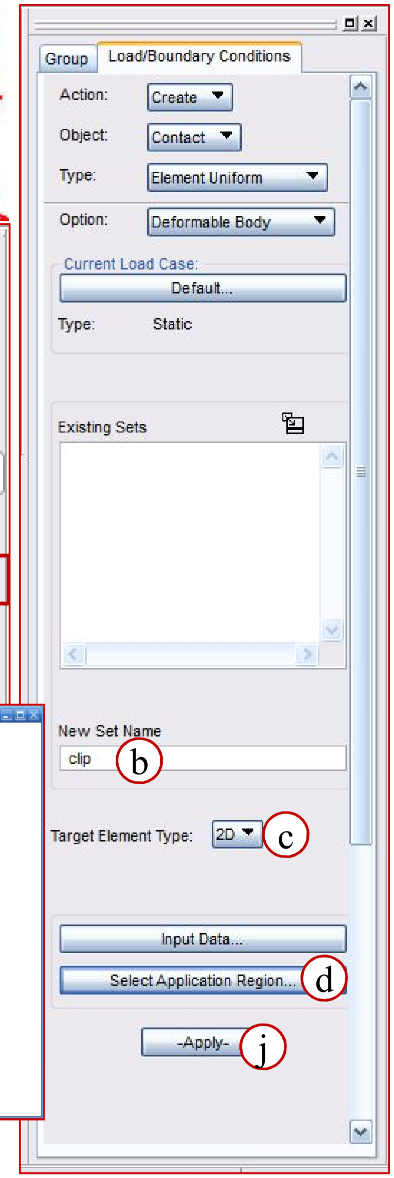
Step 2. Create Deformable Body



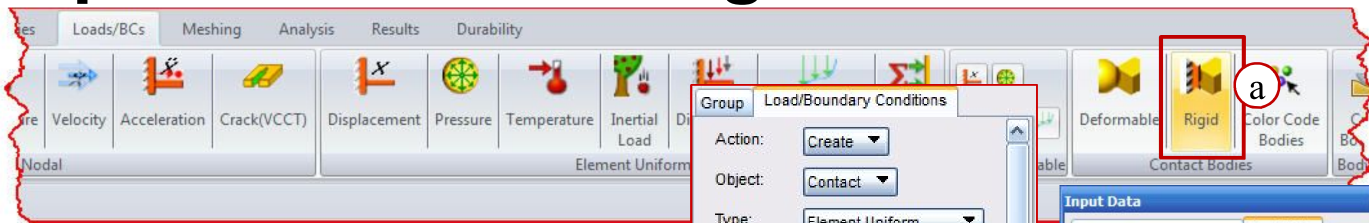
Create a deformable contact body for the spring clip:

a. Under the *Loads/BCs* tab, click **Deformable** in the *Contact Bodies* group.
b. Enter **clip** for *New Set Name*.
c. Pull down *Target Element Type* to **2D**.
d. Click **Select Application Region**.
e. Click **Pick All** to select all the surfaces.
f. Click **Add**.
g. Select the two mechanism surfaces: **Surfaces 20** and **21**.
h. Click **Remove.**
i. Click **OK**.
j. Click **Apply**.



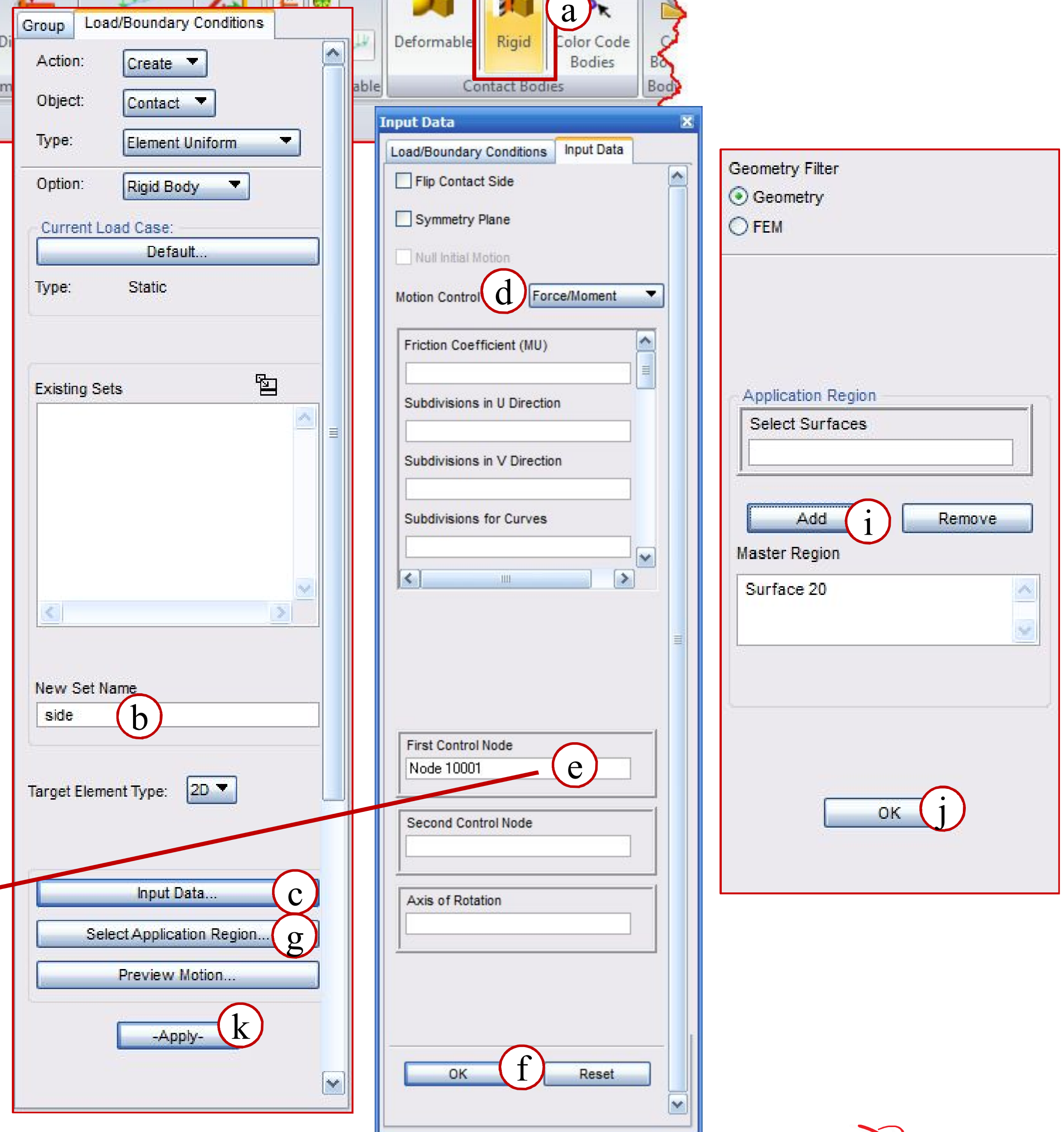


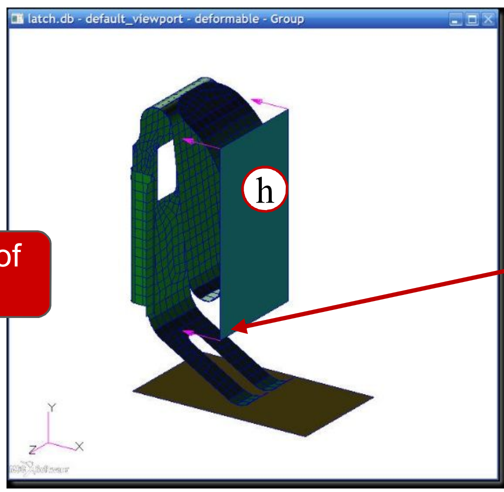
Step 3. Create Two Rigid Contact Bodies



Create rigid contact bodies for the 2 mechanism surfaces including prescribed motions from the Problem Description:

a. Click **Rigid** in the *Contact Bodies* group.
b. Enter **side** for *New Set Name*.
c. Click **Input Data.**
d. Pull down *Motion Control* to **Force/Moment**
e. Select Node on Corner of Side Surface
f. Click **OK.**
g. Click **Select Application Region.**
h. Select the side surface.
i. Click **Add.**
j. Click **OK.**
k. Click **Apply.**





Note the direction of the normal vectors

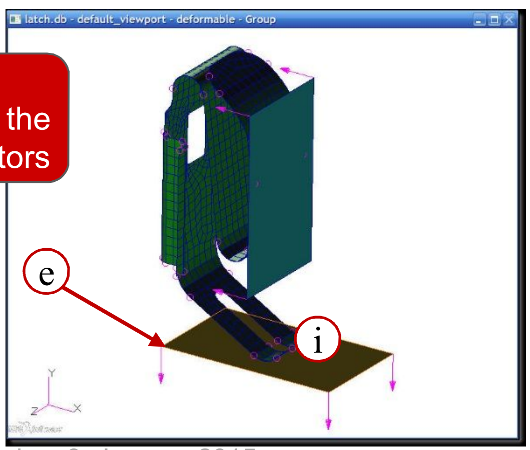
MSC Software

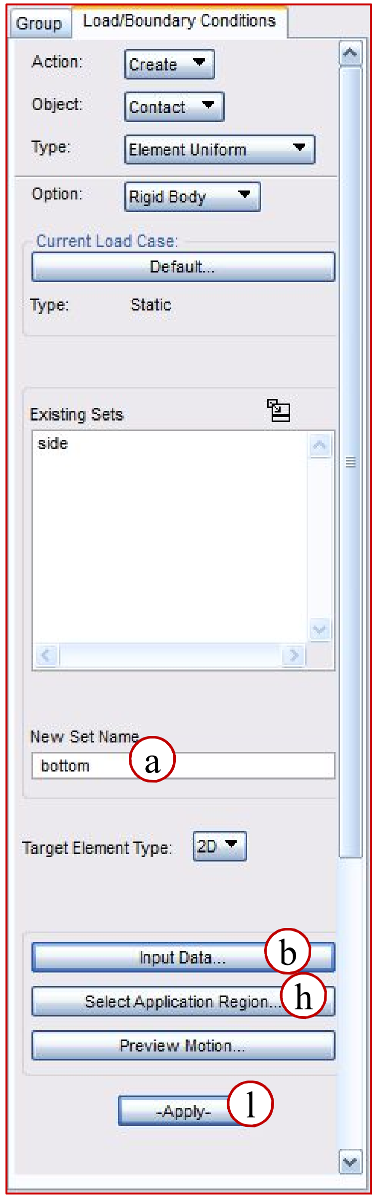
Step 3. Create Two Rigid Contact Bodies (Cont.)

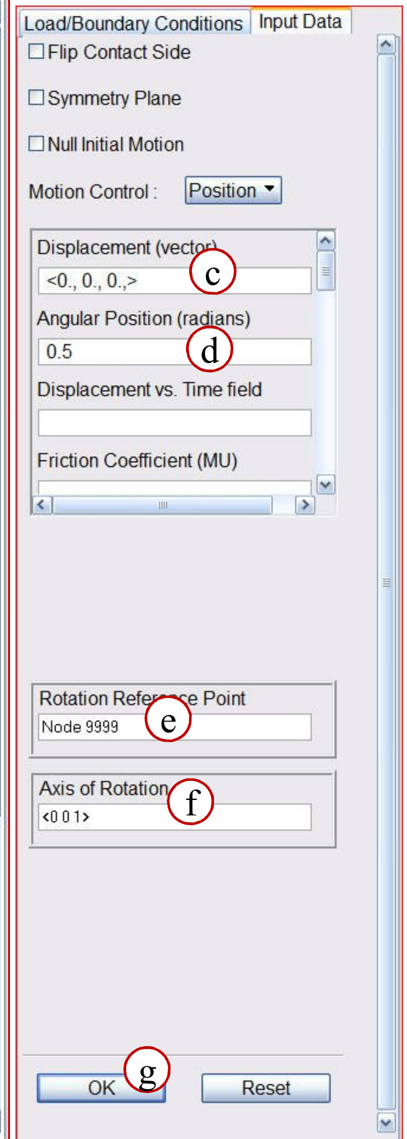
Create the rigid contact bodies (cont.):

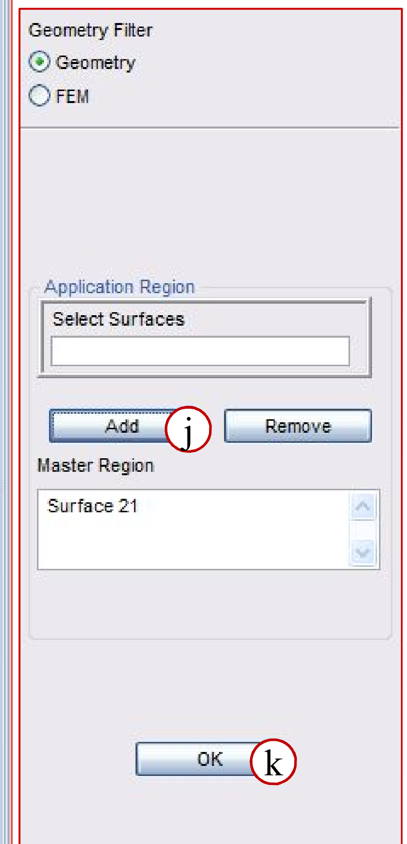
a. Enter **bottom** for *New Set Name*.
b. Click **Input Data**.
c. Pull down *Motion Control* to **Position**
d. Enter **<0, 0, 0>** for *Displacement(vector).*
e. Enter **0.5** for *Angular Position*.
f. Select **Node 9999** on the corner of the bottom surface for the *Rotation Reference Point*.
g. Enter **<0 0 1>** for the *Axis of Rotation*.
h. Click **OK.**
i. Click **Select Application Region**.
j. Select the bottom surface.
k. Click **Add**.
l. Click **OK**.
m. Click **Apply**.

Note the direction of the normal vectors

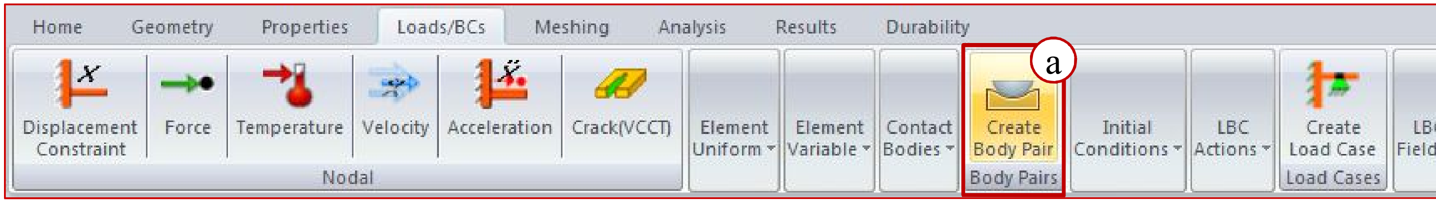






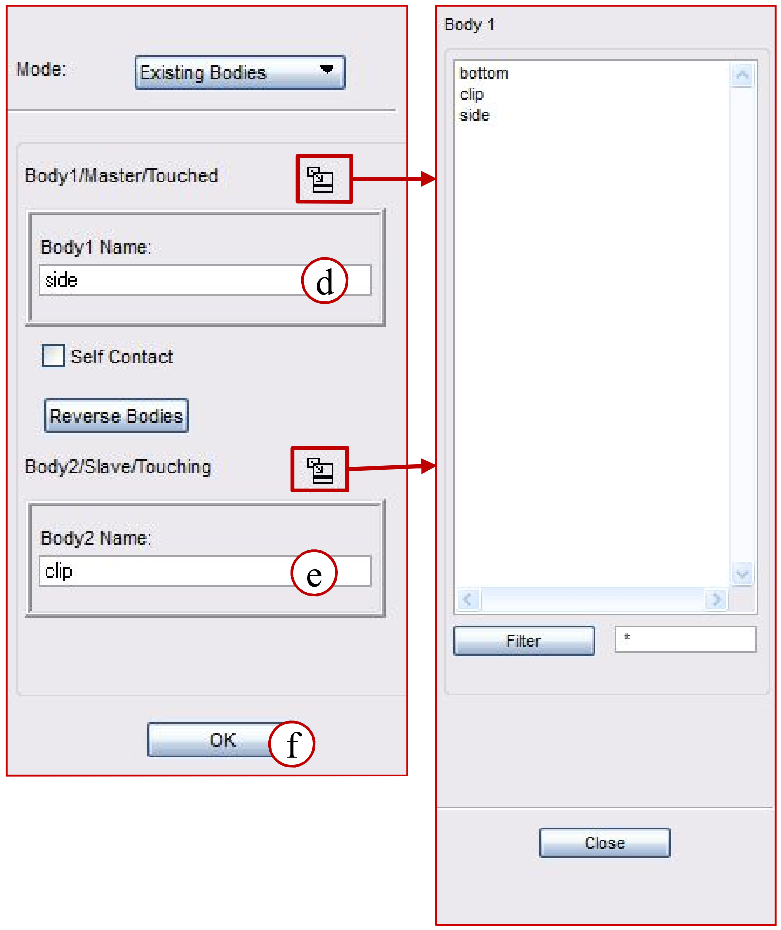


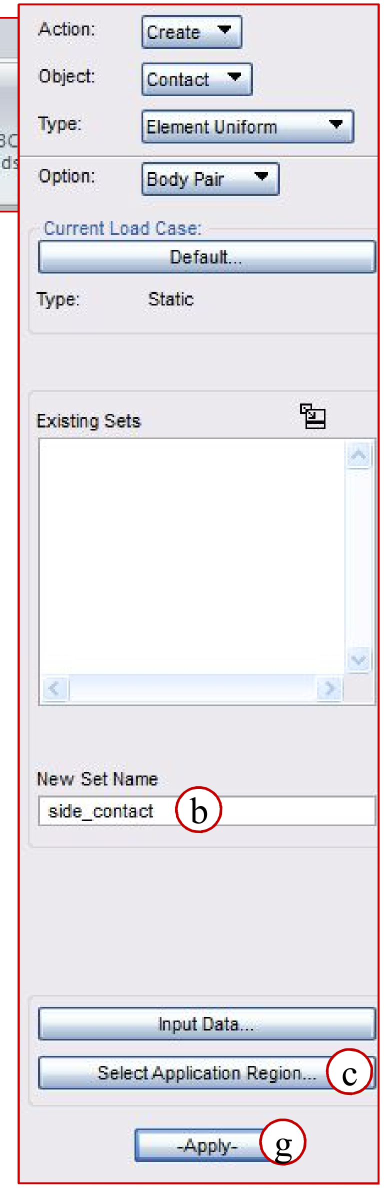
Step 4. Create Contact Pairs



Create contact pairs between the clip and rigid bodies:

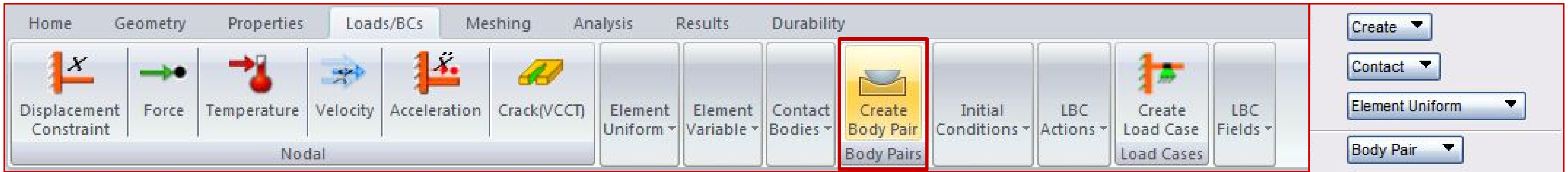
a. Click **Create Body Pair** in the *Body Pairs* group.
b. Enter **side_contact** for *New Set Name*.
c. Click **Select Application Region**.
d. Select the **side** contact body for *Body1/Master/Touched*.
e. Select the **clip** contact body for *Body2/Slave/Touching*.
f. Click **OK**.
g. Click **Apply**.





MSC Software

Step 4. Create Contact Pairs (Cont.)



Create the contact pairs between the clip and rigid bodies (cont.):

a. Enter **bottom_contact** for *New Set Name*.

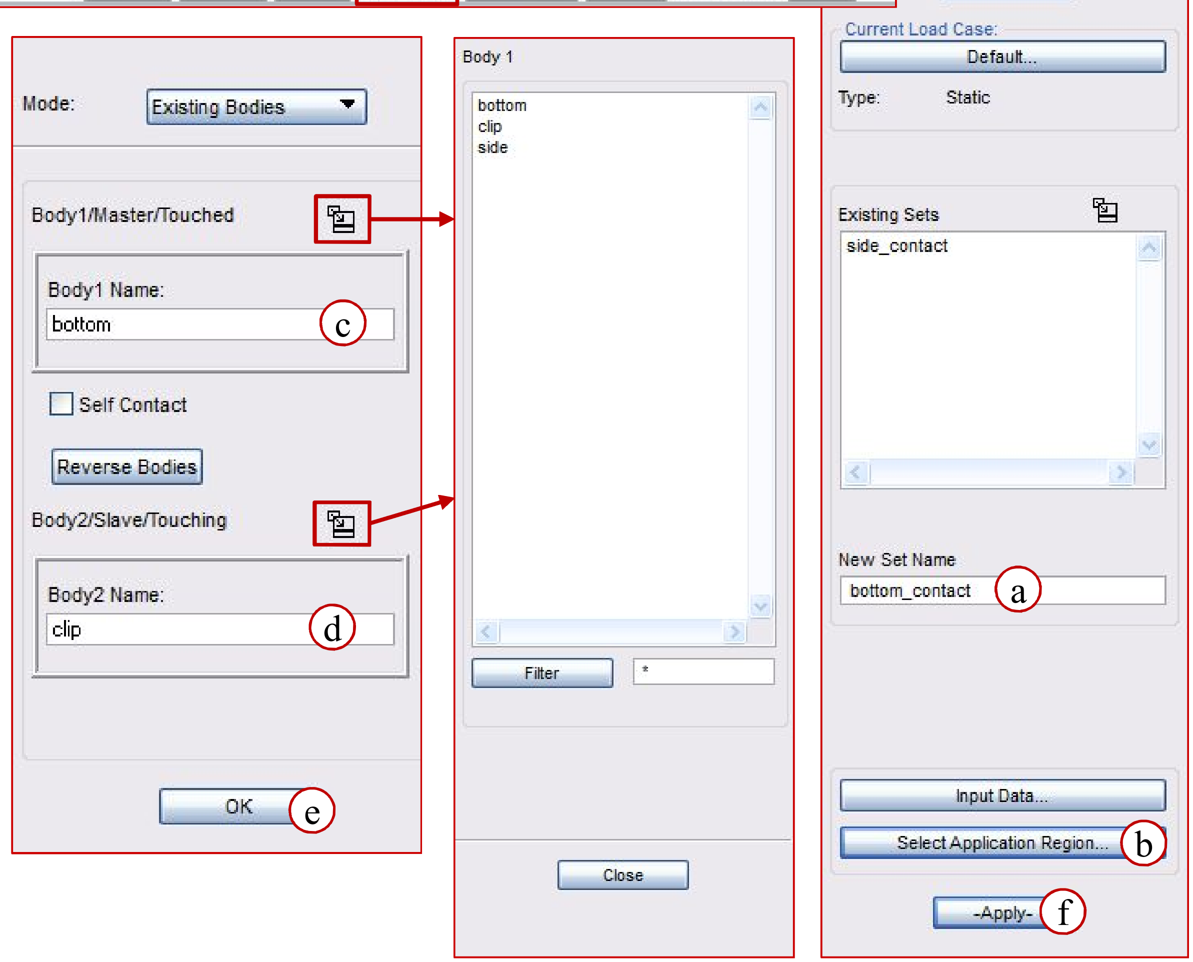
b. Click **Select Application Region**.

c. Select the **bottom** contact body for *Body1/Master/Touched*.

d. Select the **clip** contact body for *Body2/Slave/Touching*.

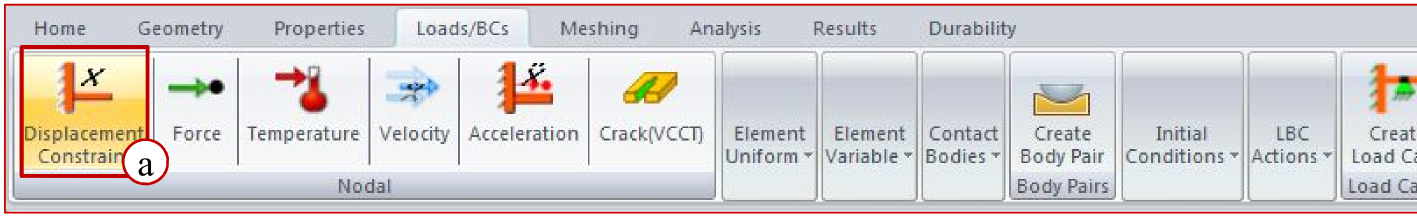
e. Click **OK**.

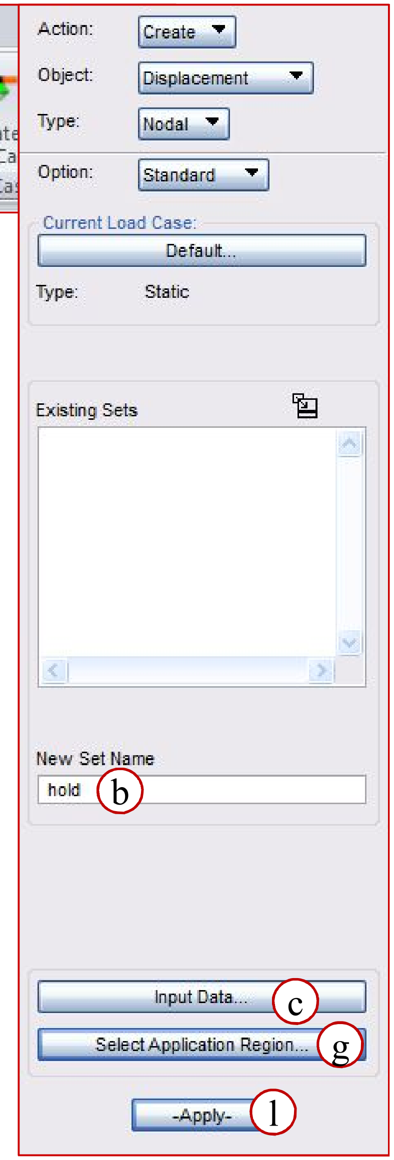
f. Click **Apply**.



MSC Software

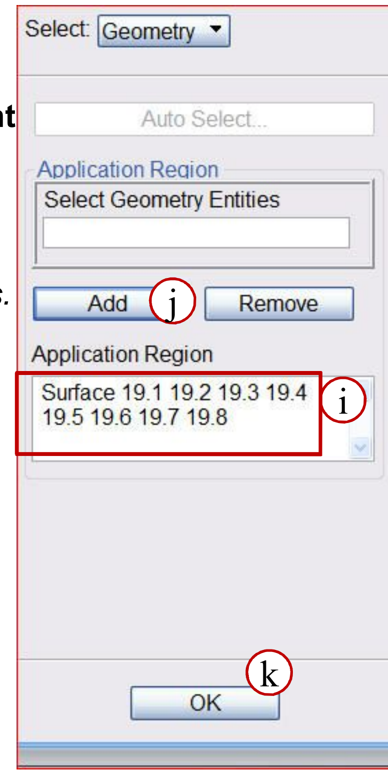
Step 5. Apply Loads and Boundary Conditions

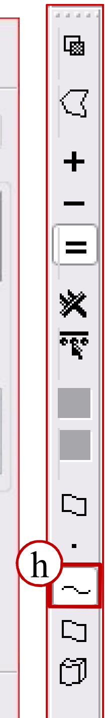


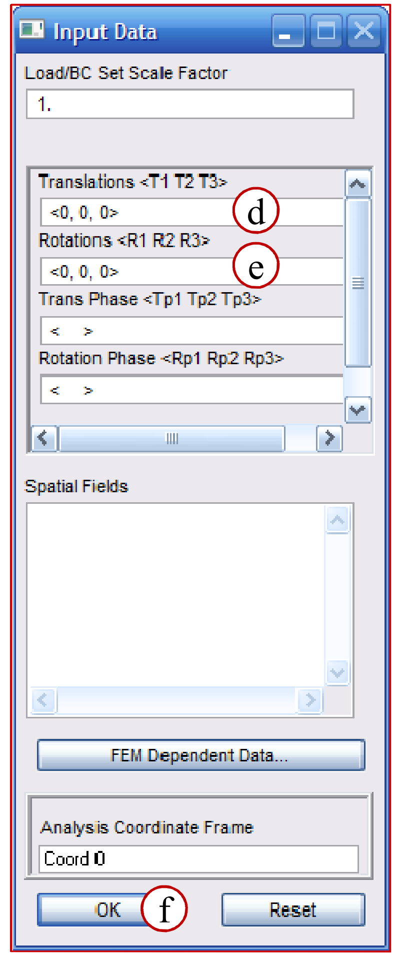


Fix the spring clip around the rectangular hole:

a. Click **Displacement Constraint** in the *Nodal* group.
b. Enter **hold** for *New Set Name*.
c. Click **Input Data**.
d. Enter **<0, 0, 0>** for *Translations*.
e. Enter **<0, 0, 0>** for *Rotations*.
f. Click **OK**.
g. Click **Select Application Region**.
h. Click **Curve or Edge** on the *Picking Filters Toolbar*.
i. Select the surface edges that bound the hole.
j. Click **Add**.
k. Click **OK**.
l. Click **Apply**.





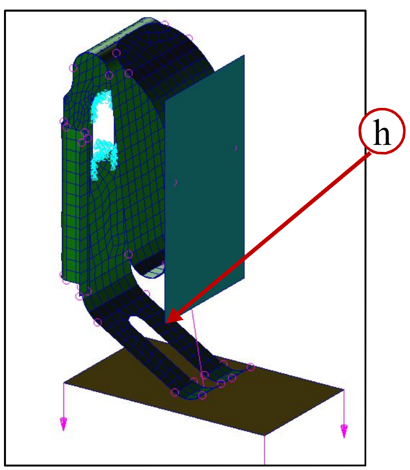


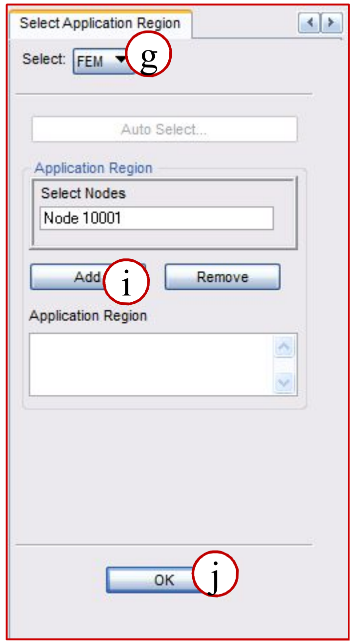
MSC Software

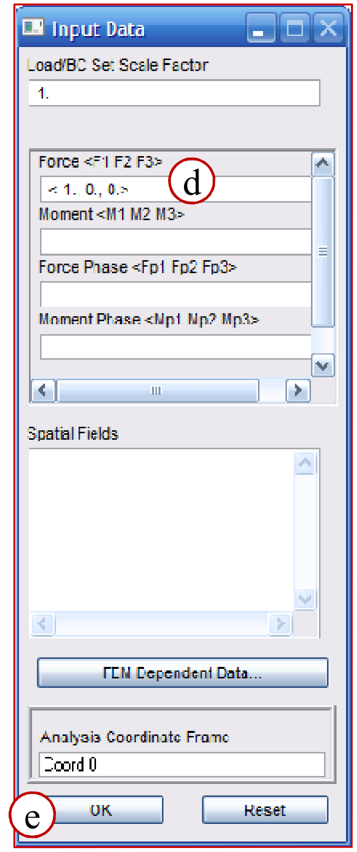
Step 5. Apply Loads & BCs (Cont.)

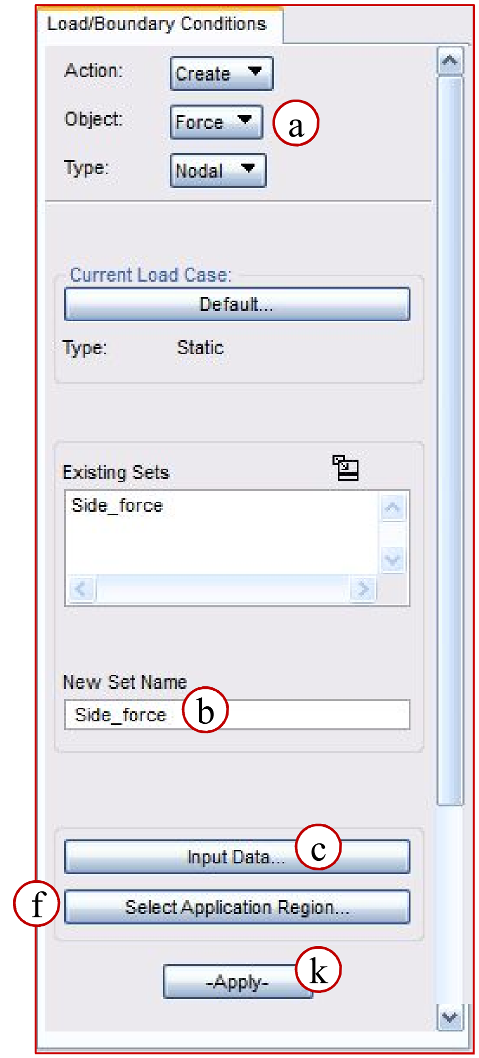
Apply Node Force <-1,0,0> to the Control Node of the Side Surface

a. Pull down *Object* > **Force**
b. Enter Name **Side_force**
c. Click Input Data
d. Enter **<-1,0,0>** for *Force*
e. Click **OK**
f. Click **Select Application Region…**
g. Pull down *Select* to **FEM**
h. Select Control Node of Side
i. Click **Add**
j. Click **OK**
k. Click **Apply**

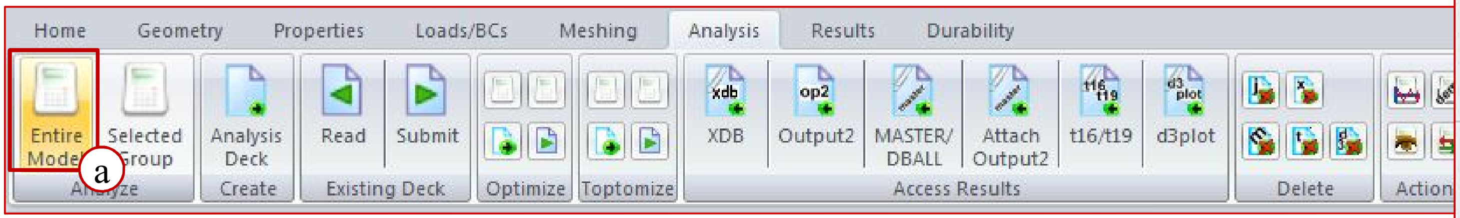


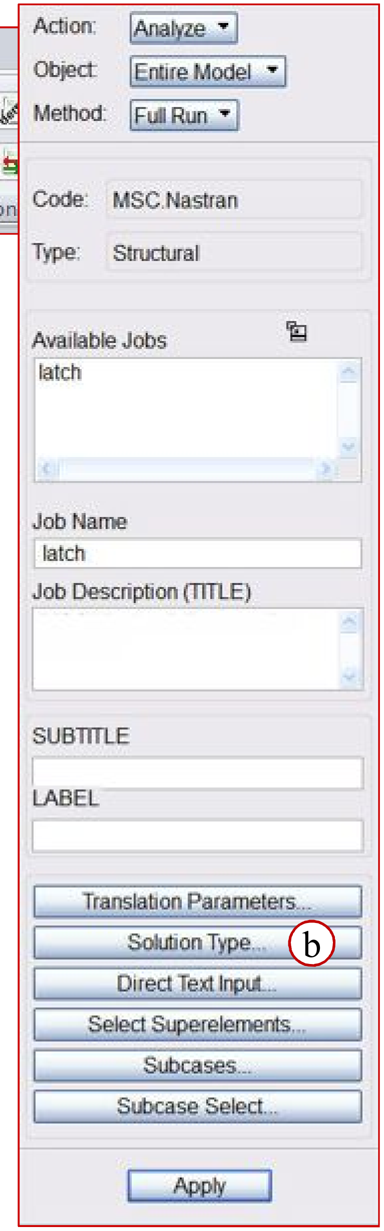






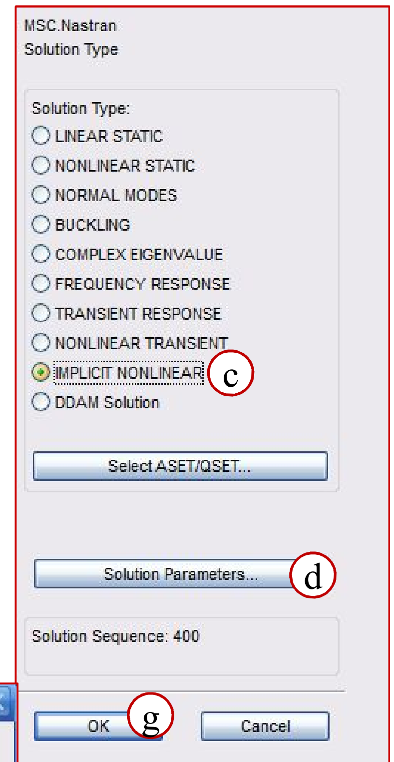
Step 6. Set Up and Run the Analysis

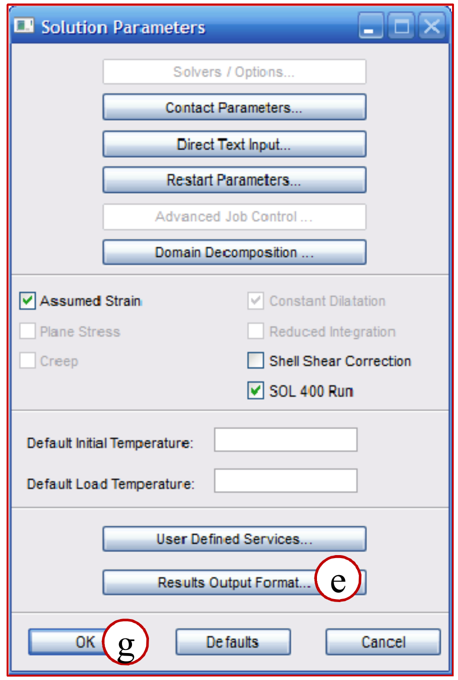




Set Solution Type to SOL400:

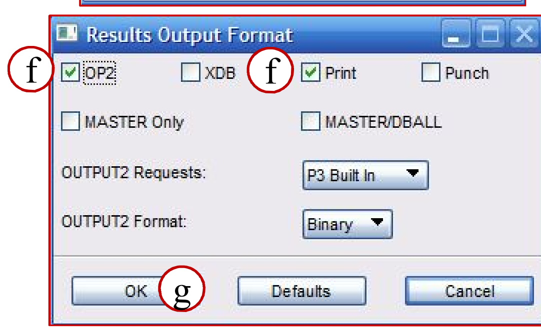
a. Under the *Analysis* tab, click **Entire Model** in the *Analyze* group.
b. Click **Solution Type….**
c. Select **Implicit Nonlinear** for *Solution Type*.
d. Click **Solution Parameters**.





Select the Results Output Format and run the analysis:

e. Click **Results Output Format**.
f. Check *OP2* and *Print*
g. Close all sub-menus by clicking **OK** .



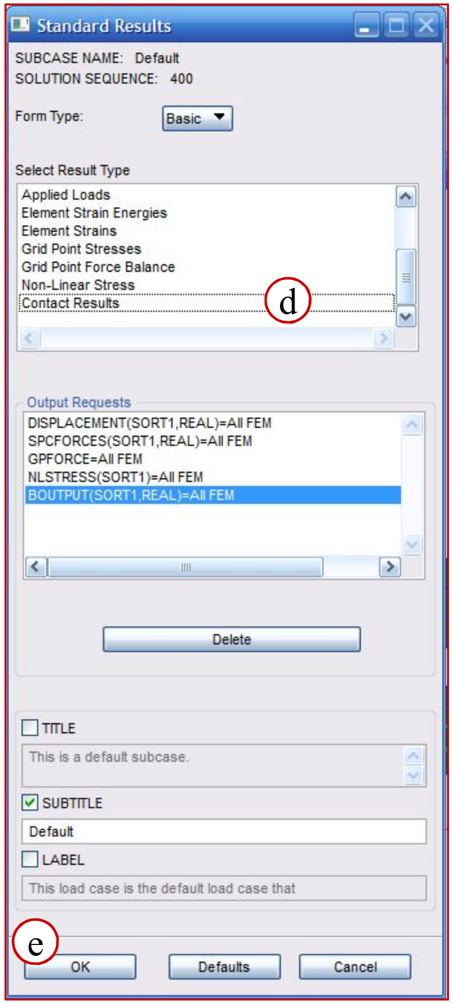
Step 6. Set Up and Run the Analysis (Cont.)

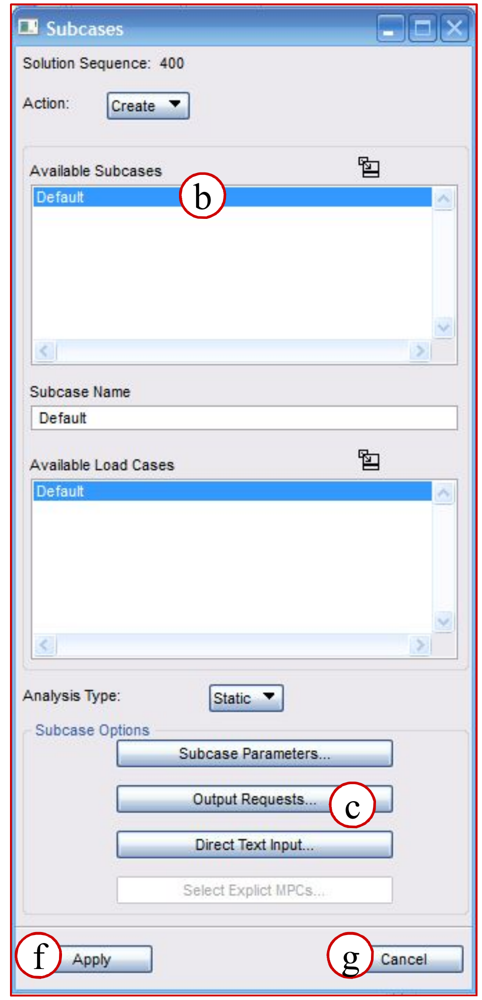
Request contact results output:

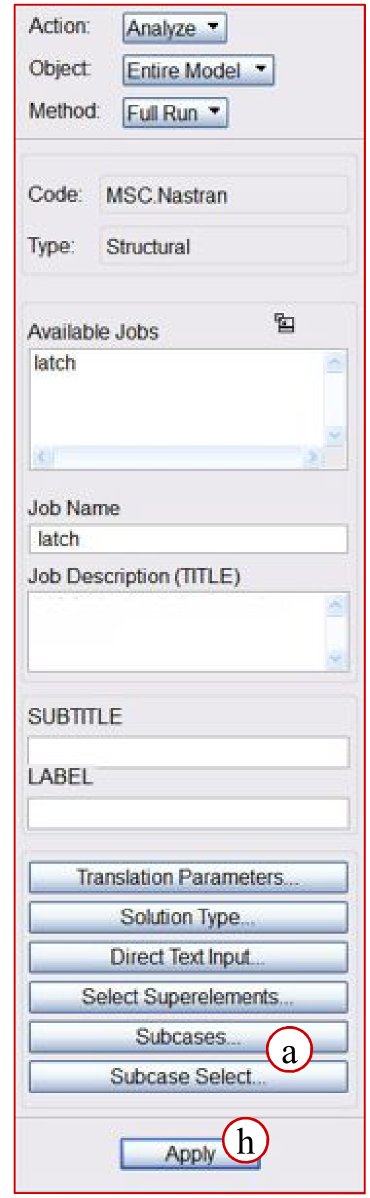
a. Click **Subcases…**
b. Click on **Default** under *Available Subcases*.
c. Click **Output Requests**.
d. Select **Contact Results**.
e. Click **OK**.
f. Click **Apply**.
g. Click **Cancel**.

To run the analysis:

h. Click **Apply**.





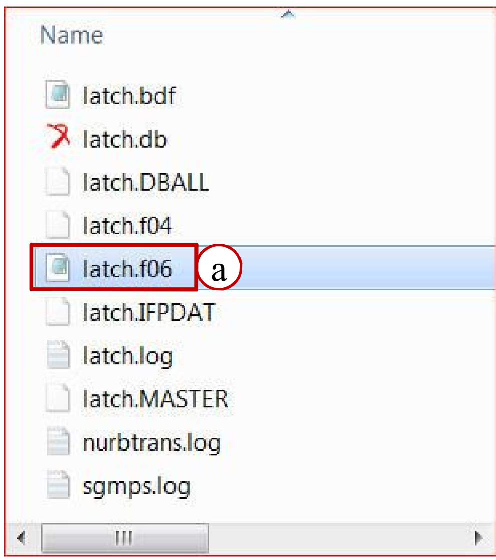


Step 7. Review the MSC Nastran f06 File

Review the MSC Nastran f06 file for errors:

a. In the training directory locate and open the file **latch.f06** with any text editor.

b. Search on the string FATAL and note the messages

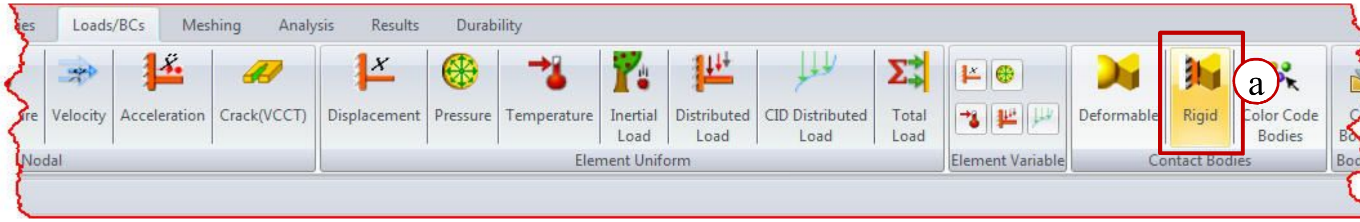


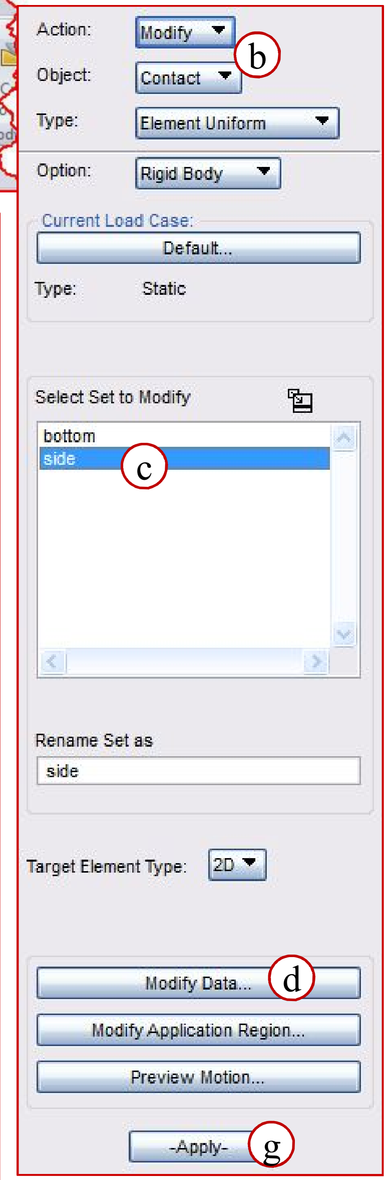
b

```
*** USER INFORMATION MESSAGE 6193 (nl3con)
    *** MAXIMUM NUMBER OF BISECTIONS OR MINIMUM TIME STEP HAS BEEN REACHED ***

 *** JOB DOES NOT CONVERGE AT THE CURRENT TIME STEP OR INCREMENT.
 *** SOLUTION DIVERGES FOR SUBCASE       1 STEP       1.

 ***
 *** USER FATAL MESSAGE 9032 (SUBDMAP NONLIN)
     RUN IS TERMINATED DUE TO MESSAGE(S) ABOVE
```

An error is encountered due to the direction of contact for the ‘side’ contact body.

Step 8. Correct the Direction of Contact

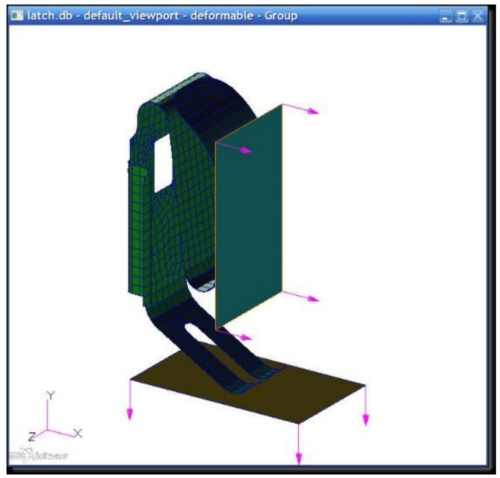




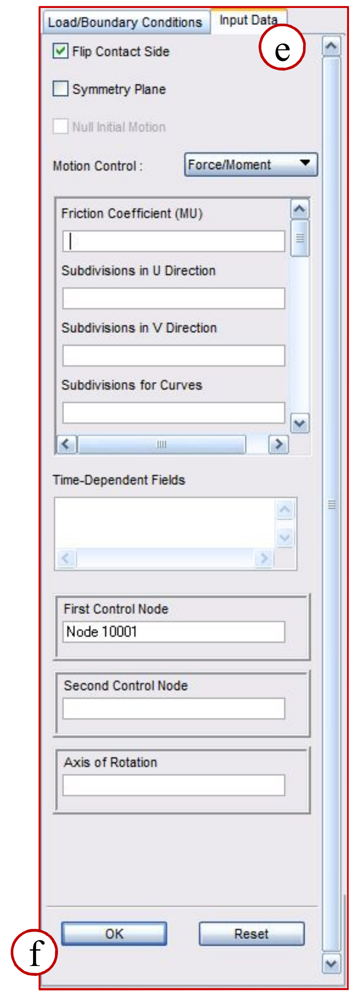
Reverse the contact side for the 'side' rigid body:

a. Under the *Loads/BCs* tab, click **Rigid** in the *Contact Bodies* group.
b. Pull down *Action* to **Modify**. Pull down *Object* to **Contact**
c. Select the **side** contact body.
d. Click **Modify Data**.
e. Check **Flip Contact Side**.
f. Click **OK**.
g. Click **Apply**.

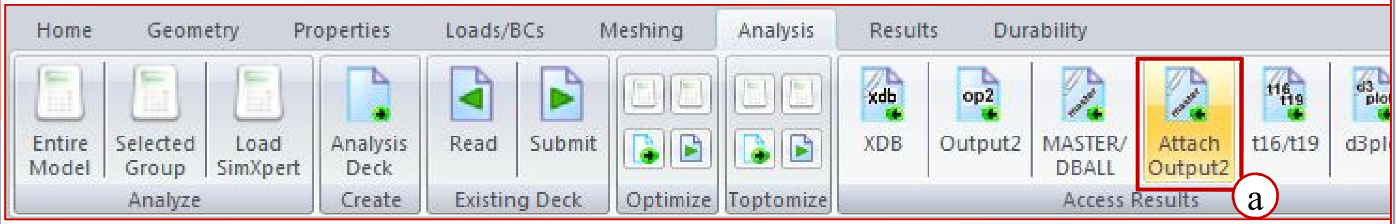
Re-run the analysis (Repeat Step 6).



Note the normal vectors now point away from the clip



Step 9. Attach .op2 Result File



Attach the Master result file:

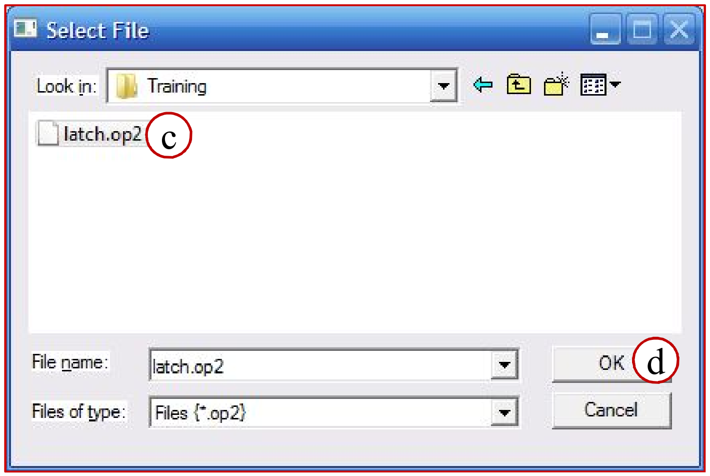
a. Under the *Analysis* tab, click **Attach Output2** in the *Access Results* group.

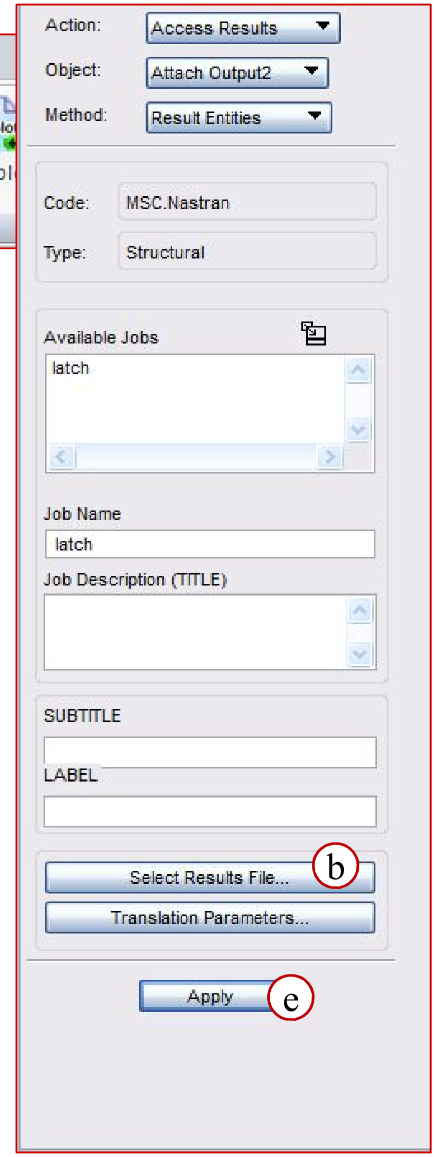
b. Click **Select Results File…**

c. Select the **latch.op2** result file.

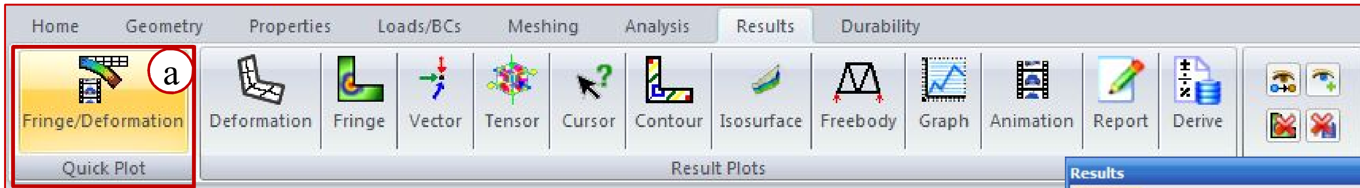
d. Click **OK**.

e. Click **Apply**.





Step 10. Plot the Deformation and Stress Results



Plot deformation and von Mises stress:

a. Under the *Results* tab click **Fringe/Deformation** in the *Quick Plot* group.

b. Select **SC1:…A1:Time = 1.0** for *Select Result Cases.*

c. Select **Nonlinear Stresses, Stress Tensor** for *Select Fringe Result*.

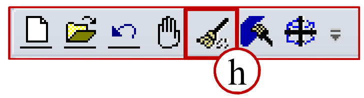
d. Select **Displacements, Translational** for *Select Deformation Result*.

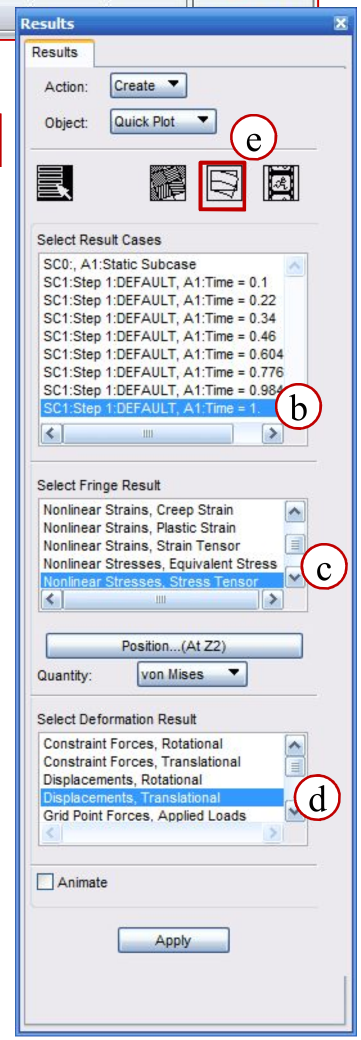
e. Click the **Deform Attributes** icon

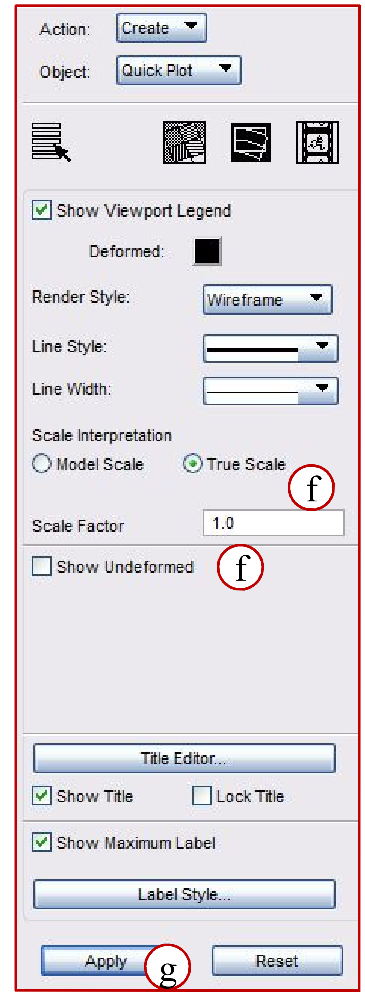
f. Select **True Scale** and Uncheck **Show Undeformed**

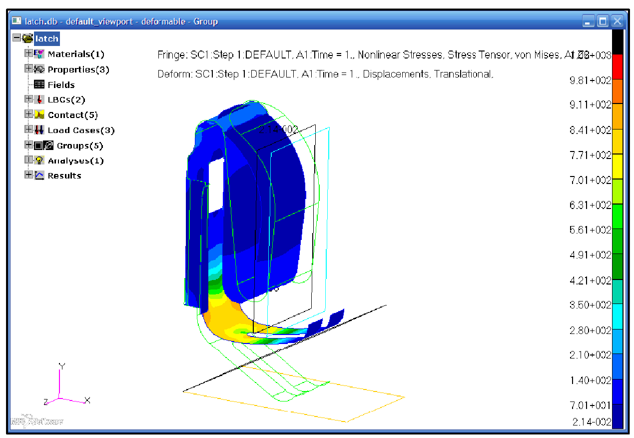
g. Click **Apply**.

h. When finished, click **Reset Graphics**.







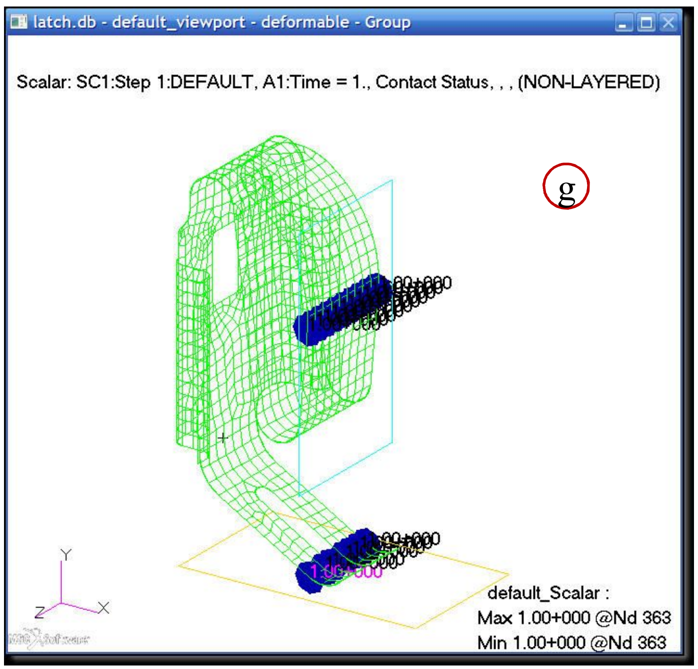


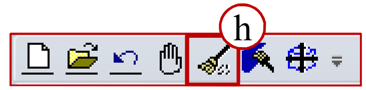
MSC Software

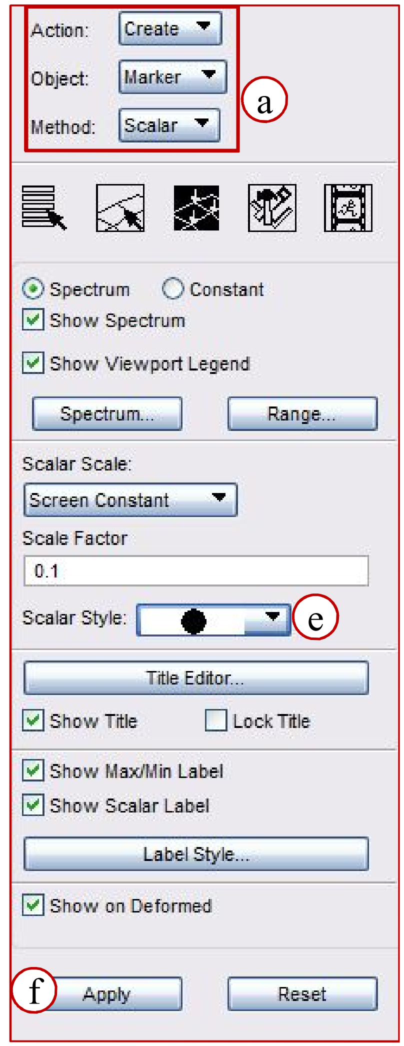
Step 11. Creating Marker Plot of Contact Status

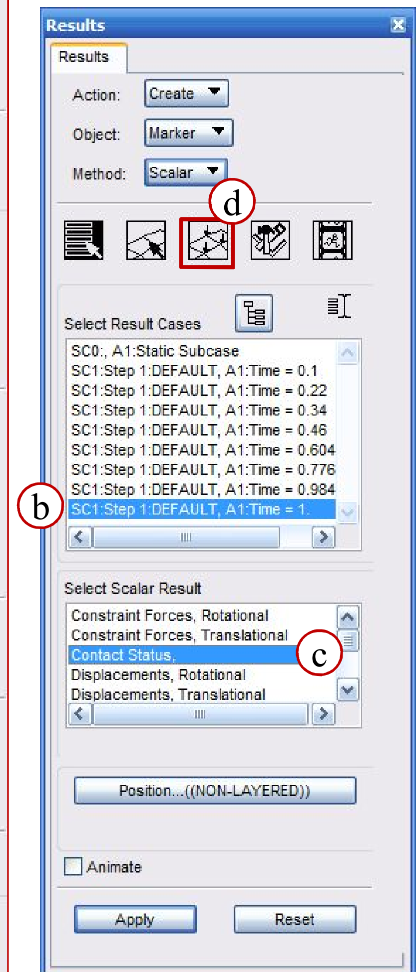
Create a Marker Plot of the Contact status:

a. Pull down **Create** > **Marker** > **Scalar**.
b. Select **SC1:…A1:Time = 1.0** for *Select Result Cases.*
c. Select **Contact Status** for the *Scalar Result*.
d. Click the **Display Attributes** icon.
e. Pull down *Scalar Style* and pick the **Solid Sphere**.
f. Click **Apply**.
g. The solid sphere markers appear on the clip as shown.
h. When finished, click **Reset Graphics**.





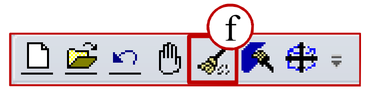


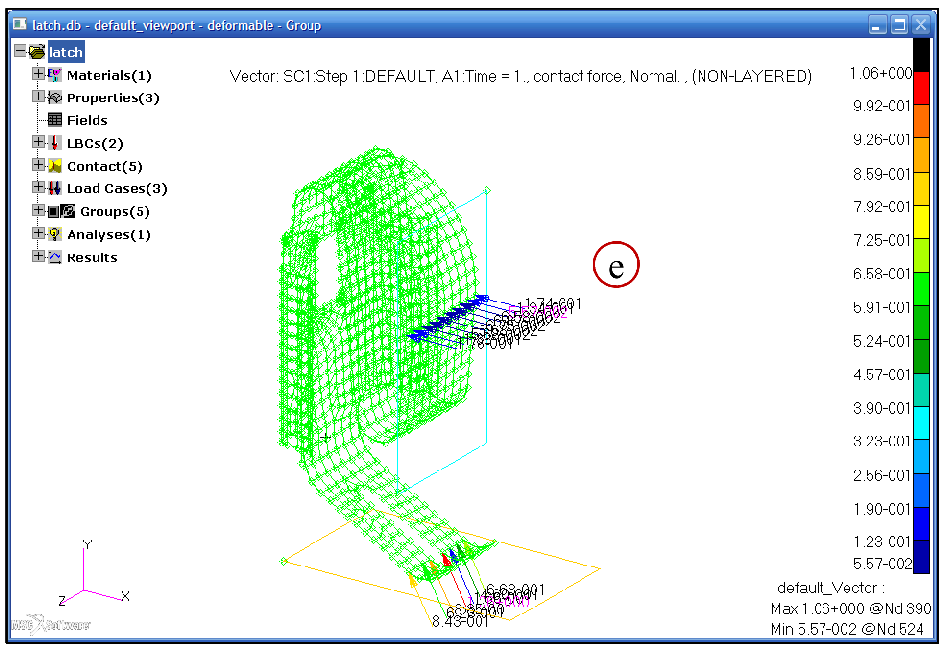


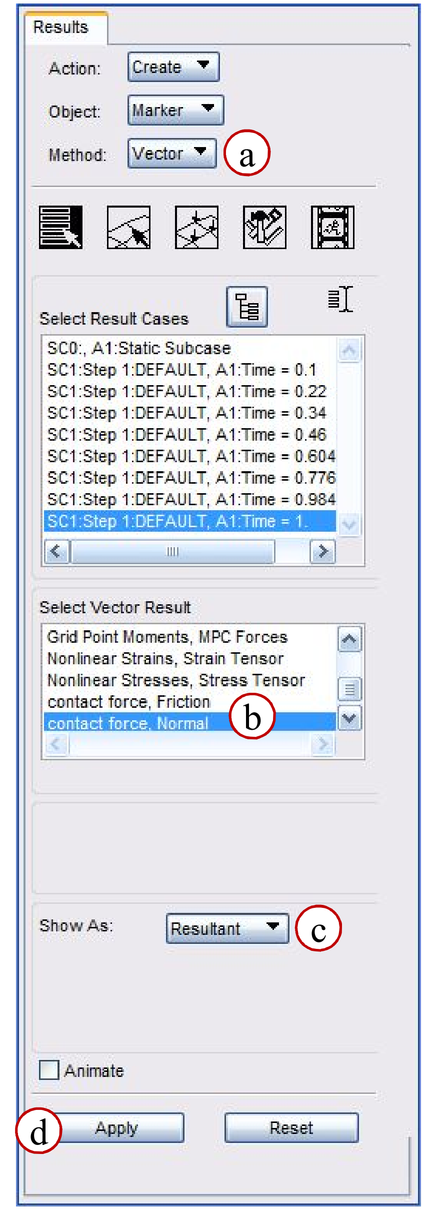
Step 12. Plot Normal Contact Force

Plot the Normal Contact Force Vector for the clip:

a. Pull down **Vector** for *Method*
b. For the *Vector Result* select **Contact Force, Normal**.
c. Pull down **Resultant** for *Show As*.
d. Click **Apply**.
e. The Normal Force Vectors will appear on the clip as shown.
f. When finished, click **Reset Graphics**.

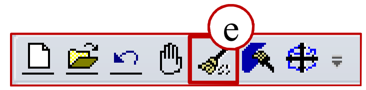






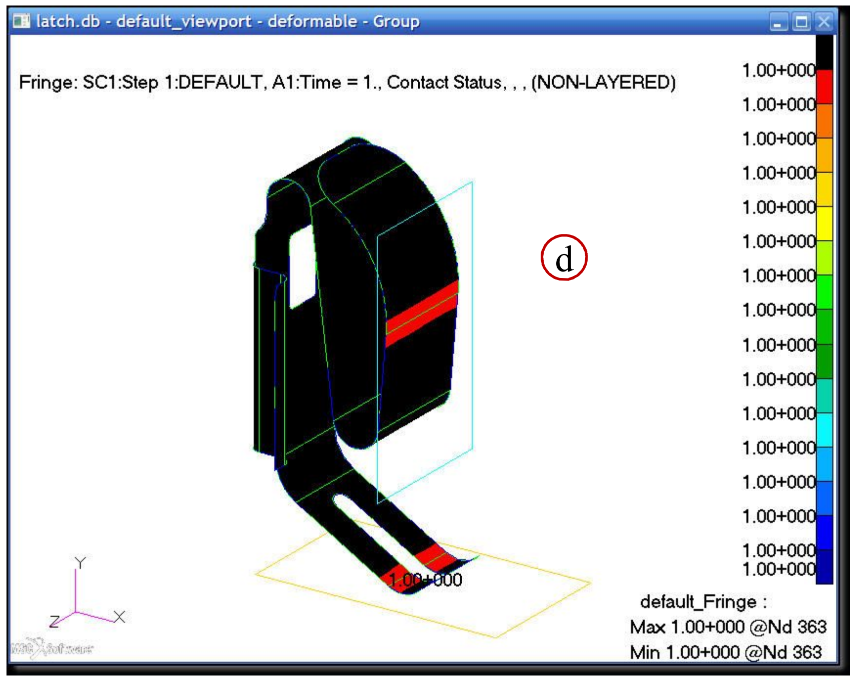
MSC Software

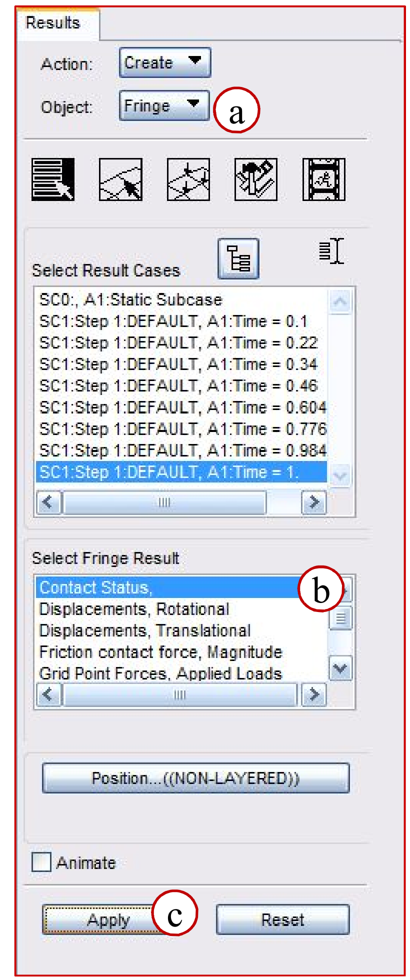
Step 13. Display a Fringe Plot of Contact Status



Display a Fringe Plot of the Contact status for the clip:

a. Pull down *Object* to **Fringe.**
b. For the *Fringe Result* select **Contact Status**.
c. Click **Apply**.
d. The fringe plot will appear on the clip as shown.
e. When finished, click **Reset Graphics**.

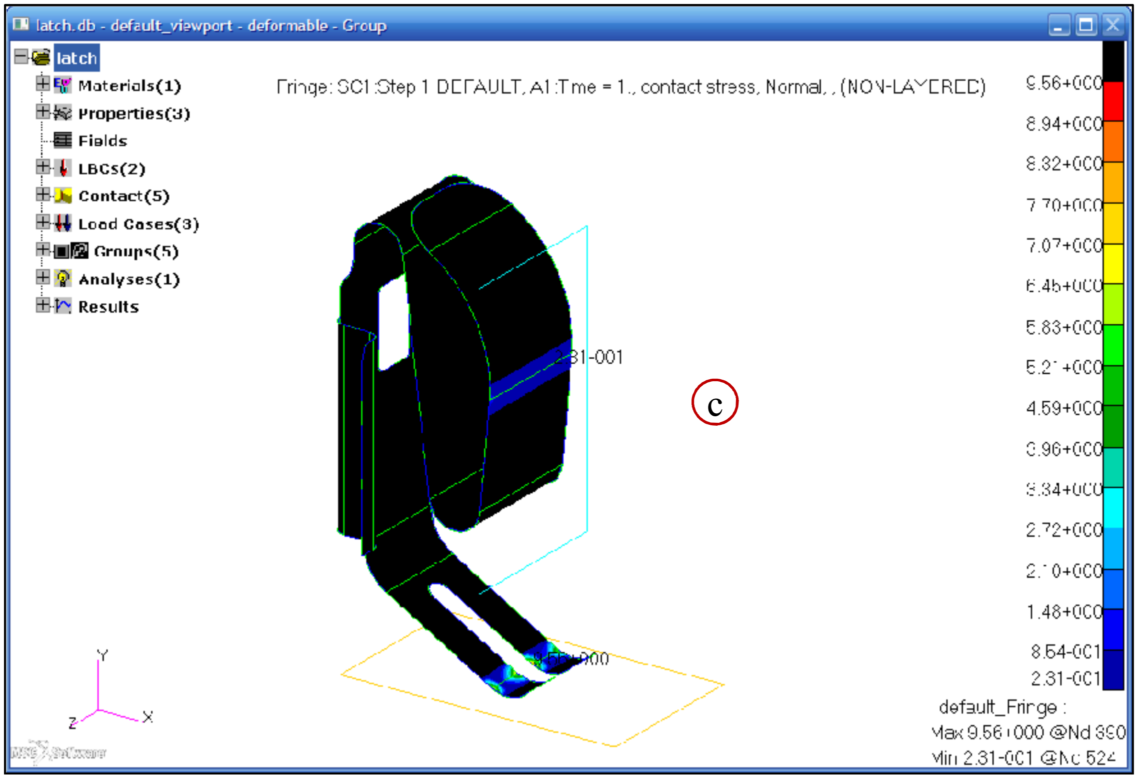


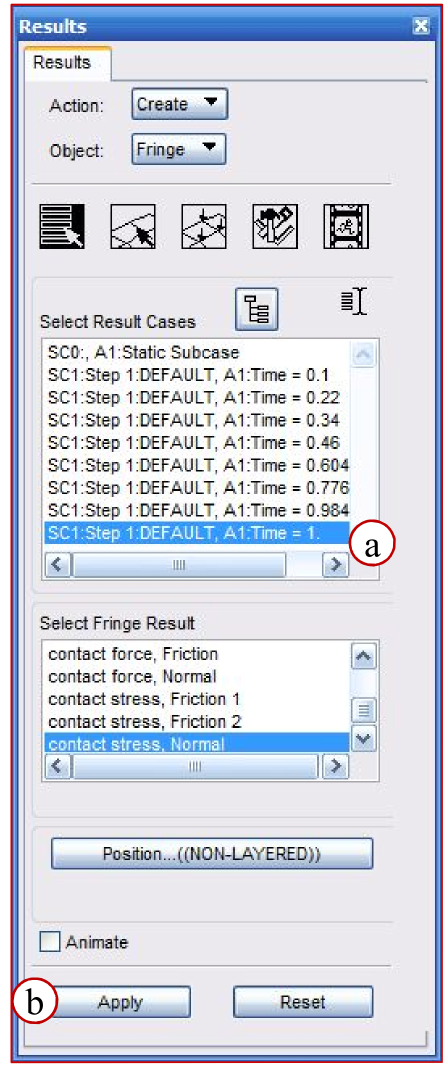


Step 14. Display a Fringe Plot of Normal Contact Stress

Display a fringe plot of Normal Contact Stress on the clip:

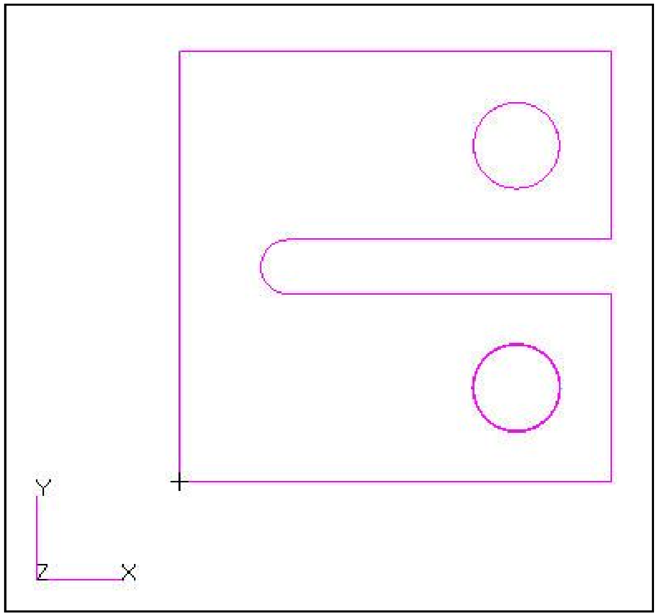
a. For the *Fringe Result* select **Contact Stress, Normal**.
b. Click **Apply**.
c. The normal stress fringe plot will appear on the clip as shown.

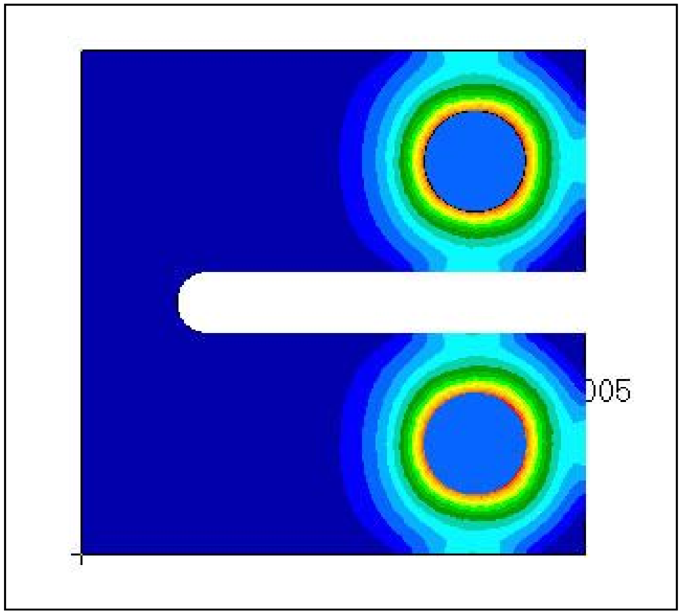




WORKSHOP 4

INTERFERENCE FIT





- **Workshop Objectives:**
  - Perform interference fit analysis using deformable-deformable contact
  - Modify contact table parameters
  - Understand the two methods of modeling interference fit
  - Switch between Node to Segment and Segment to Segment contact
- **Software Version:**
  - Patran 2013
  - MSC Nastran 2013.1
- **Files Required:**
  - **interference_fit.igs**
- **Problem Description**
  - Two types of interference fit modeling are demonstrated in this workshop. The interference can be modeled geometrically, or you can use congruent geometry and specify an Interference Closure in the contact table. This workshop will show that the same results are achieved for both methods.

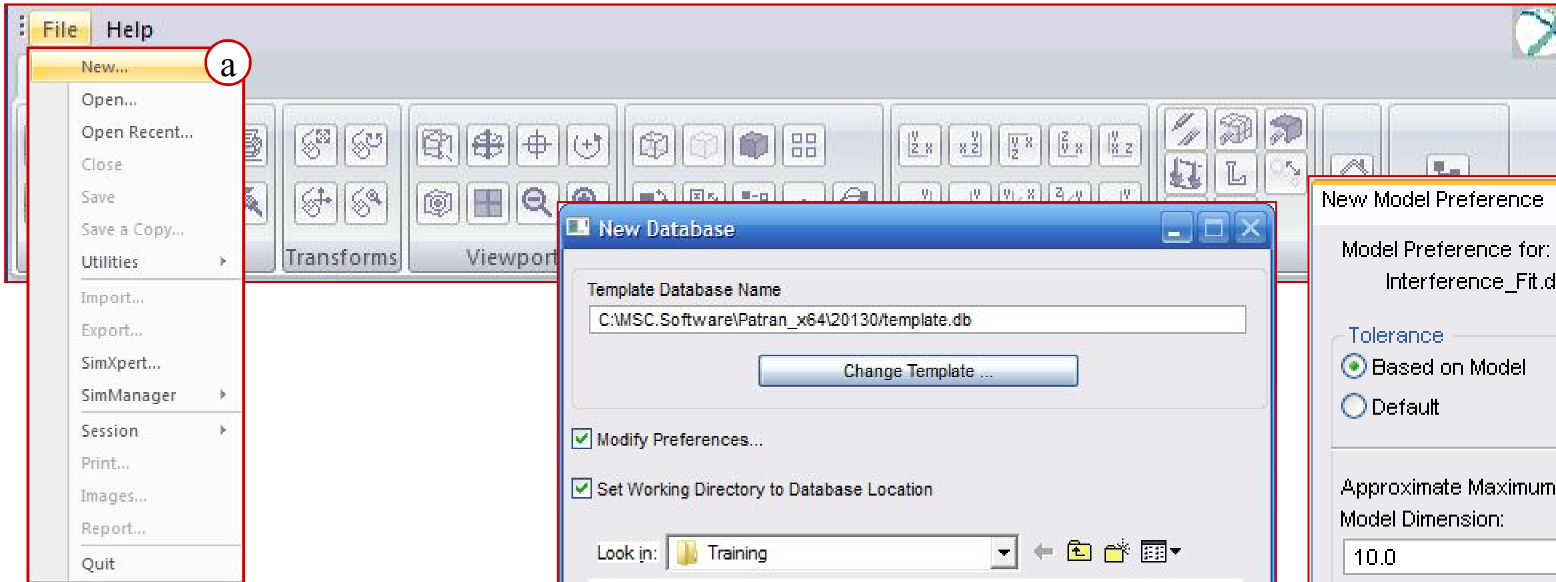
- **Suggested Exercise Steps**
  1. Create a new database called **Interference_Fit**.
  2. Import the Geometry **interference_fit.igs**.
     - Uncheck Import to Parasolid.
  3. Create a 0.1 inch Mesh with 8 noded quads.
  4. Fix the left edge of the lug in the X and Y direction.
  5. Define deformable contact bodies for the lug, upper pin, and lower pin with friction coefficients of **0.3**.
  6. Define Contact pairs
     - Turn off all self-contact and contact between the Upper and Lower Pin.
     - Create a Distance Tolerance of **0.015** between the Lug and Lower Pin.
     - Create an Interference Closure of **0.01** between the Lug and Upper Pin.
  7. Define material properties for steel with an elastic modulus of **30E6** and a Poisson's ratio of **0.3**.
  8. Define plane strain element properties.
     - Select Large Strain for Nonlinear Formulation (**SOL400**)

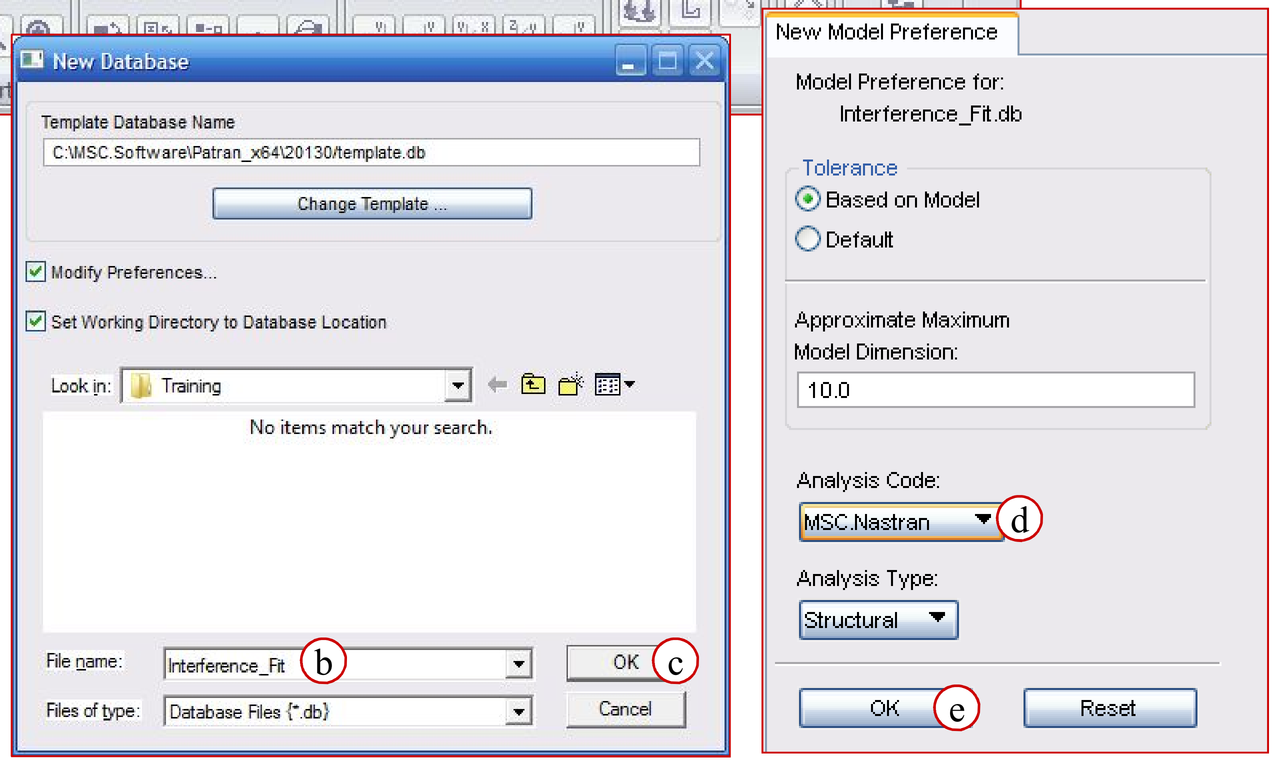
- **Suggested Exercise Steps (Cont.)**
  9. Set up and submit an implicit nonlinear analysis.
     - Do not use Assumed Strain.
     - Define Contact Parameters
       - Distance Tolerance (ERROR) of **0.015**.
       - Activate quadratic contact (**LINQUAD**).
       - Friction type is bilinear coulomb.
     - Use OP2 for the results output format.
     - Define Subcase Parameters
       - Use Adaptive Increment Type and Trial Time Step Size = **0.1**.
       - Use Pure Full Newton for the Matrix Update Method.
     - Select to Output Element Strain and Contact results.
  10. Attach the .op2 Results File.
  11. Plot the Cauchy Stress Fringe Results superimposed on the true scale deformed shape for the final load increment.
  12. Plot Contact Status and Contact Force results.

- **Suggested Exercise Steps (Cont.)**

13. Run the analysis again using Segment to Segment contact in place of Node to Segment contact
14. Attach the Segment to Segment .op2 Result file
15. Plot the Cauchy Stress Fringe Results superimposed on the true scale deformed shape.
16. Plot Contact Status and Contact Force results for Segment to Segment Contact
17. Quit Patran.

Step 1. Create a New Database

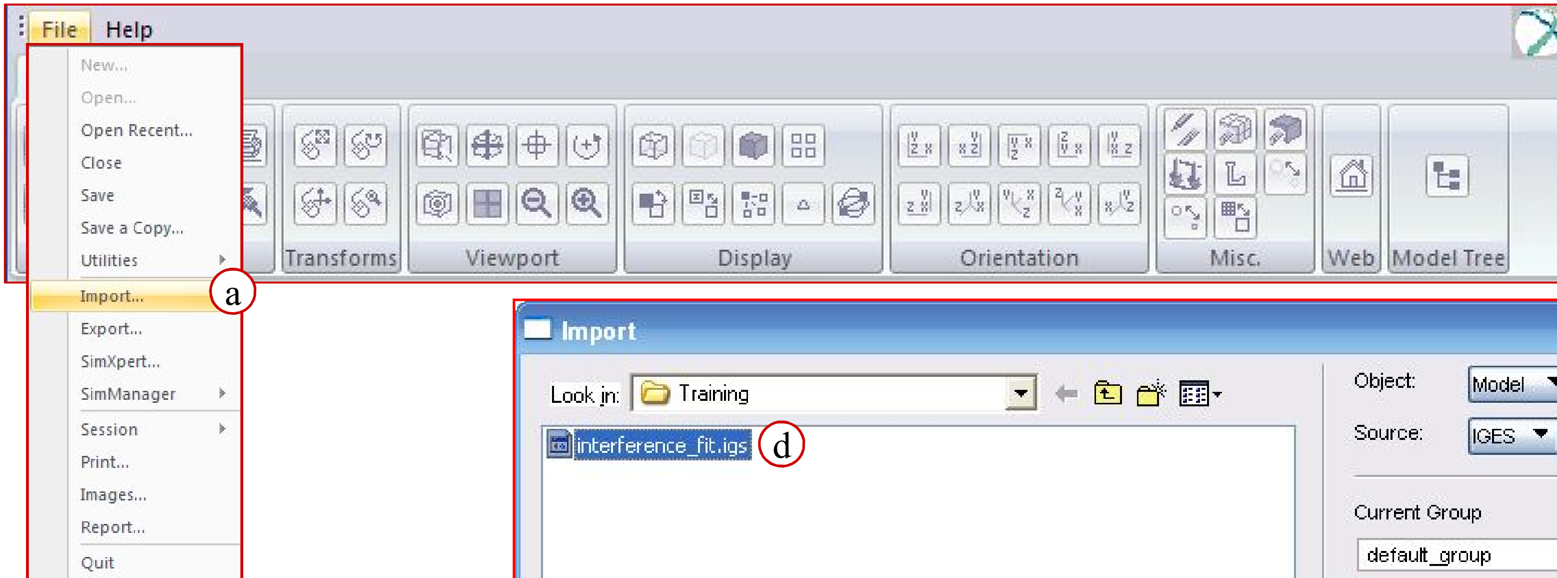


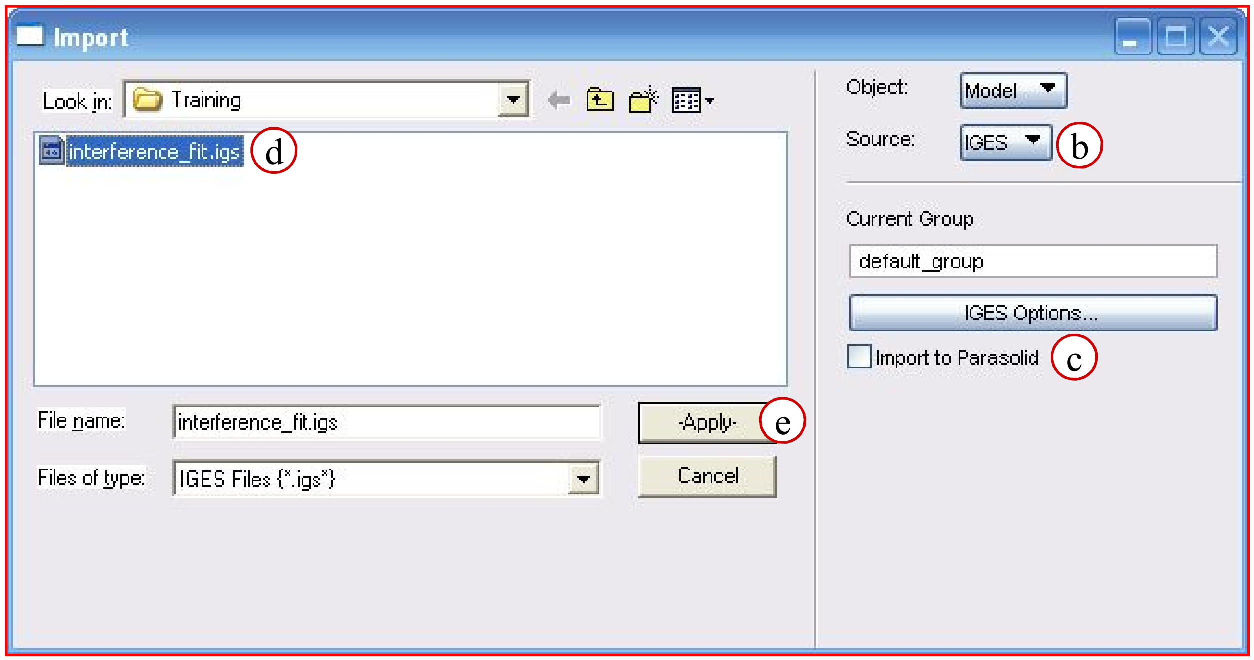


Create a new database. Name it **Interference_Fit.db**.

a. Pull down *File* **> New**.
b. Enter **Interference_Fit** as the *File name*.
c. Click **OK**.
d. Confirm **MSC Nastran** as the *Analysis Code*.
e. Click **OK**.

Step 2. Import the Geometry



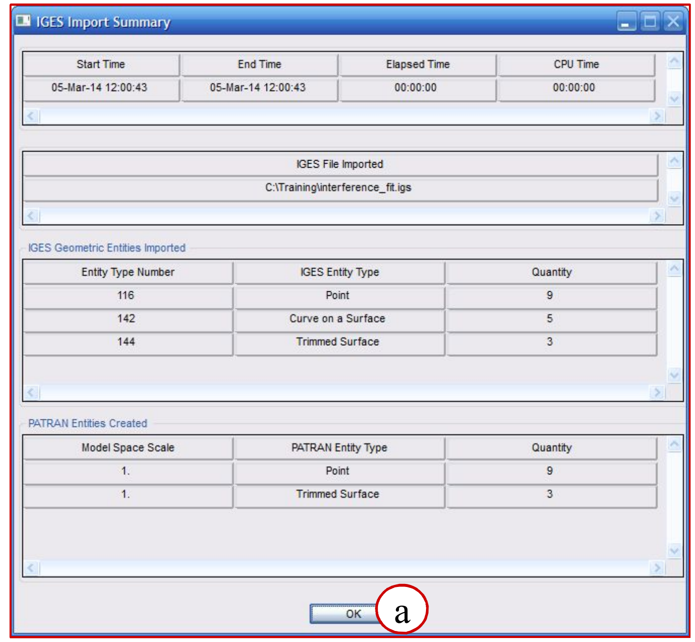


Import the geometry.

a. Pull down *File* **> Import**.
b. Pull down *Source* to **IGES**.
c. Uncheck **Import to Parasolid**.
d. Navigate to and select **interference_fit.igs**.
e. Click **Apply**.

Step 2. Import the Geometry (Cont.)

a. The *IGES Import Summary* form shows the types and numbers of entities imported into the Patran database. Click **OK**.



IGES Import Summary

Start Time	End Time	Elapsed Time	CPU Time
05-Mar-14 12:00:43	05-Mar-14 12:00:43	00:00:00	00:00:00

IGES File Imported
C:\Training\interference_fit.igs

IGES Geometric Entities Imported

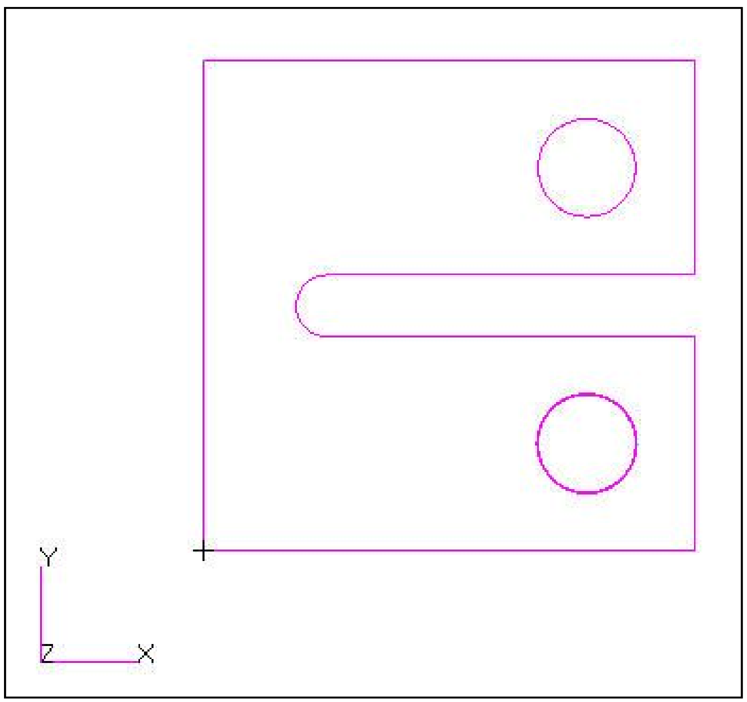
Entity Type Number	IGES Entity Type	Quantity
116	Point	9
142	Curve on a Surface	5
144	Trimmed Surface	3

PATRAN Entities Created

Model Space Scale	PATRAN Entity Type	Quantity
1.	Point	9
1.	Trimmed Surface	3

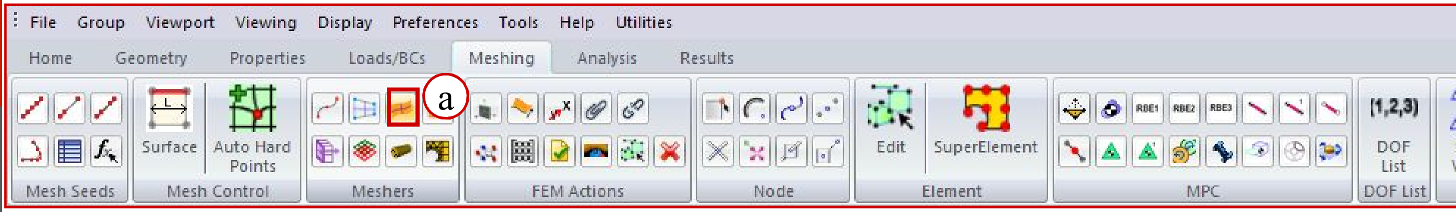
OK

a



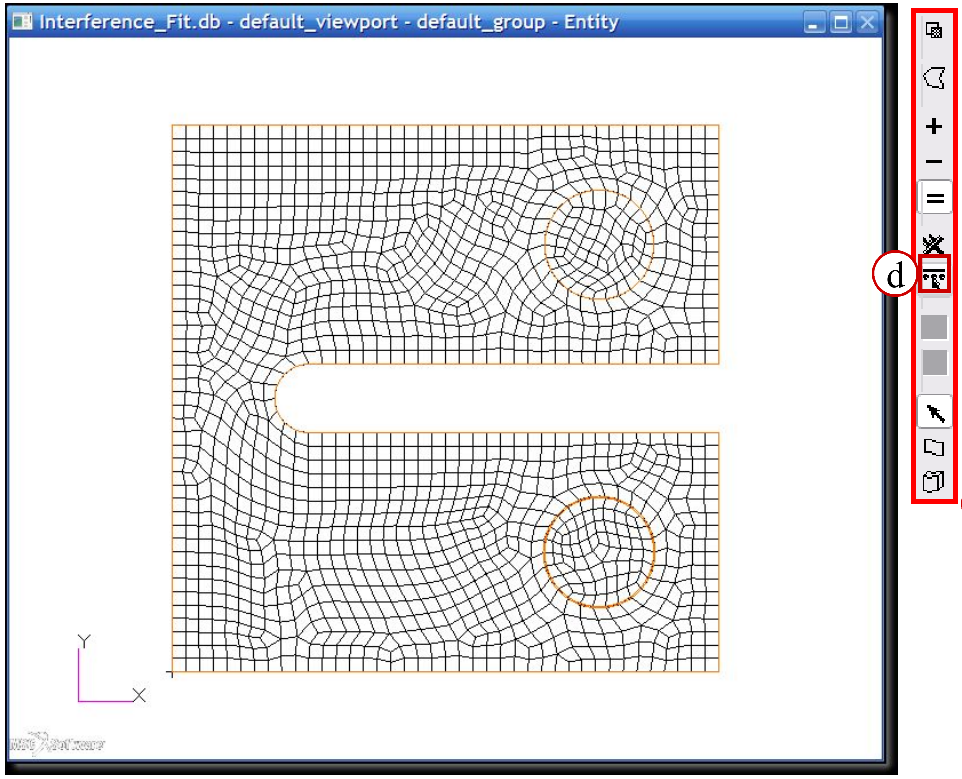
The imported geometry consists of 3 surfaces: one large lug with 2 pins. The lower pin has an interference of 0.01 inches built into the geometry. The upper pin geometry is congruent with the Lug geometry. The interference fit in this case will be created by the Interference Closure contact parameter which will force a gap between the two bodies.

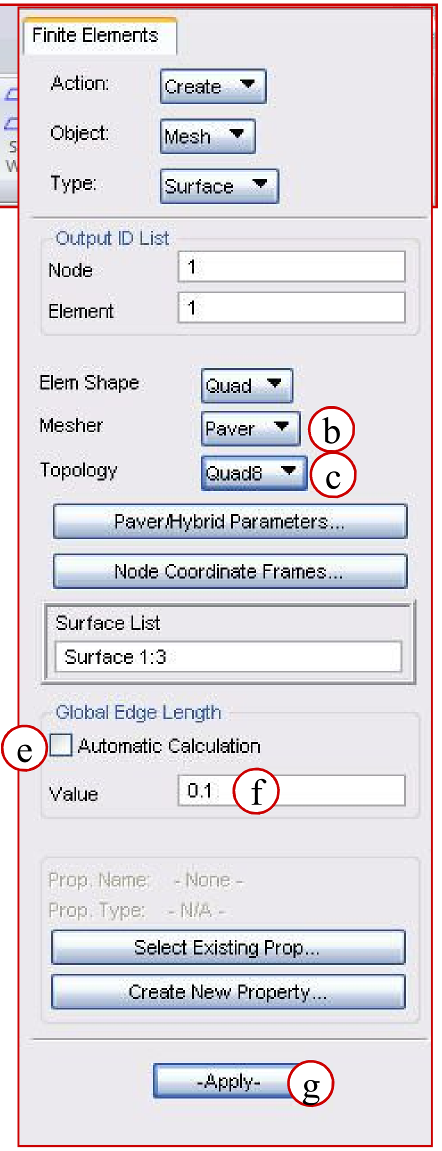
Step 3. Create the Elements



Create a 0.1 inch Mesh with 8 noded quads.

a. Under the *Meshing* tab, click **Surface** in the *Meshers* group.
b. Pull down **Paver** for *Mesher*.
c. Pull down **Quad8** for *Topology*. Quad8 elements are higher-order elements with mid-side nodes.
d. Click **Pick All** on the *Picking Filters* toolbar to select all three Surfaces.
e. Uncheck **Automatic Calculation**.
f. Enter **0.1** for the *Global Edge Length Value.*
g. Click **Apply**.



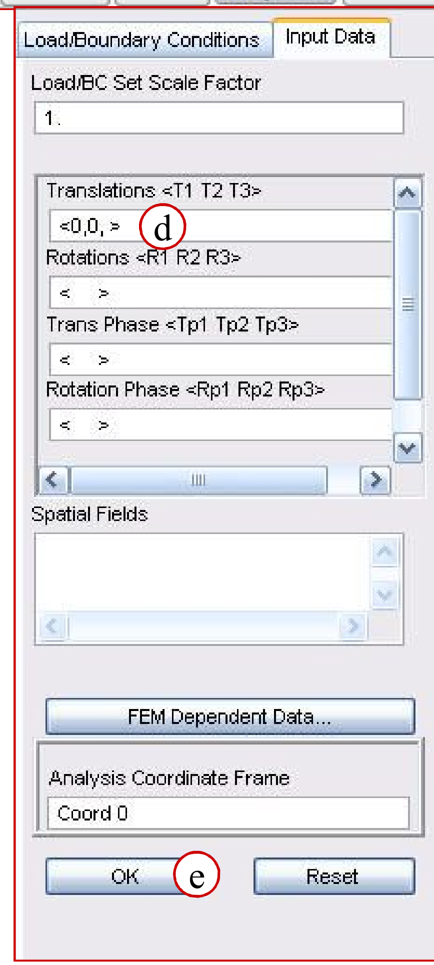


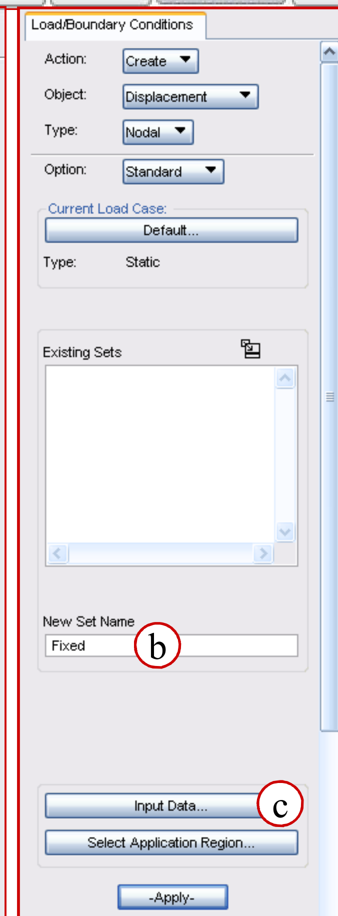
MSC Software

Step 4. Fix the Left Edge of the Lug

Fix the left edge of the lug in the X and Y direction.

a. Under the *Loads/BCs* tab, click **Displacement Constraint** in the *Nodal group.*
b. Enter **Fixed** as the *New Set Name.*
c. Click **Input Data**.
d. Enter **<0,0, >** for the *Translations*.
e. Click **OK**.

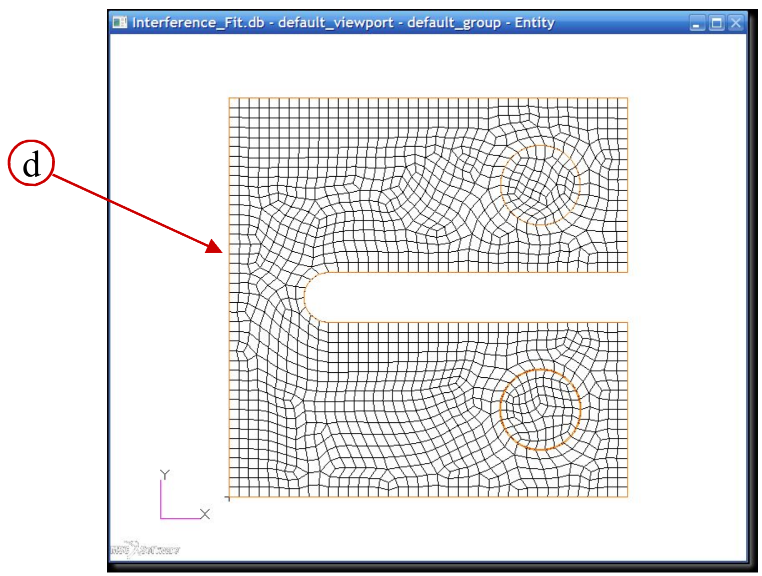


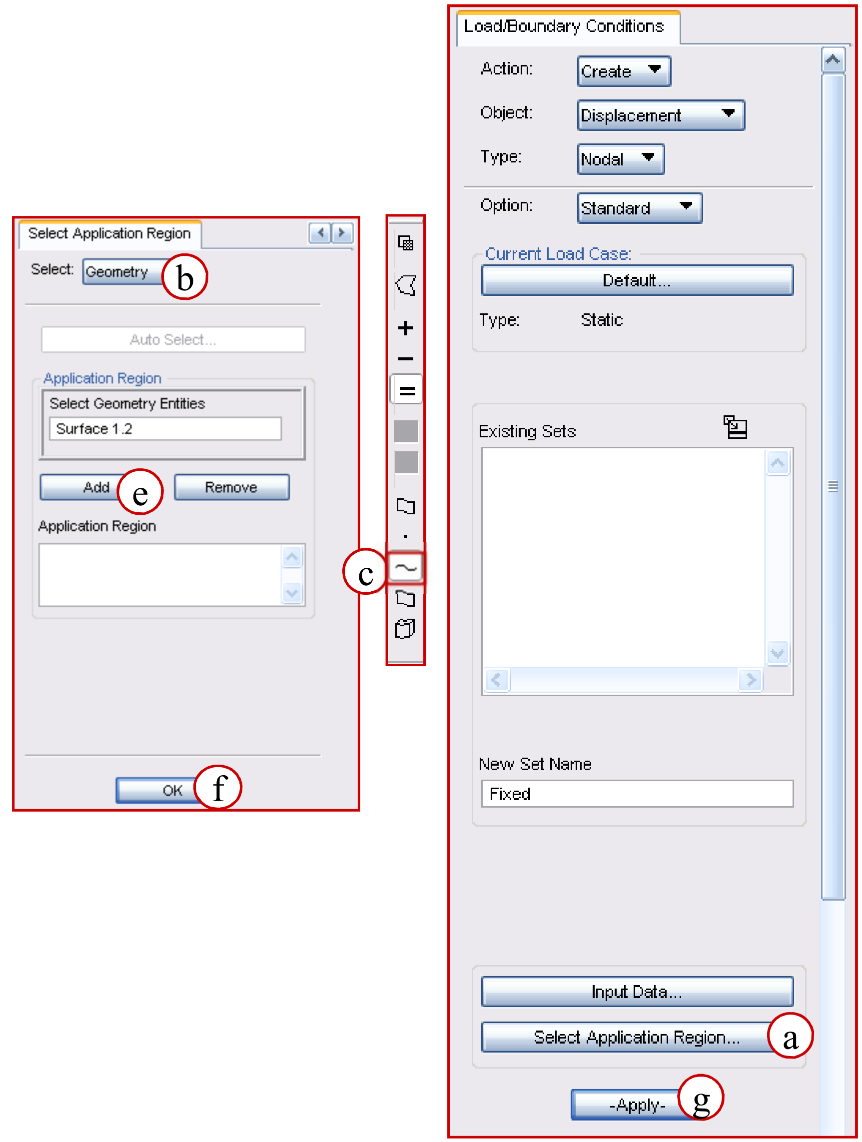


MSC Software

Step 4. Fix the Left Edge of the Lug (Cont.)

a. Click **Select Application Region**.
b. Confirm the *Select* filter is set to **Geometry**.
c. Click **Curve or Edge** on the *Picking Filters* toolbar.
d. Select the left edge of the lug.
e. Click **Add**.
f. Click **OK**.
g. Click **Apply**.



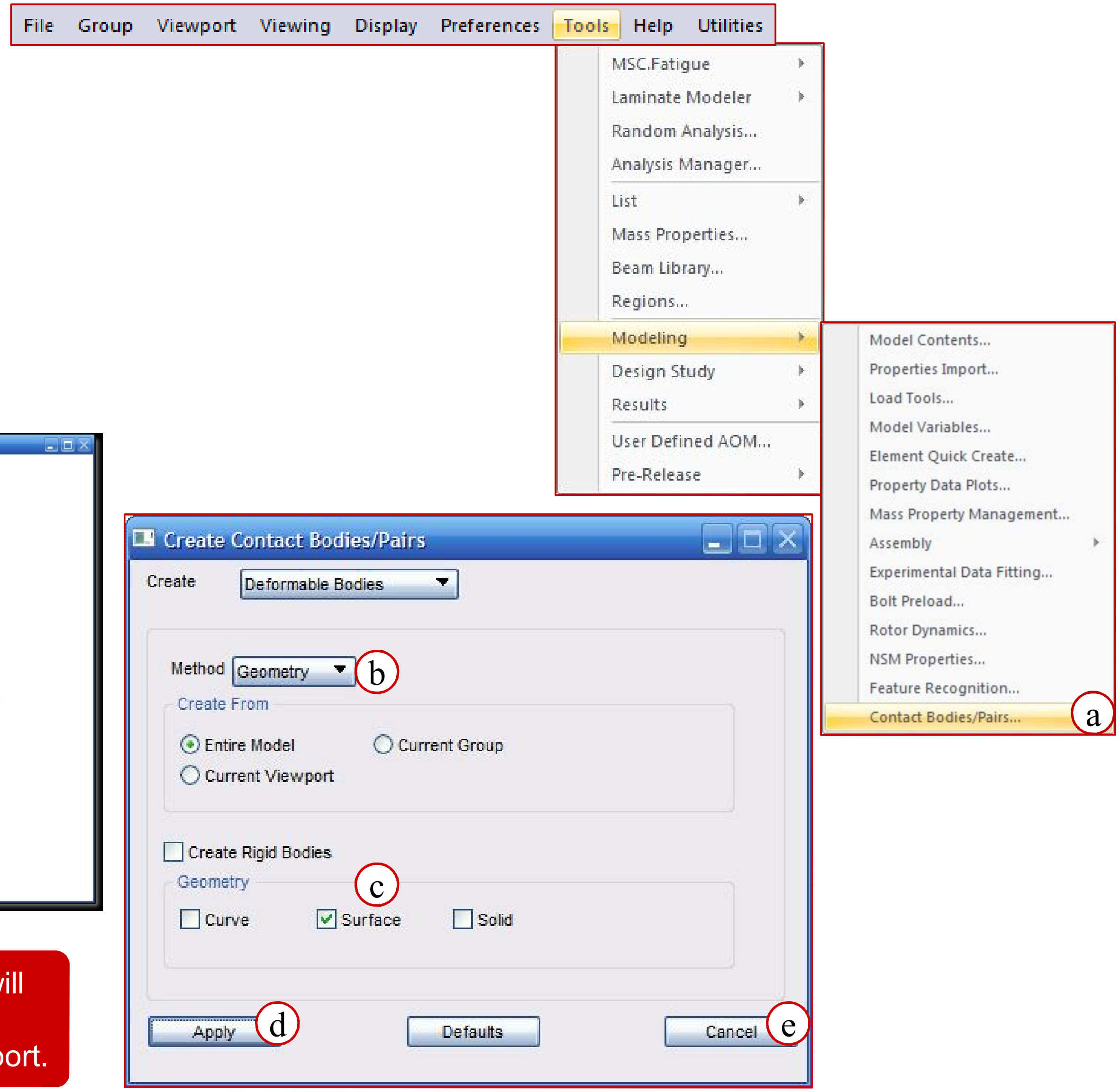


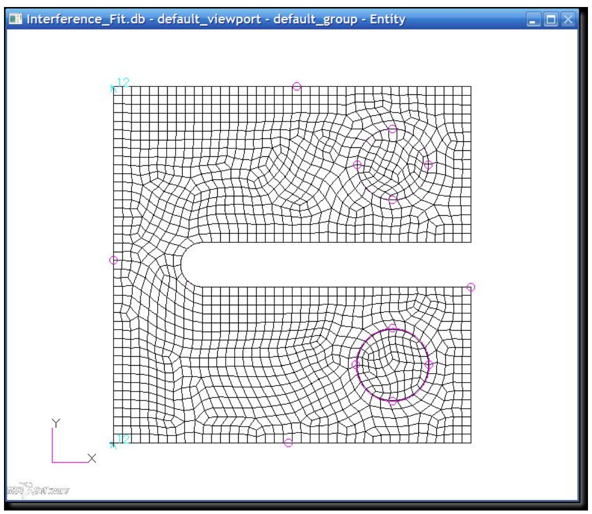
Step 5. Define the Deformable Contact Bodies

Create three deformable contact bodies

.

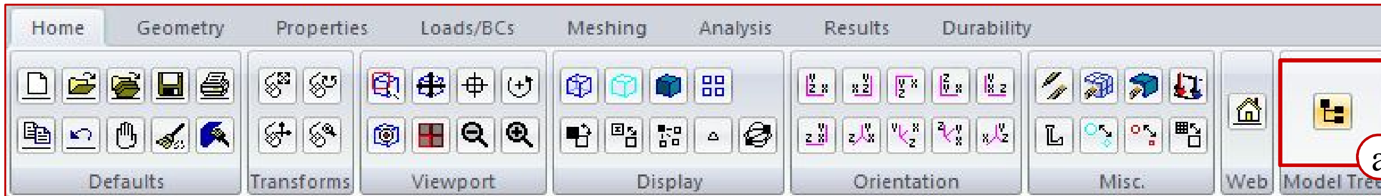
a. Pull down *Tools* > *Modeling* and select **Contact Bodies/Pairs…**
b. Pull down *Method* to **Geometry**
c. Select **Surface** under *Geometry*
d. Click **Apply**
e. Click **Cancel**





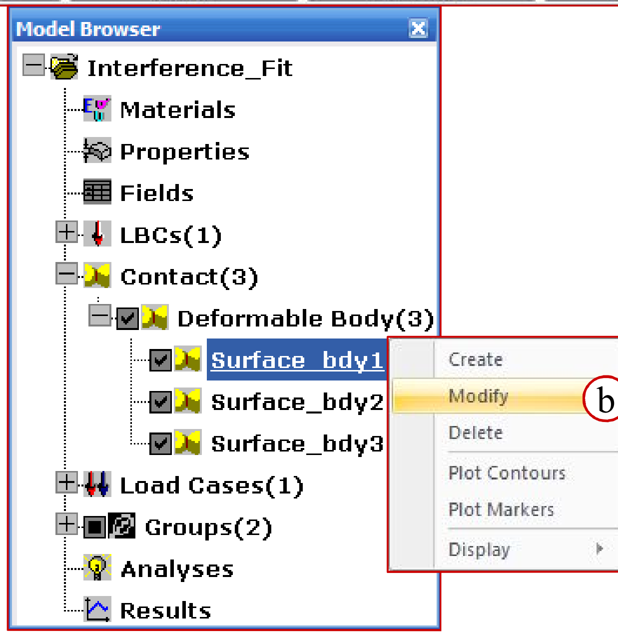
After you click **Apply**, you will see the contact markers (magenta circles) in the viewport.

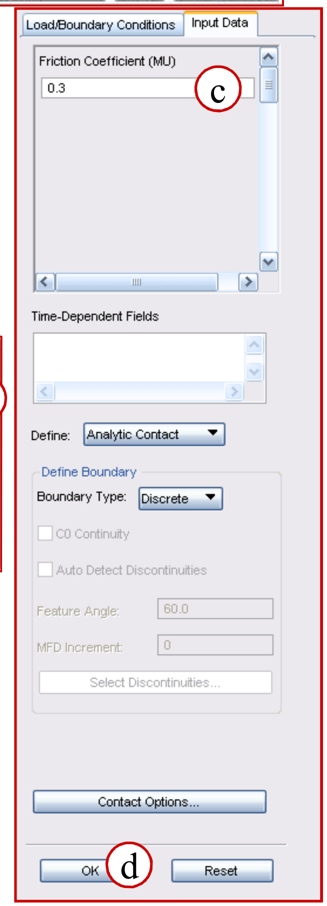
Step 5. Define the Deformable Contact Bodies (Cont.)

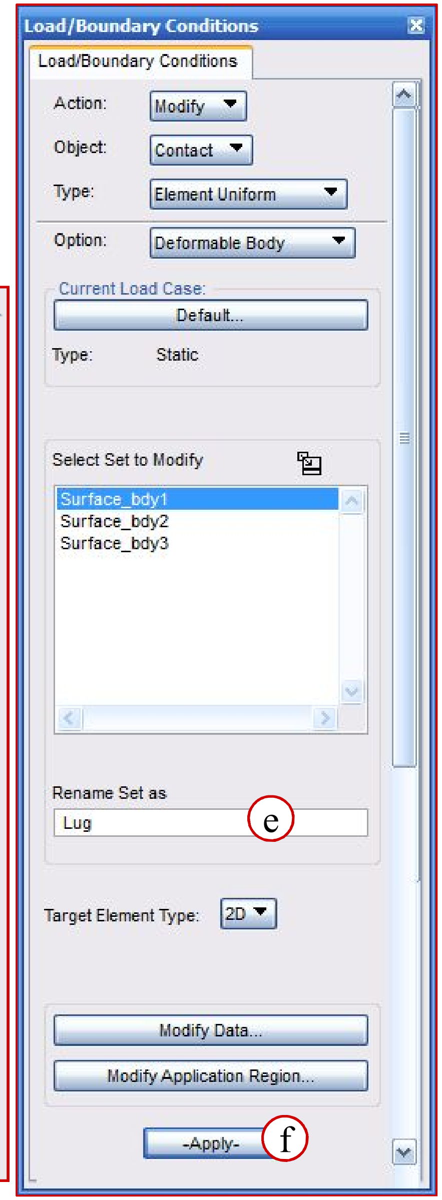


Rename the deformable contact bodies:

a. Under the *Home* tab click **Model Tree**
b. Right-click on **Surface_bdy1** under *Contact→Deformable Body* group in the *Model Browser* and *select* **Modify**.
c. Enter **0.3** for the *Friction Coefficient.*
d. Click **OK.**
e. Enter **Lug** for the *Rename Set as.*
f. Click **Apply**.



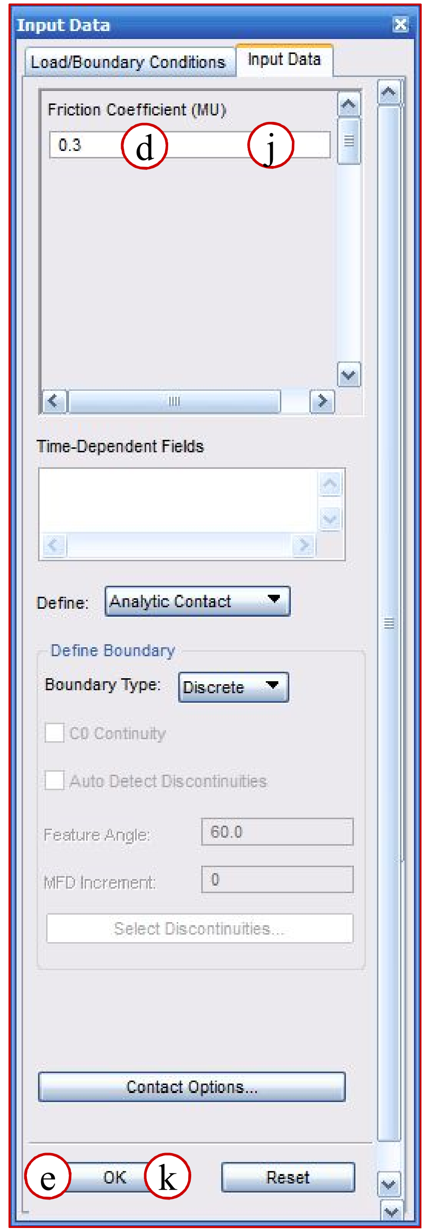


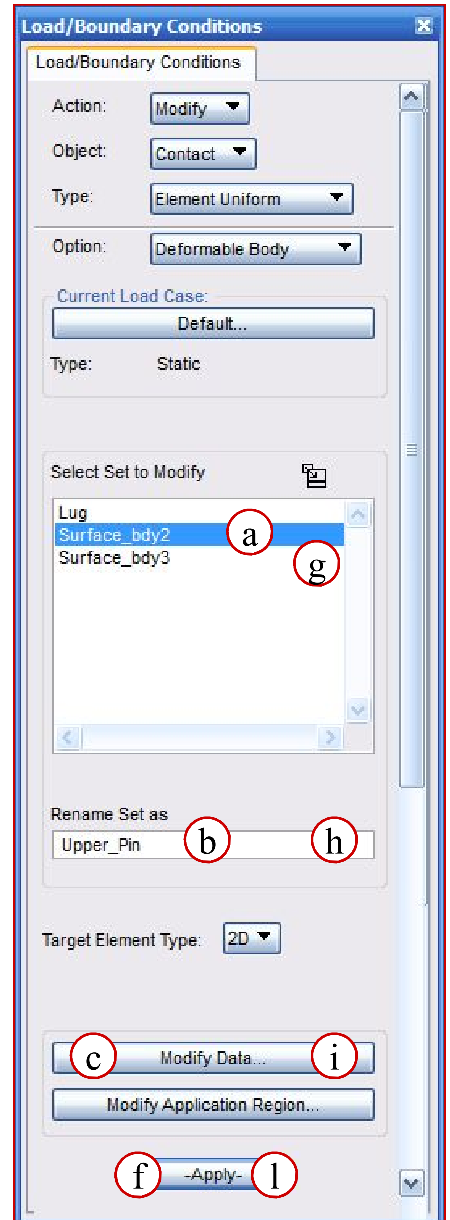


Step 5. Define the Deformable Contact Bodies (Cont.)

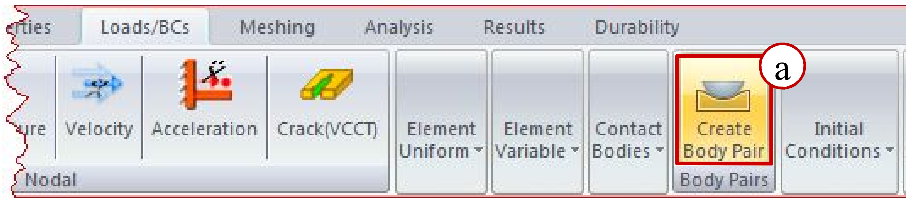
Rename the deformable contact bodies (cont.):

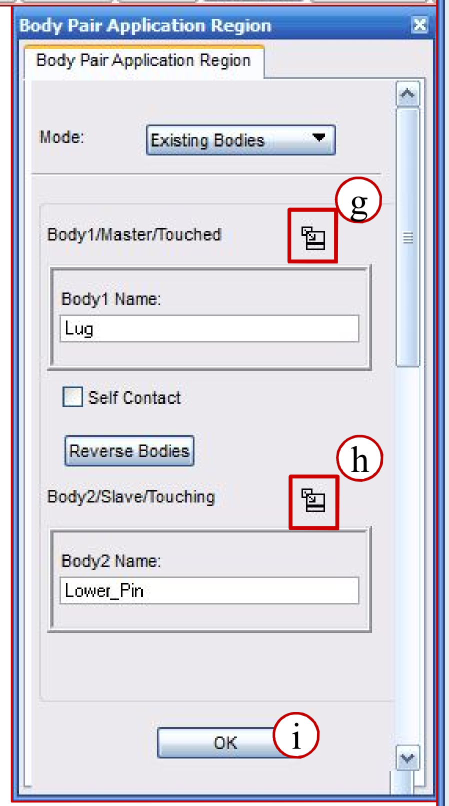
a. Click **Surface_bdy2** under *Select Set to Modify.*
b. Enter **Upper_Pin** for the *Rename Set as.*
c. Click **Modify Data**
d. Enter **0.3** for the *Friction Coefficient.*
e. Click **OK.**
f. Click **Apply.**
g. Click **Surface_bdy3** under *Select Set to Modify.*
h. Enter **Lower_Pin** for the *Rename Set as.*
i. Click **Modify Data**
j. Enter **0.3** for the *Friction Coefficient.*
k. Click **OK.**
l. Click **Apply.**

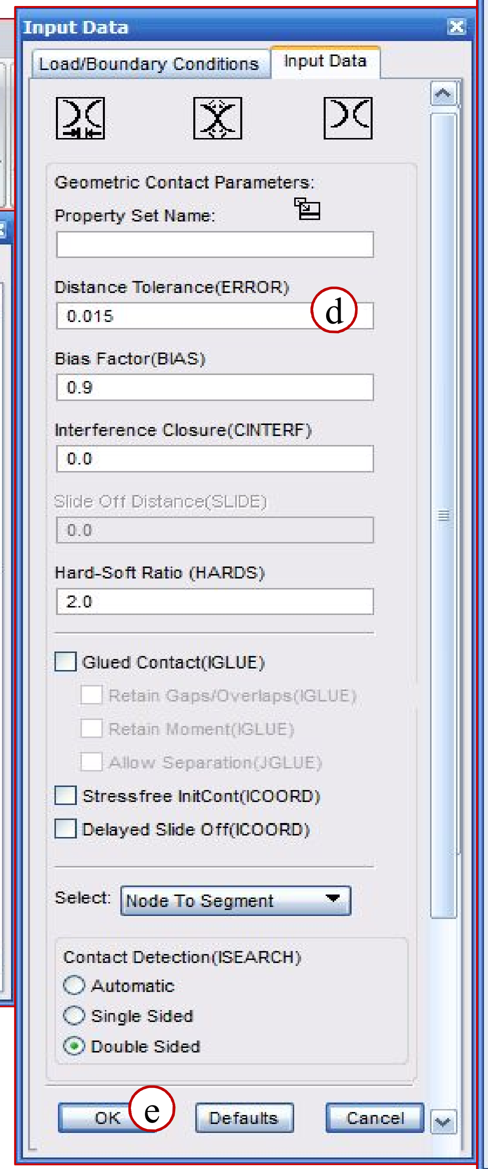


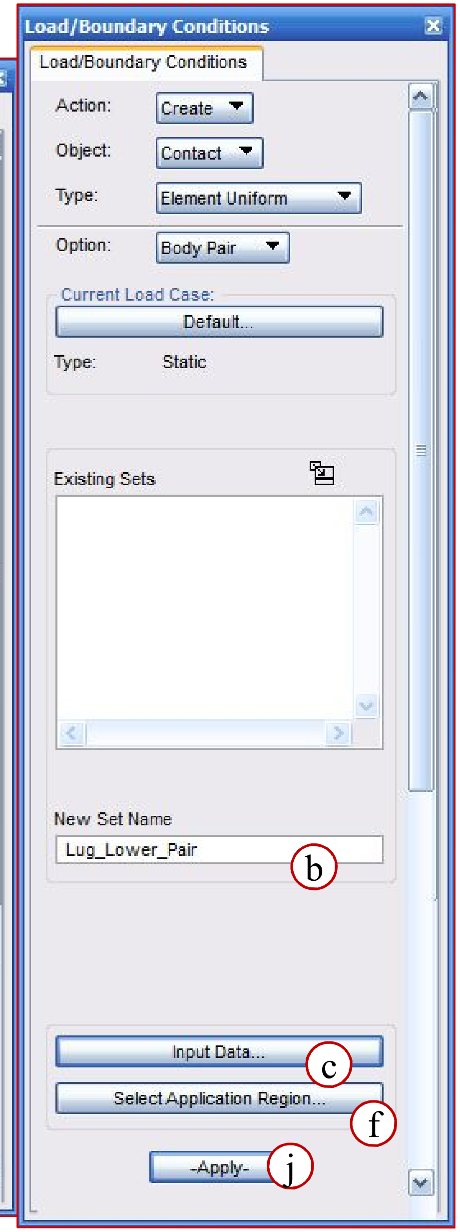


Step 6. Create Contact Pairs









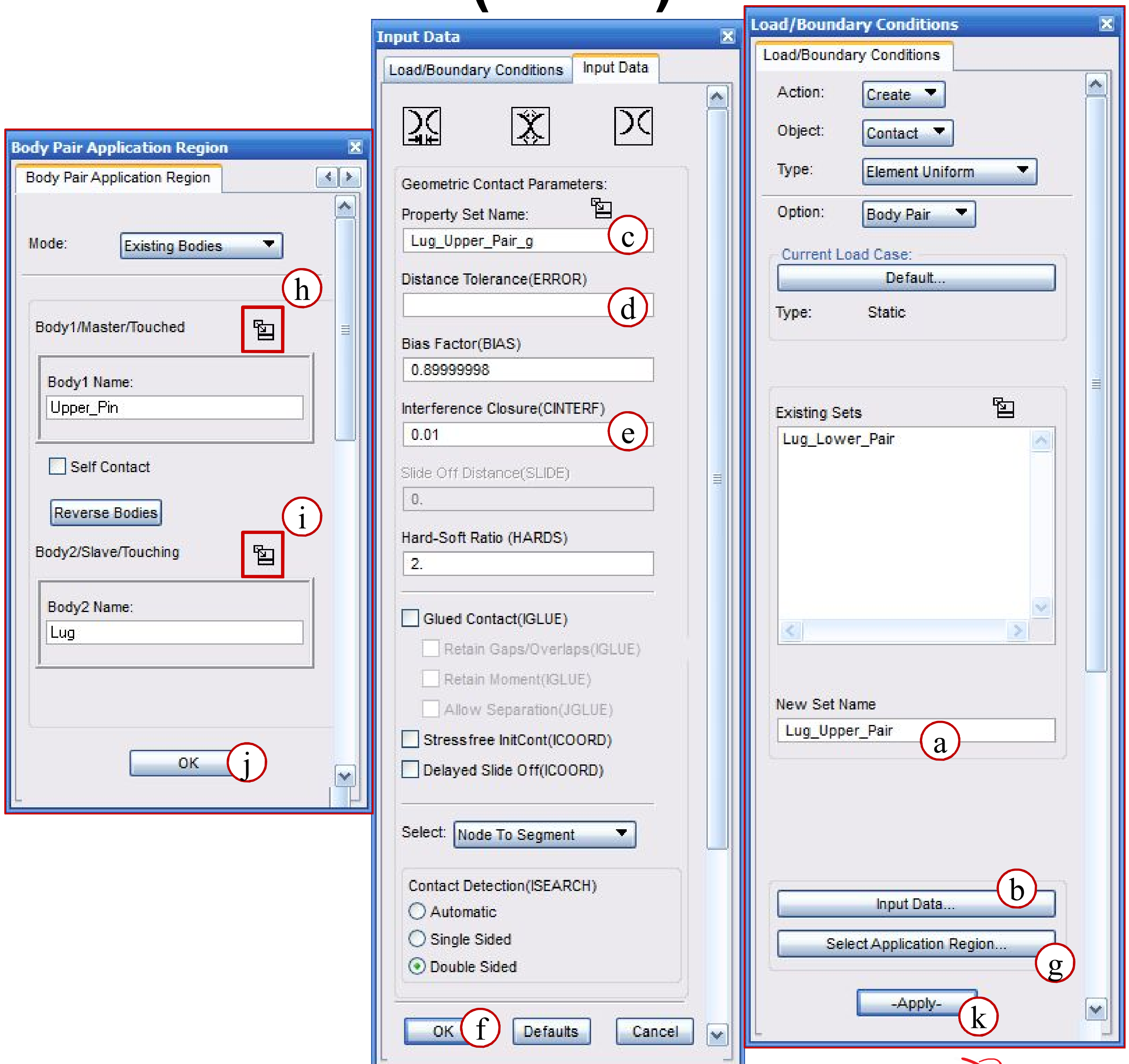
Create Contact Pair between Lower pin and the Lug:

a. Under the *Loads/BCs* tab click **Create Body Pair** in the *Body Pairs* group
b. Enter **Lug_Lower_Pair** for *New Set Name*
c. Click **Input Data.**
d. Enter **0.015** for the *Distance Tolerance*
e. Click **OK**
f. Click **Select Application Region**
g. Click **Select Existing Touched Body** icon near *Body1/Master/Touched* and select **Lug**
h. Click **Select Existing Touching Body** icon near *Body2/Slave/Touching* and select **Lower_Pin**
i. Click **OK**
j. Click **Apply**

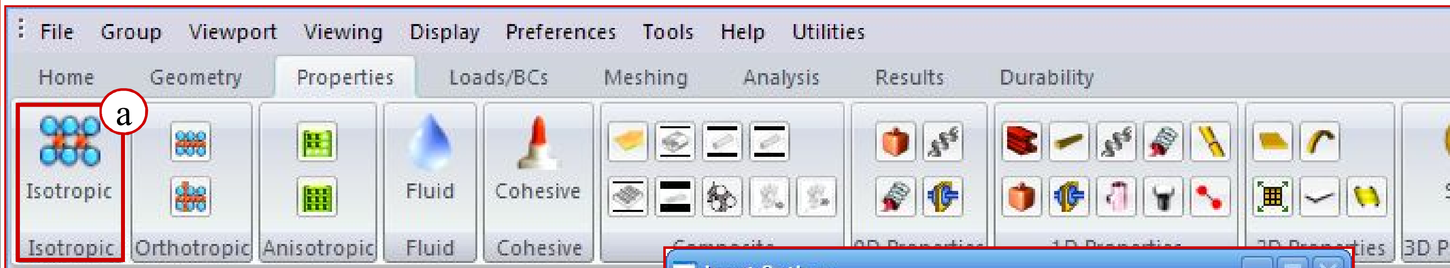
Step 6. Create Contact Pairs (Cont.)

Create Contact Pair between Lug and the Upper Pin:

a. Enter **Lug_Upper_Pair** for *New Set Name*
b. Click **Input Data**.
c. Enter **Lug_Upper_Pair_g** for *Property Set Name*.
d. Enter **0** for the *Distance Tolerance*
e. Enter **0.01** for the *Interference Closure*
f. Click **OK**
g. Click **Select Application Region**
h. Click **Select Existing Touched Body** icon near *Body1/Master/Touched* and select **Upper_Pin**
i. Click **Select Existing Touching Body** icon near *Body2/Slave/Touching* and select **Lug**
j. Click **OK**
k. Click **Apply**

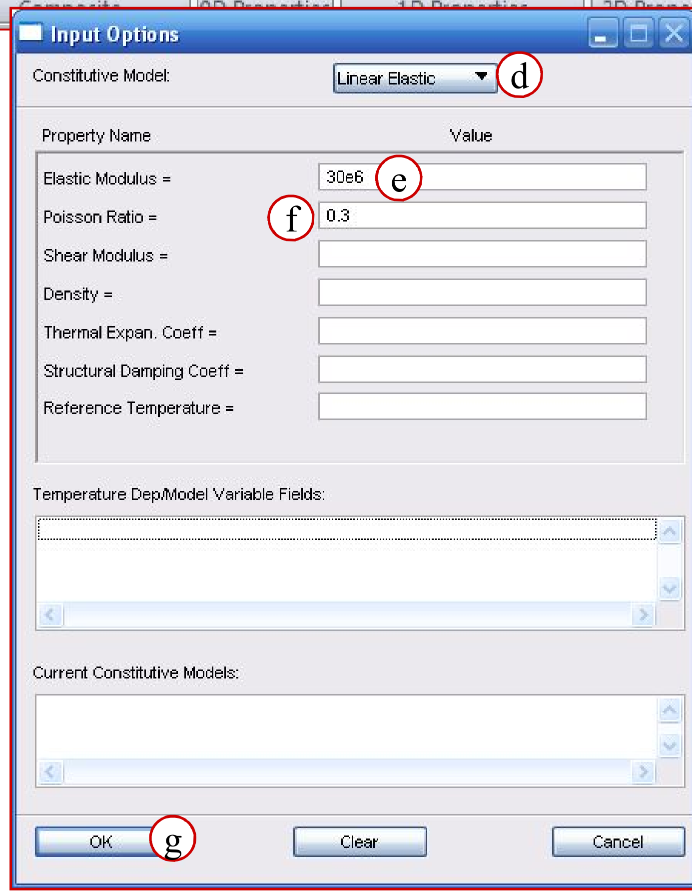


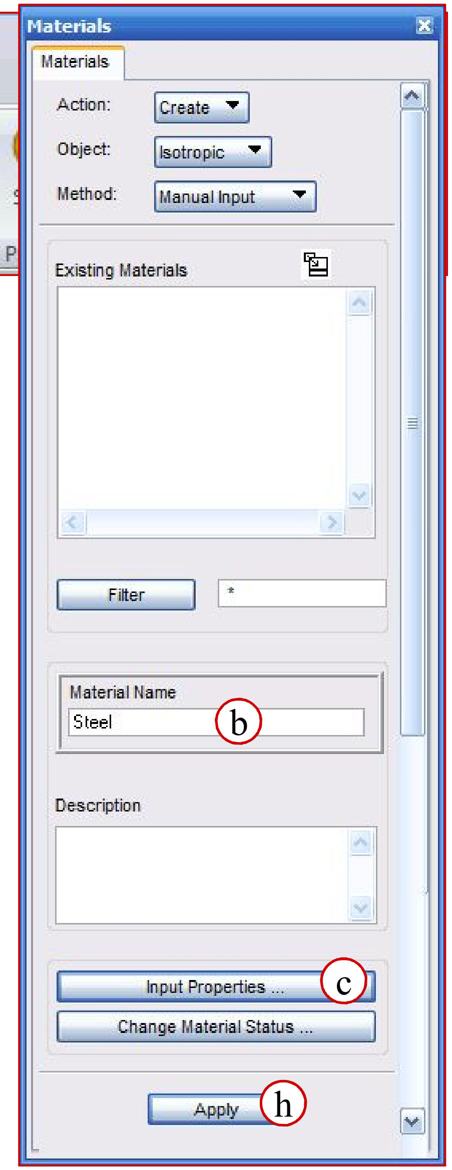
Step 7. Define the Material Properties



Define material properties for steel with an elastic modulus of **30E6** and a Poisson's ratio of **0.3** :

a. Under the *Properties* tab, click **Isotropic** in the *Isotropic* group.
b. Enter **Steel** for the *Material Name.*
c. Click **Input Properties**.
d. Confirm **Linear Elastic** is selected for the *Constitutive Model*.
e. Enter **30e6** for the *Elastic Modulus*.
f. Enter **0.3** for the *Poisson Ratio*.
g. Click **OK**.
h. Click **Apply**.

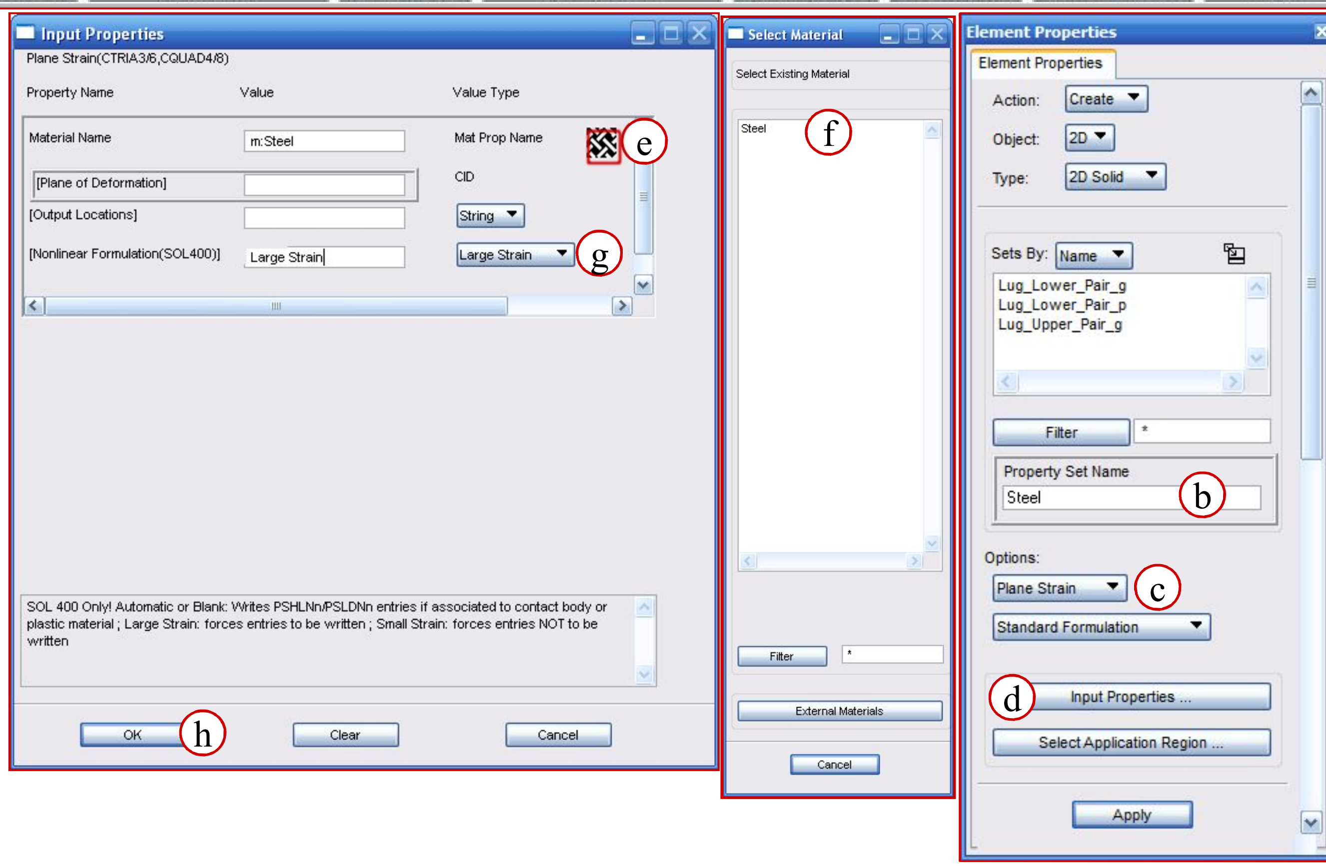




Step 8. Define the Element Properties

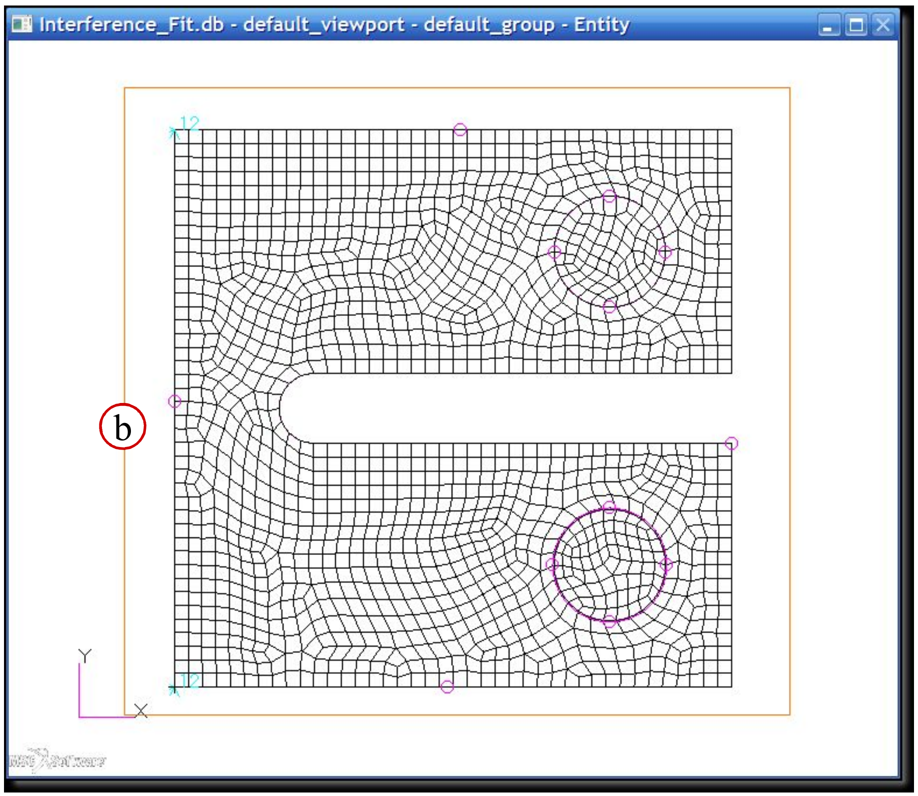
Define Plane Strain element properties:

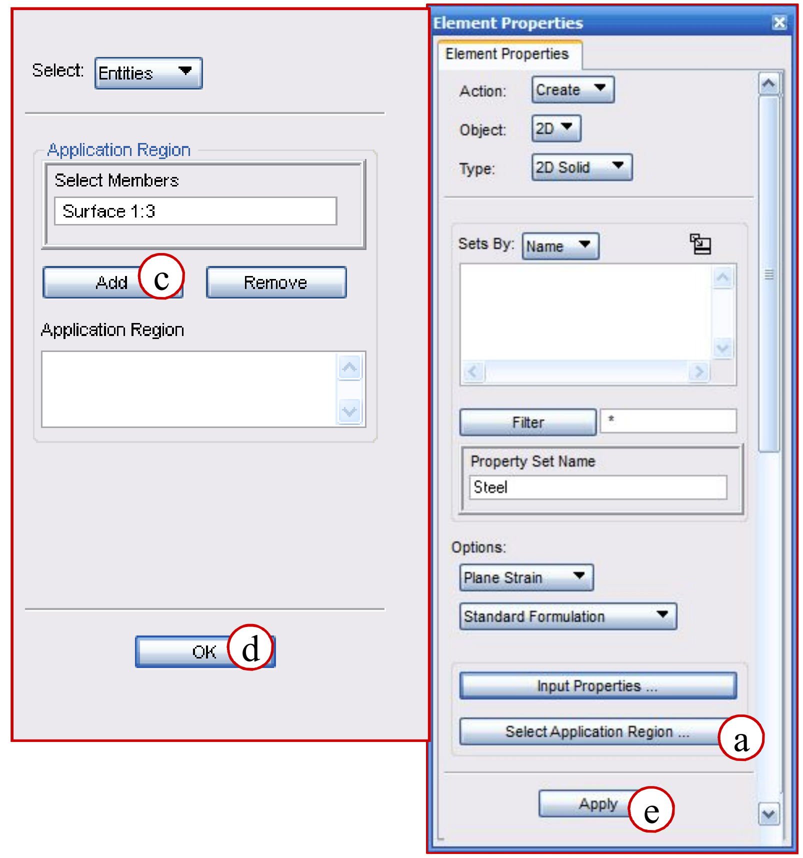
a. Click **2D Solid** in the *2D Properties* group.
b. Enter **Steel** for the *Property Set Name*.
c. Confirm that **Plane Strain** and **Standard Formulation** are selected under *Options*.
d. Click **Input Properties**.
e. Click the **Mat Prop Name** icon.
f. Choose **Steel** from the Existing Material list.
g. Pull down **Large Strain** for *Nonlinear Formulation (SOL400)*.
h. Click **OK**.



Step 8. Define the Element Properties (Cont.)

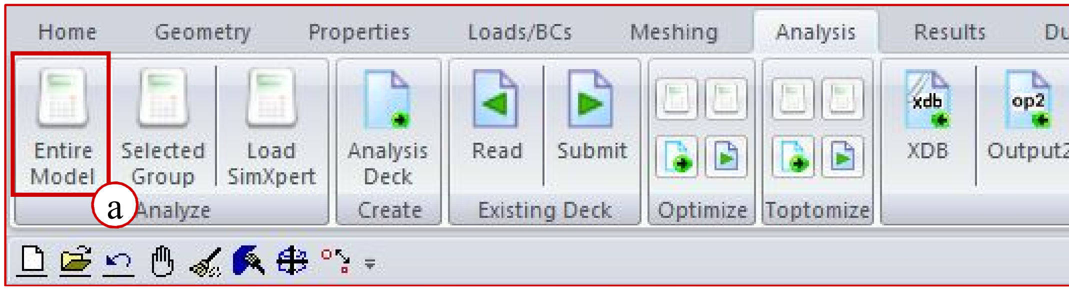
a. Click **Select Application Region**.
b. Drag a rectangle to select all three Surfaces.
c. Click **Add**.
d. Click **OK**
e. Click **Apply**





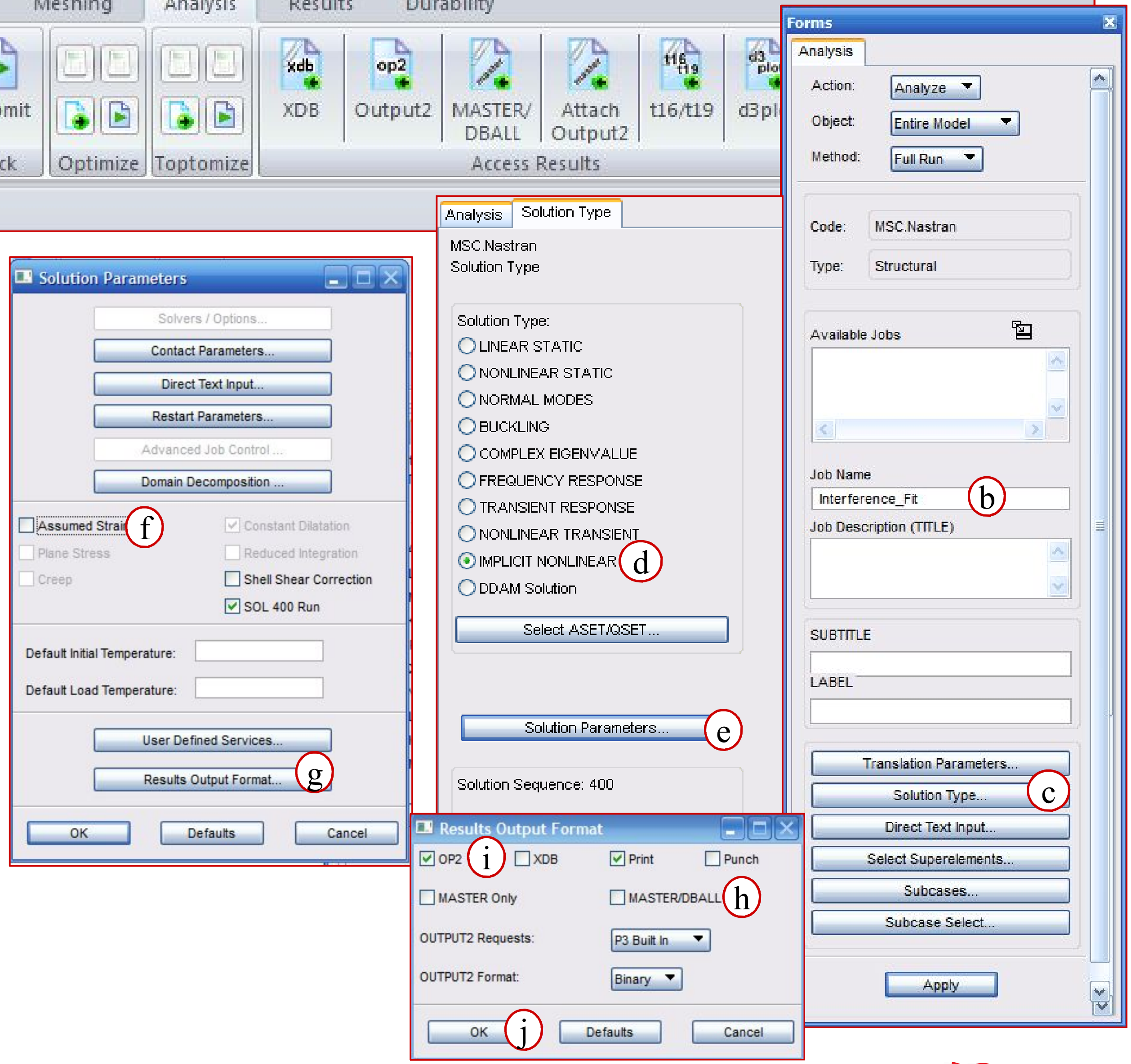
MSC Software

Step 9. Set up and Submit the Analysis



Set up the Analysis:

a. Under the *Analysis* tab, click **Entire Model** in the *Analyze* group.
b. Enter **Interference_Fit** as the *Job Name*.
c. Click **Solution Type**.
d. Select **Implicit Nonlinear** for *Solution Type.*
e. Click **Solution Parameters**.
f. Uncheck **Assumed Strain**
g. Click **Results Output Format…**
h. Uncheck **MASTER/DBALL.**
i. Select **OP2**
j. Click **OK**

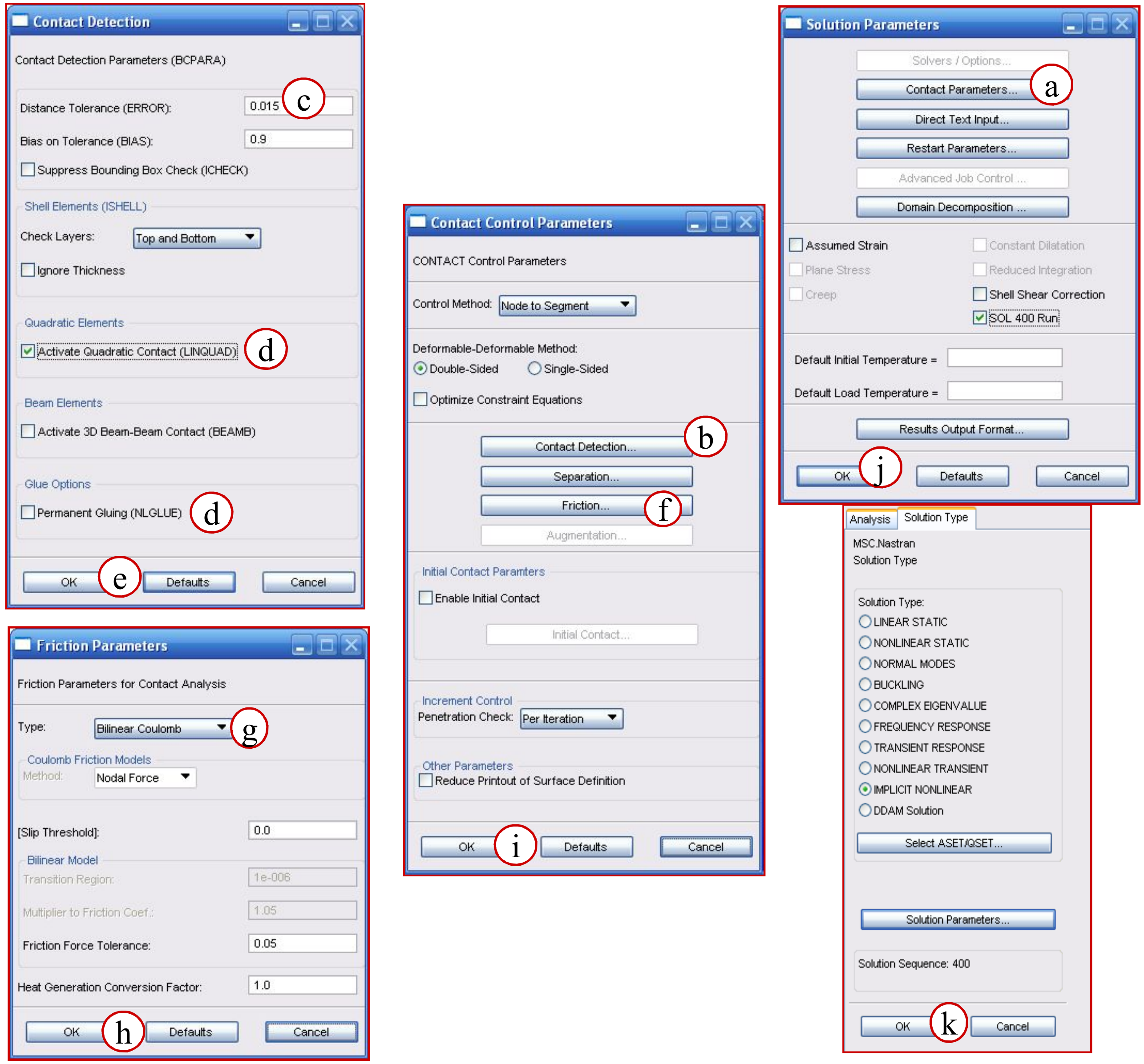


MSC Software

Step 9. Set up and Submit the Analysis (Cont.)

Define Contact Parameters:

a. Click **Contact Parameters**.
b. Click **Contact Detection**.
c. Enter **0.015** for *Distance Tolerance (ERROR).*
d. Check **Activate Quadratic Contact (LINQUAD)**. Uncheck **Permanent Gluing**
e. Click **OK**.
f. Click **Friction**.
g. Pull down **Bilinear Coulomb** for *Type*.
h. Click **OK**.
i. Click **OK**.
j. Click **OK**.
k. Click **OK**.

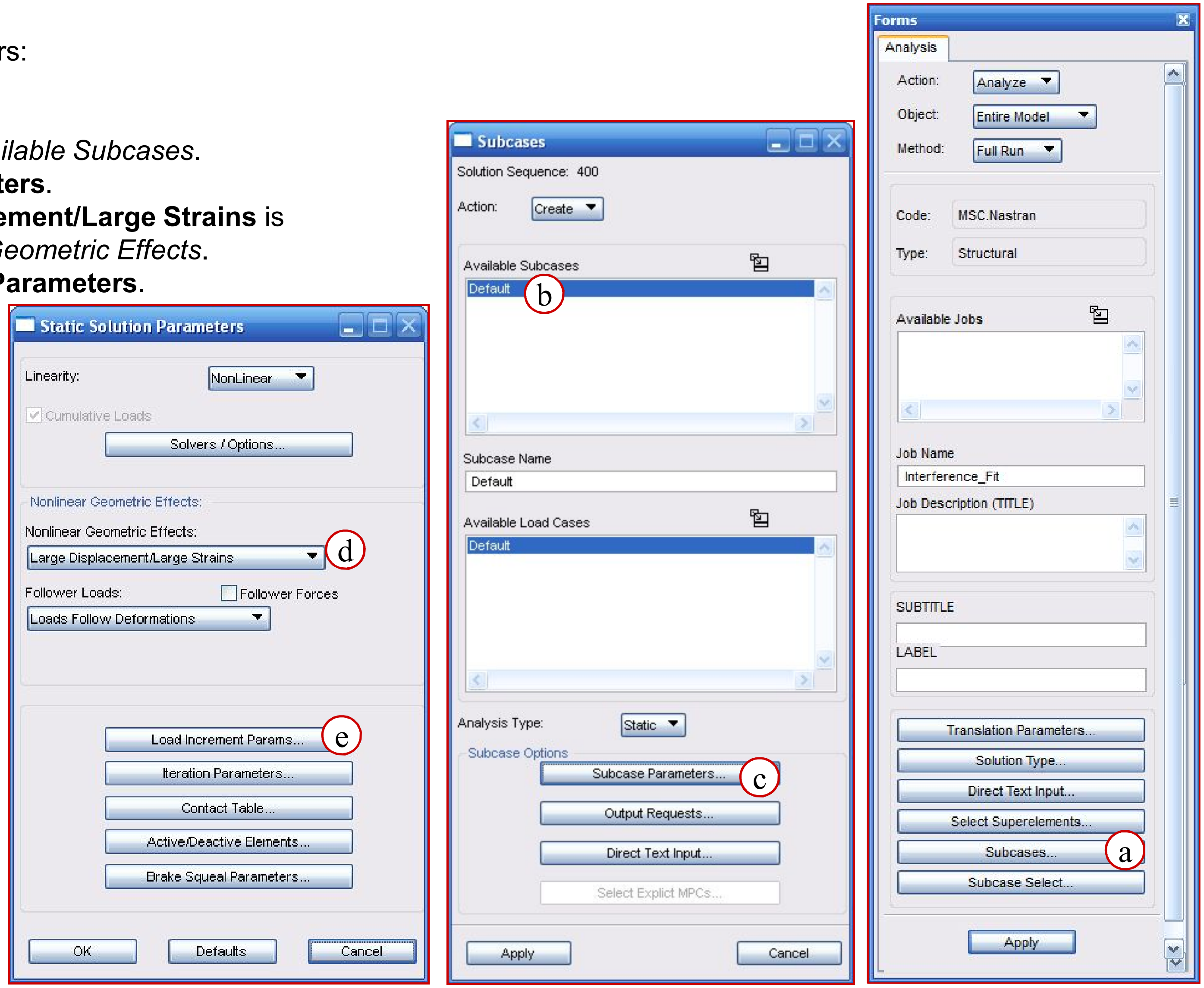


MSC Software

Step 9. Set up and Submit the Analysis (Cont.)

Define Subcase Parameters:

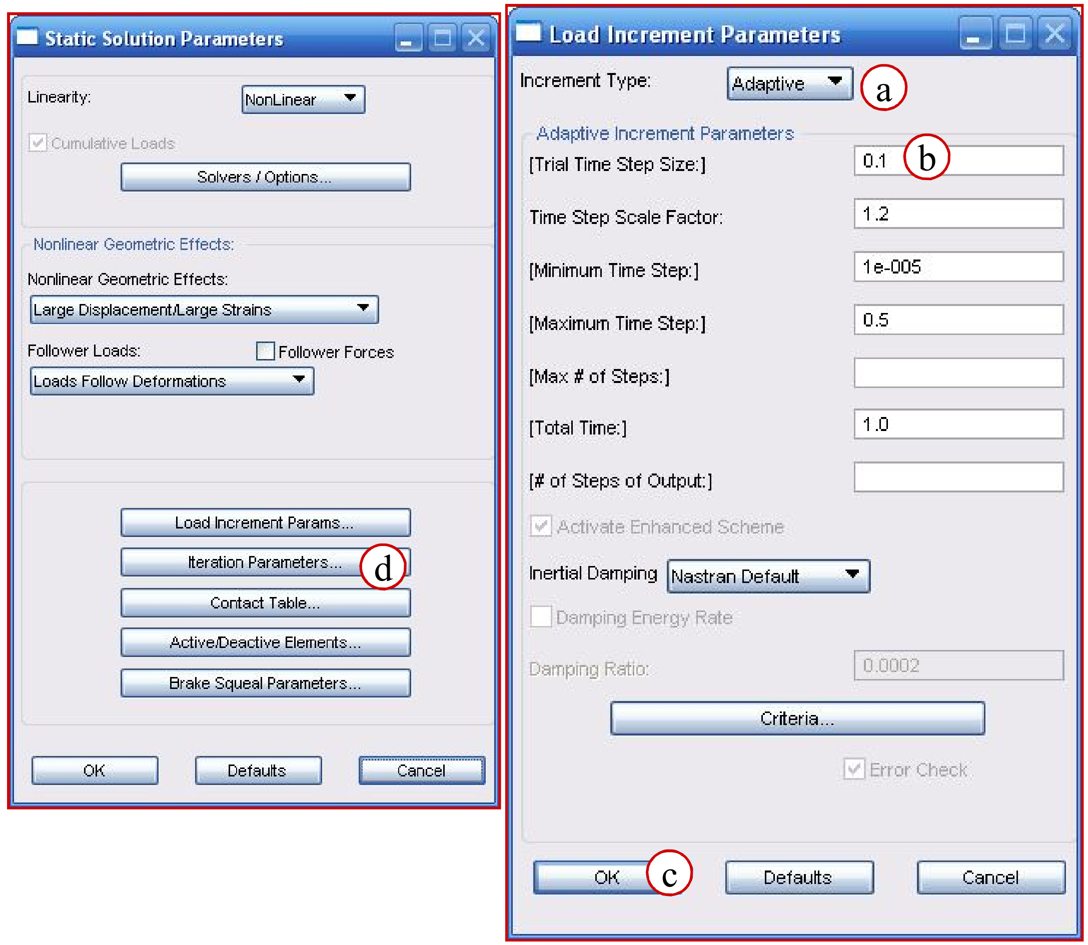
a. Click **Subcases**.
b. Click **Default** under *Available Subcases*.
c. Click **Subcase Parameters**.
d. Confirm **Large Displacement/Large Strains** is selected for *Nonlinear Geometric Effects*.
e. Click **Load Increment Parameters**.



Step 9. Set up and Submit the Analysis (Cont.)

Set Increment Type to Adaptive and Trial Time Step Size = **0.1** :

a. Pull down *Increment Type* to **Adaptive**.
b. Enter ***0.1*** for *Trial Time Step Size.*
c. Click **OK**.
d. Click **Iteration Parameters**.

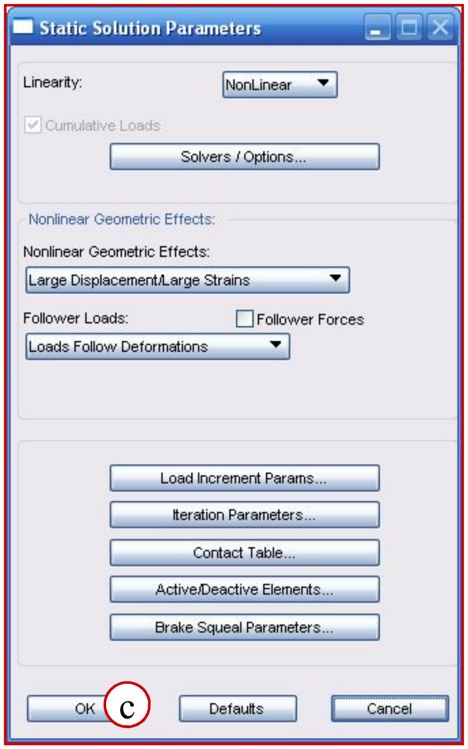


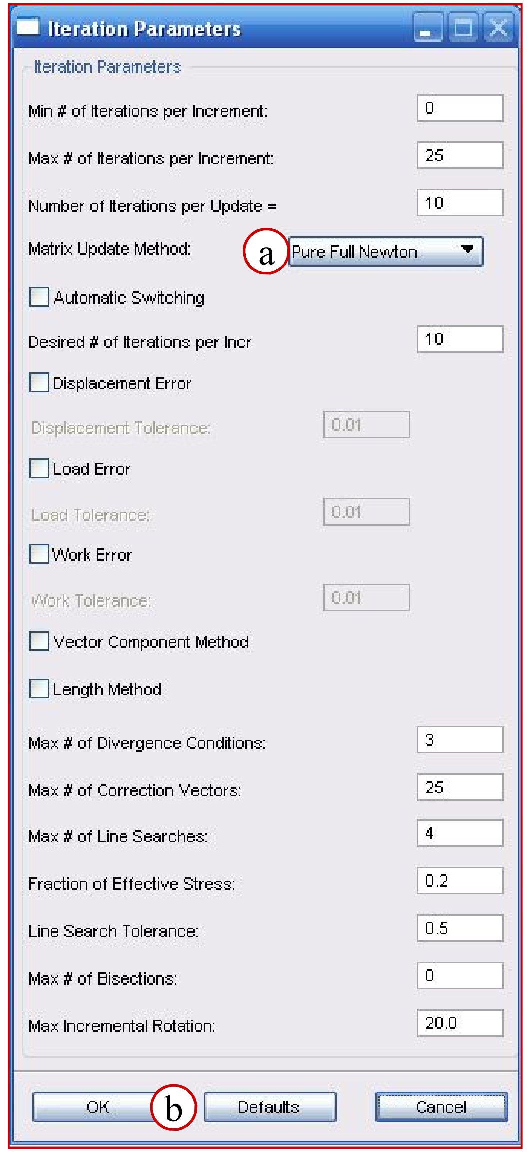
MSC Software

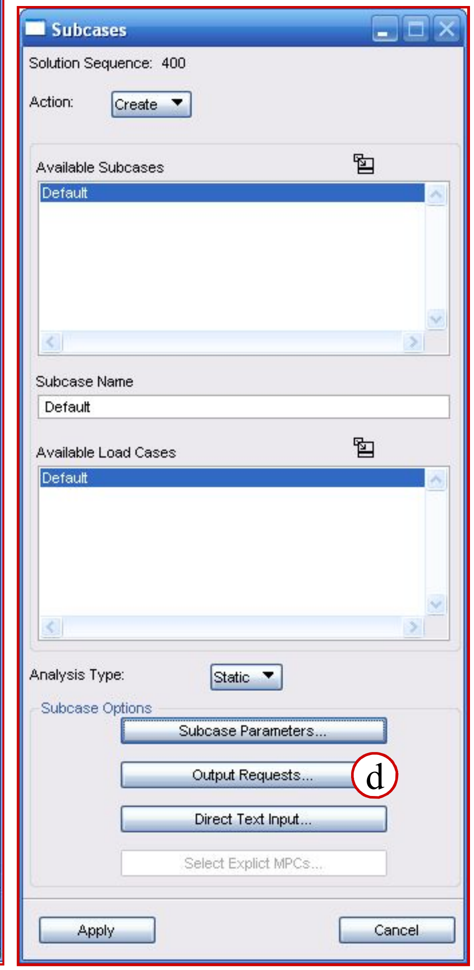
Step 9. Set up and Submit the Analysis (Cont.)

Select Pure Full Newton for the Matrix Update Method

a. Pull down **Pure Full Newton** for the *Matrix Update Method*.
b. Click **OK**.
c. Click **OK** on the *Static Solution Parameters* form.
d. Click **Output Requests**.





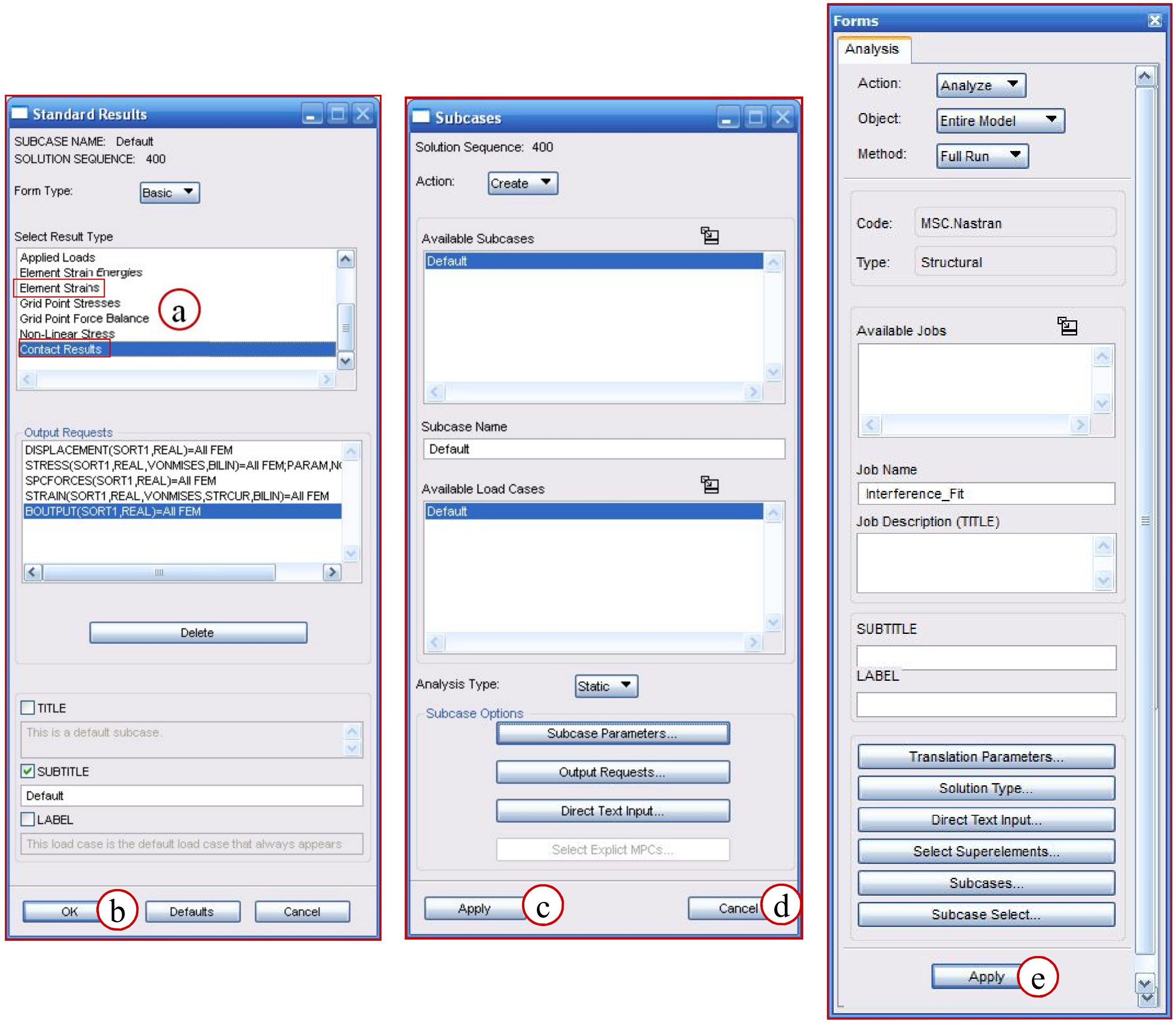


MSC Software

Step 9. Set up and Submit the Analysis (Cont.)

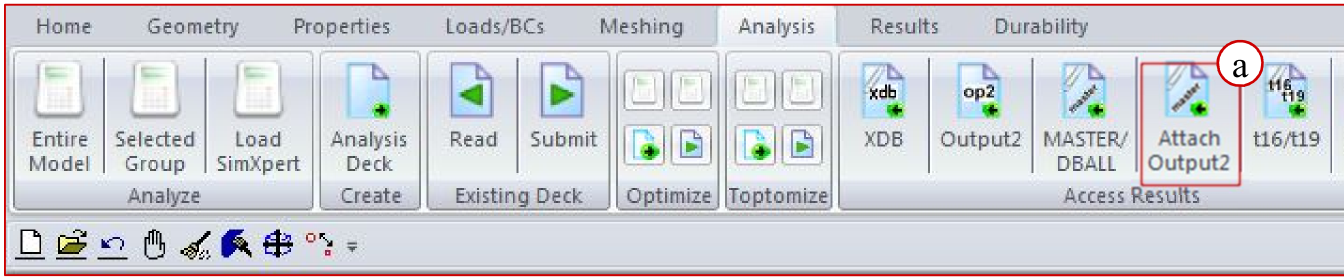
Select to Output Element Strain and Contact results (cont.):

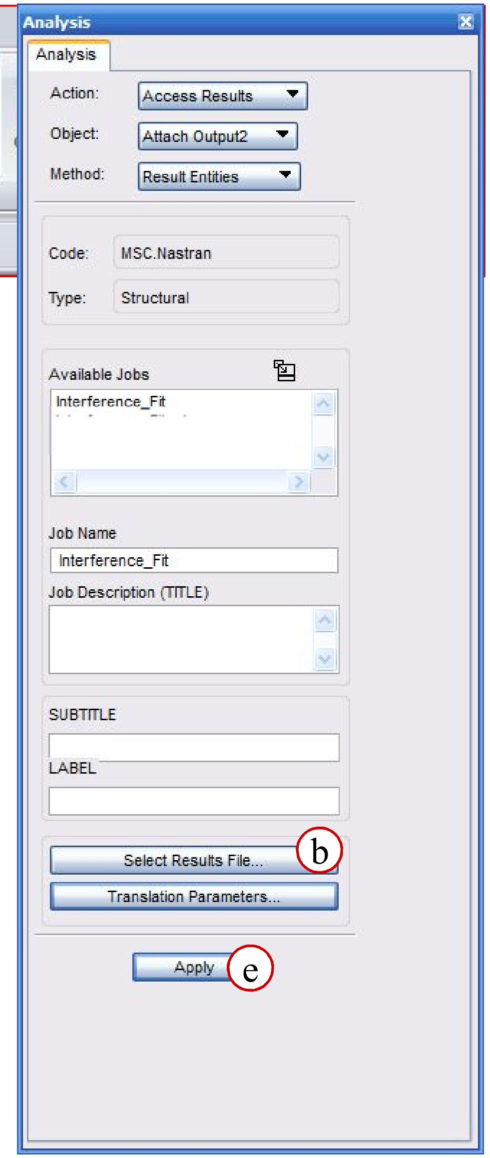
a. Select **Element Strains** and **Contact Results**.
b. Click **OK**.
c. Click **Apply**.
d. Click **Cancel**.
e. Click **Apply**.



MSC Software

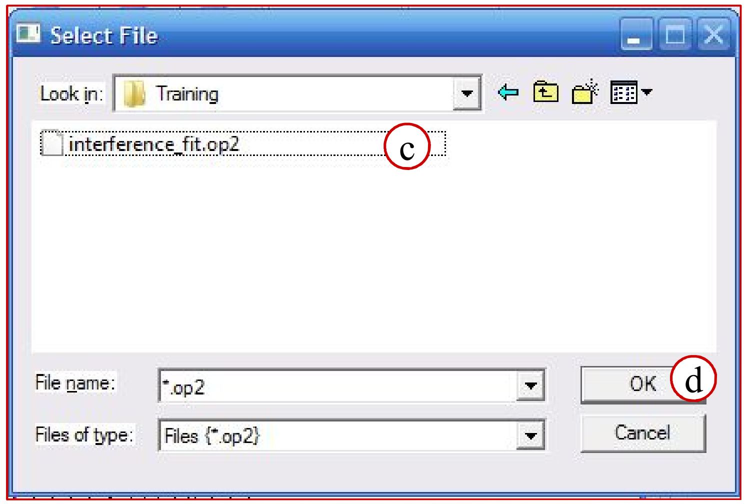
Step 10. Attach the .op2 Results File



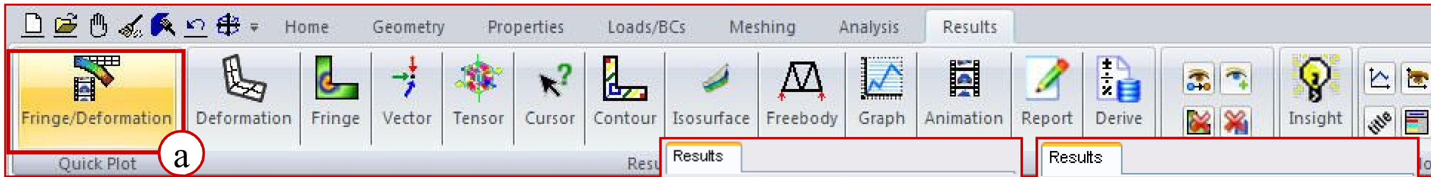


Attach the .OP2 Results file.

a. Click **Attach Output2** in the *Access Results* group.
b. Click **Select Results File…**
c. Select **interference_fit.op2**
d. Click **OK**.
e. Click **Apply**.

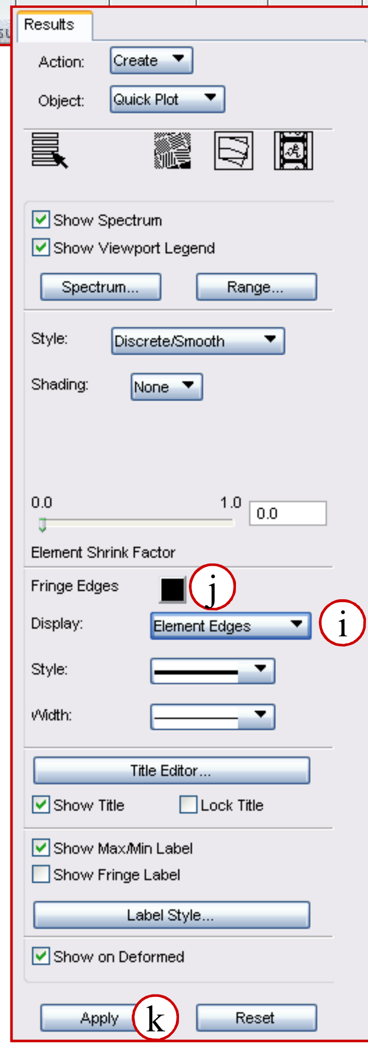


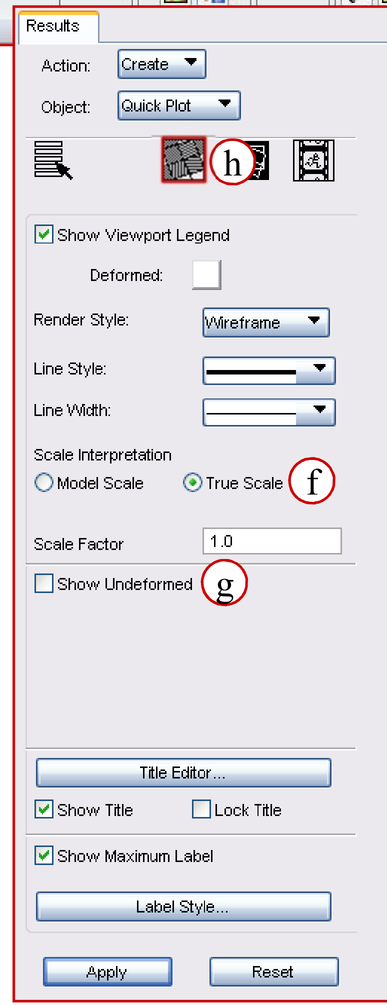
Step 11. View Deformation and Stress Results

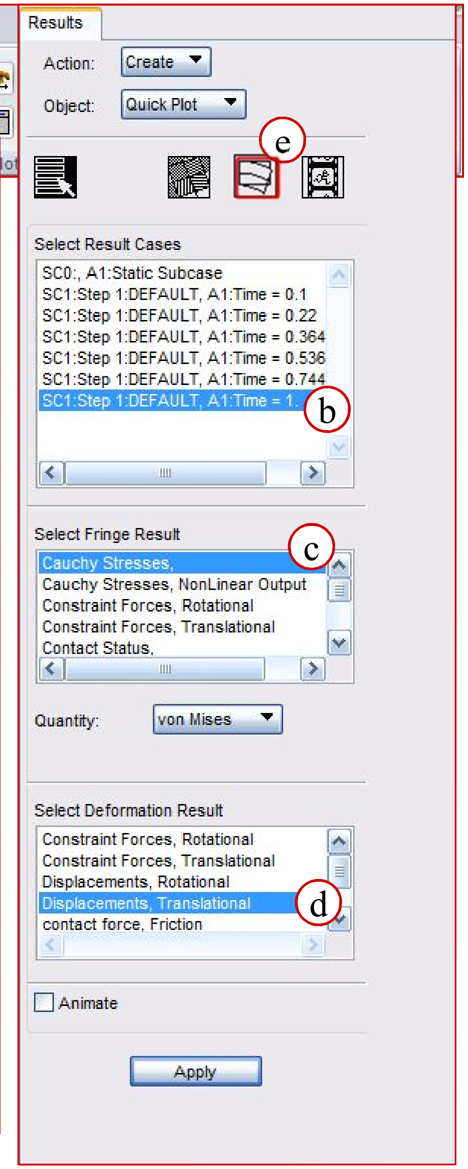


Plot the Cauchy Stress Fringe Results superimposed on the true scale deformed shape for the final load increment.

a. Under the *Results* tab, click **Fringe/Deformation** in the *Quick Plot* group.
b. Select the final result case.
c. Select **Cauchy Stresses** for Fringe Result.
d. Select **Displacements, Translational** for Deformation Result.
e. Click **Deform Attributes**.
f. Select **True Scale**.
g. Uncheck **Show Undeformed**.
h. Click **Fringe Attributes**.
i. Pull down **Element Edges** for *Display*.
j. Select **Black** for *Fringe Edges*.
k. Click **Apply**.

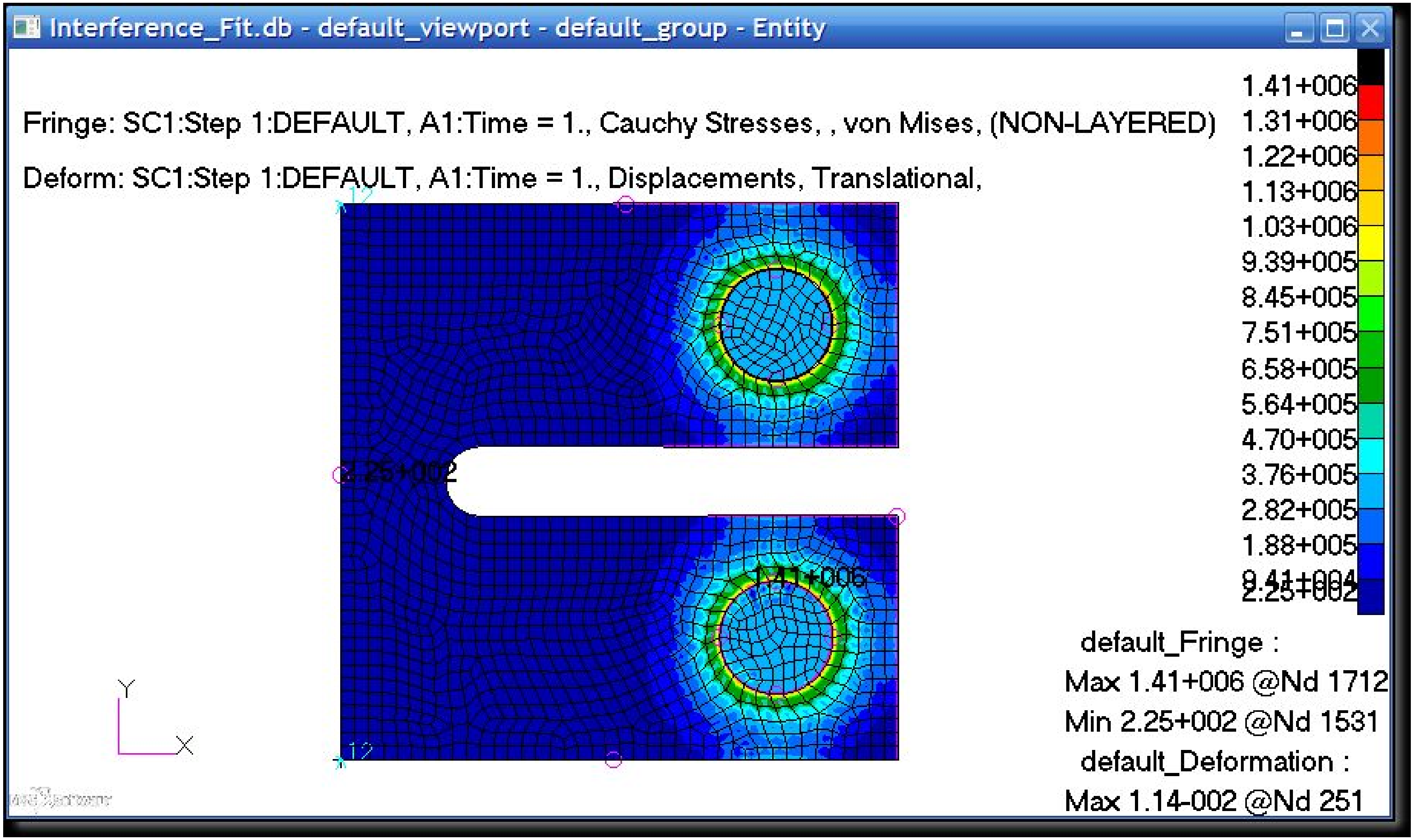




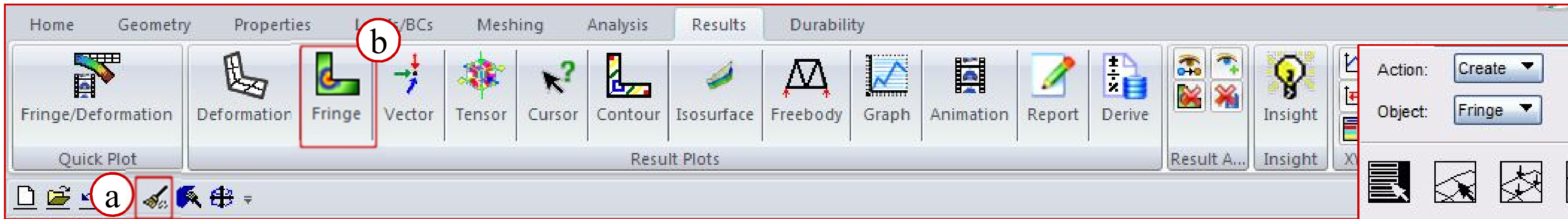


MSC Software

Step 11. View Deformation and Stress Results (Cont.)

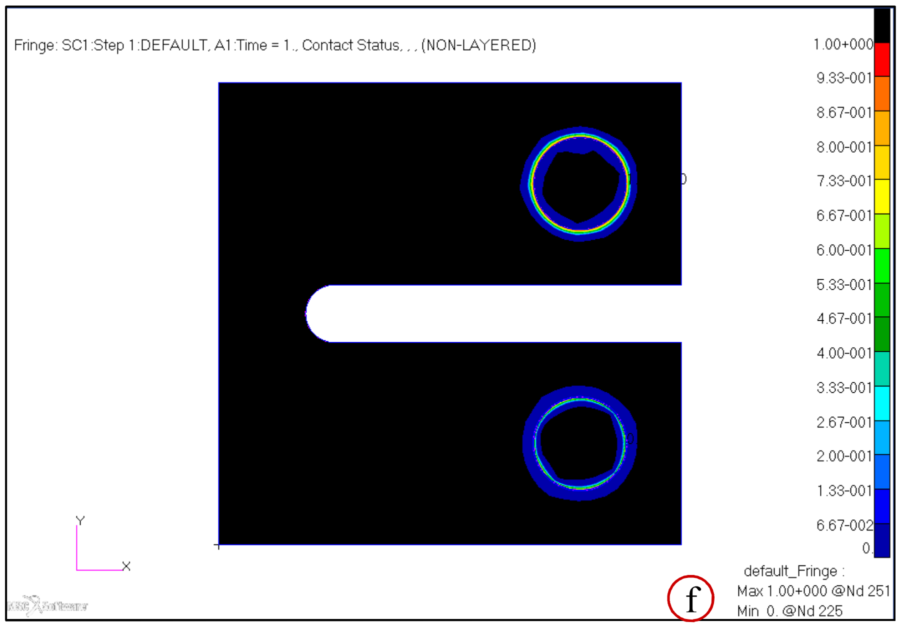


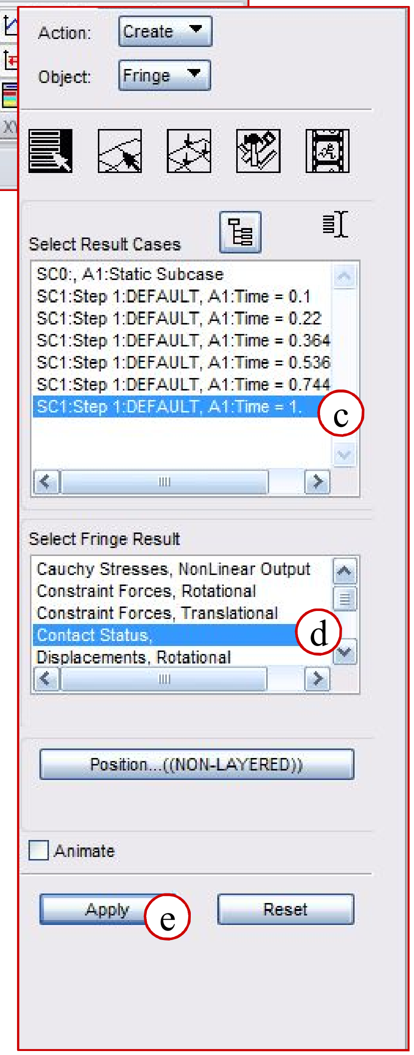
Step 12. View Contact Results



Plot Contact Status results.

a. Click **Reset Graphics** to clear the current results plot.
b. Click **Fringe** in the *Result Plots* group.
c. Select the final result case.
d. Select **Contact Status** under *Select Fringe Result*.
e. Click **Apply**.
f. Note the range from 0 to 1. Zero indicating the master body and 1, the slave body.

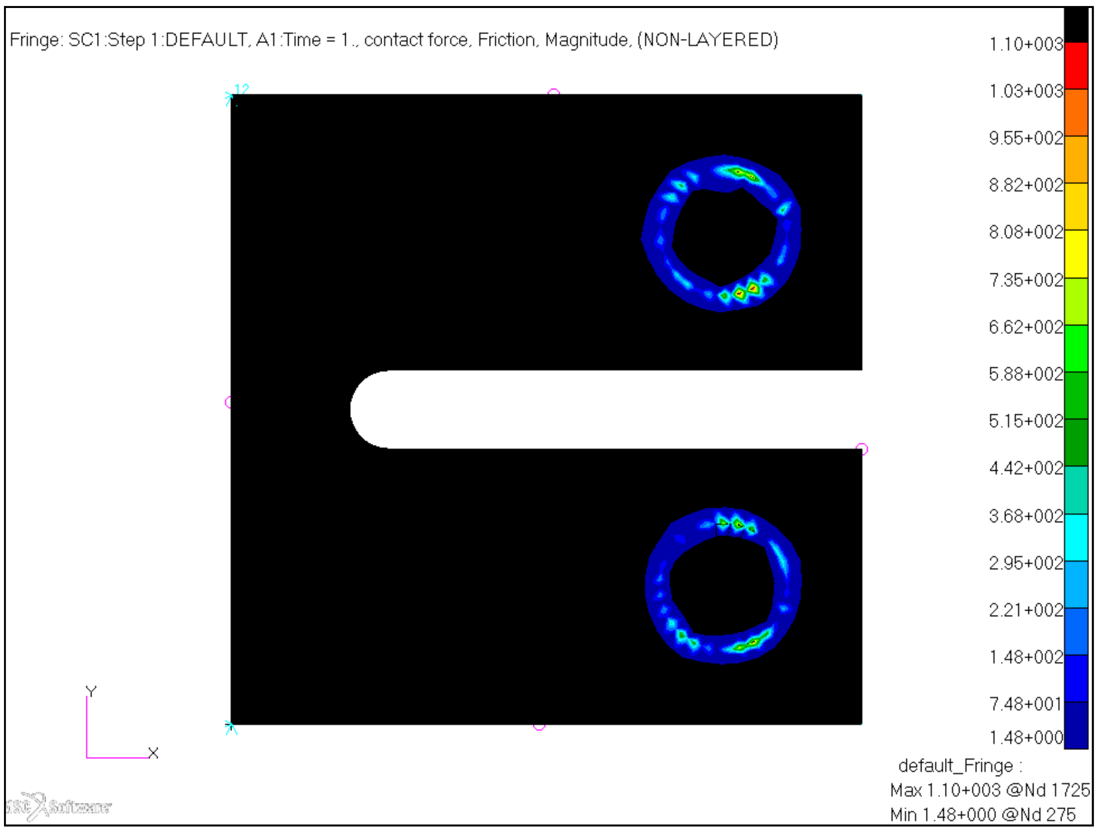


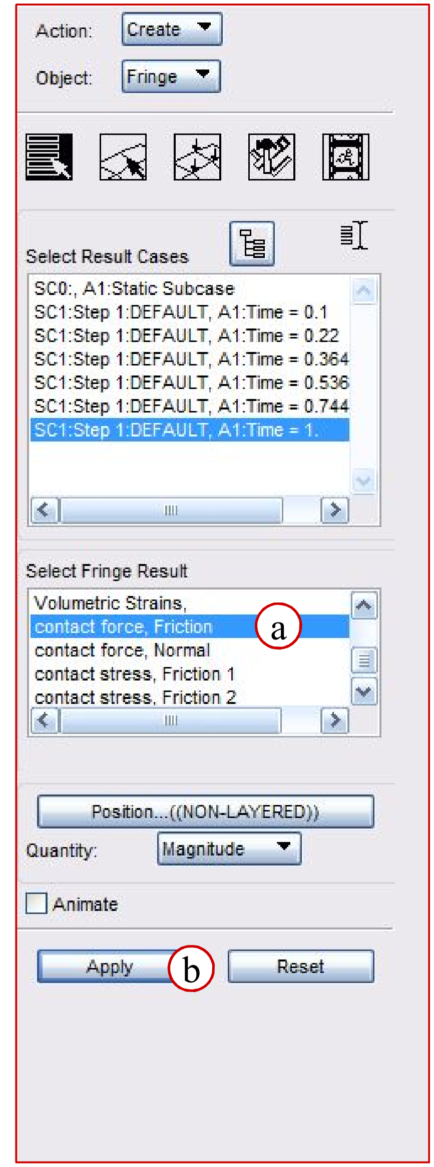


Step 12. View Contact Results (Cont.)

Plot contact friction force.

a. Select **contact force, Friction** under *Select Fringe Result*.
b. Click **Apply**.



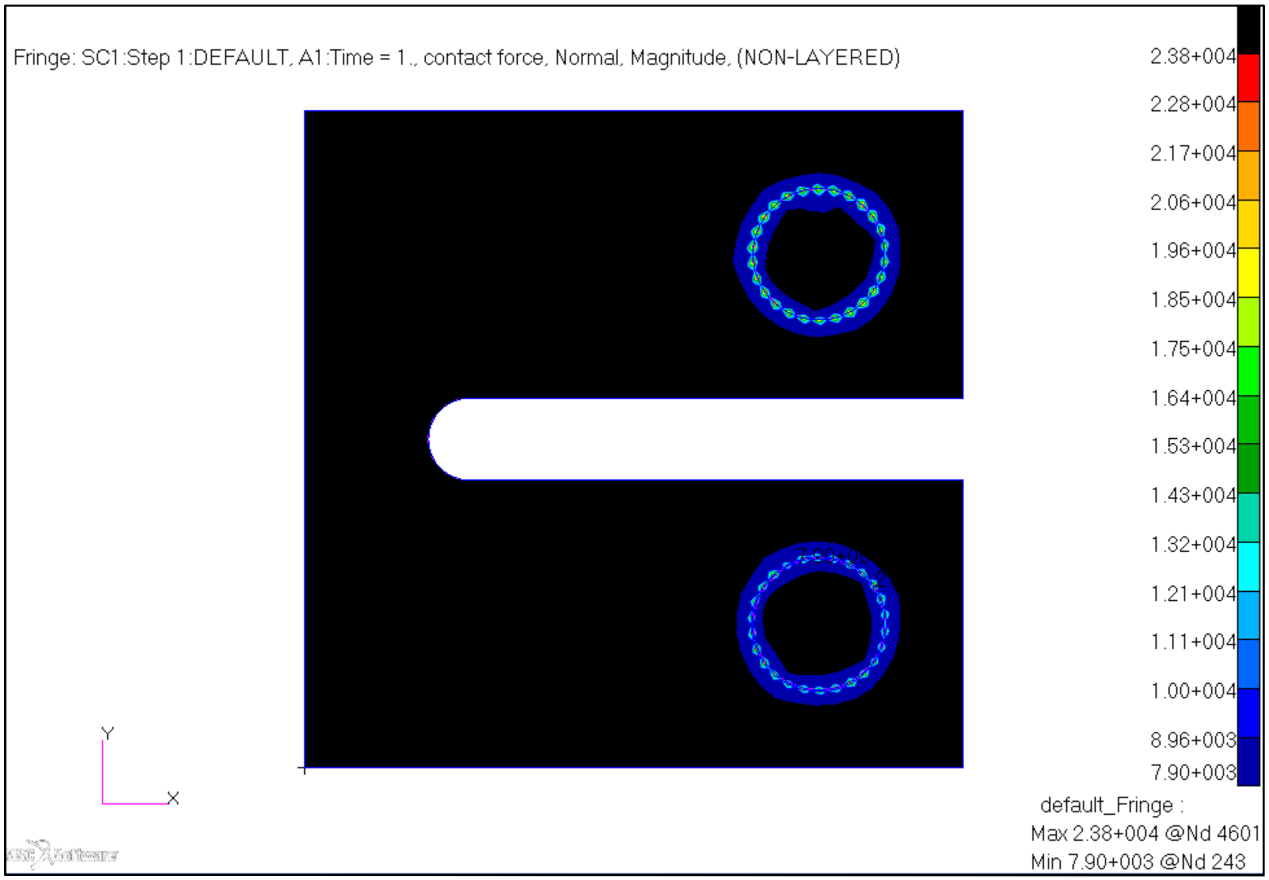


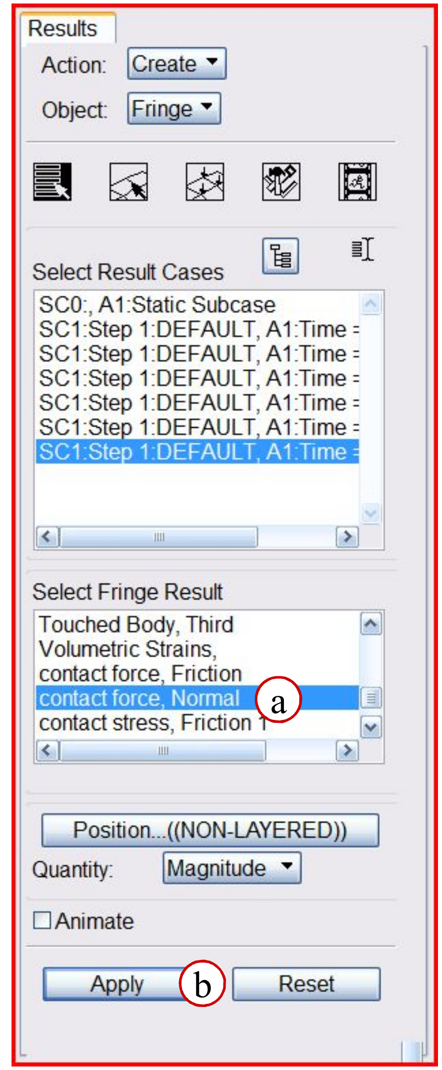
Step 12. View Contact Results (Cont.)

Plot contact normal force.

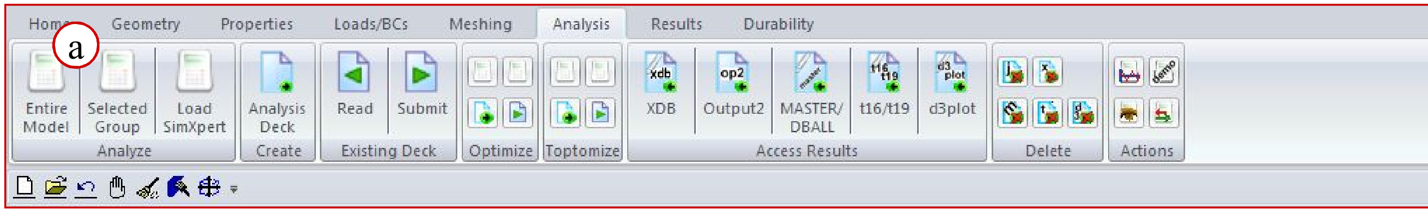
a. Select **contact force, Normal** under *Select Fringe Result*.

b. Click **Apply**.



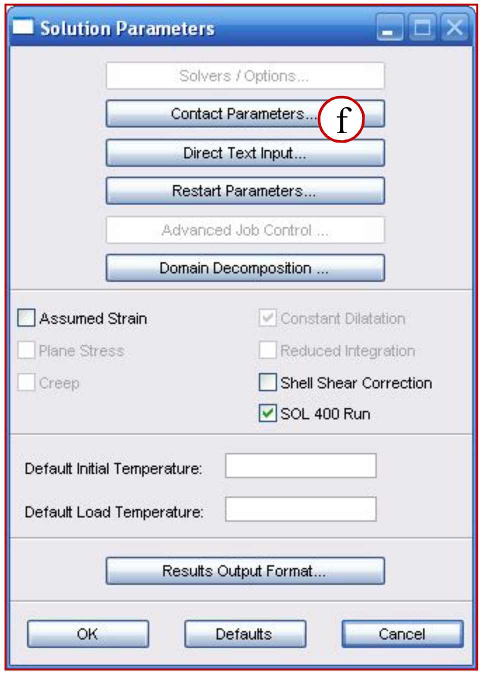


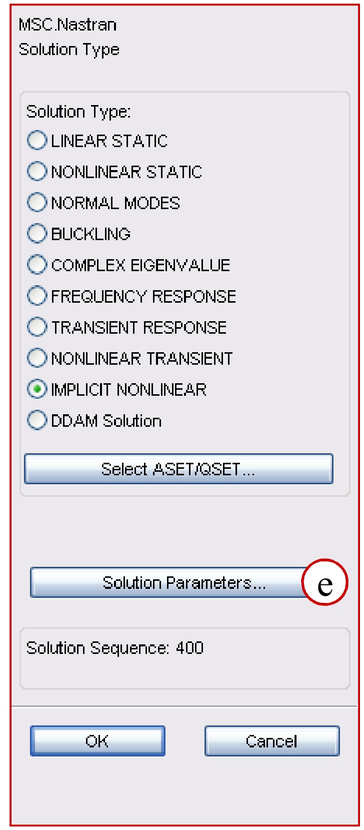
Step 13. Run Analysis with Segment to Segment

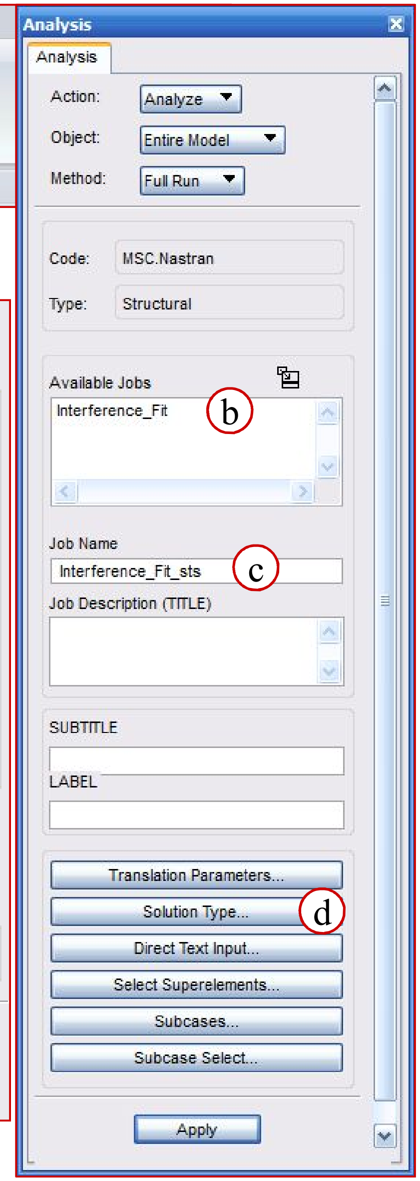


Run the Analysis using Segment to Segment contact in place of Node to Segment contact.

a. Under the *Analysis* tab, click **Entire Model** in the *Analyze* group.
b. Select **Interference_Fit** from the list of *Available Jobs*.
c. Change the *Job Name* to **Interference_Fit_sts**.
d. Click **Solution Type…**
e. Click **Solution Parameters…**
f. Click **Contact Parameters…**



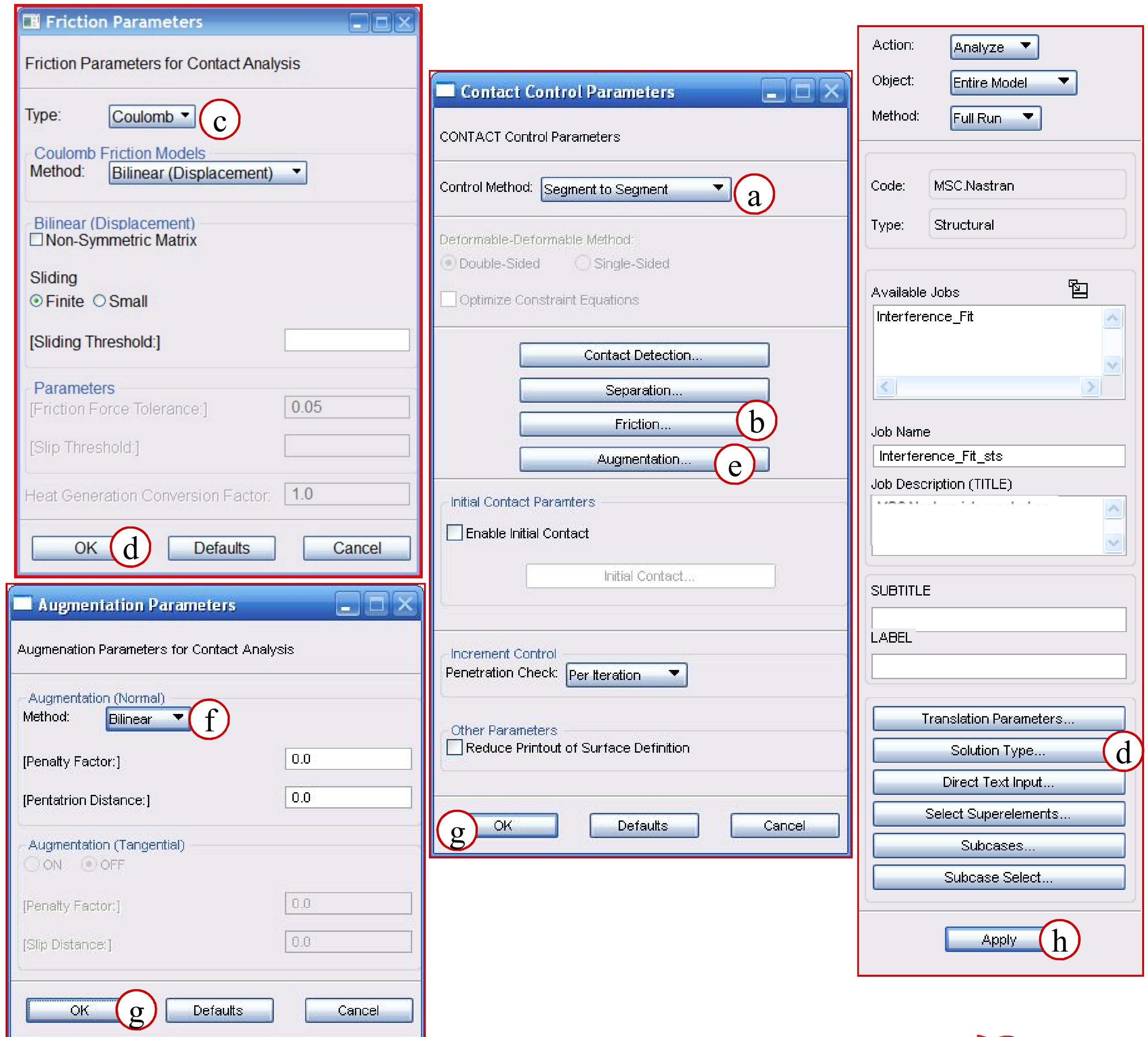




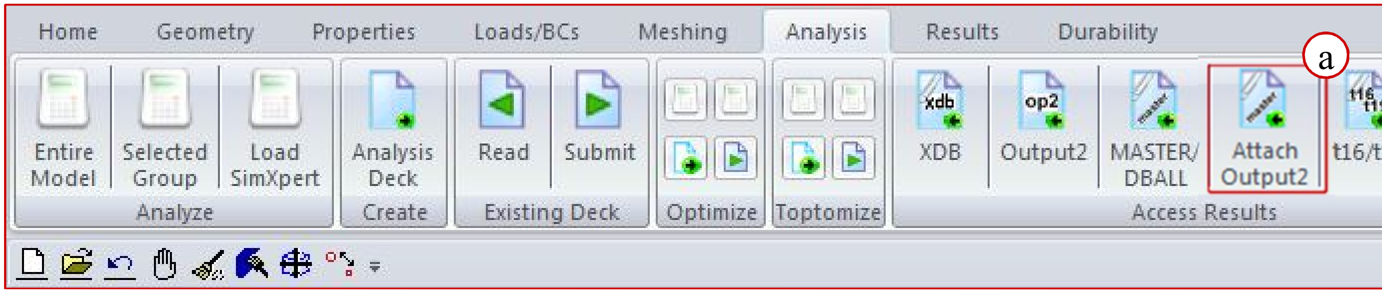
Step 13. Run Analysis with Segment to Segment (Cont.)

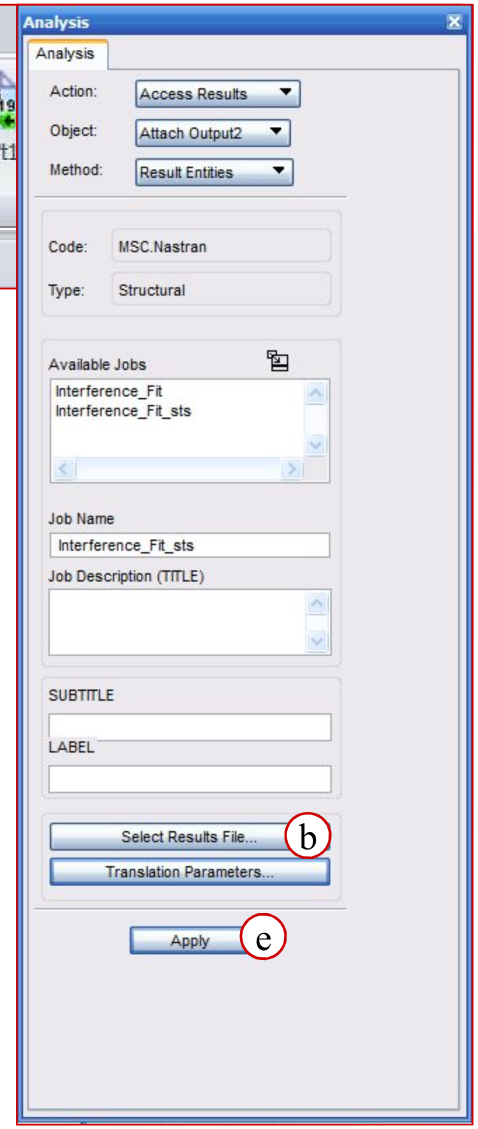
Run the Analysis using Segment to Segment contact in place of Node to Segment contact.

a. Pull down *Control Method* to **Segment to Segment**.
b. Click **Friction**
c. Pull down *Type* to **Coulomb**
d. Click **OK**
e. Click **Augmentation…**
f. Pull down *Augmentation Method* to **Bilinear**.
g. Click **OK** on all forms.
h. Click **Apply**.



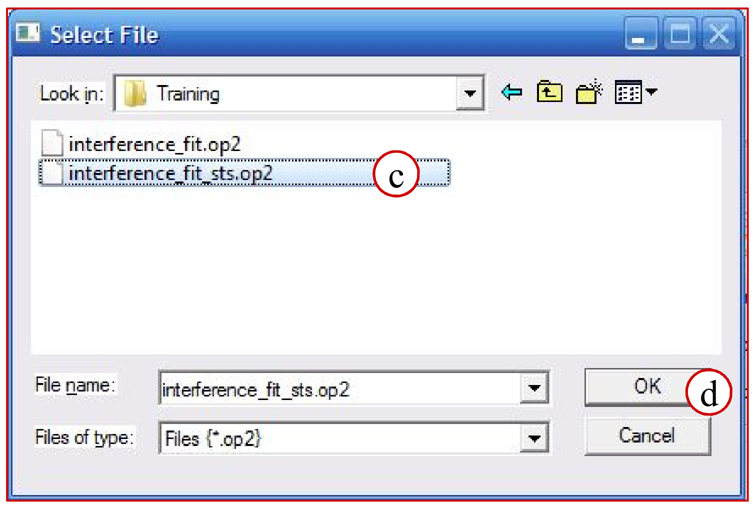
Step 14. Attach Segment to Segment Result File



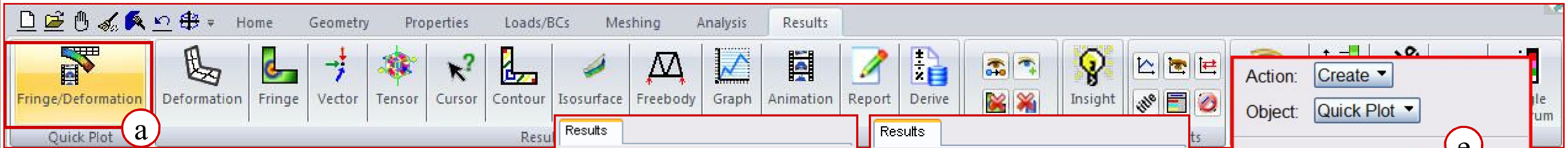


Attach the Segment to Segment .op2 Result file

a. Click **AttachOutput2** in the *Access Results* group.
b. Click **Select Results File…**
c. Select **interference_fit_sts.op2**.
d. Click **OK**.
e. Click **Apply**.

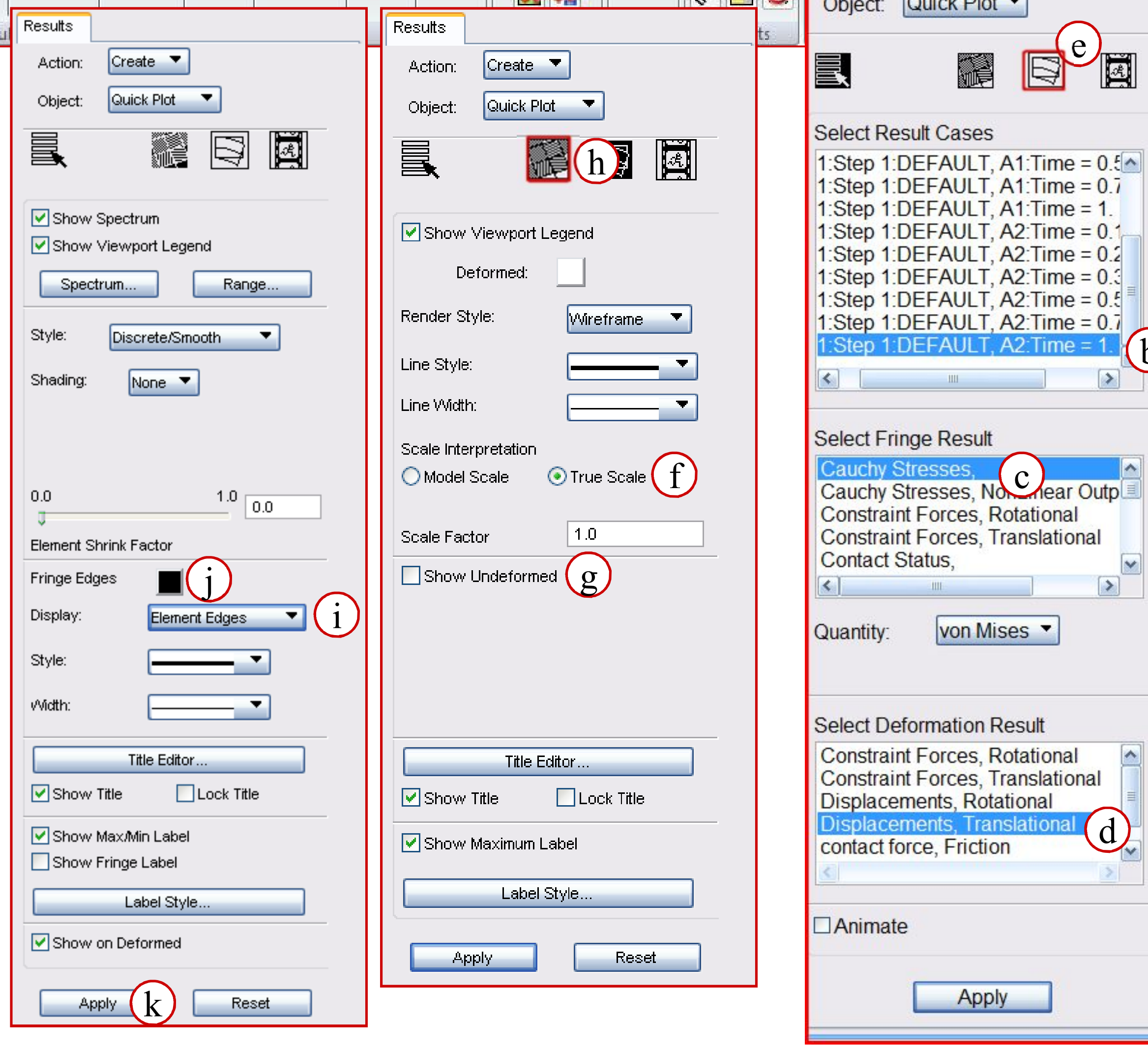


Step 15. View Segment to Segment Results

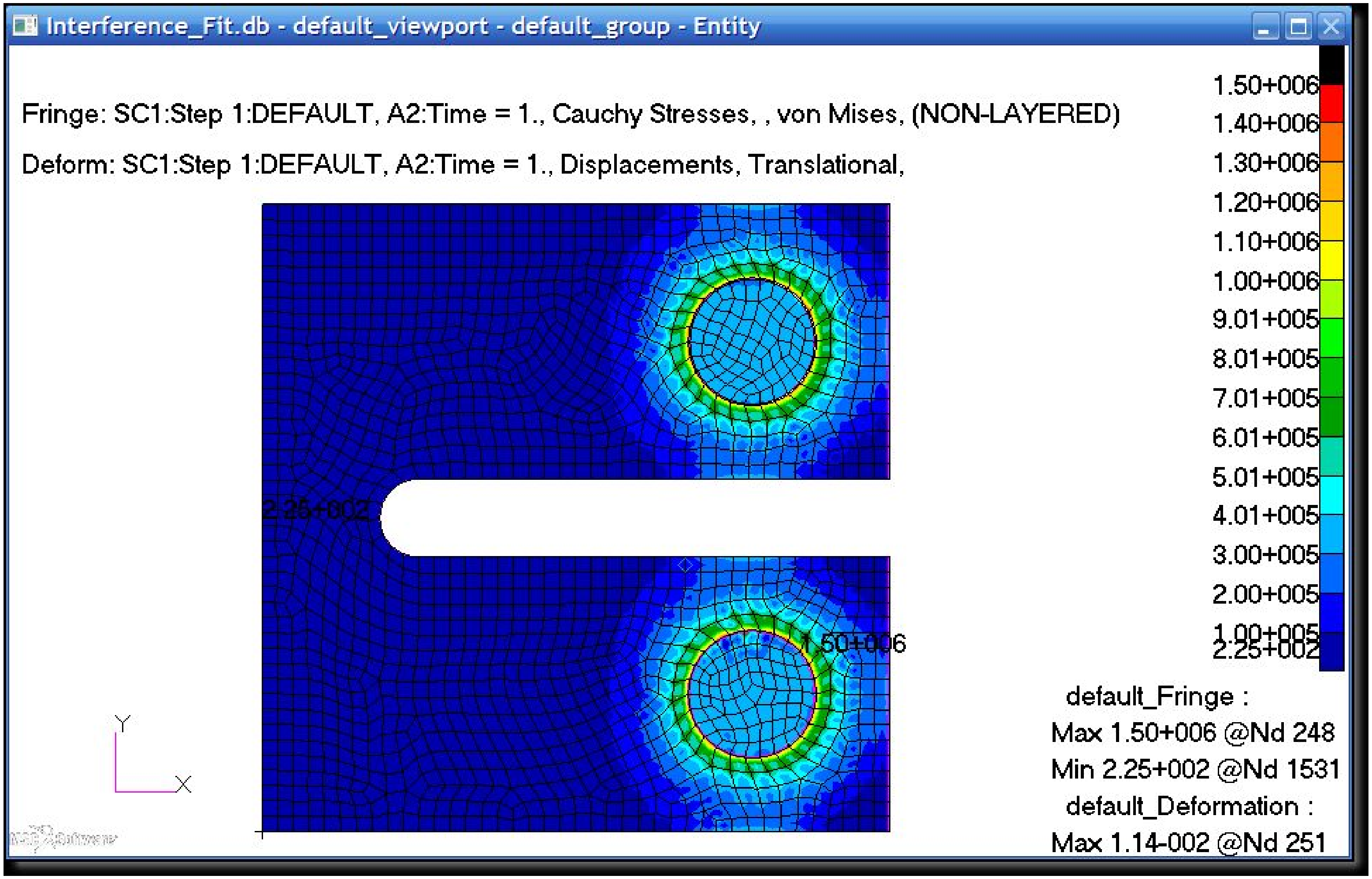


Plot the Cauchy Stress Fringe Results superimposed on the true scale deformed shape.

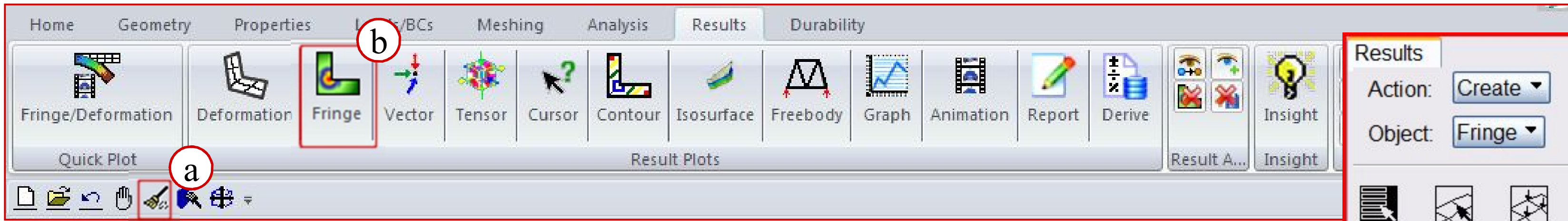
a. Under the *Results* tab, click **Fringe/Deformation** in the *Quick Plot* group.
b. Select the final result case.
c. Select **Cauchy Stresses** for Fringe Result.
d. Select **Displacements, Translational** for Deformation Result.
e. Click **Deform Attributes**.
f. Select **True Scale**.
g. Uncheck **Show Undeformed**.
h. Click **Fringe Attributes**.
i. Pull down **Element Edges** for *Display*.
j. Select **Black** for *Fringe Edges*.
k. Click **Apply**.



Step 15. View Segment to Segment Results (Cont.)

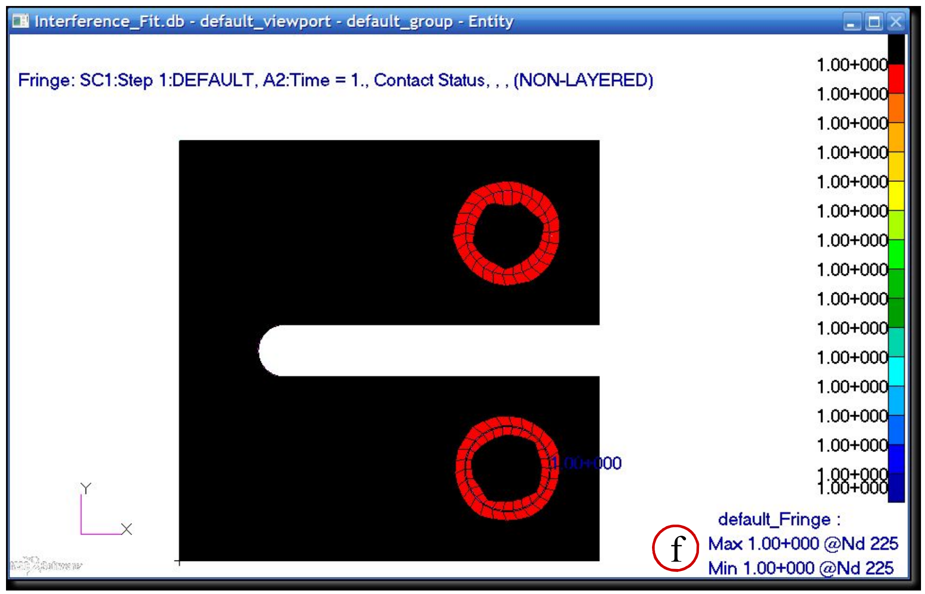


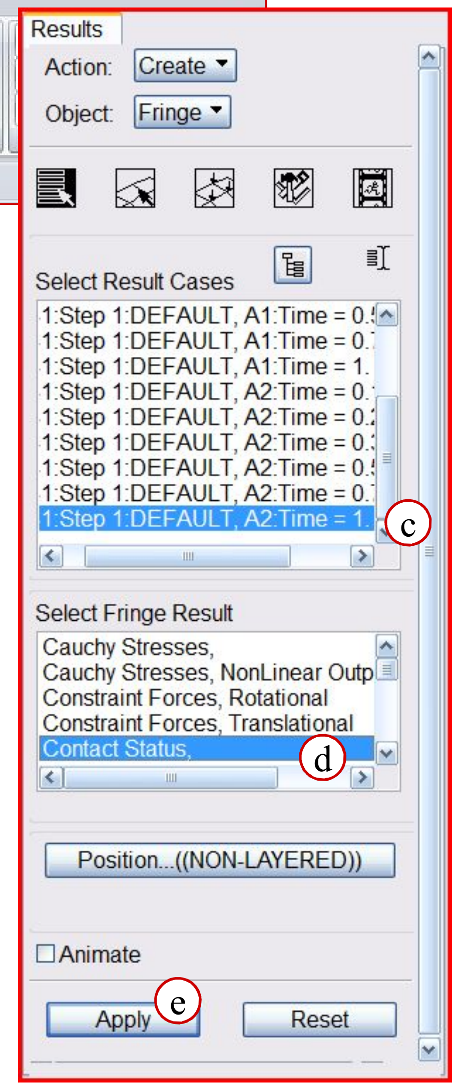
Step 16. View Segment to Segment Contact Results



Plot Contact Status results.

a. Click **Reset Graphics** to clear the current results plot.
b. Click **Fringe** in the *Result Plots* group.
c. Select the final result case.
d. Select **Contact Status** under *Select Fringe Result*.
e. Click **Apply**.
f. Note the range is now uniformly 1. For segment to segment contact, there is no slave and master relationship.

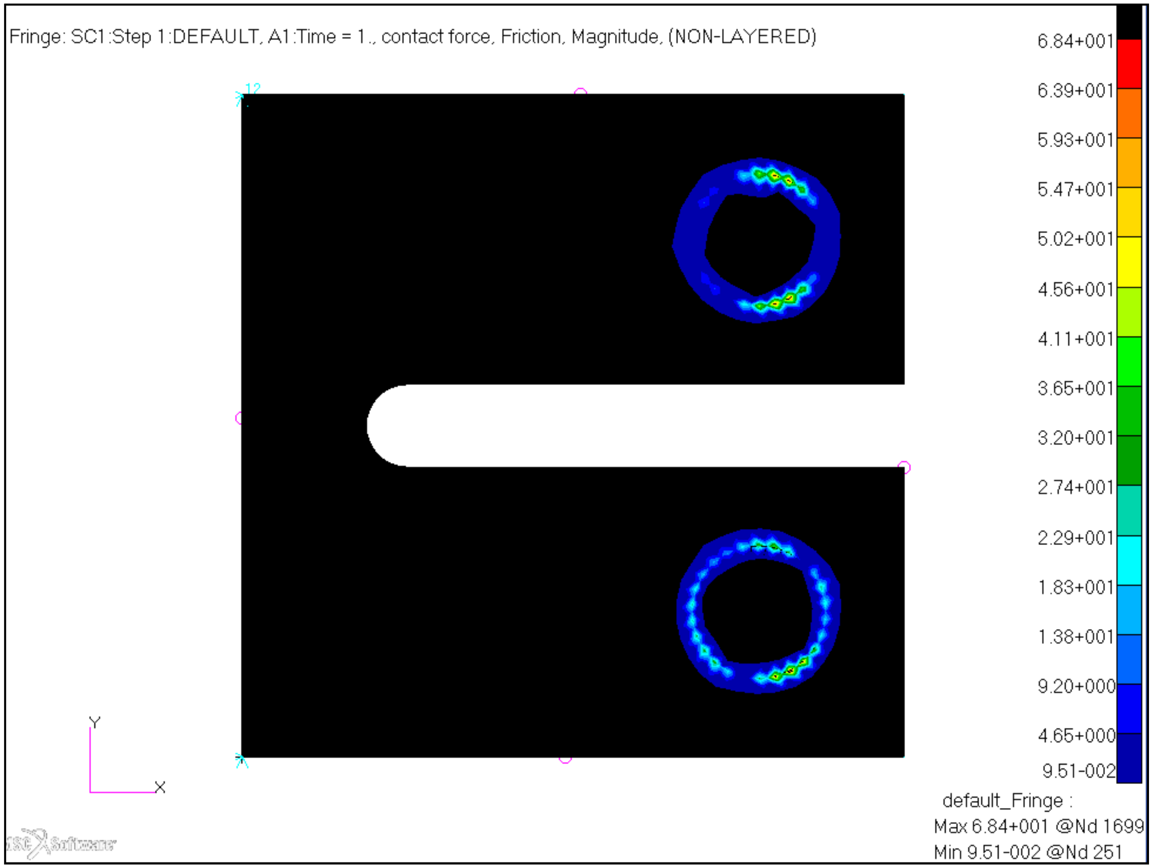


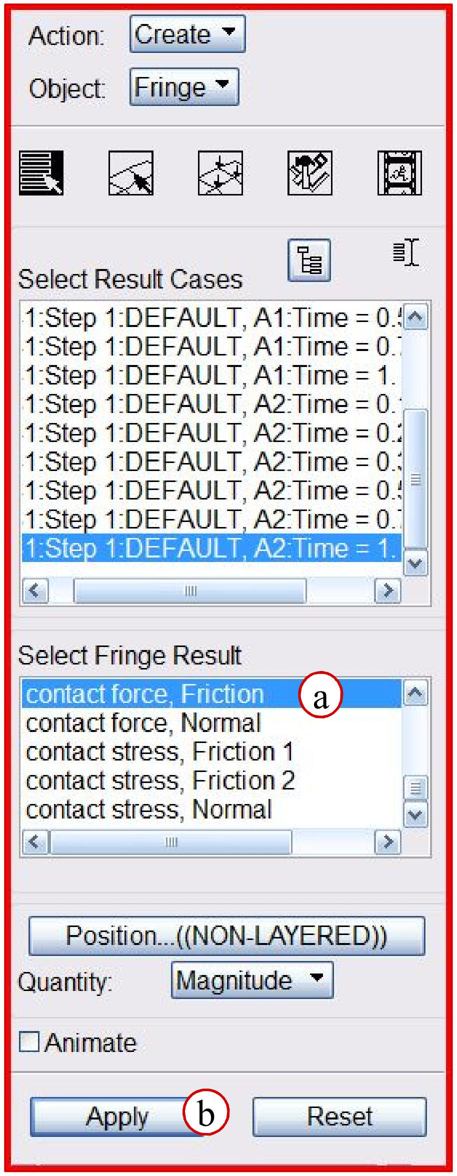


Step 16. View Segment to Segment Contact Results (Cont.)

Plot contact friction force.

a. Select **contact force, Friction** under *Select Fringe Result*.
b. Click **Apply**.



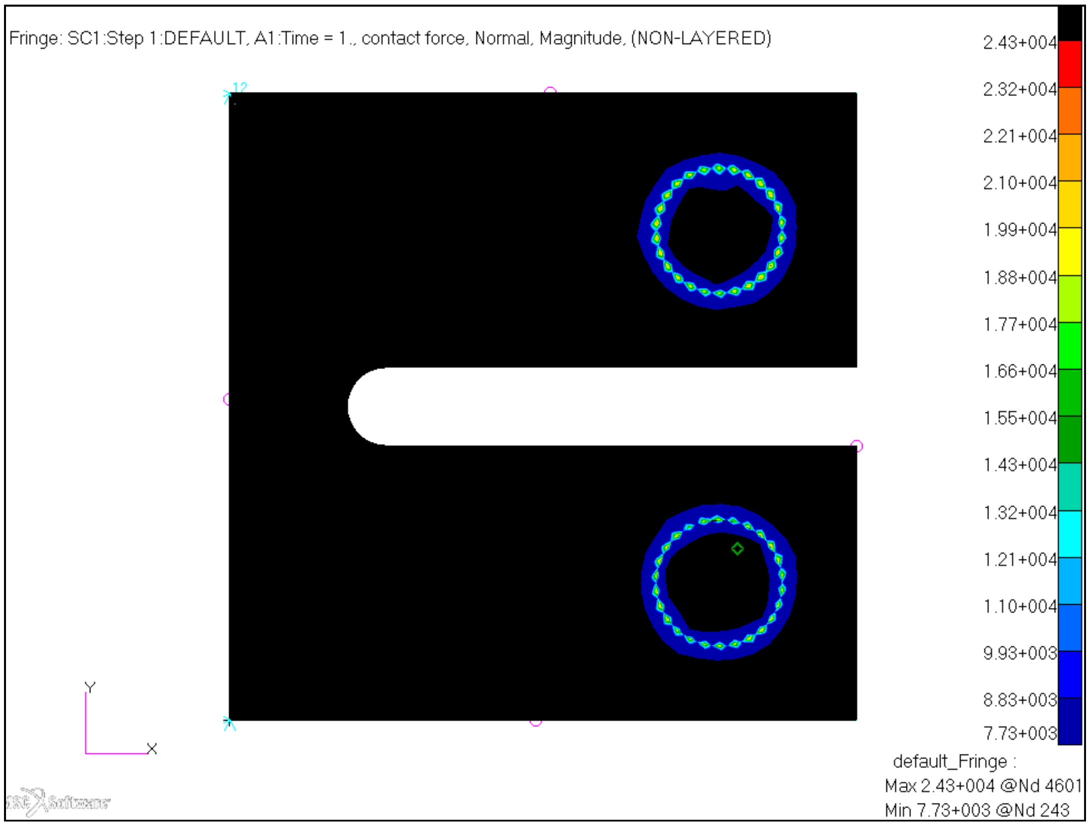


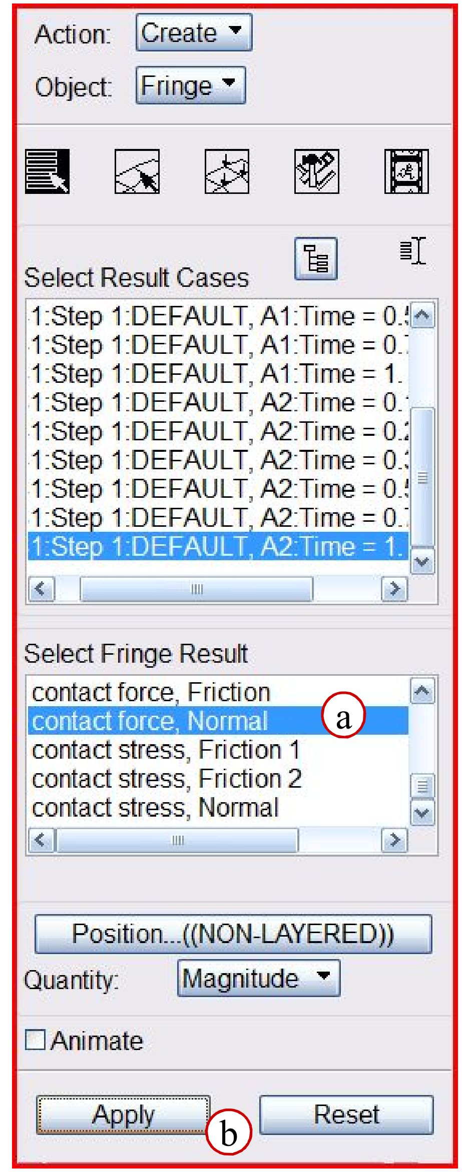
Step 16. View Segment to Segment Contact Results (Cont.)

Plot contact normal force.

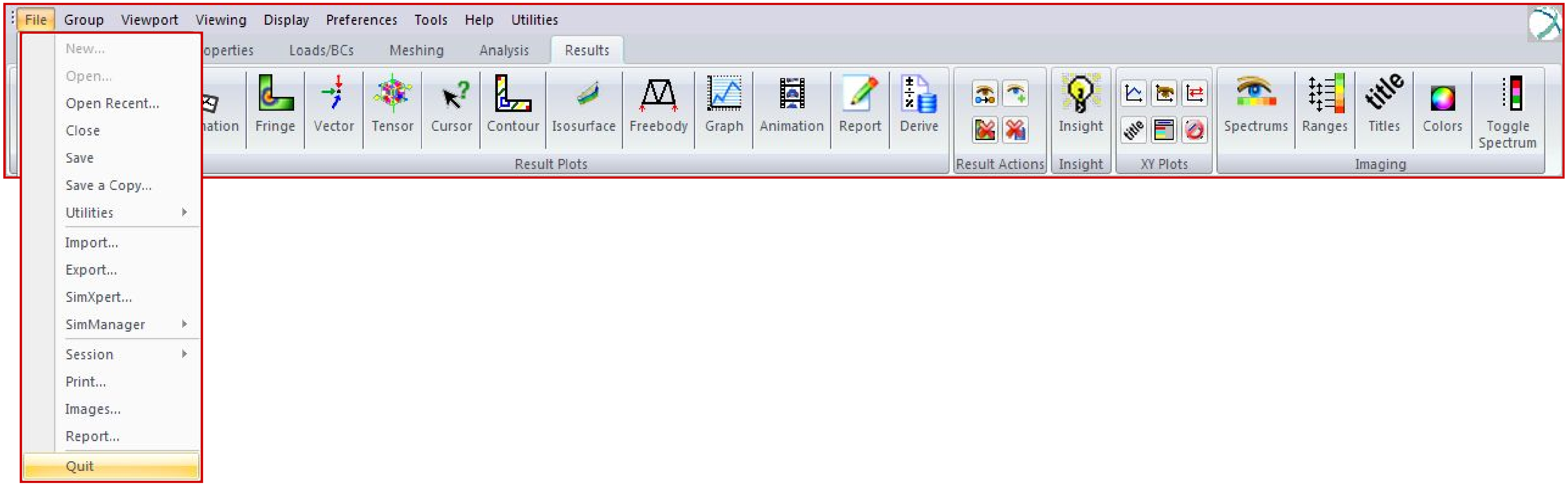
a. Select **contact force, Normal** under *Select Fringe Result*.

b. Click **Apply**.

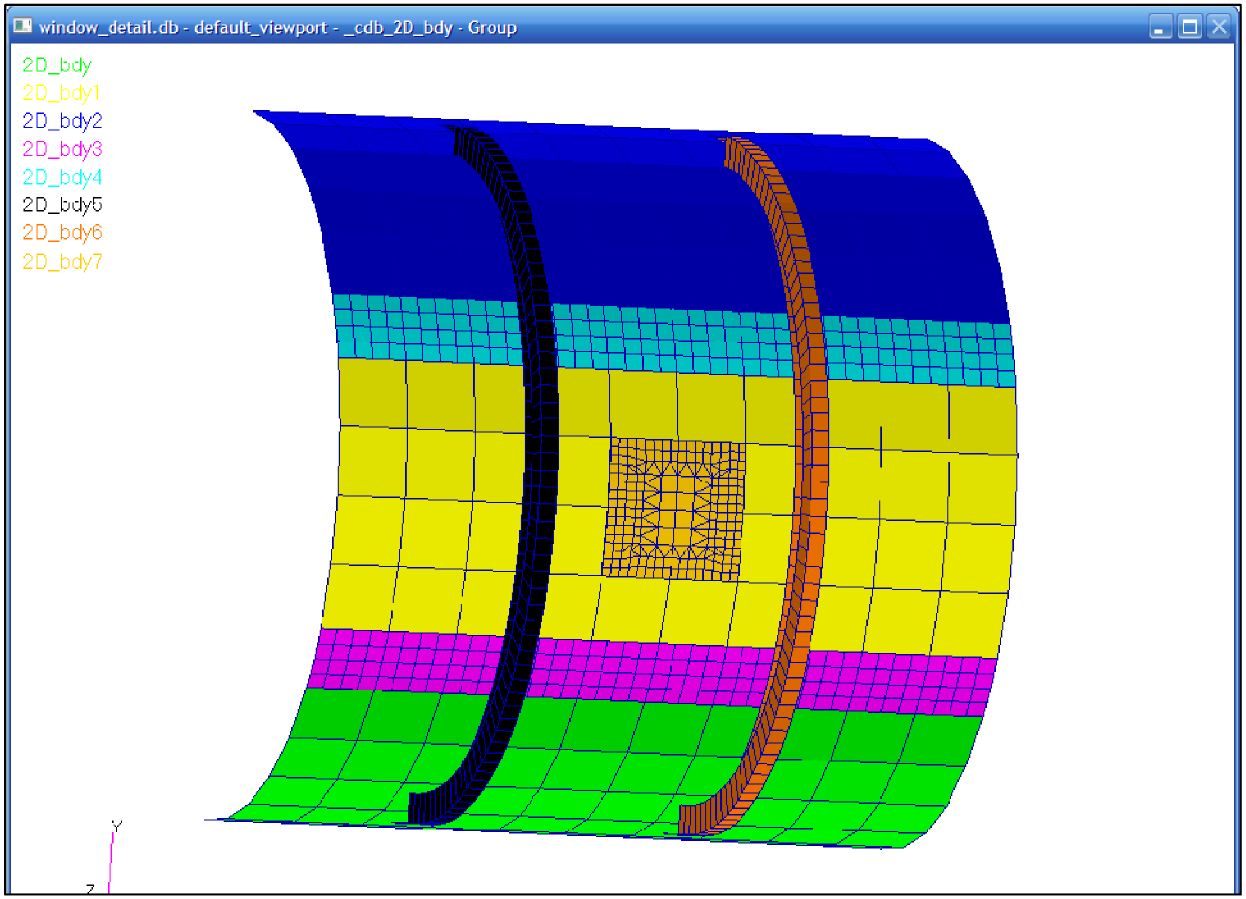




Step 17. Quit Patran



WORKSHOP 5
CONTACT PAIRS



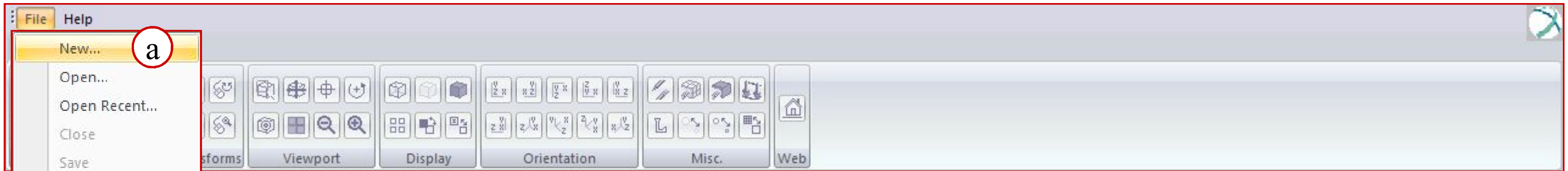
- **Objective**
  - Include a fine mesh locally in a coarser-meshed model.
  - Use Patran's Automatic Contact Body and Body Pair detection to detect and create multiple contact bodies.
  - Use the Distance Tolerance (ERROR) to achieve glued contact between bodies with gaps.

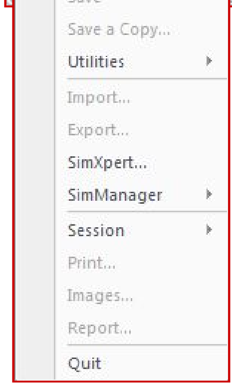
- **Files Required**
  - wingskin_fe.bdf

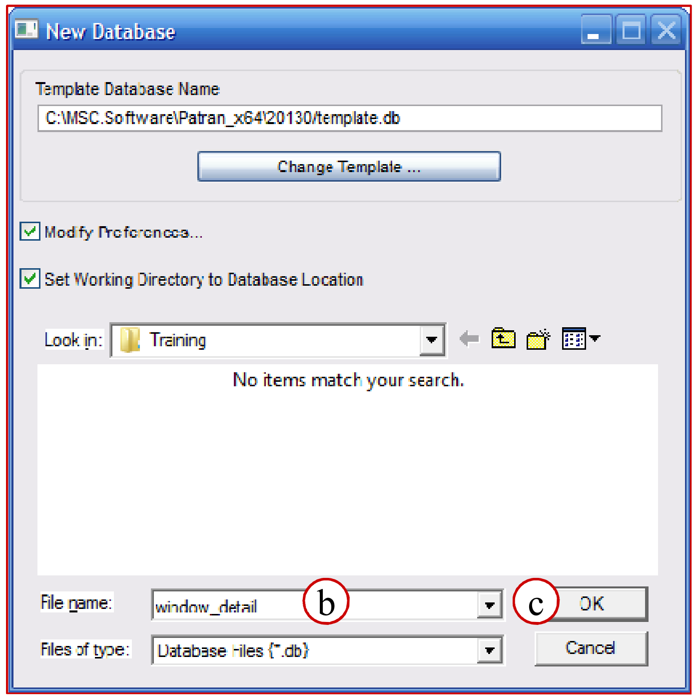
- **Software Version**
  - MSC Nastran 2013.1
  - Patran 2013

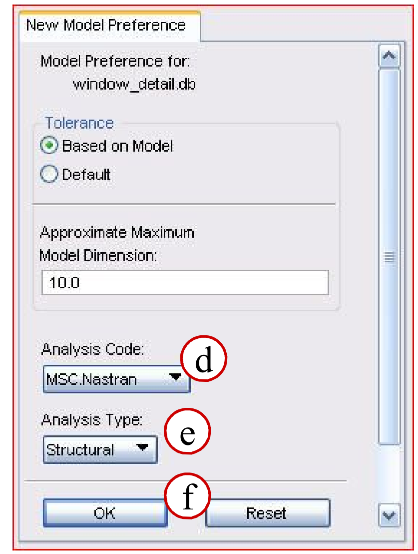
- **Suggested Steps:**
  1. Create a new Patran database.
  2. Import the MSC Nastran input file, **wingskin_fe.bdf**.
  3. Define Contact Bodies using Patran's auto detect tool.
  4. Define Contact Pairs.
     - Contact Pair between the Panels with tolerance of **0.003** and glued contact.
     - Contact Pair between the Rib and panels with tolerance of **0.02** and glued contact.
  5. Modify the Load Case
  6. Verify the Boundary Conditions.
  7. Set up an implicit Nonlinear analysis (SOL400) with contact results. Run the analysis.
  8. Attach the OP2 results.
  9. Review the deformed shape.
  10. Verify contact status.

Step 1. Create a New Database





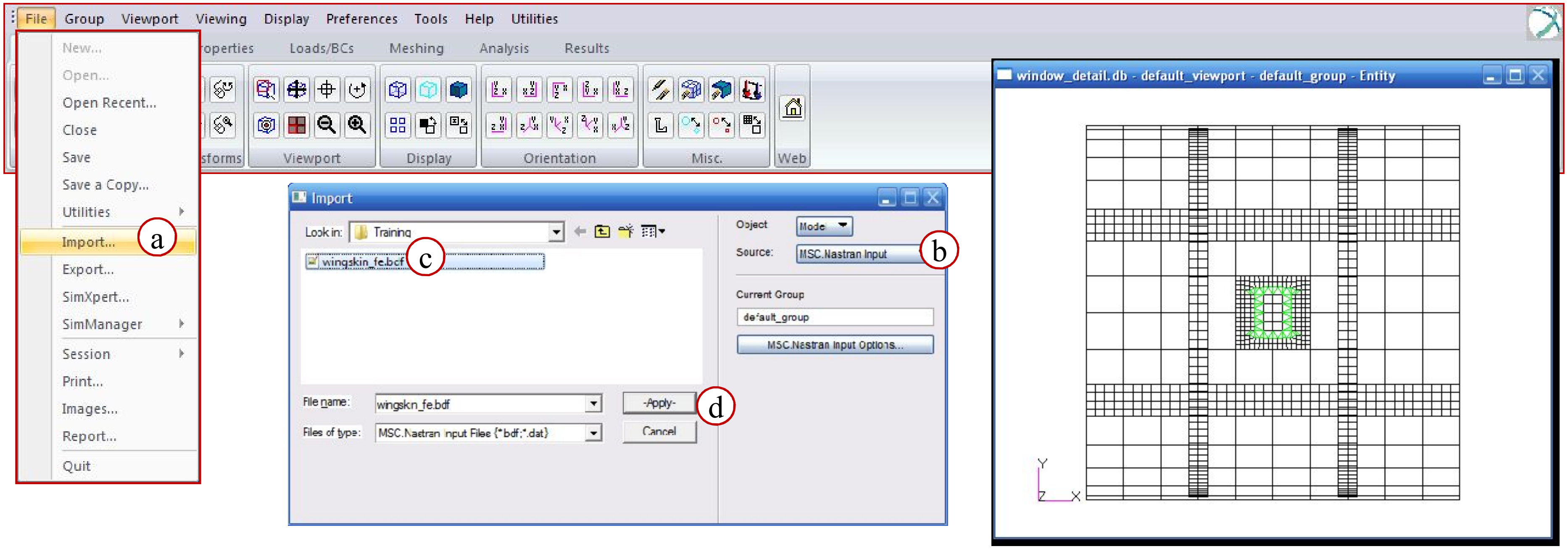




Create a new database. Name it window_detail.db

a. Pull down **File** > **New**.
b. Enter **window_detail** as the *File name*.
c. Click **OK**.
d. Pull down **MSC.Nastran** as the *Analysis Code*.
e. Pull down **Structural** as *Analysis Type*.
f. Click **OK**.

Step 2. Import the Model



Import the MSC Nastran Input file, wingskin_fe.bdf.

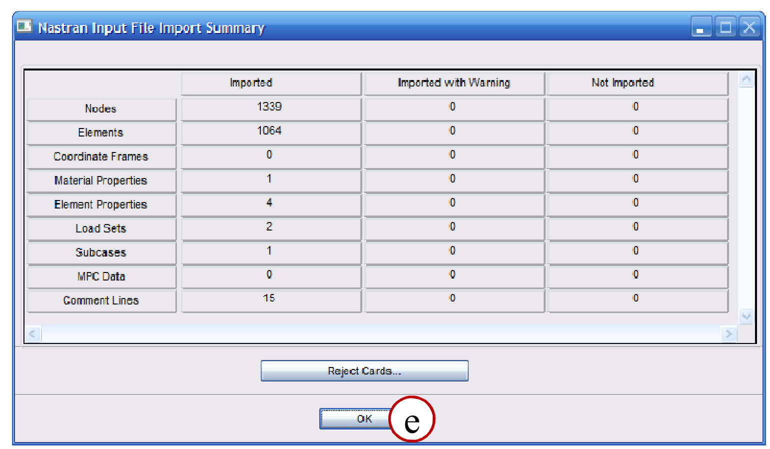
a. Pull down **File** > **Import.**

b. Pull down **MSC.Nastran Input** for *Source*.

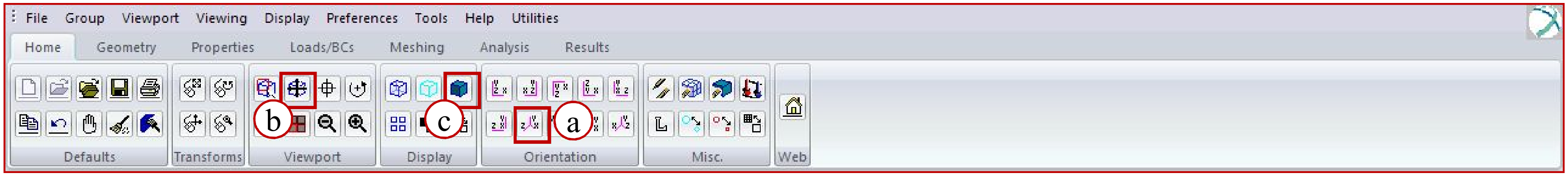
c. Select **wingskin_fe.bdf**

d. Click **Apply.**

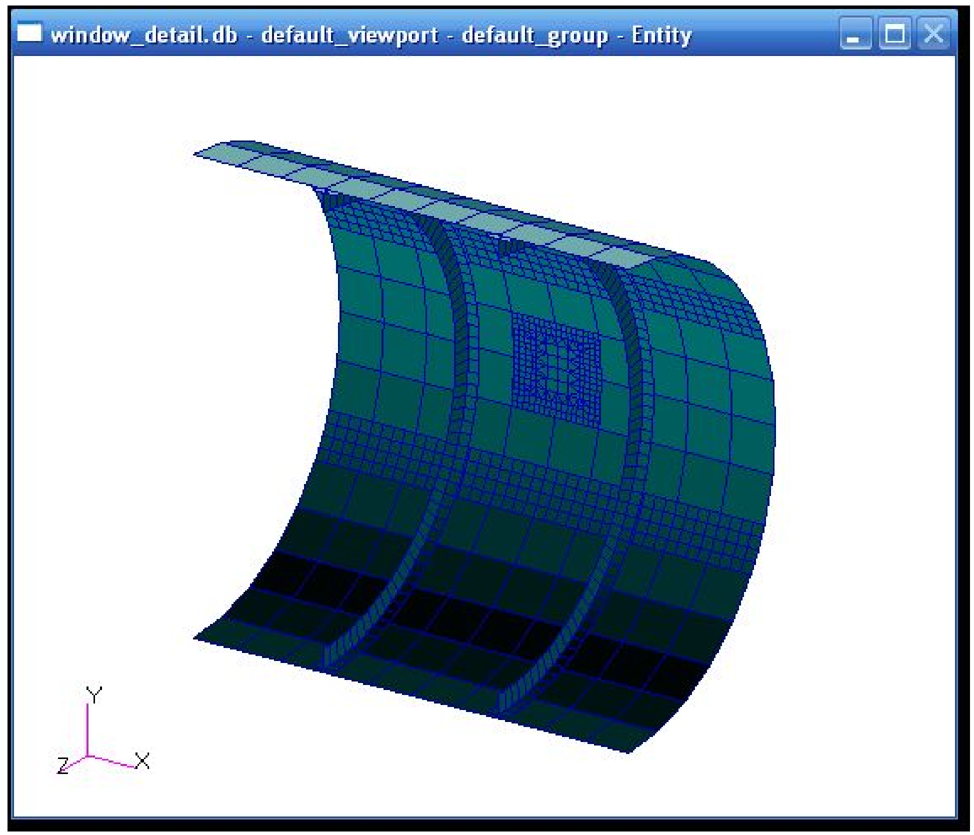
e. Review the *Import Summary* and click **OK.**



Step 2. Import the Model (Cont.)



a. Click **Iso 1 View** in the *Orientation* group.
b. Click **Fit View** in the *Viewport* group.
c. Click **Smooth Shaded** in the *Display* group.



NOTE: Notice that here we have an airframe model that has a very fine mesh in the window and some of its panels. Other panels have a comparatively coarser mesh.

MSC Software

Step 3. Define Contact Bodies

Define Contact bodies using the Create Contact Body/Pairs tool.

a. Pull down *Tools* > *Modeling* > **Contact Bodies/Pairs.**

b. Pull down *Method* to **Connectivity.**

c. Check **2D** for Elements.

d. Click **Apply.**

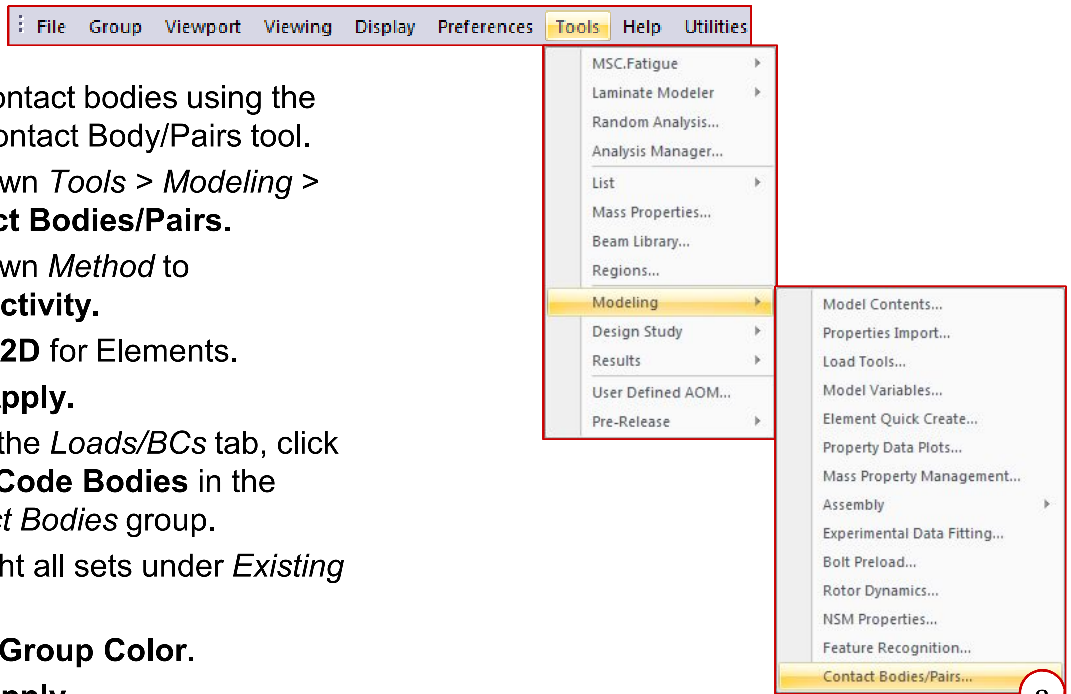
e. Under the *Loads/BCs* tab, click **Color Code Bodies** in the *Contact Bodies* group.

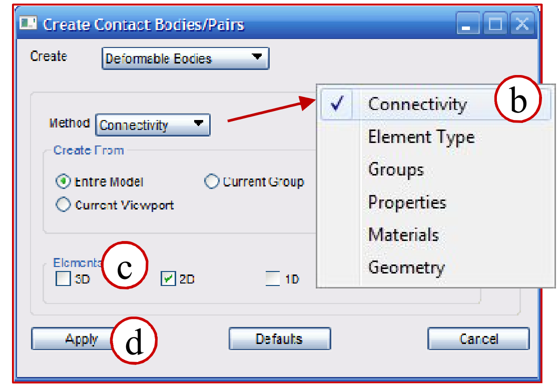
f. Highlight all sets under *Existing Sets.*

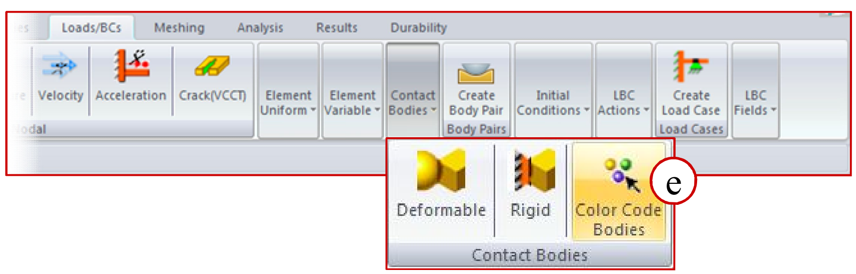
g. Select **Group Color.**

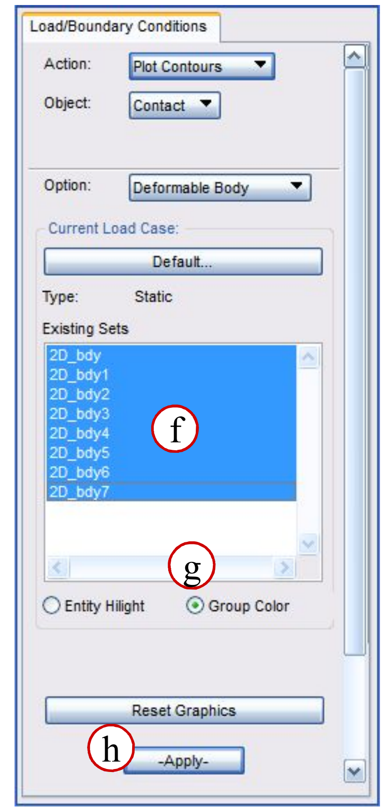
h. Click **Apply.**

All Contact bodies will be detected and generated with one click. As an alternative to the Connectivity Method, you can use Properties and Contact bodies will be generated based on their associated Properties and named accordingly.

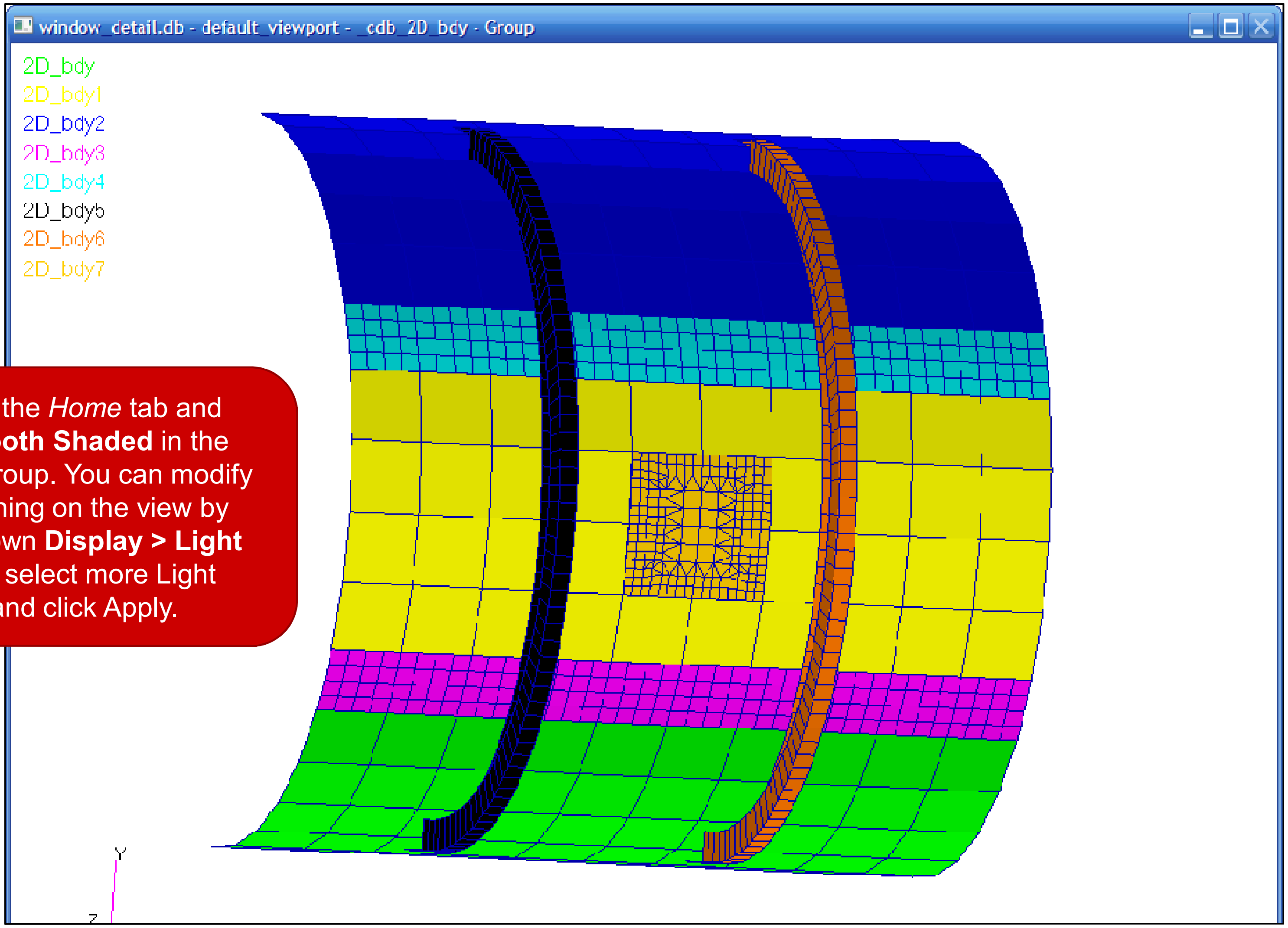








Contact Bodies Defined

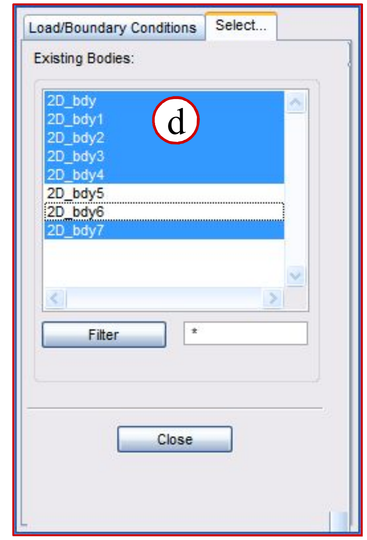


Return to the *Home* tab and click **Smooth Shaded** in the *Display* group. You can modify the lightening on the view by Pulling down **Display > Light Sources,** select more Light Sources and click Apply.

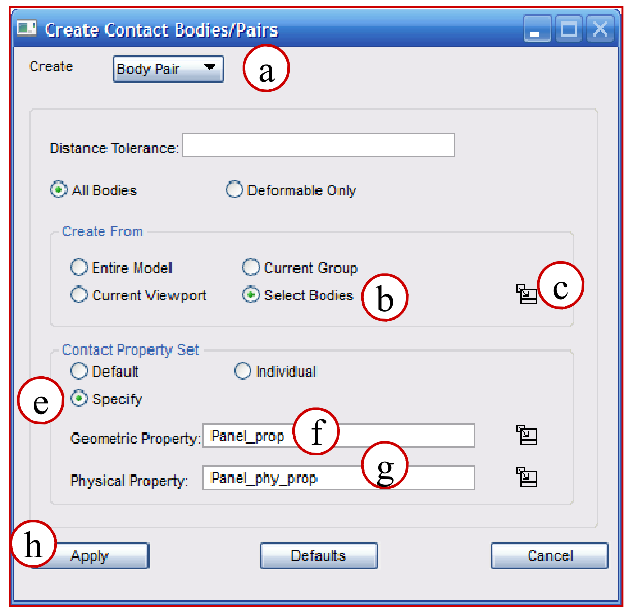
Step 4. Define Contact Pairs (Cont.)

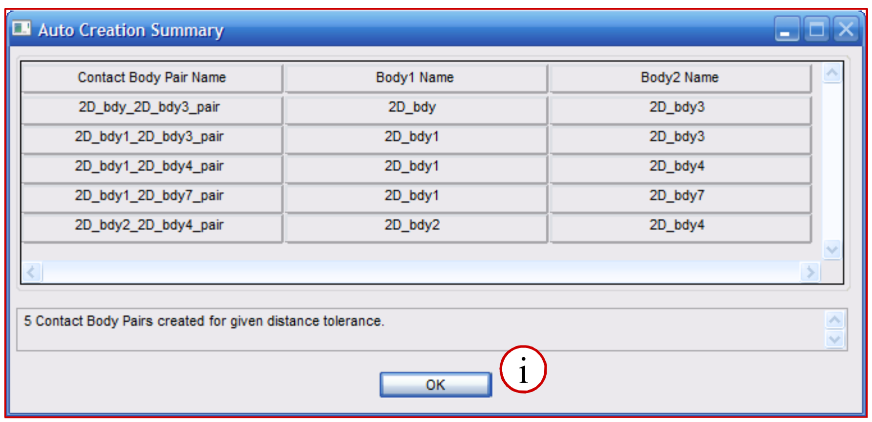
Define Contact Pairs using Create Contact Body/Pair tool.

a. Pull down *Create* to **Body Pair.**
b. Select **Select Bodies** on the *Create Contact Bodies/Pairs* form.
c. Click the *Select Existing Bodies* **icon.**
d. Highlight **all Sets** under *Existing Bodies, except 2D_bdy5 and 2D_bdy6* (the ribs).
e. Select **Specify** in the *Contact Properties Set* group
f. Enter **Panel_prop** for *Geometric Property.*
g. Enter **Panel_phy_prop** for*Physical Property.*
h. Click **Apply.**
i. Click **OK** to close the *Auto Creation Summary* form.



If you closed the Create Contact Body/pair tool earlier, you can reopen it by pulling down Tools > Modeling > Contact Bodies/Pairs…

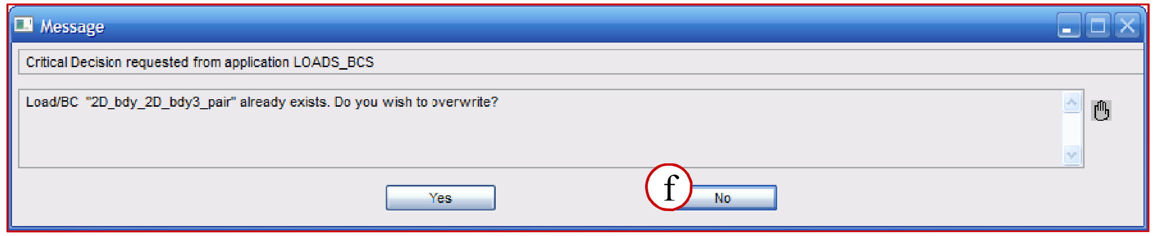


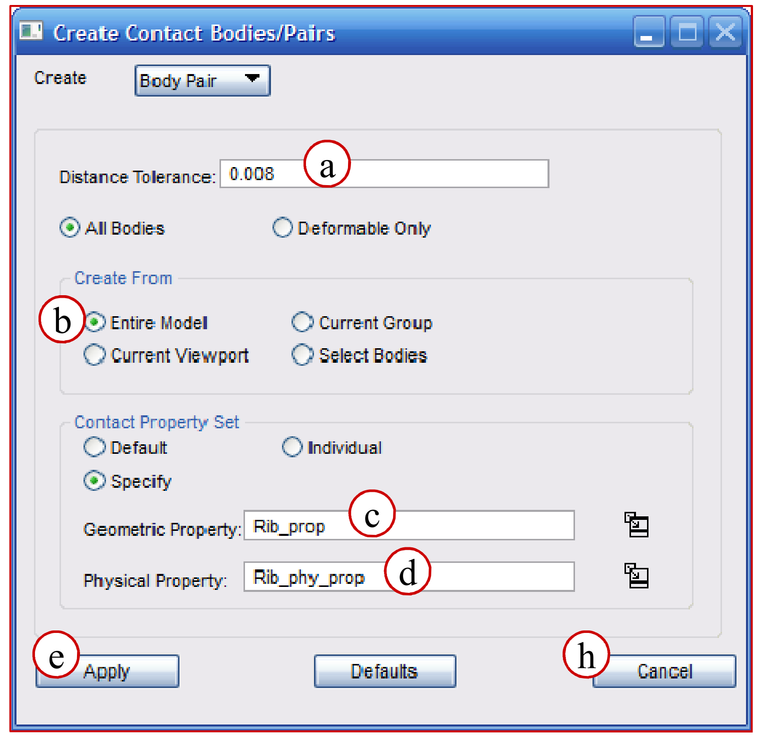


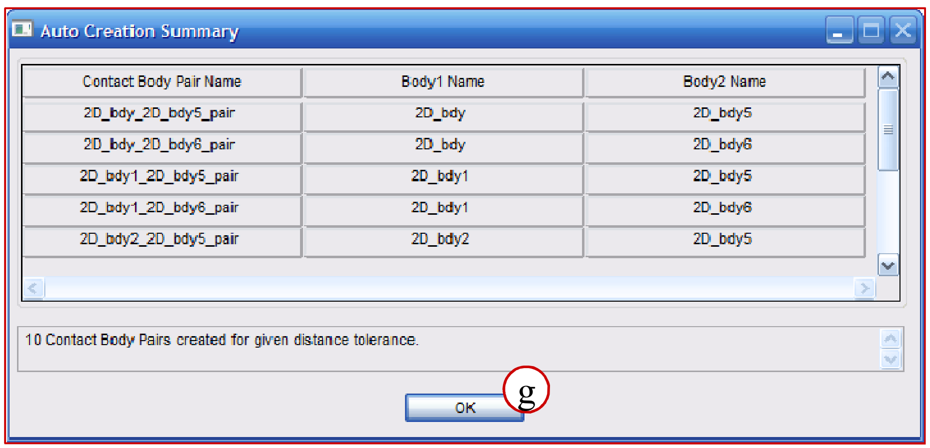
Step 4. Define Contact Pairs (Cont.)

Define the contact pair between the rib and all the panels.

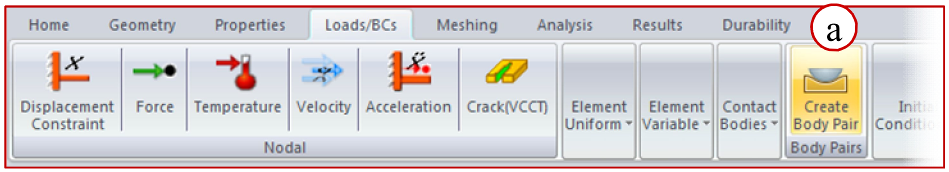
a. Enter **0.008** for *Distance Tolerance.* This is a gap between the rib and panels.
b. Select **Entire Model** in *Create From*.
c. Enter **Rib_prop** for *Geometric Property.*
d. Enter **Rib_phy_prop** for *Physical Property.*
e. Click **Apply.**
f. Click **No** on all the Messages for duplicate pairs.
g. Click **OK.**
h. Click **Cancel**.





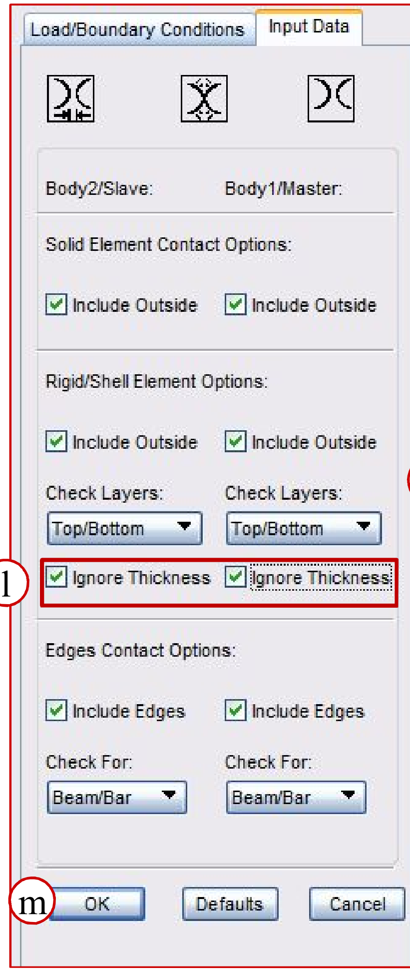


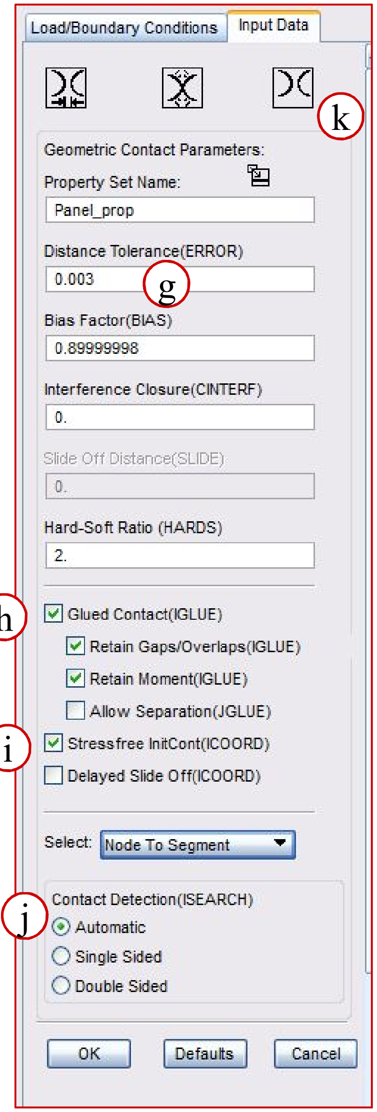
Step 4. Define Contact Pairs (Cont.)

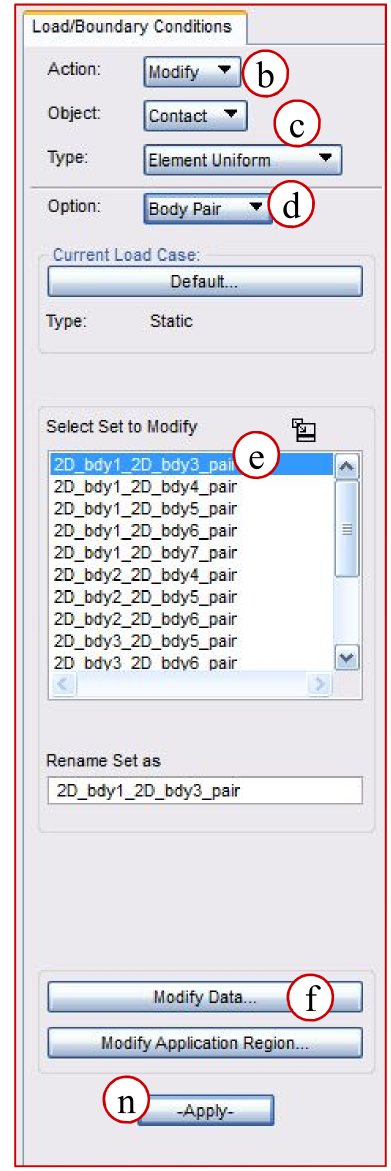


Modify the contact pair parameters.

a. Click *Create Body Pair*
b. Pull down Action to **Modify.**
c. Pull down Object to **Contact.**
d. Pull down Option to **Body Pair.**
e. Highlight **2D_bdy1_2D_bd3_pair.**
f. Click **Modify Data…**
g. Enter **0.003** for *Distance Tolerance(ERROR)*
h. Check **Glued Contact, Retain Gaps/Overlaps,** and **Retain Moment.**
i. Check **Stress Free InitCont.**
j. Select **Automatic** under *Contact Detection.*
k. Click **Contact Option.**
l. Check **Ignore Thickness** for both Bodies.
m. Click **OK.**
n. Click **Apply.**



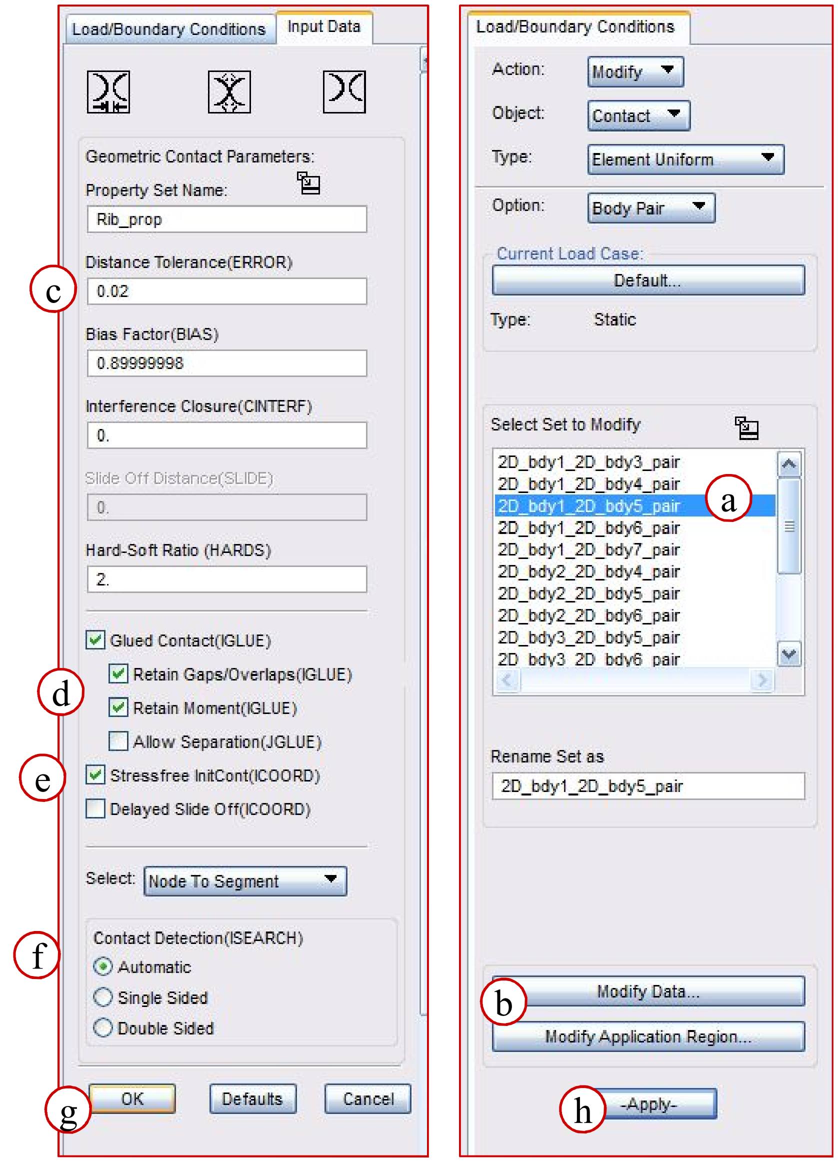




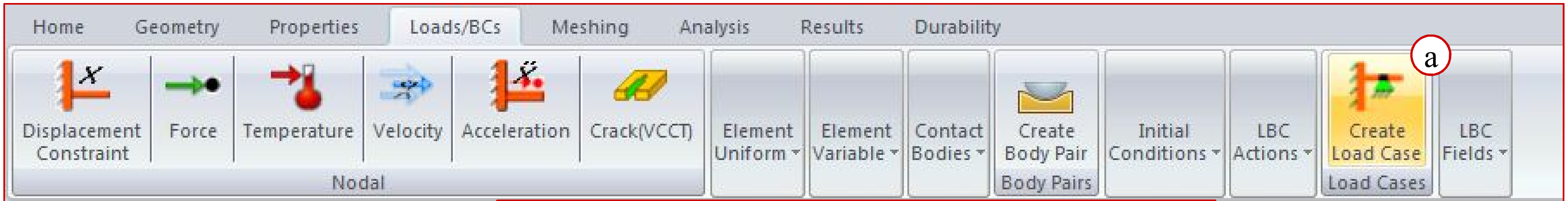
Step 4. Define Contact Pairs (Cont.)

Modify the contact pair parameters for Ribs

a. Highlight **2D_bdy1_2D_bdy5_pair**
b. Click **Modify Data…**
c. Enter **0.02** for *Distance Tolerance*
d. Check **Glued Contact, Retain Gaps/Overlaps,** and **Retain Moment.**
e. Check **Stress Free InitCont**
f. Select **Automatic** under *Contact Detection*
g. Click **OK**
h. Click **Apply**



Step 5. Modify the Load Case



Modify the default load case to add Displacement and Load applied to the current loadcase. (They have been previously defined in the input deck imported in this workshop)

a. On the *Loads/BCs* tab, click **Create Load Case**.

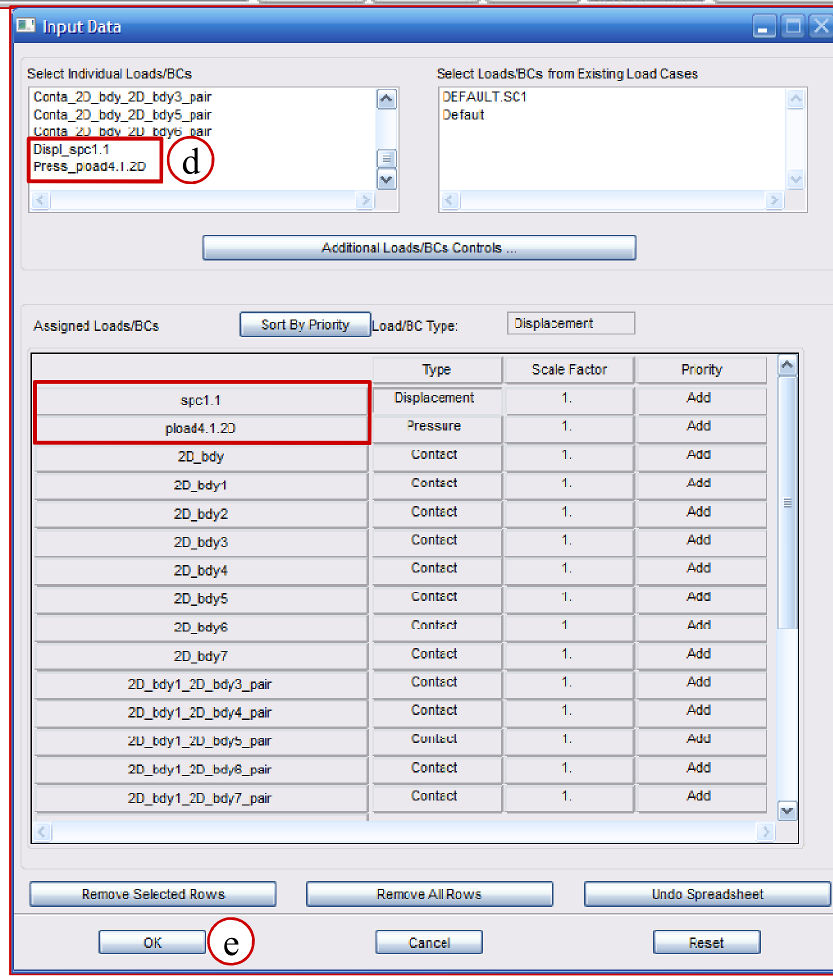
b. Pull down *Action* > **Modify**.

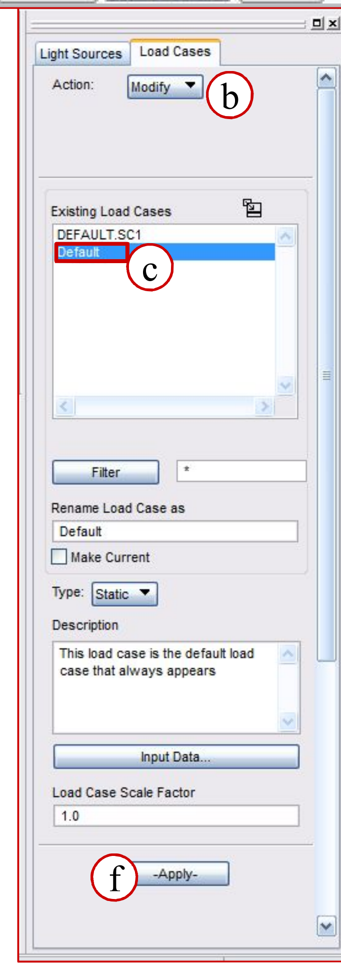
c. Select the **Default** load case.

d. Click on the **Displ_spc1.1** and **Press_pload4.1.2D** Loads/BCs. They should appear on the list of Loads/BCs below.

e. Click **OK**.

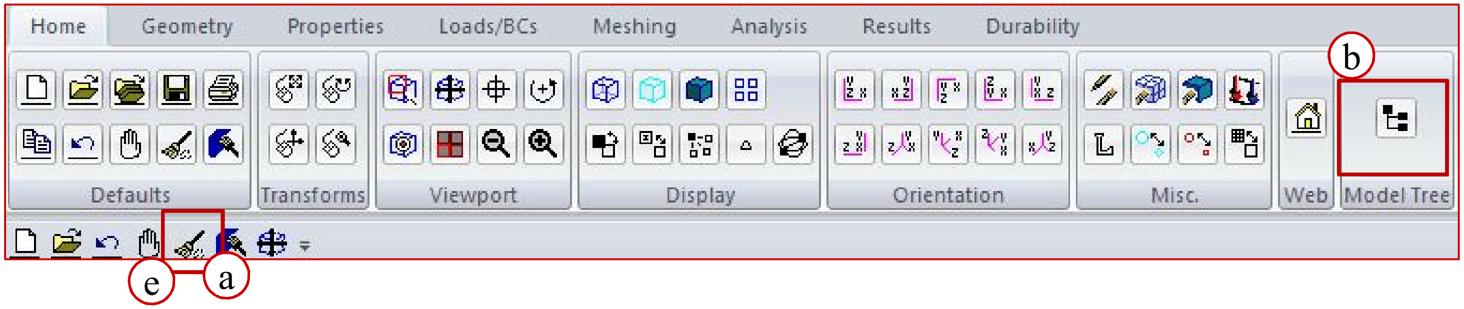
f. Click **Apply**.





MSC Software

Step 6. View Boundary Conditions



Create a pinned constraint on the axial edges:

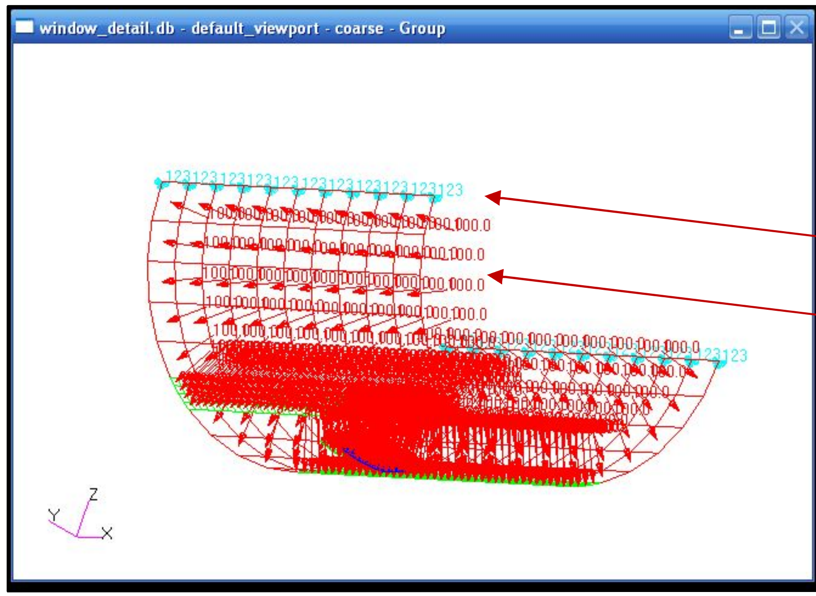
a. Click **Reset Graphics.**

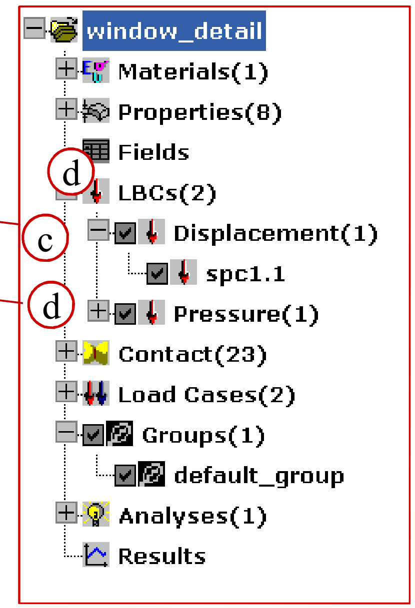
b. Click on **Model Tree** in **Home** tab.

c. Click **Displacement** in Model Tree to view this BC on model.

d. Click **Press_pload4.1.2D.** in Model Tree to view this BC on model.

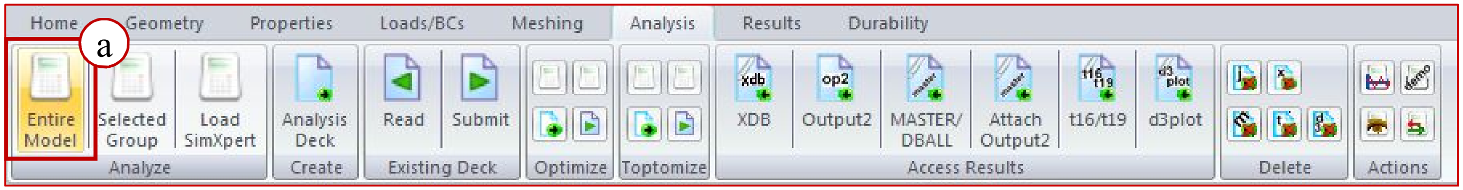
e. Click **Reset Graphics.**





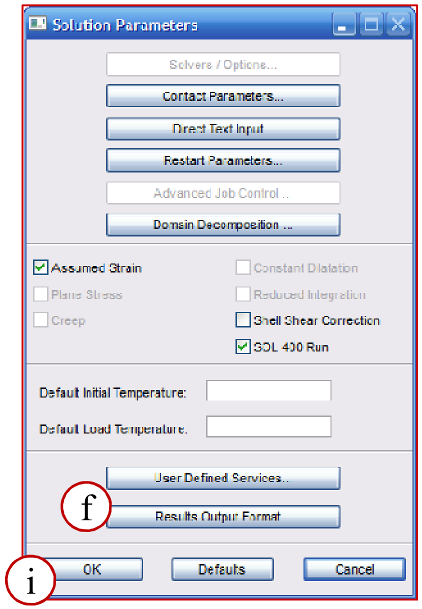
MSC Software

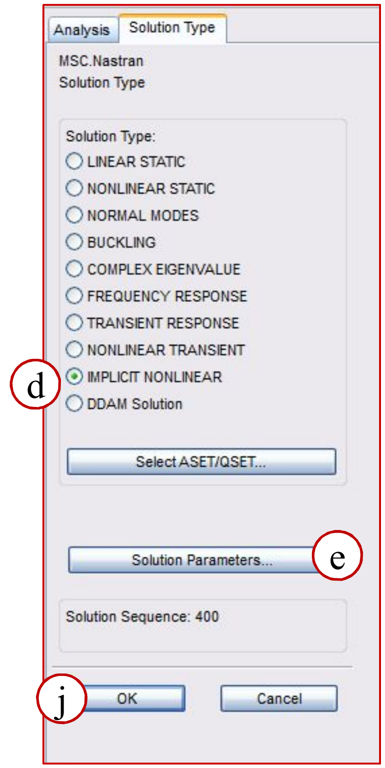
Step 7. Set up Implicit Nonlinear Analysis

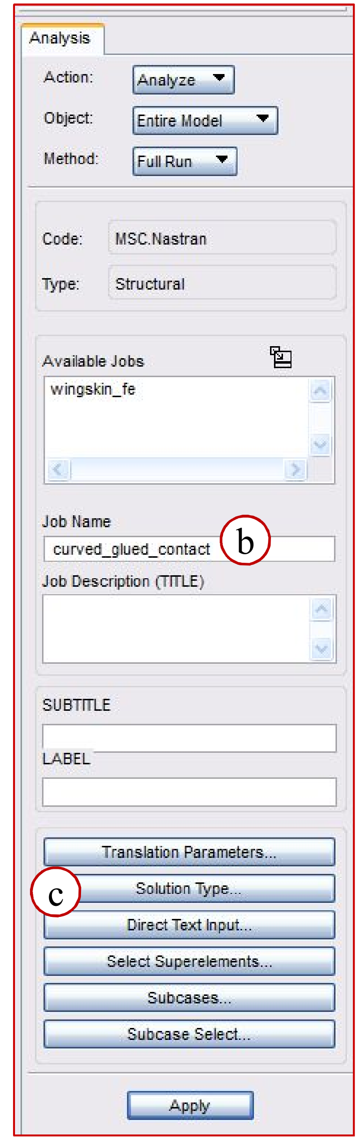


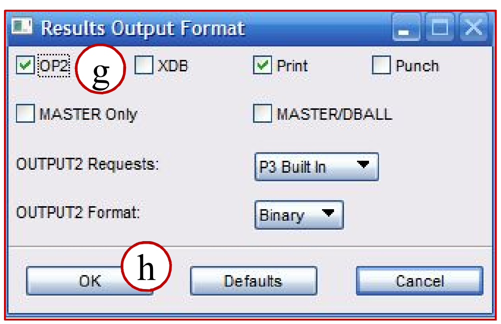
Set up the Nonlinear static analysis.

a. Under the *Analysis* tab, click **Entire Model** in the *Analyze* group.
b. Enter **curved_glued_contact** for *Job Name*.
c. Click **Solution Type**.
d. Select **Implicit Nonlinear** for *Solution Type*.
e. Click **Solution Parameters**.
f. Click **Results Output Format**.
g. Uncheck **XDB** and check **OP2.**
h. Click **OK.**
i. Click **OK.**
j. Click **OK.**





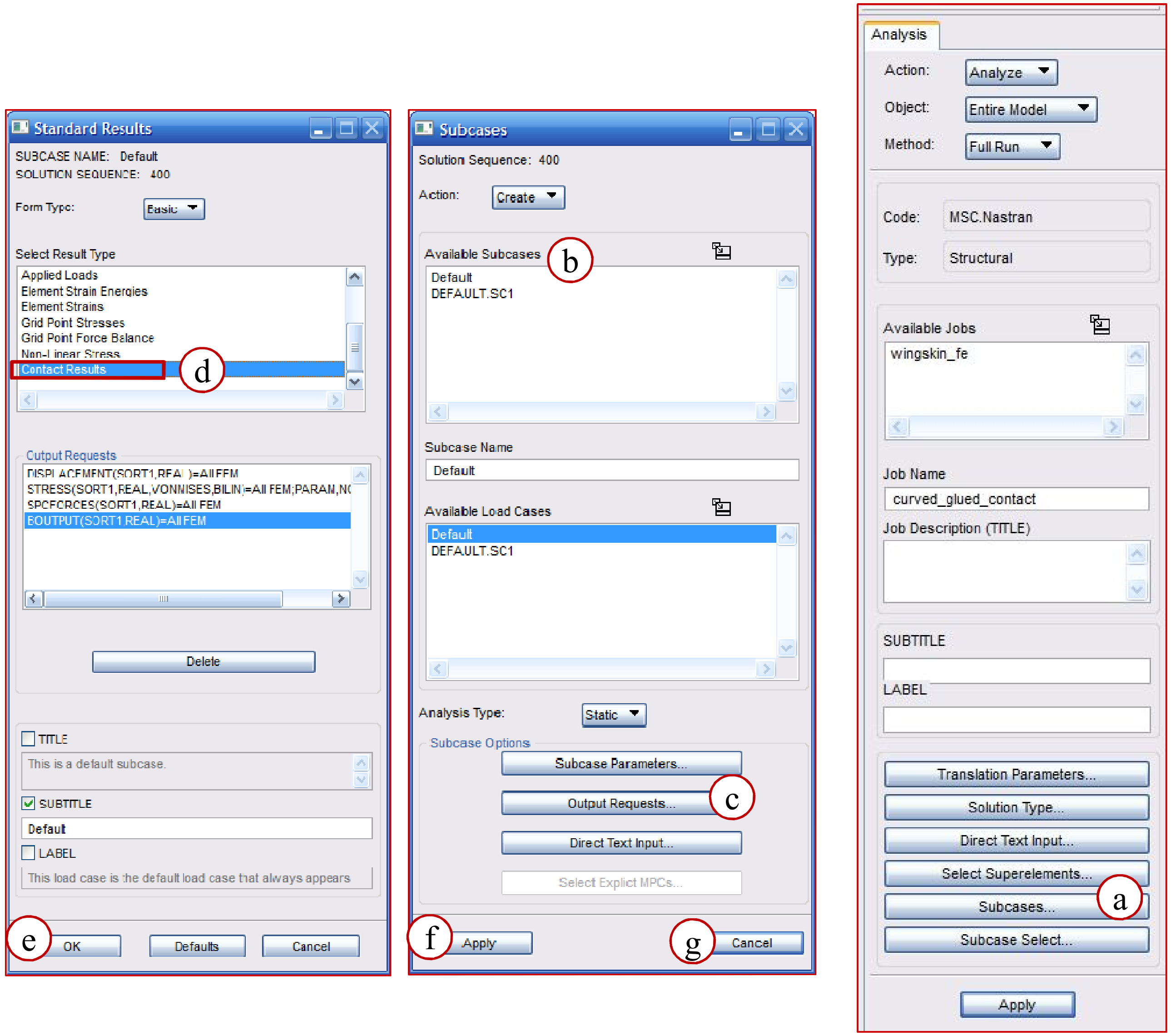




Step 7. Set up Implicit Nonlinear Analysis (Cont.)

Request Contact output.

a. Click **Subcases**.
b. Highlight **Default** under *Available Subcases*.
c. Click **Output Requests**.
d. Highlight **Contact Results** under *Select Result Type*.
e. Click **OK**.
f. Click **Apply** on the *Subcases* form.
g. Click **Cancel**.



MSC Software

Step 7. Set up Implicit Nonlinear Analysis (Cont.)

Run the analysis:

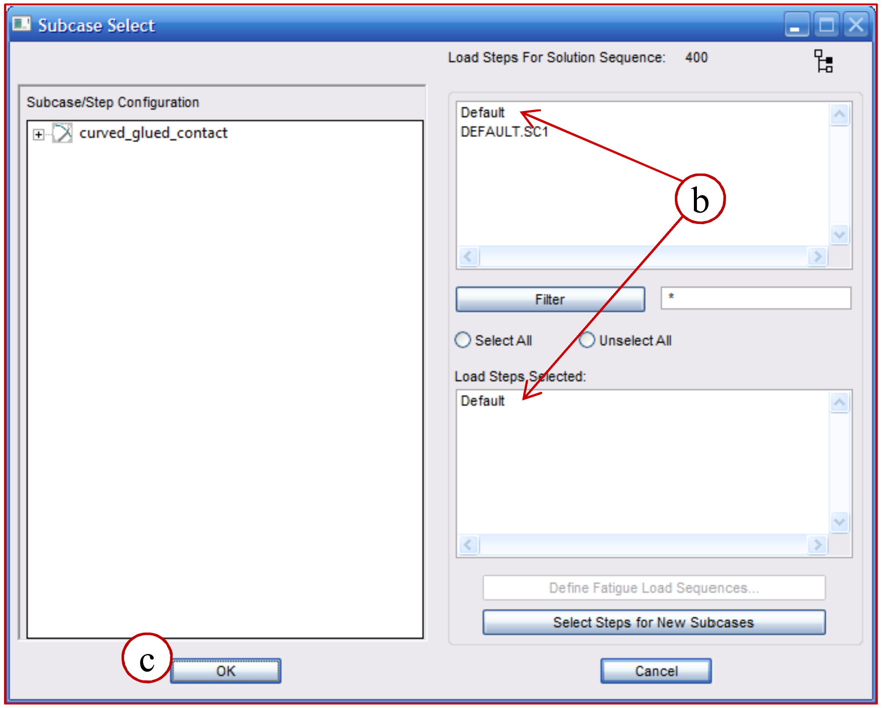
a. Click **Subcase Select**.

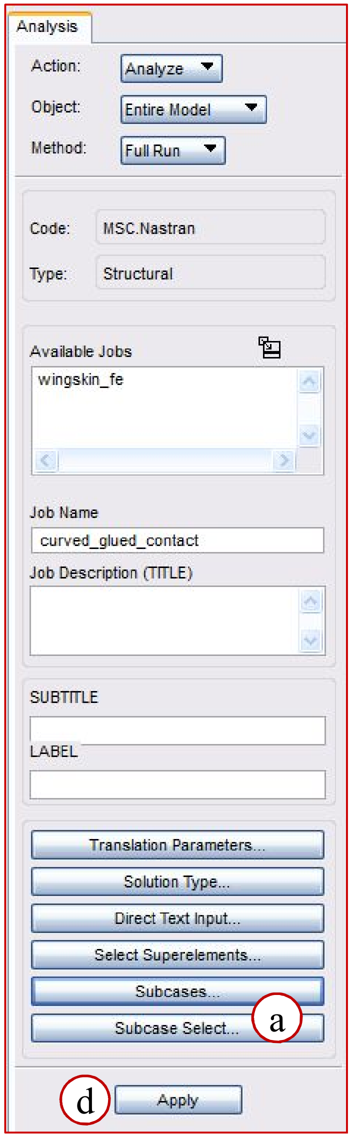
b. Highlight **Default** in the top pane. Ensure *Default* is the only subcase present under *Load Steps Selected*.

c. Click **OK**.

d. Click **Apply**.

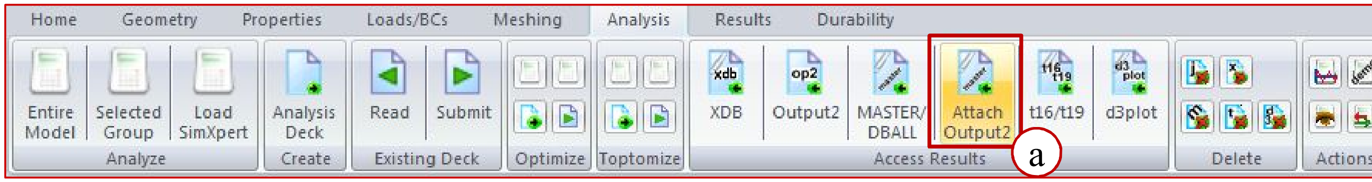
The job is written to *curved_glue_contact.bdf* and the analysis is run.





MSC Software

Step 8. Attach the OP2 Results



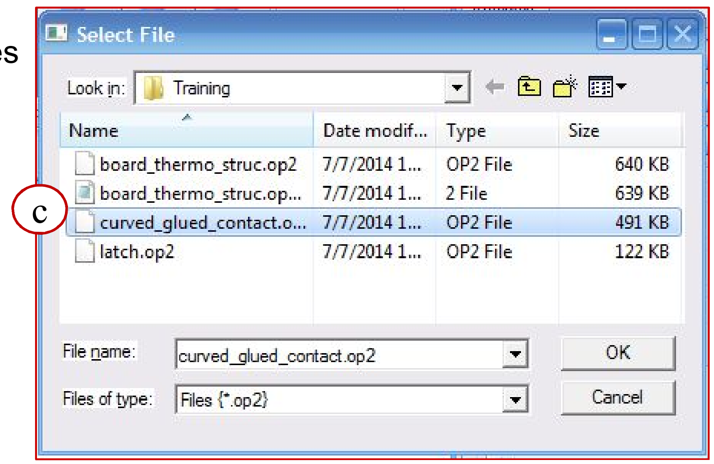
Attach the OP2 results:

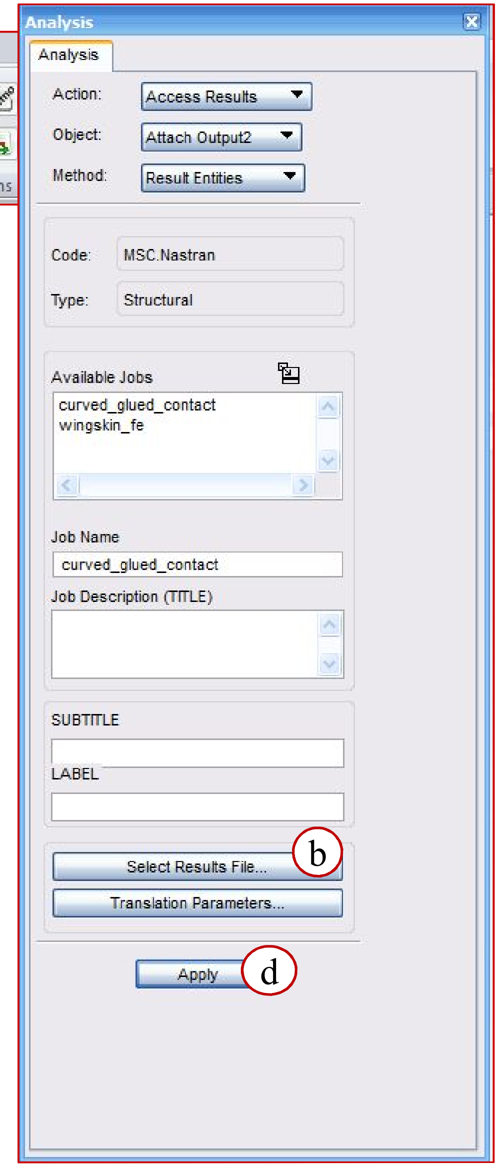
a. Click **AttachOutput2** in the *Access Results* group.

b. Click **Select Results File**

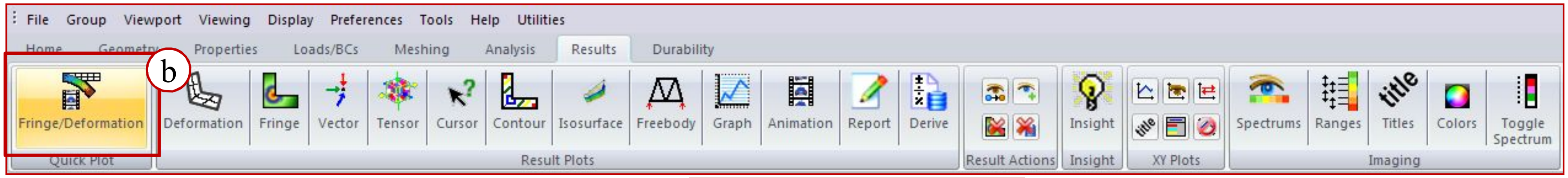
c. Select **curved_glued_contact.op2** from the list of *Available Jobs.*

d. Click **Apply**. Patran will look for the results file with a name that matches the *Job Name*.





Step 9. Post Process with Patran



Review the deformation results:

a. Click **Reset Graphics.**

b. Under the *Results* tab, click **Fringe/Deformation** in the *Quick Plot* group.

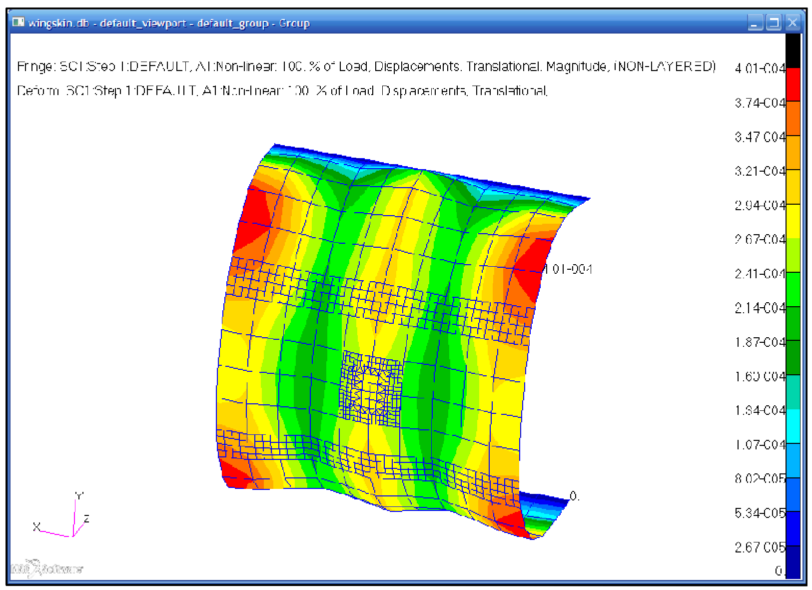
c. Highlight the second subcase **SC1:DEFAULT, A1 Static Subcase**.

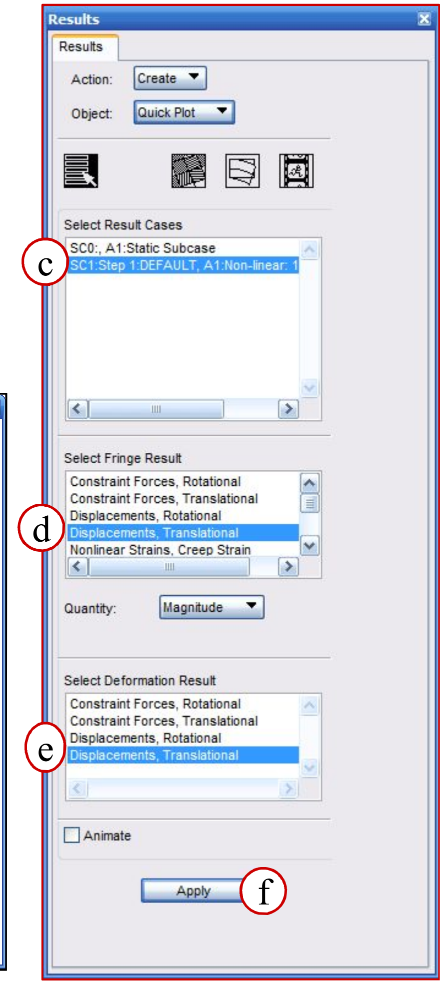
d. Highlight **Displacements, Translational** for *Fringe Result*.

e. Highlight **Displacements, Translational** for *Deformation Result*.

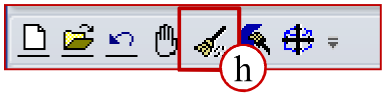
f. Click **Apply**.

Check that the results are reasonable. Note that the displacement contours flow across the glued boundaries with different mesh sizes.





Step 10. Verify Contact Status



Verify the contact status for each of the contact bodies

a. Pull down **Create/Marker/Scalar** for *Action*, *Object*, and *Method*, respectively.

b. Highlight the first Result Case **SC0:, A1:Static Subcase** which contains the contact results

c. Select **Contact Status** under *Select Scalar Result*.

d. Click **Display Attributes**.

e. Enter **0.03** for *Scale Factor*.

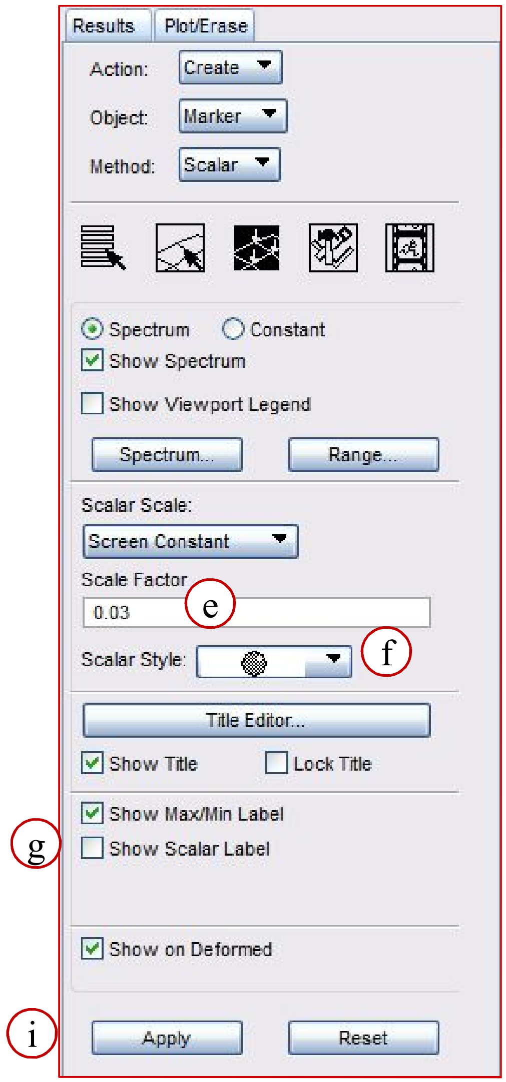
f. Pull down the **Scalar Style** to your preference.

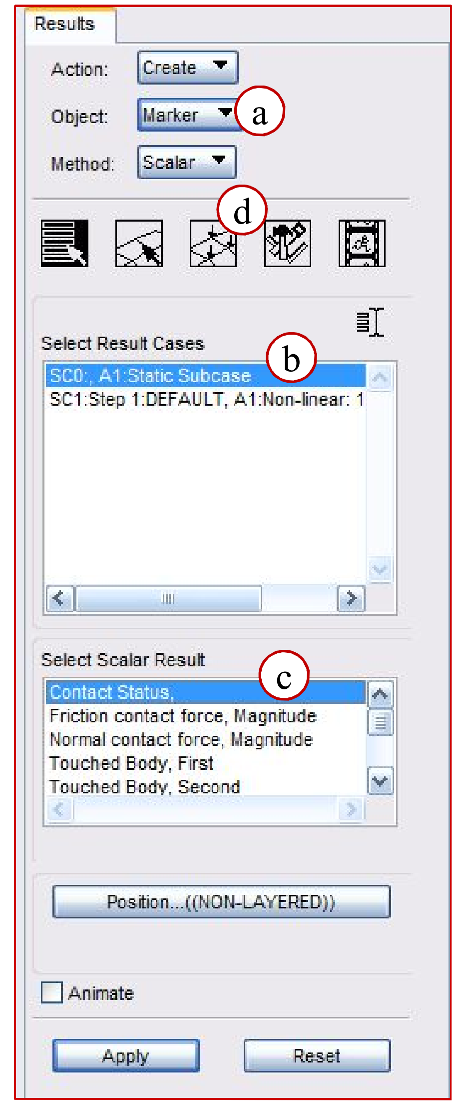
g. Uncheck **Show Scalar Label.**

h. Click **Reset Graphics.**

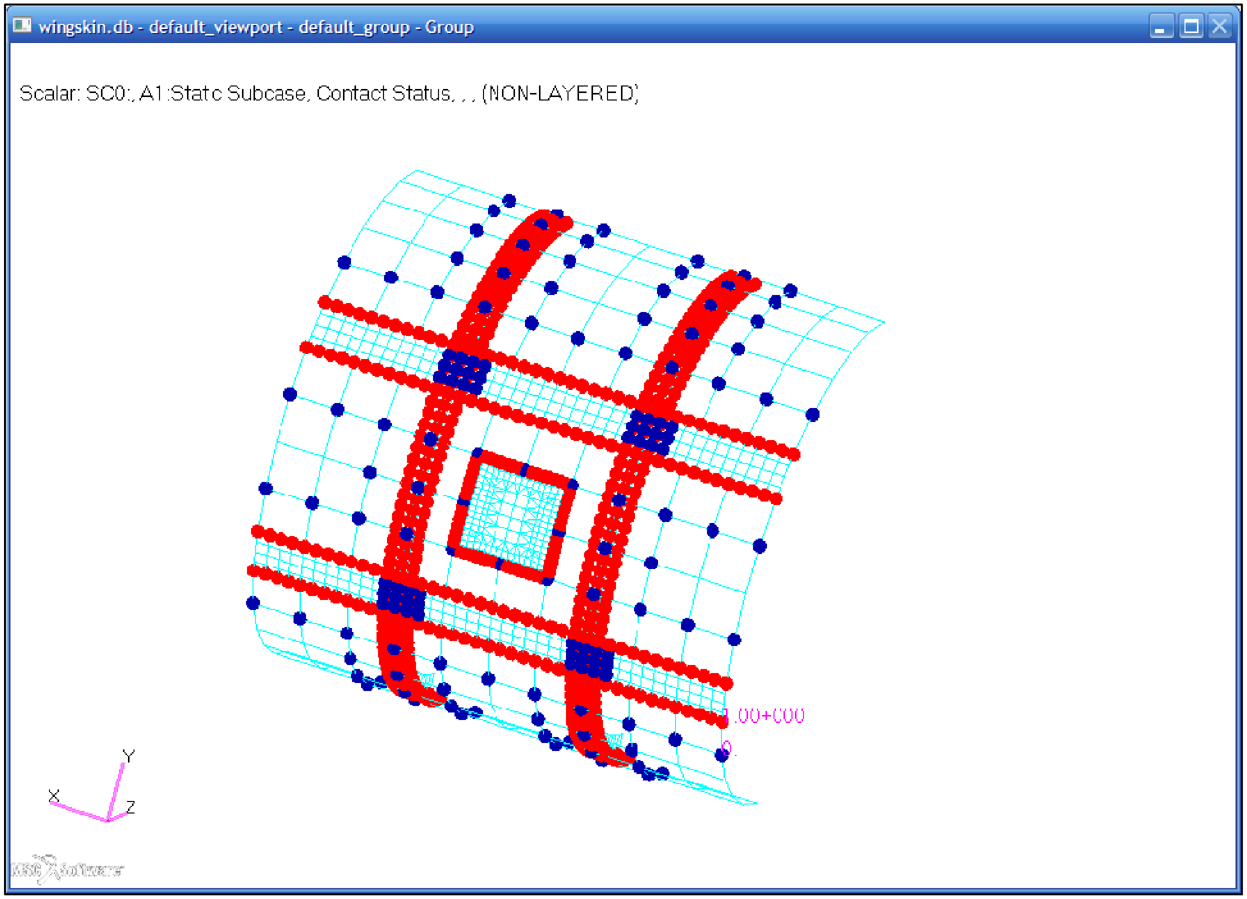
i. Select **Apply.**

The contact status plot for the fine mesh is displayed on the next page.



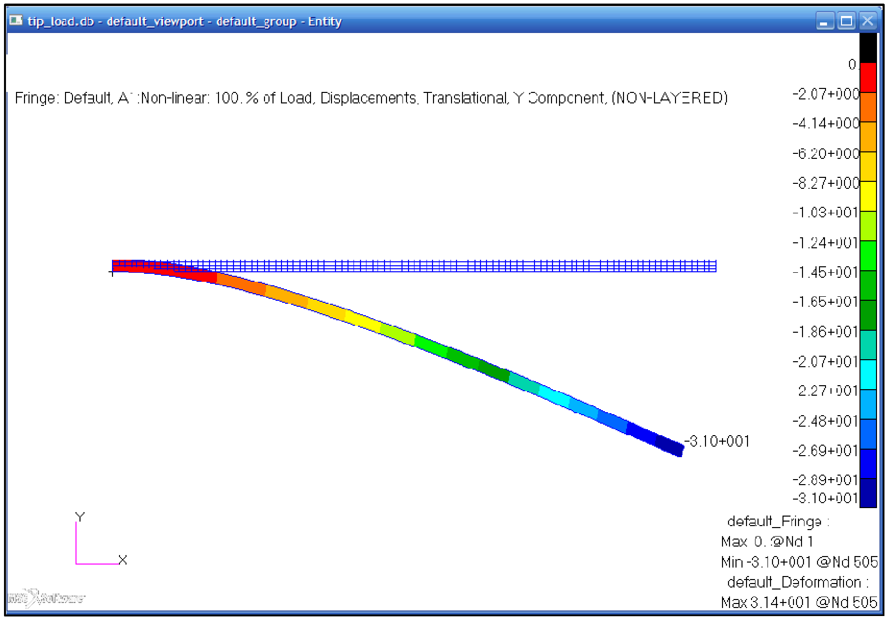


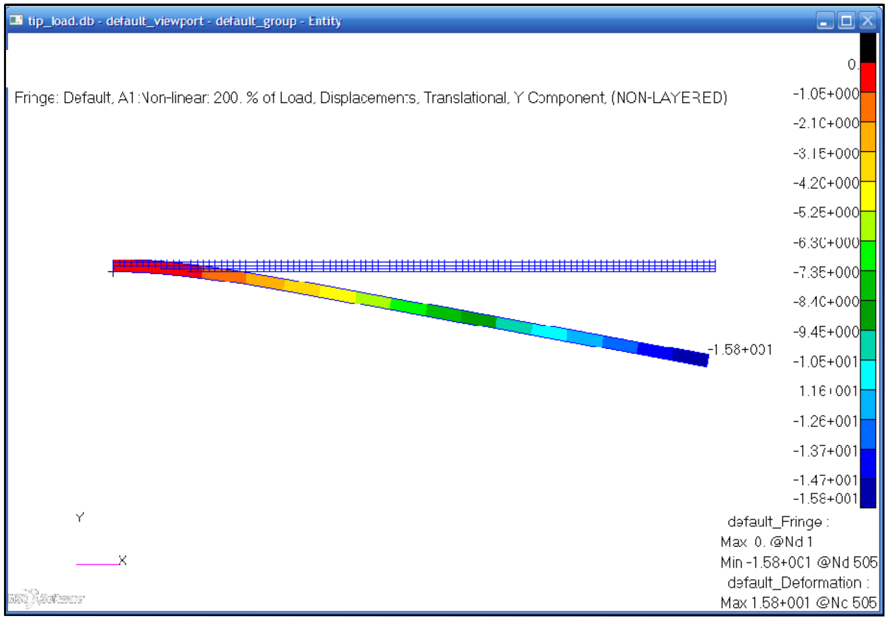
Step 10. Verify Contact Status (Cont.)



Patran will place a scalar symbol on every node that is in contact.

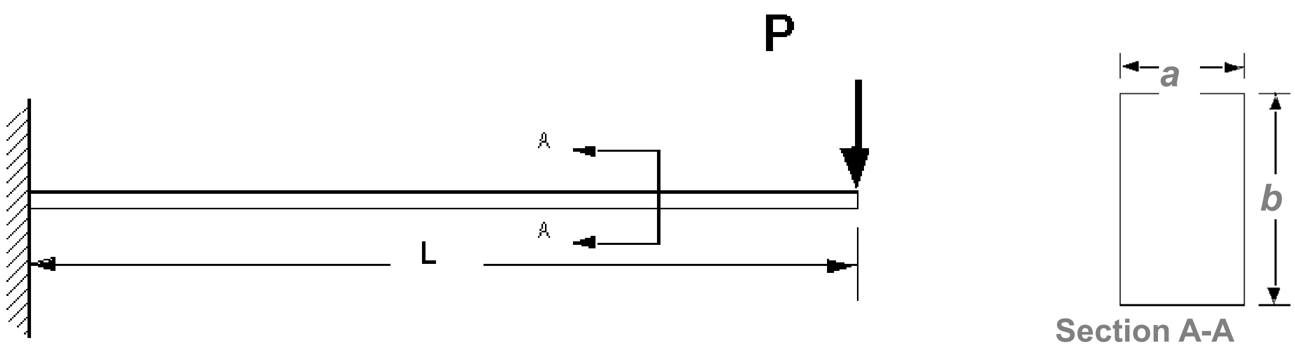
WORKSHOP 6
PLASTIC DEFORMATION





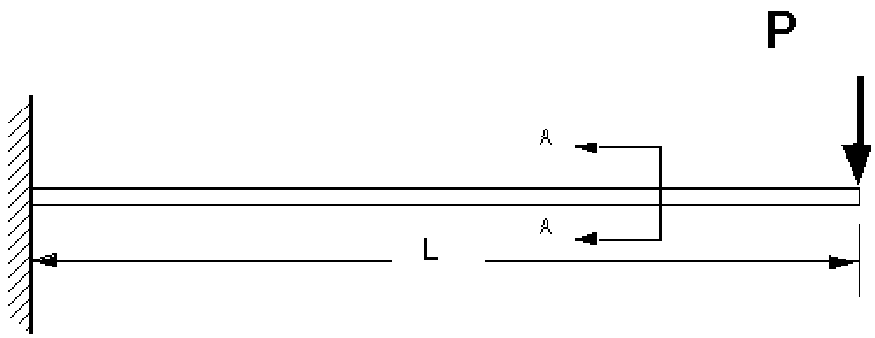
- **Workshop Objectives**
  - Plastic Deformation
- **Software Version**
  - Patran 2013
  - MSC Nastran 2013.1
- **Required Files**
  - Tip_load.db
- **Problem Description**
  - The simulation is to use 3D solid finite elements. The elements are uniformly spaced along the length of the beam (i.e. a mesh 100 elements wide and four elements deep).

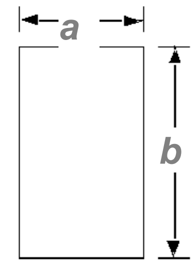
- **Problem Description (Cont.)**
  - A cantilever beam is subjected to a static load. In the previous workshop, the beam is initially analyzed using small deformation theory. However, after reviewing the results, it becomes apparent that small deformation theory is not appropriate for this problem. A large deformation analysis using plastic properties is performed.



(Data in next page)

- **Problem Description (Cont.)**



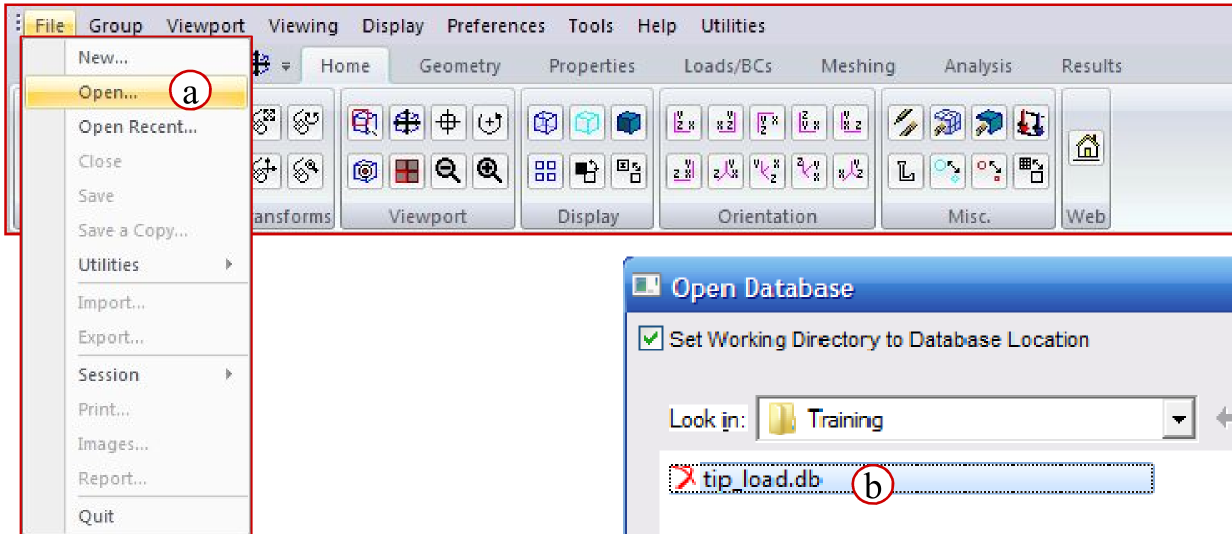


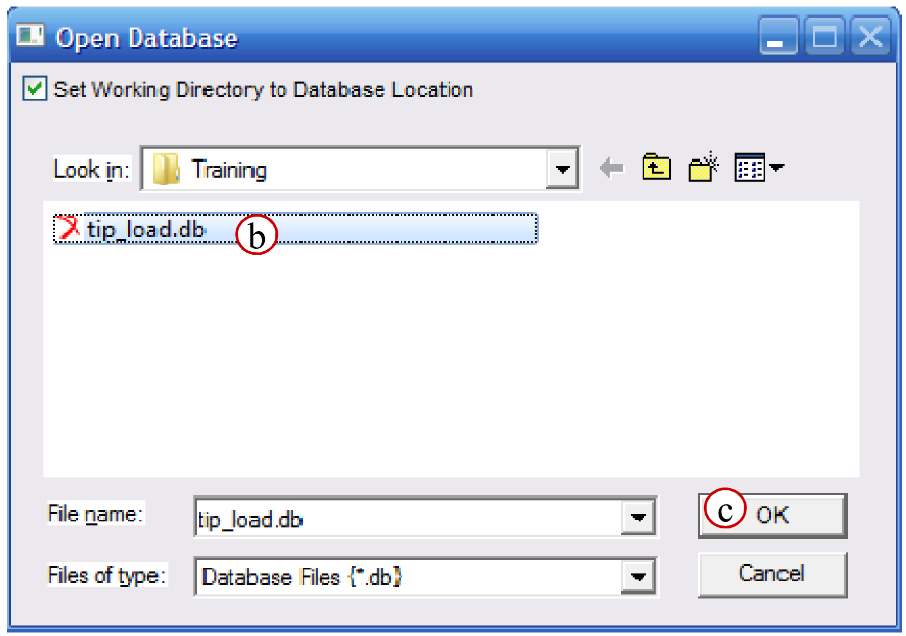
Section A-A

Length, L	100.0 in	2.54 m
a	1.0 in	25.4 mm
b	2.0 in	50.8 mm
Young's Modulus	30.0 x 10^6 lb/in^3	207 GPa
Poisson's Ratio	0.3	0.3
P	1200 lb	5338 N

- **Suggested Steps**
  1. Open the database **tip_load** from the Linear and Nonlinear Analysis of a Cantilever Beam workshop.
  2. Create a Plastic Property Table.
  3. Add Plastic Property to the material, steel.
  4. Change the element properties of the beam to use reduced integration.
  5. Modify the Boundary condtions:
     - Load tip load to 300 in Y direction. Applied to 4 nodes, 300x4=1200lbs.
     - Create a 2nd loadcase for Unloading. Set scale factor of tipload to 0.
  6. Create and run an implicit non-linear analysis.
     - Select **SOL400** Run in Solution Parameters.
     - Select **.OP2** as the result output format.
     - In Load Increment parameters, set increment type to **Adaptive**
     - Run the analysis.
     - Attach the **.OP2** result file.
  7. Post-processing with Patran.
     - Create a quickplot with the fringe of Plastic deformation to see where and if Cantilever beams goes plastic.
     - Create a quickplot with the fringe and deformation of the **y** component of the displacement results.

Step 1. Open the Database



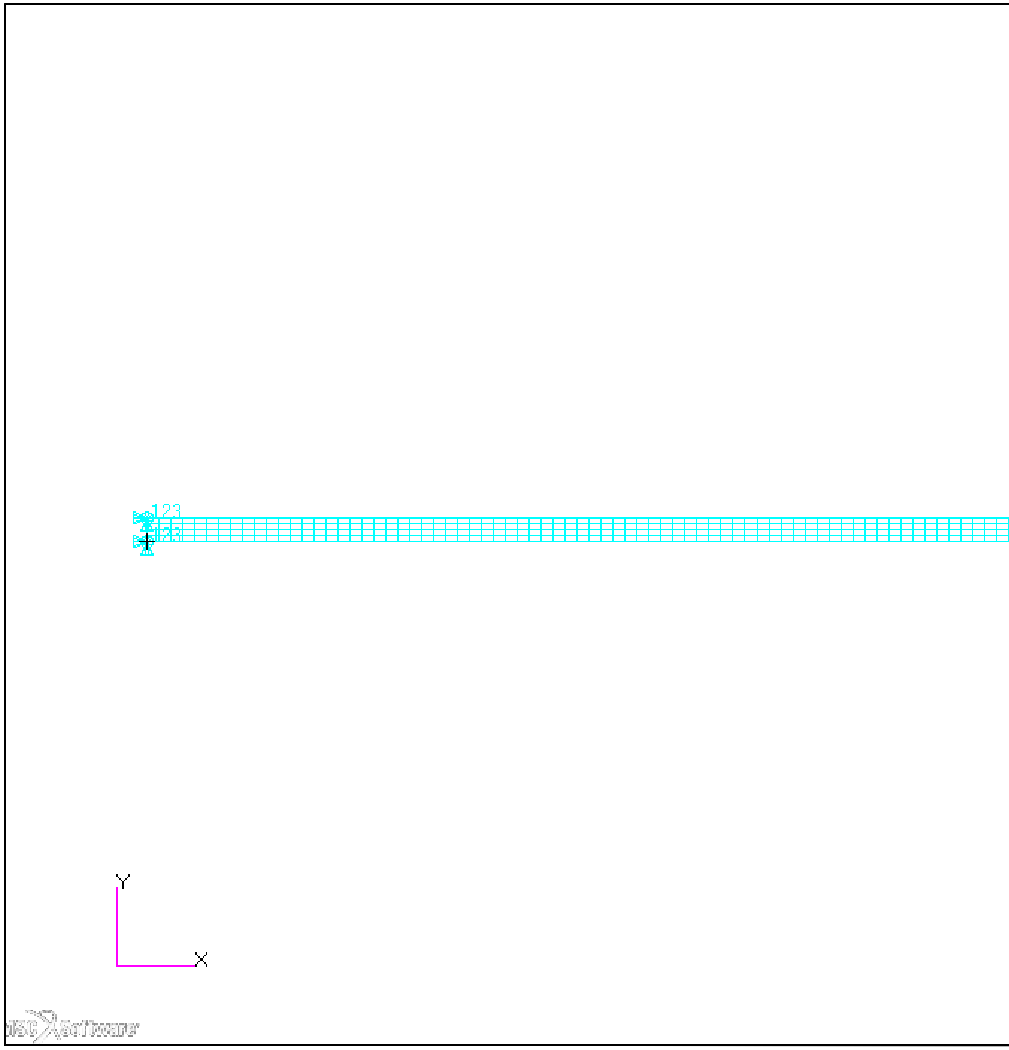


Open the database Tip_Load.db.

a. Pull down *File* > **Open**.
b. Select **Tip_Load** as the *file name*.
c. Click **OK**.

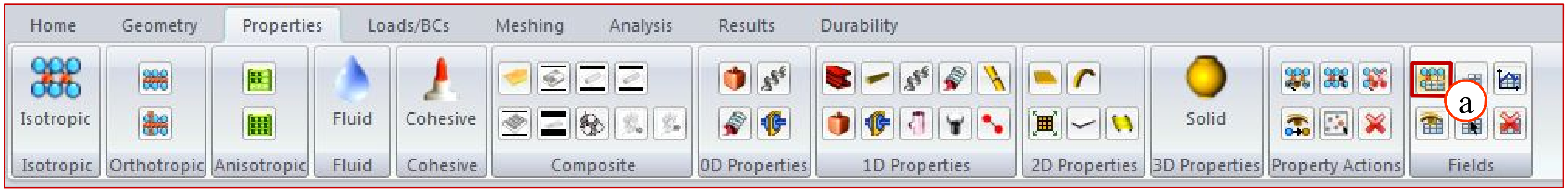
If you deleted Tip_Load.db, you may download a copy of it in the Files folder of this course.

Step 1. Open the Database (Cont.)



Change the viewport display of the beam to front view. Press the **Front View** icon in the *Home* tab under the *Orientation* group.

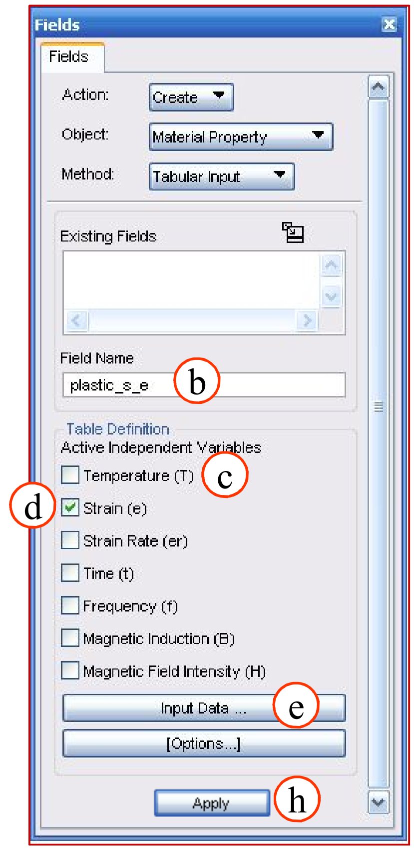
Step 2. Create Plastic Property Table

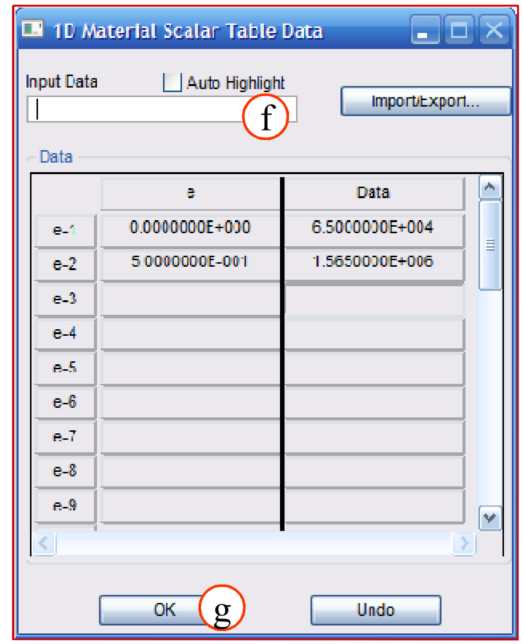


Create the Plastic Property Table.

a. Under the *Properties* tab, click **Material** in the *Fields* group.
b. Enter **plastic_s_e** as the *Field* name.
c. Uncheck the **Temperature (T).**
d. Check **Strain (e)**.
e. Click **Input Data**.
f. Enter the data shown in the table.
g. Click **OK**.
h. Click **Apply**.

(65000 is Yield Stress and 30E6 is Young Modulus, considering plastic range slope as 10% slope of Elastic range so another point on the curve can be calculated as follows for the table (0.1*30E6*0.5+65E3 = 1.565E6)

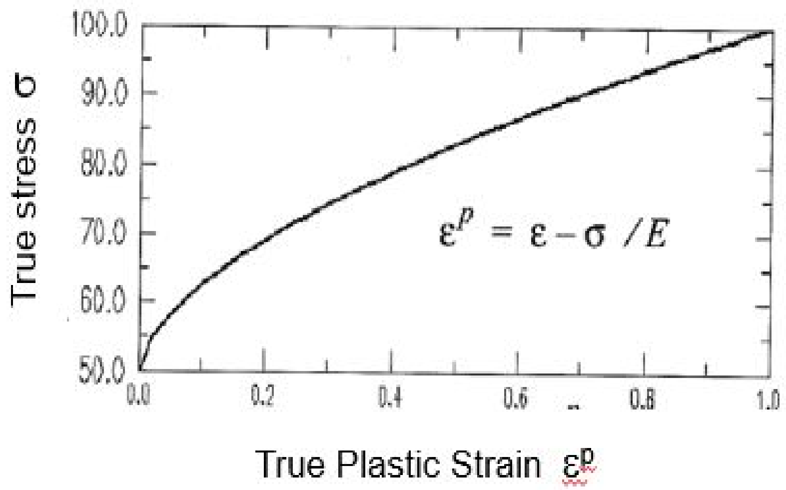




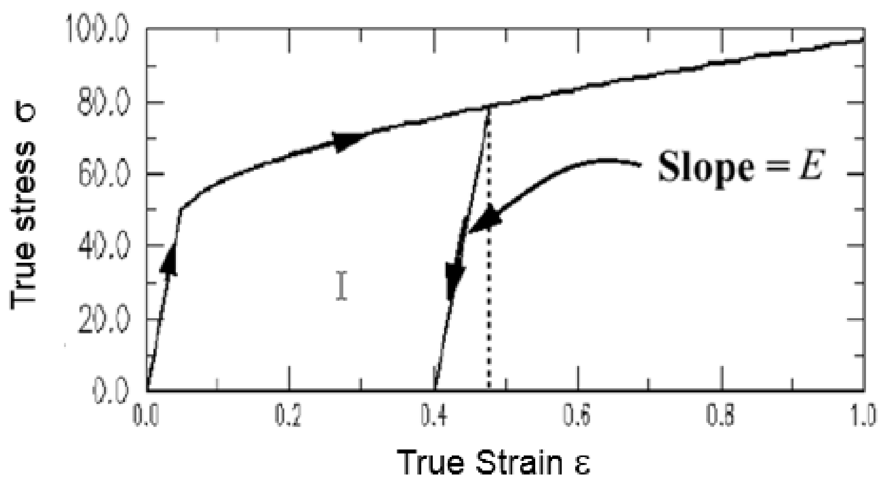
Strain	Stress
0	65000
0.5	1.565E6

Note on Material Plastic Property entry.

- **Elasto-Plastic material property can be entered in two ways:**
  - ***True Plastic Strain vs. True Stress:*** (Figure 1)
    - Starts from Yielding Stress with zero plastic strain.
  - ***Total True Strain vs. True Stress:*** (Figure 2)
    - Starts from Zero Stress and Strain. Note that the 2nd point in the table definition has to be the yield point and the slope should match exactly with the Young Modulus.

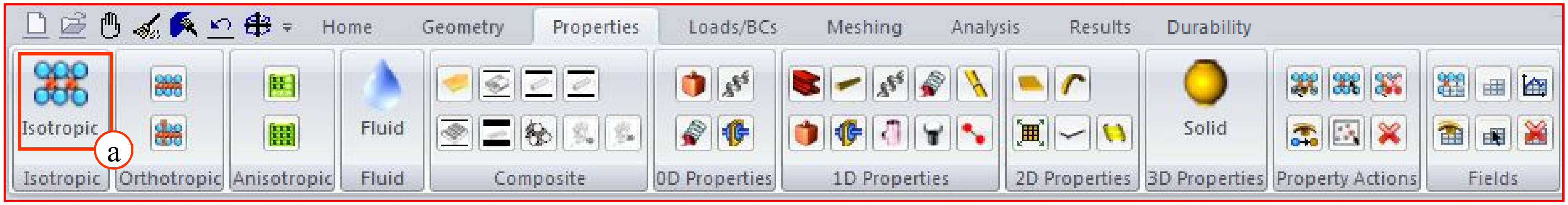


(Figure 1)



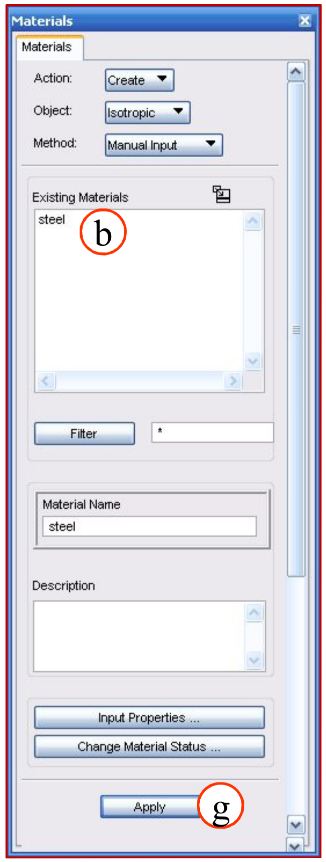
(Figure 2)

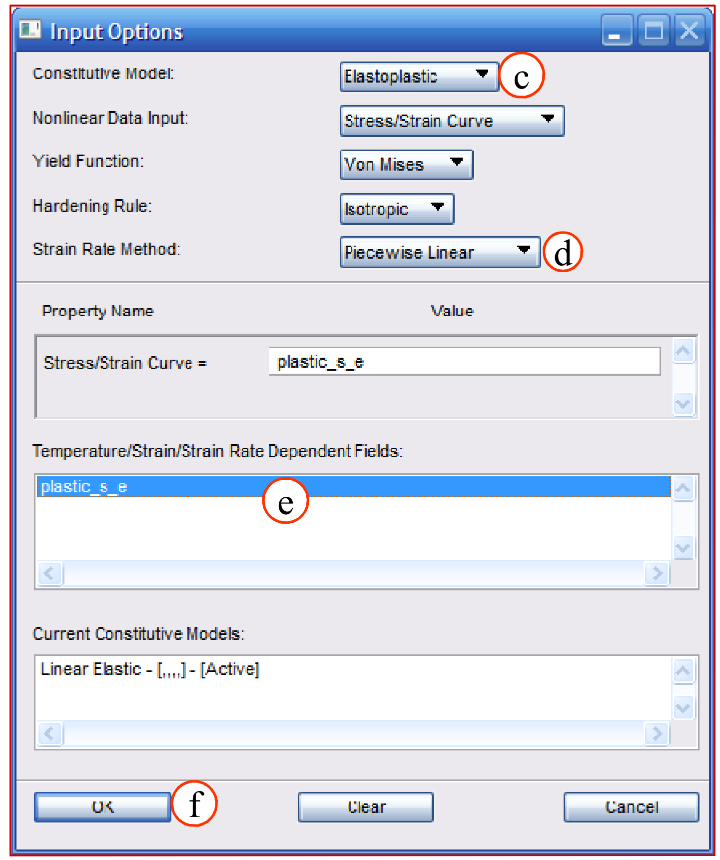
Step 3. Apply Plastic Property to the Material



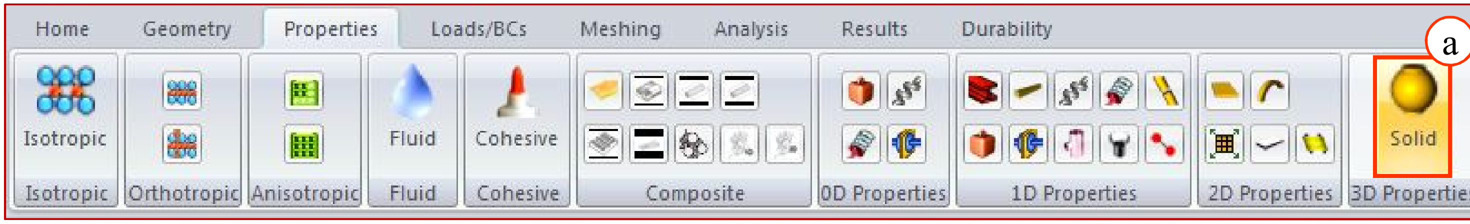
Modify the existing material.

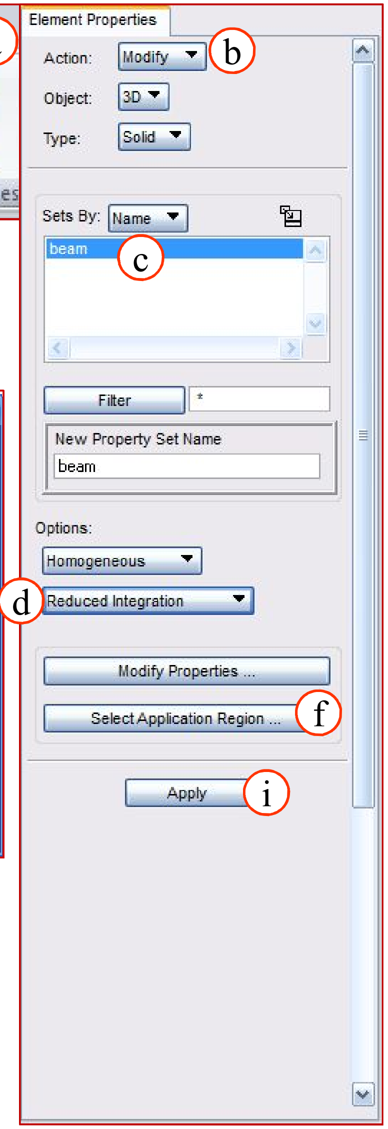
a. Under the *Properties* tab, click **Isotropic** in the *Isotropic* group.
b. Select **steel** from the *Existing Materials*
c. Pull down *Constitutive Model* to **Elastoplastic.**
d. Pull down *Strain Rate Method* to **Piecewise Linear**.
e. Select **plastic_s_e** from the *Temperature/Strain/Strain Rate Dependent Fields.*
f. Click **OK**
g. Click **Apply**





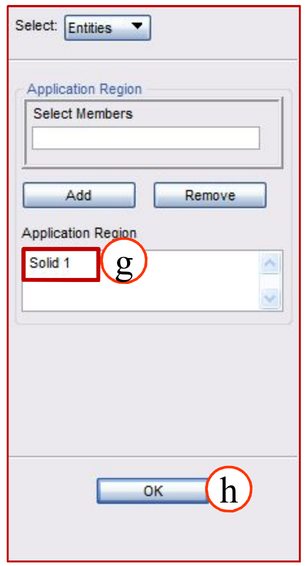
Step 4. Modify Physical Properties

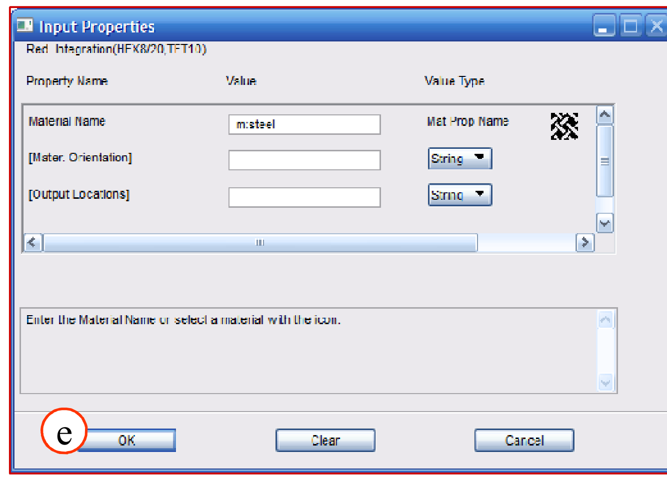




Now modify the properties of the beam.

a. Under the *Properties* tab, click **Solid** in the *3D Properties* group.
b. Pull down *Action* to **Modify**.
c. Select the property set name **beam**.
d. Pull down *Options:* **Reduced Integration**.
e. Check that steel is the *Material Name* and press **OK**.
f. Click **Select Application Region**.
g. Select **Solid 1**.
h. Click **OK**.
i. Click **Apply**.



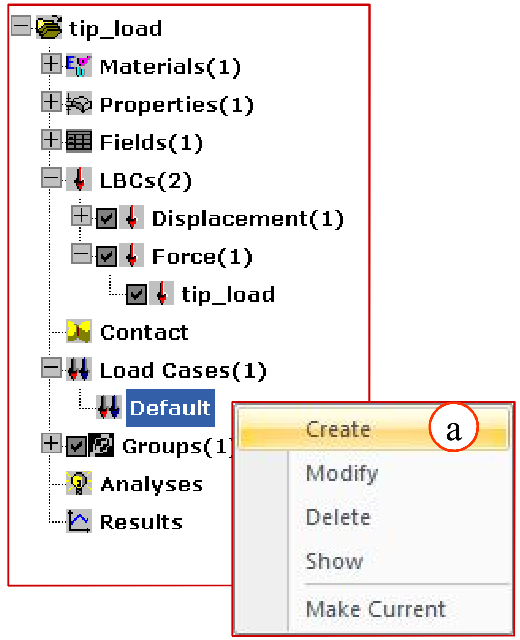


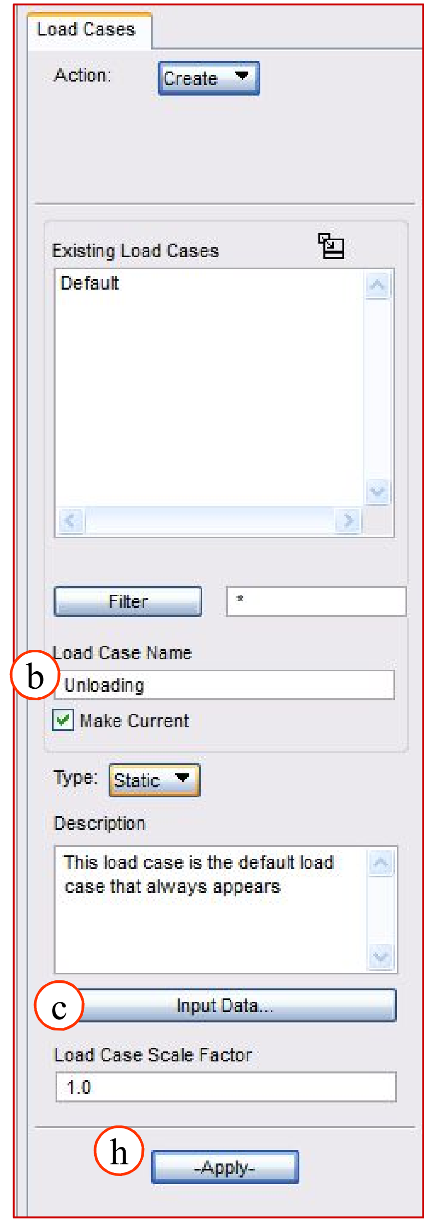
MSC Software

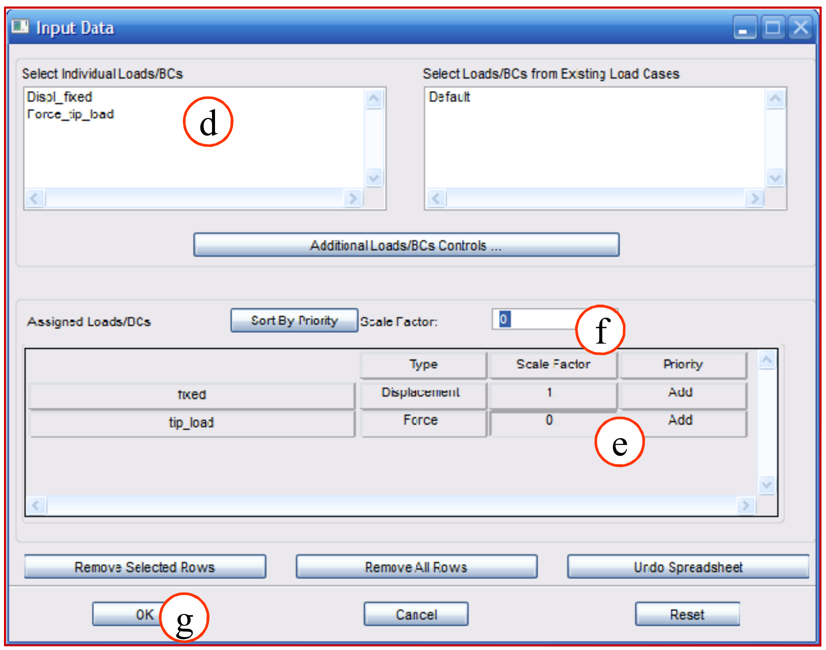
Step 5. Modify Loads/BCs

Create a 2nd Load Case

a. Right click **Default** in the *Model Tree* and select **Create.**
b. Enter *Name*: **Unloading**
c. Click **Input Date**
d. Select **Displ_fixed** and **Force_tip_load** from *Select Indivudual Loads/BCs*
e. Select Scale Factor for tip_load
f. Enter **0** for Scale Factor of *tip_load.*
g. Click **OK**
h. Click **Apply**



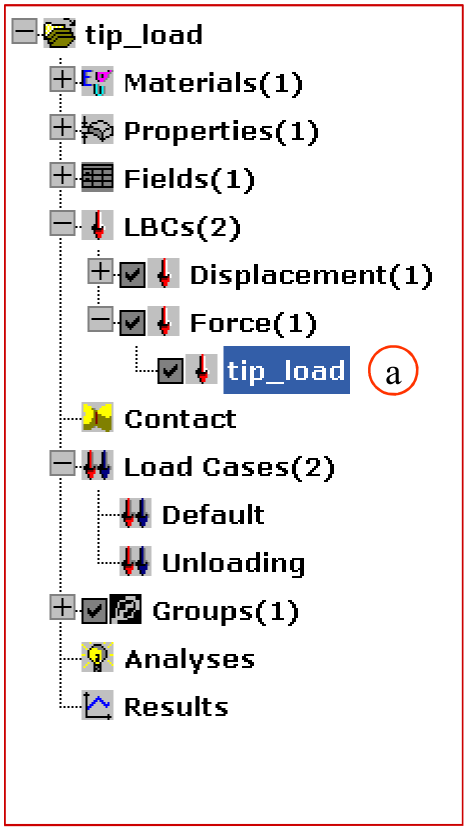


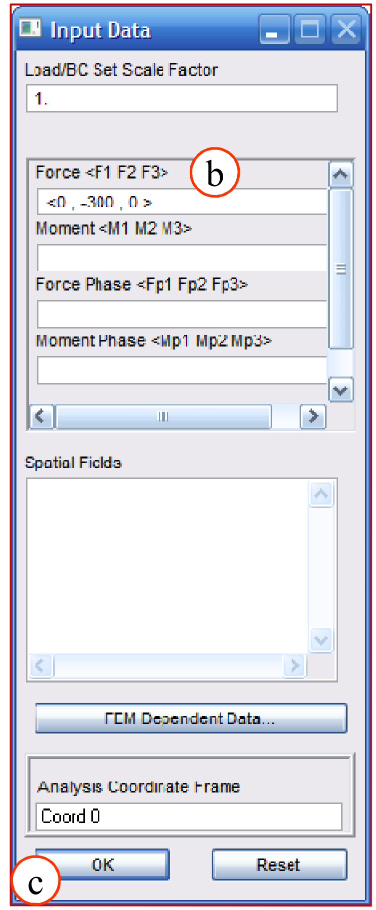


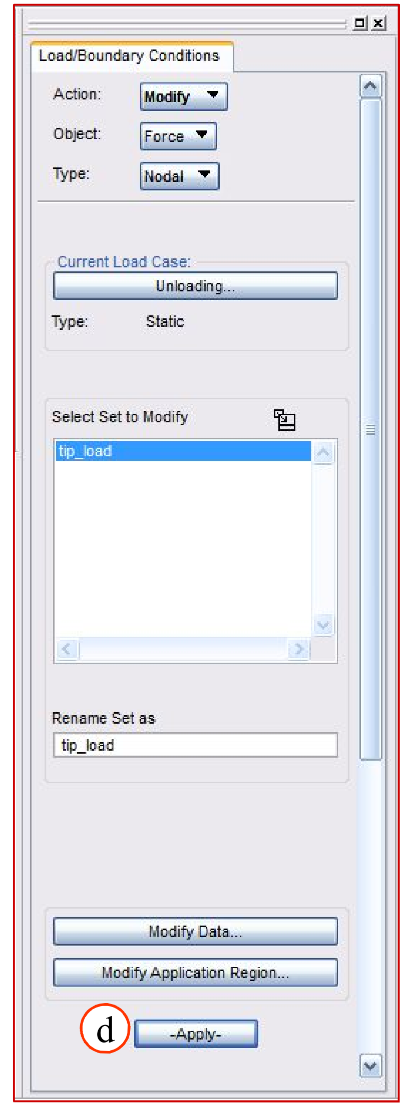
Step 5. Modify Loads/BCs (Cont.)

Modify the applied tip load to each node to be -300

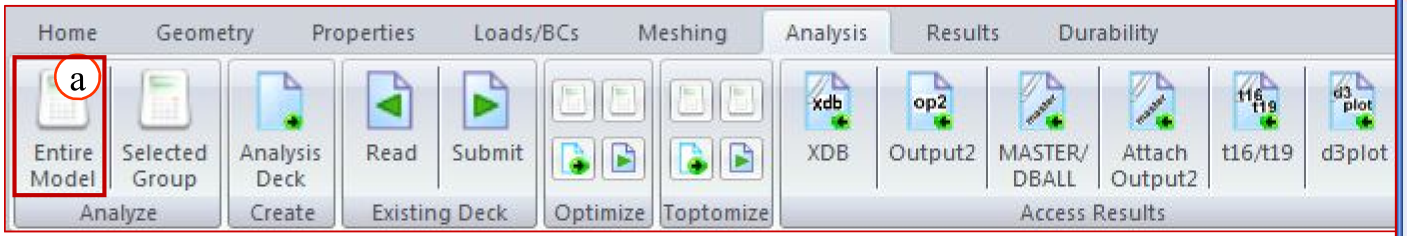
a. Double click **tip_load** in Model Tree
b. Modify Force **<0,-300,0>.**
c. Click **OK**
d. Click **Apply**





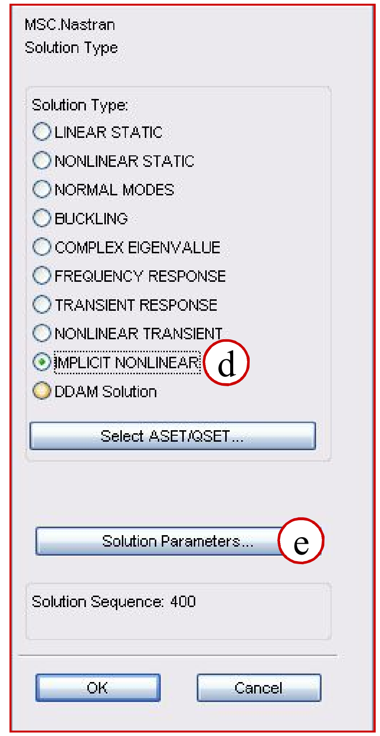


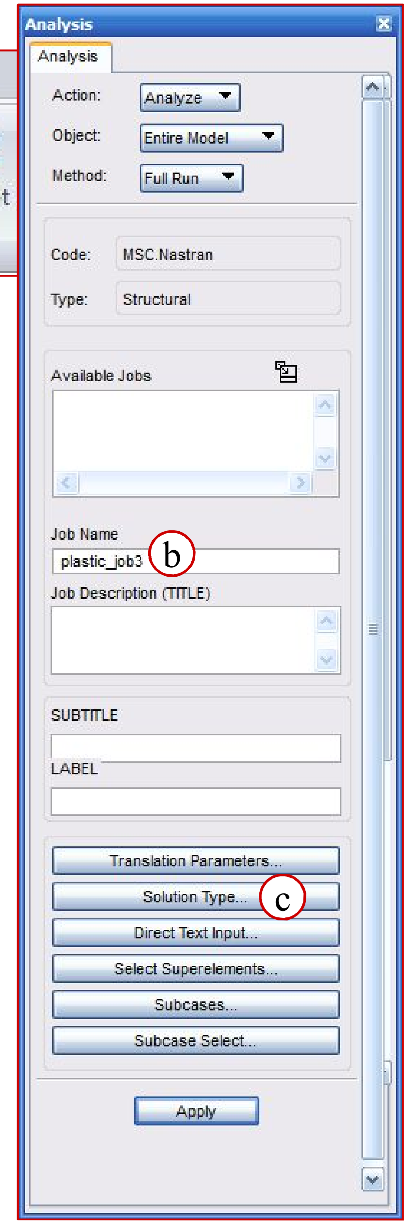
Step 6. Analyze the Model



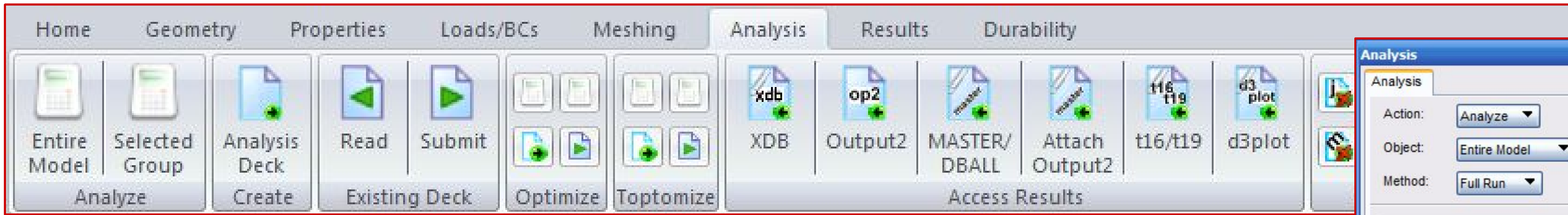
Run a nonlinear analysis.

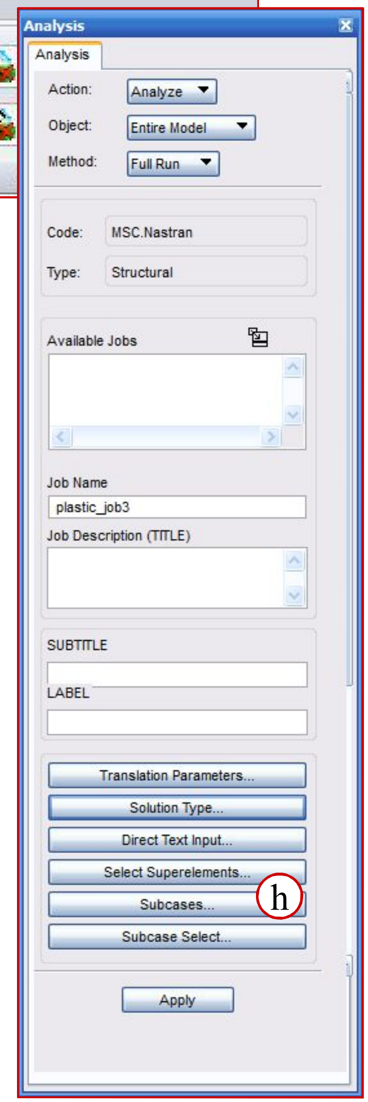
a. Under the *Analysis* tab, Click **Entire Model** in the *Analyze* group.
b. Enter *Job Name*: **plastic_job3**.
c. Click **Solution Type…**
d. Select **IMPLICIT NONLINEAR**.
e. Click on **Solution Parameters**.





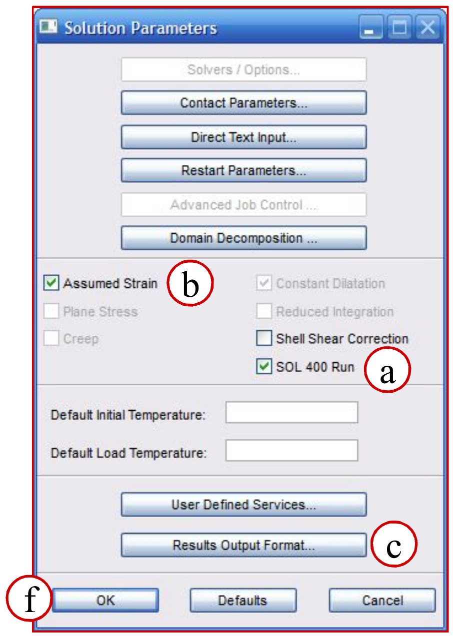
Step 6. Analyze the Model (Cont.)

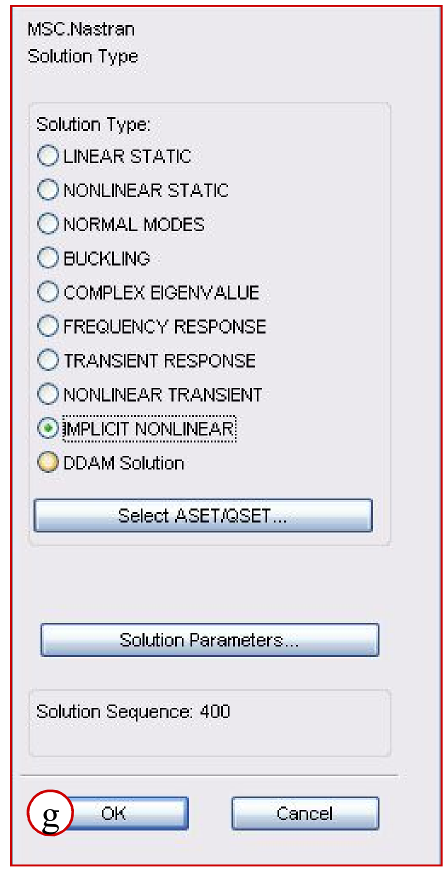


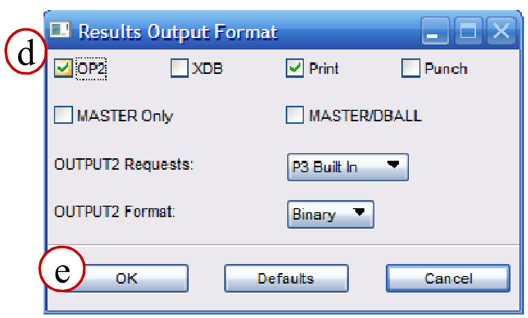


Run Nonlinear analysis.

a. Confirm **SOL 400 Run**.
b. Confirm **Assumed Strain**.
c. Click on **Results Output Format**.
d. Select **OP2**.
e. Click **OK**.
f. Click **OK**.
g. Click **OK**.
h. Click on **Subcases**.

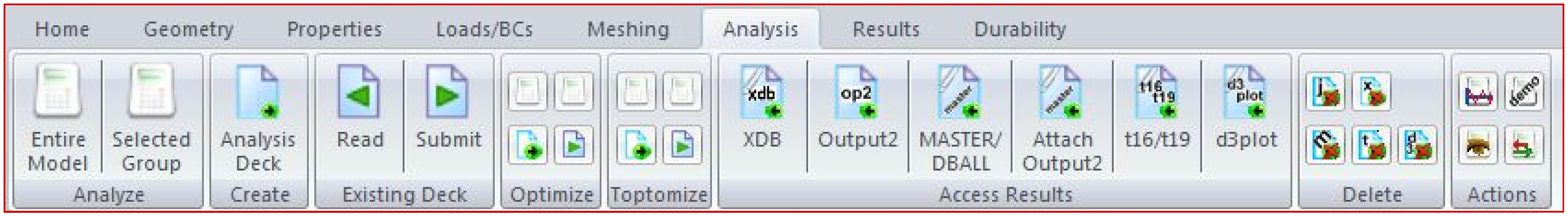






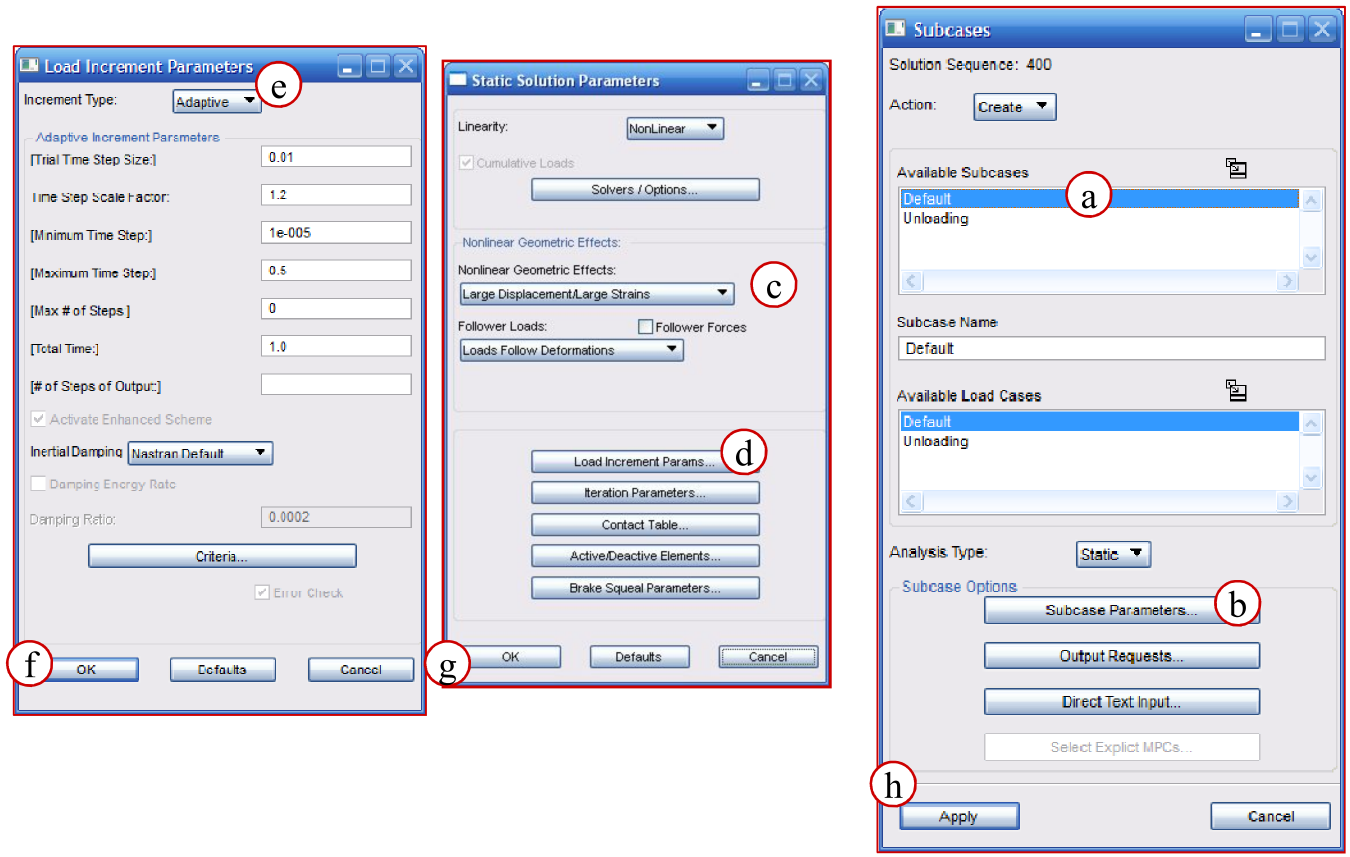
MSC Software

Step 6. Analyze the Model (Cont.)

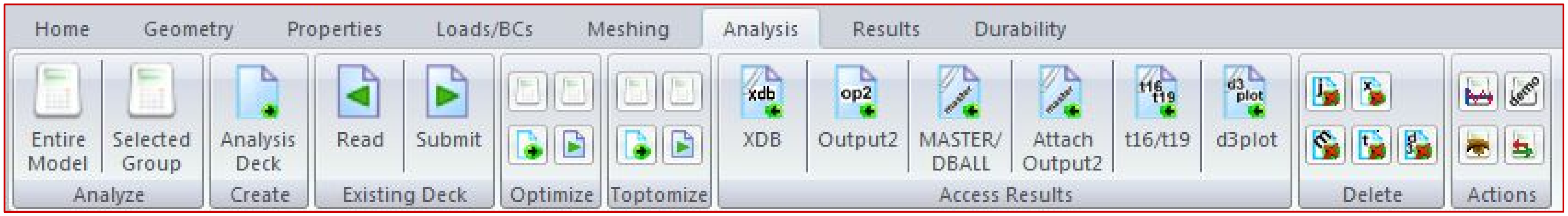


Change Increment Type to Adaptive

a. Select **Default Subcase**
b. Click on **Subcase Parameters**.
c. Confirm that *Nonlinear Geometric Effects* is set to **Large Displacements/Large Strains**.
d. Click on **Load Increment Parameters**.
e. Pull down *Increment Type* is set to **Adaptive**.
f. Click **OK**.
g. Click **OK**
h. Click **Apply**

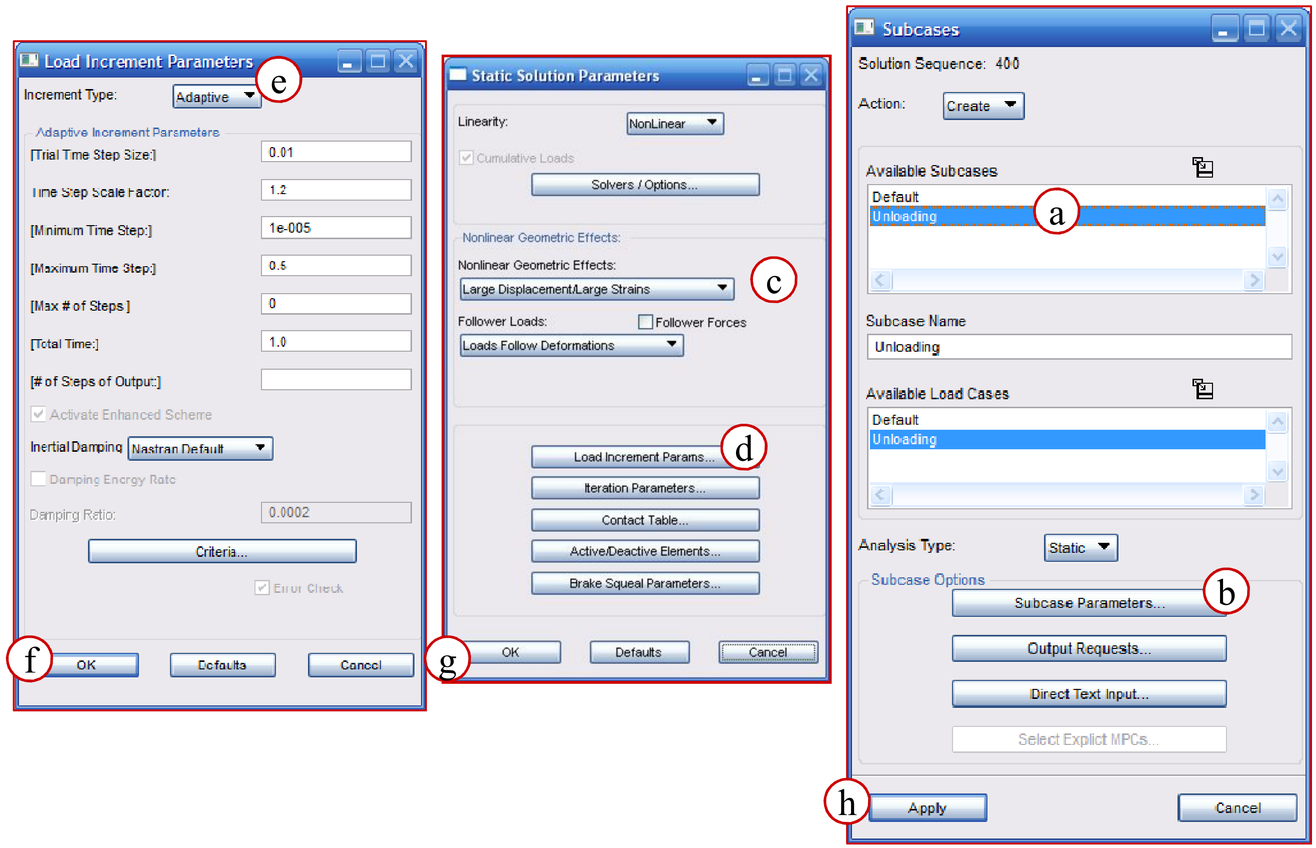


Step 6. Analyze the Model (Cont.)



Change Increment Type to Adaptive

a. Select **Unloading subcase**
b. Click on **Subcase Parameters**.
c. Confirm that *Nonlinear Geometric Effects* is set to **Large Displacements/Large Strains**.
d. Click on **Load Increment Parameters**.
e. Confirm *Increment Type* is set to **Adaptive**.
f. Click **OK**.
g. Click **OK**
h. Click **Apply**

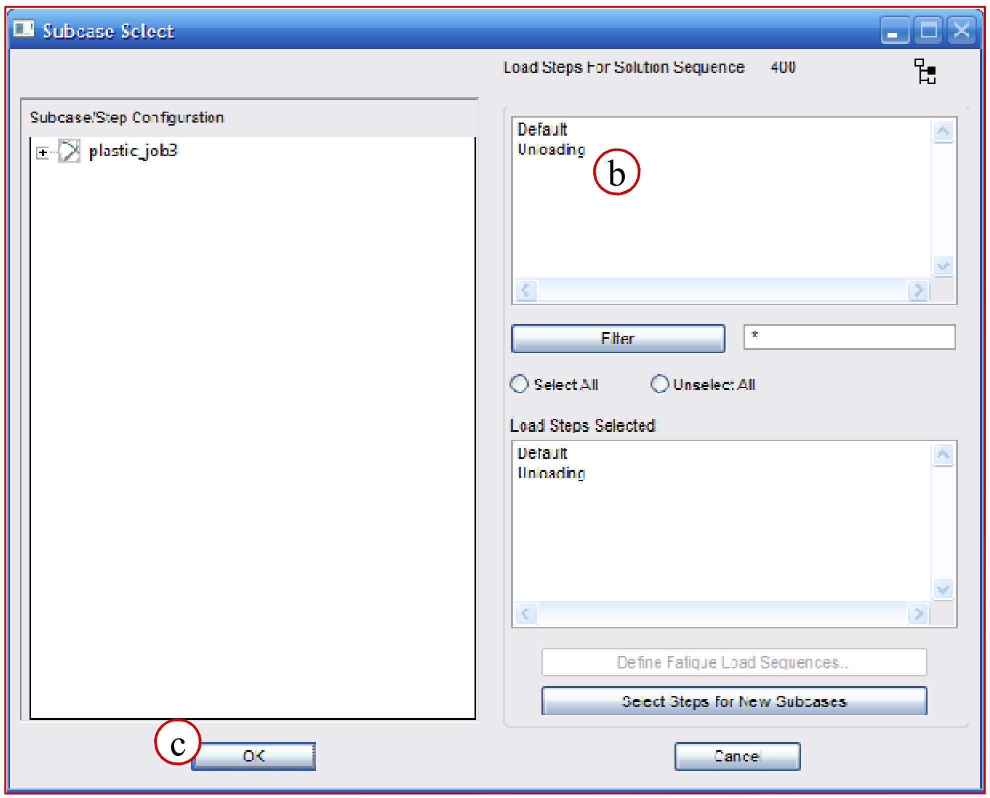


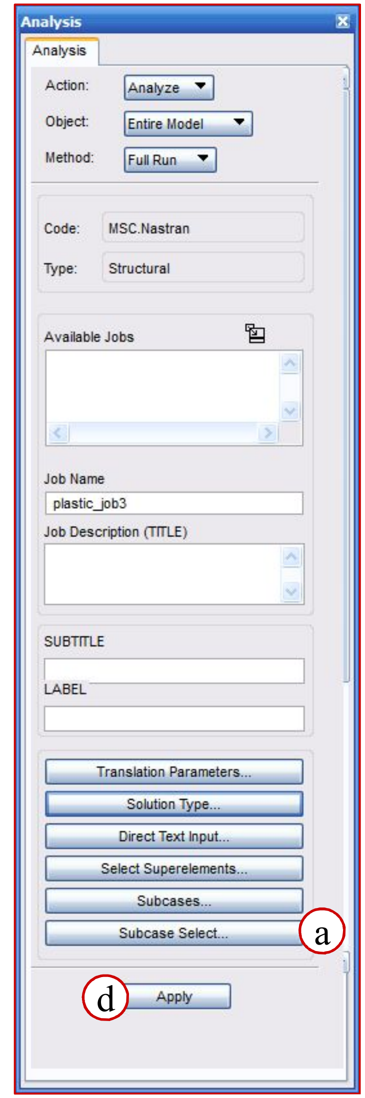
MSC Software

Step 6. Analyze the Model (Cont.)

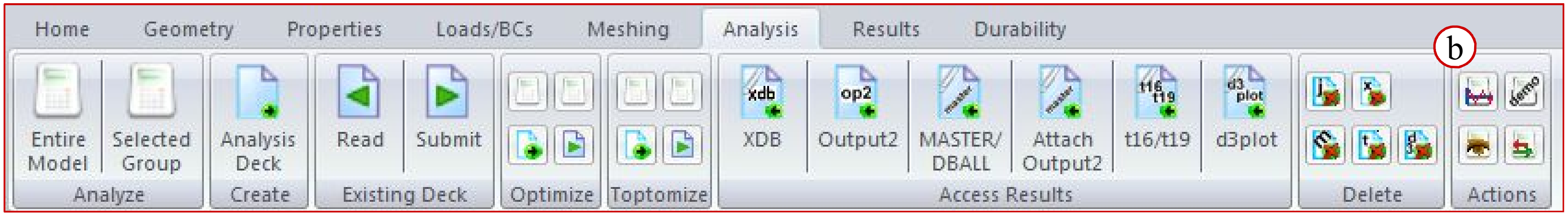
Add both Steps for analysis, and run the job.

a. Click **Subcase Select**
b. Select **Unloading**
c. Click **OK**
d. Click **Apply** to run the job.



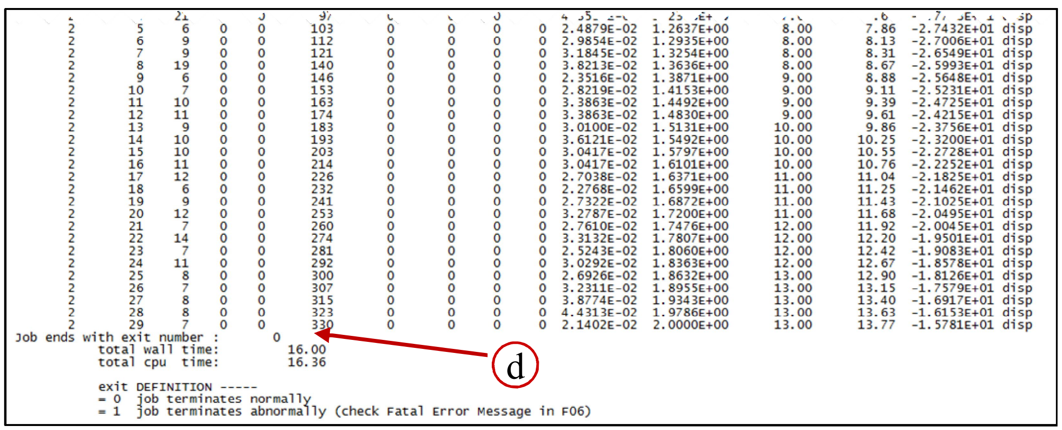


Step 6. Analyze the Model (Cont.)

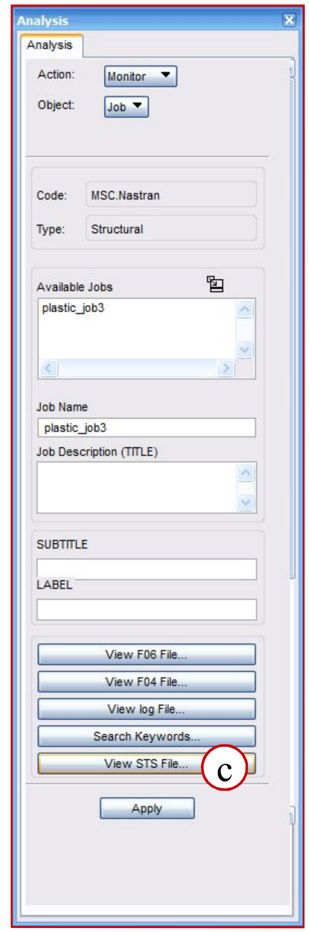


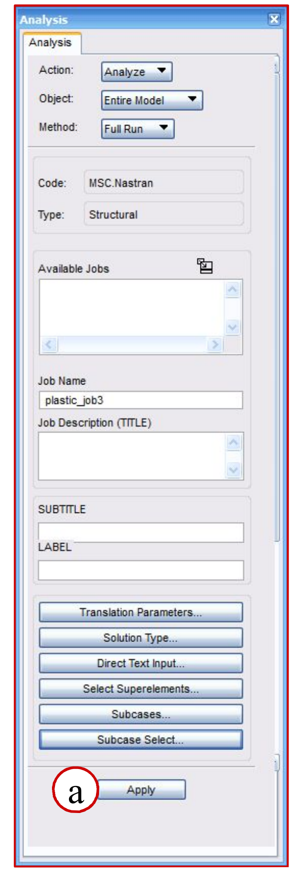
Run the Job.

a. Click **Apply**.
b. Click **Monitor**
c. Click **View STS file** to check the job is done.
d. Check Job terminate normally. **Exit number to be 0.**



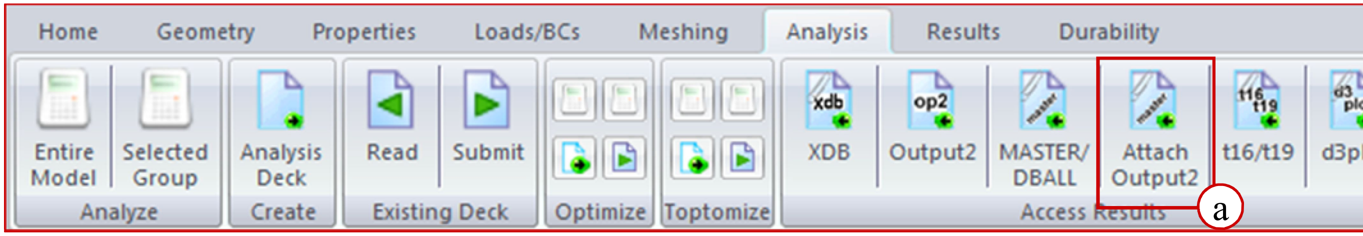
```
2     5    6   0   0    103   0   0   0   0  2.4879E-02  1.2637E+00    8.00    7.86  -2.7432E+01 disp
2     6    9   0   0    112   0   0   0   0  2.9854E-02  1.2935E+00    8.00    8.13  -2.7006E+01 disp
2     7    9   0   0    121   0   0   0   0  3.1845E-02  1.3254E+00    8.00    8.31  -2.6549E+01 disp
2     8   19   0   0    140   0   0   0   0  3.8213E-02  1.3636E+00    8.00    8.67  -2.5993E+01 disp
2     9    6   0   0    146   0   0   0   0  2.3516E-02  1.3871E+00    9.00    8.88  -2.5648E+01 disp
2    10    7   0   0    153   0   0   0   0  2.8219E-02  1.4153E+00    9.00    9.11  -2.5231E+01 disp
2    11   10   0   0    163   0   0   0   0  3.3863E-02  1.4492E+00    9.00    9.39  -2.4725E+01 disp
2    12   11   0   0    174   0   0   0   0  3.3863E-02  1.4830E+00    9.00    9.61  -2.4215E+01 disp
2    13    9   0   0    183   0   0   0   0  3.0100E-02  1.5131E+00   10.00    9.86  -2.3756E+01 disp
2    14   10   0   0    193   0   0   0   0  3.6121E-02  1.5492E+00   10.00   10.25  -2.3200E+01 disp
2    15   10   0   0    203   0   0   0   0  3.0417E-02  1.5797E+00   10.00   10.55  -2.2728E+01 disp
2    16   11   0   0    214   0   0   0   0  3.0417E-02  1.6101E+00   10.00   10.76  -2.2252E+01 disp
2    17   12   0   0    226   0   0   0   0  2.7038E-02  1.6371E+00   11.00   11.04  -2.1825E+01 disp
2    18    6   0   0    232   0   0   0   0  2.2768E-02  1.6599E+00   11.00   11.25  -2.1462E+01 disp
2    19    9   0   0    241   0   0   0   0  2.7322E-02  1.6872E+00   11.00   11.43  -2.1025E+01 disp
2    20   12   0   0    253   0   0   0   0  3.2787E-02  1.7200E+00   11.00   11.68  -2.0495E+01 disp
2    21    7   0   0    260   0   0   0   0  2.7610E-02  1.7476E+00   12.00   11.92  -2.0045E+01 disp
2    22   14   0   0    274   0   0   0   0  3.3132E-02  1.7807E+00   12.00   12.20  -1.9501E+01 disp
2    23    7   0   0    281   0   0   0   0  2.5243E-02  1.8060E+00   12.00   12.42  -1.9083E+01 disp
2    24   11   0   0    292   0   0   0   0  3.0292E-02  1.8363E+00   12.00   12.67  -1.8578E+01 disp
2    25    8   0   0    300   0   0   0   0  2.6926E-02  1.8632E+00   13.00   12.90  -1.8126E+01 disp
2    26    7   0   0    307   0   0   0   0  3.2311E-02  1.8955E+00   13.00   13.15  -1.7579E+01 disp
2    27    8   0   0    315   0   0   0   0  3.8774E-02  1.9343E+00   13.00   13.40  -1.6917E+01 disp
2    28    8   0   0    323   0   0   0   0  4.4313E-02  1.9786E+00   13.00   13.63  -1.6153E+01 disp
2    29    7   0   0    330   0   0   0   0  2.1402E-02  2.0000E+00   13.00   13.77  -1.5781E+01 disp
Job ends with exit number :      0
          total wall time:        16.00
          total cpu  time:        16.36

          exit DEFINITION -----
          = 0  job terminates normally
          = 1  job terminates abnormally (check Fatal Error Message in F06)
```



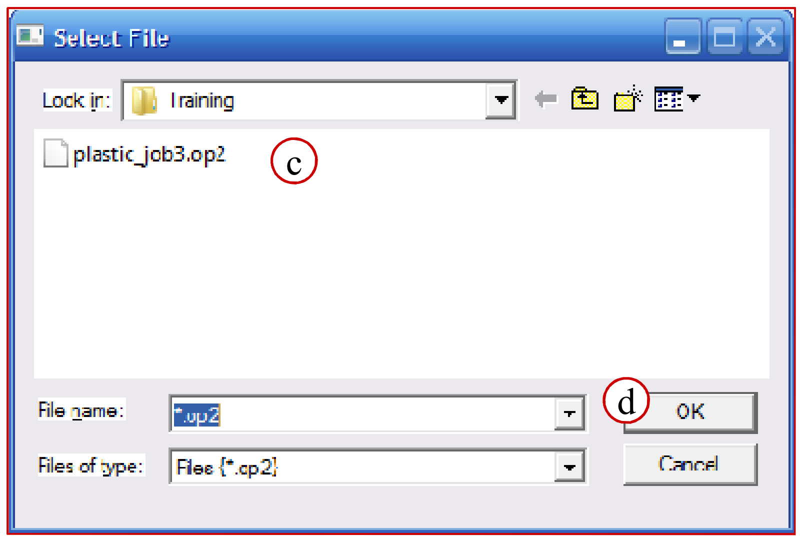
MSC Software

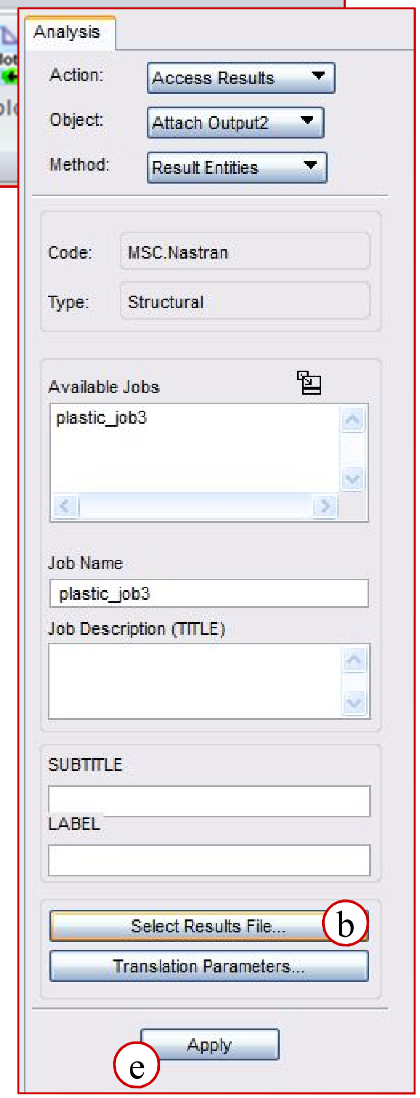
Step 6. Analysis: Access Results



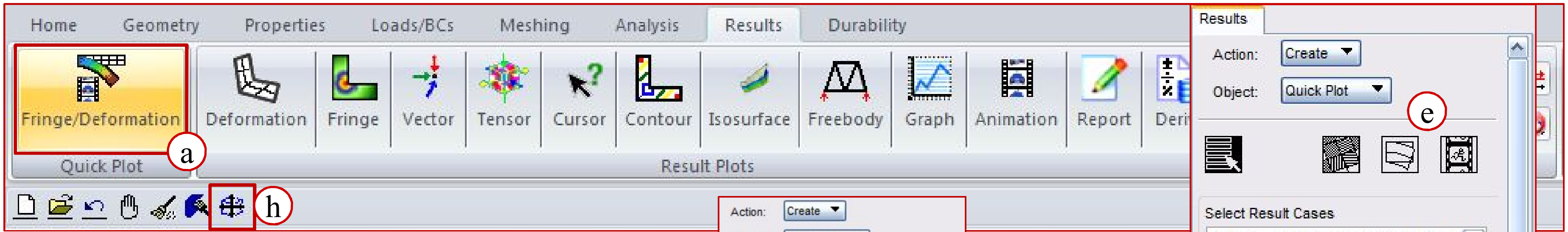
Access the results from the nonlinear analysis.

a. Under the *Analysis tab*, click **Attach Output2** in the *Access Results* group.
b. Click on **Select Results File**.
c. Select the file ***plastic_job3***.
d. Click **OK**.
e. Click on **Apply**.



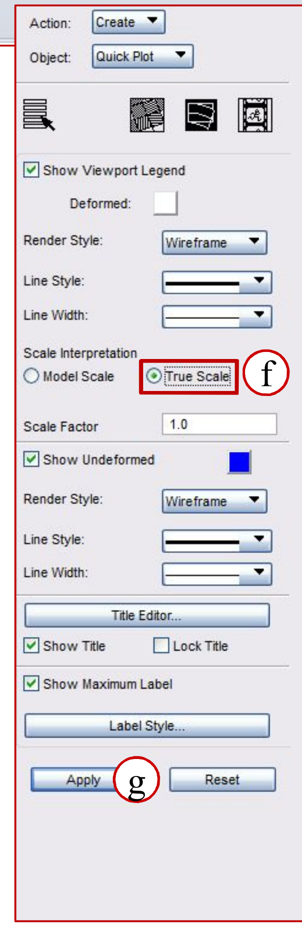


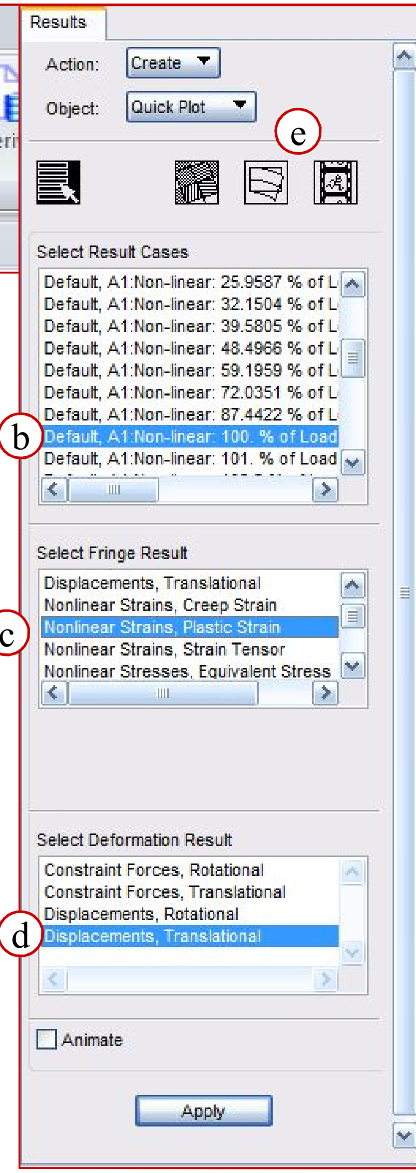
Step 7. Create a Fringe Deformation Plot



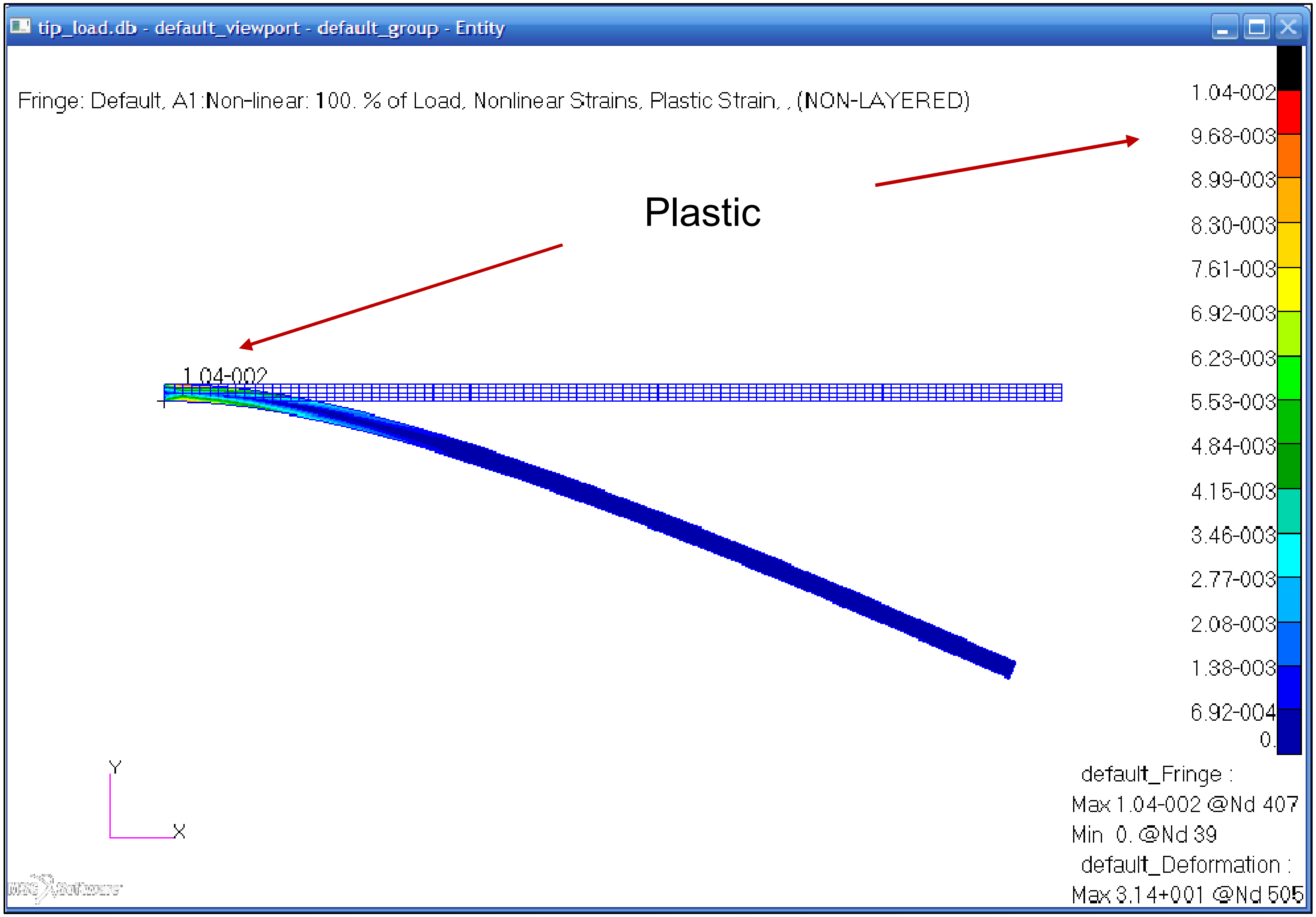
Post-process the job results.

a. Under the *Results* tab, click **Fringe/Deformation** in the *Quick Plot* group.
b. Select **Default A1: Non-linear: 100. of Load** from *Select Results Cases.*
c. Select **Nonlinear Stress, Plastic Strain** from *Select Fringe Result*.
d. Select **Displacements, Translational** from the *Select Deformation Result*.
e. Click on **Deform Attributes** icon.
f. Select **True Scale**.
g. Click **Apply**.
h. Click **Fit View** to get a view of the entire beam.





Step 7. Create a Fringe Deformation Plot (Cont.)

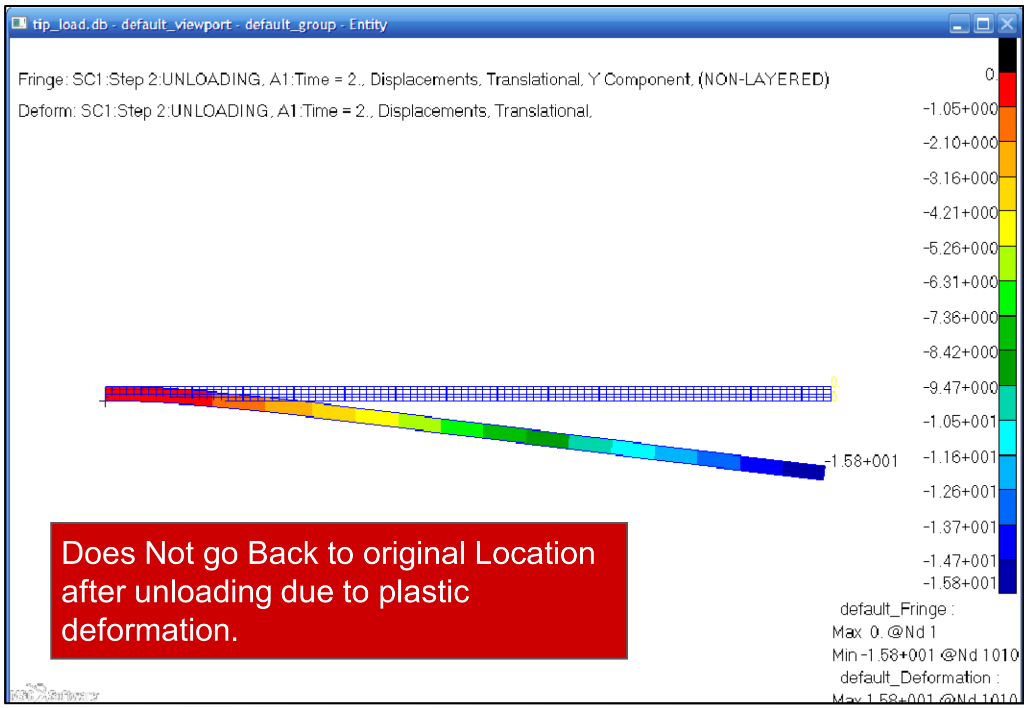


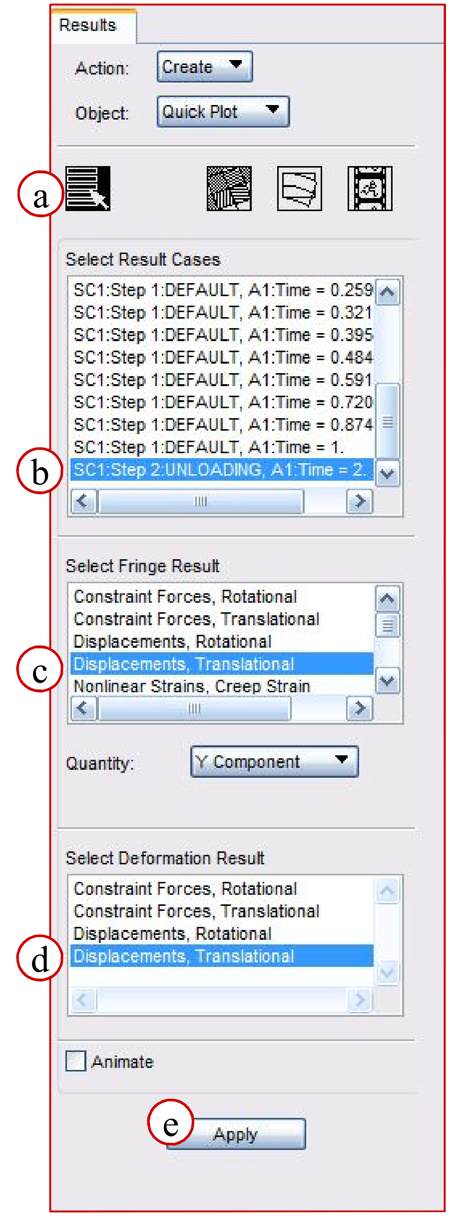
Step 7. Create a Fringe Deformation Plot

Create Fringe deformation plot with deformation to better evaluate the deformation after loading and unloading

a. Click on **Select Results** icon.
b. Select **A1:Non-linear: 200. % of Load**
c. Select **Displacement Translational** in Fringe results
d. Pull down *Quantity* to **Y Component**.
e. Click **Apply**

Create Plot at 100% of Load (At complete Loading) and compare the plots (Next Page)

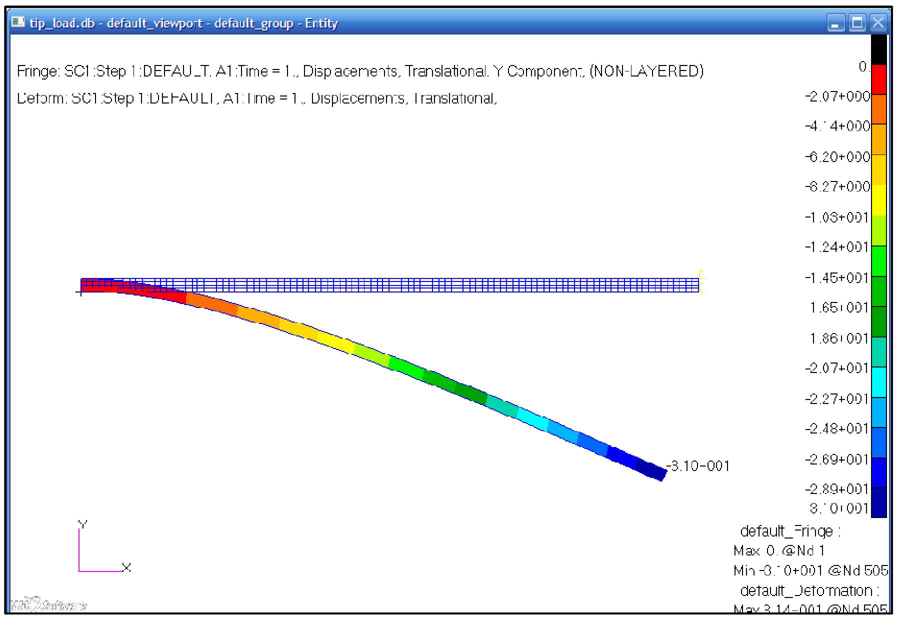




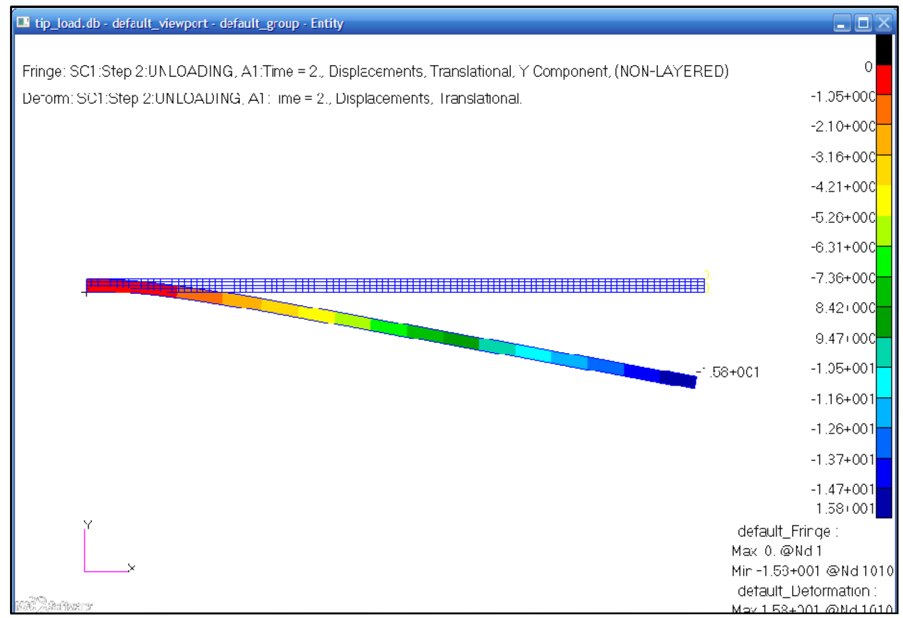
MSC Software

Compare Results of Deformed shape

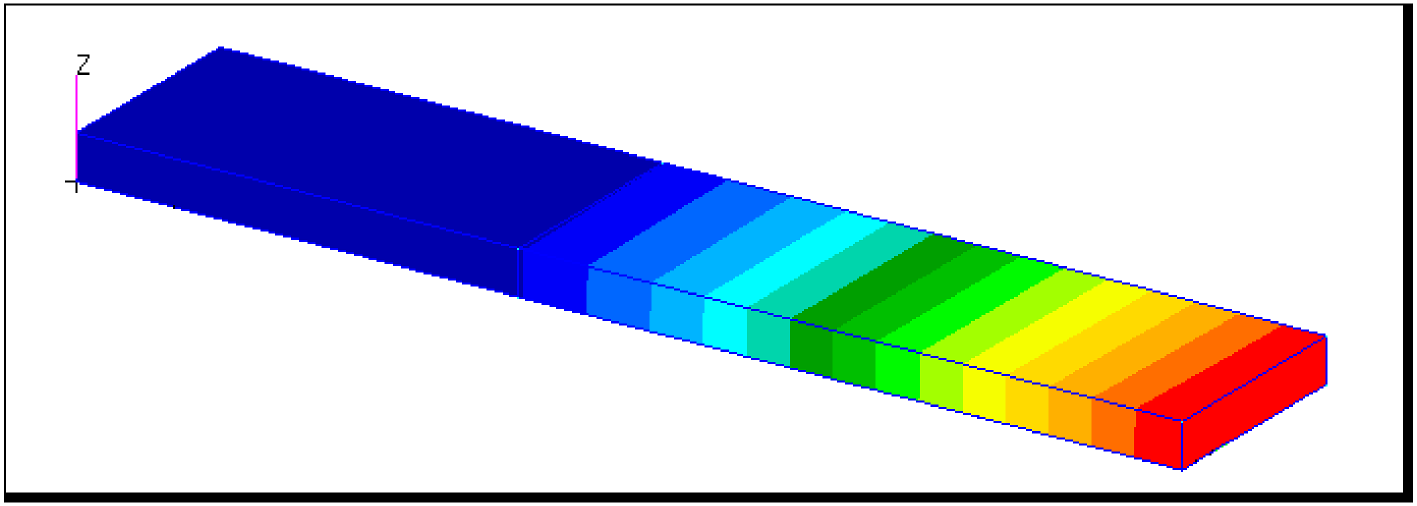
Loaded



Unloaded



WORKSHOP 7
SOLID SHELL COMPOSITES MODELING

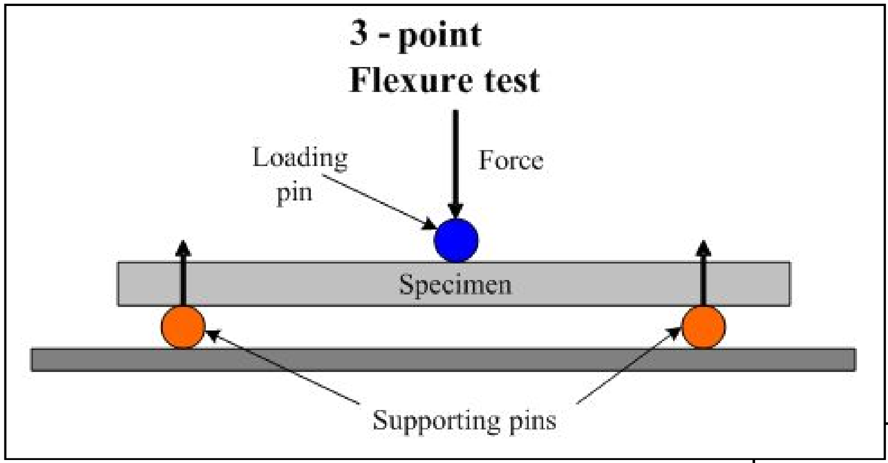


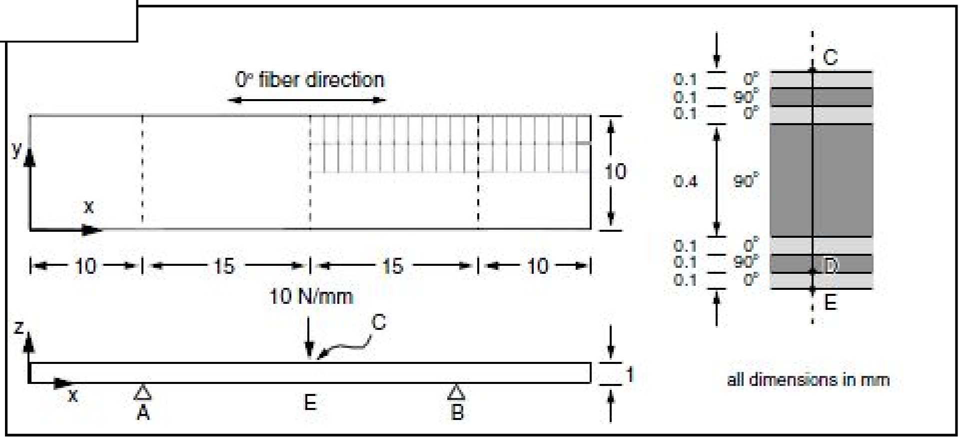
MSC Software

- **Workshop Objectives**
  – Become familiar with setting up a solid composite shell model
- **Software Version**
  – Patran 2013
  – MSC Nastran 2013.1
- **Files Required**
  – solid_shell.db

Problem Description

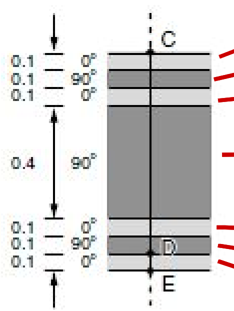
– Simulate a three-point-bending composite test specimen using solid composite shell elements
– Due to symmetry, only a quarter of the specimen is modeled.





- **Problem Description (cont.)**
  - You will build a laminate by entering each ply thickness as a percentage of the total thickness
  - Two very thin outer plies are used to pick up stresses at the outside faces of the part.

Stress recovery ply



all dimensions in mm

Stress recovery ply

	Material Name	% Thickness	Orientation
1	Ortho1	1e-3	0
2	Ortho1	9.999	0
3	Ortho1	10	90
4	Ortho1	10	0
5	Ortho1	40	90
6	Ortho1	10	0
7	Ortho1	10	90
8	Ortho1	9.999	0
9	Ortho1	1e-3	0

- **Suggested Steps**
  1. Open solid_shell.db
  2. Review load and boundary conditions by plotting markers
  3. Verify element coordinate systems
  4. Create Orthotropic material properties
  5. Create a laminated composite
  6. Create a 3D solid properties
  7. Set up the Analysis
  8. Attach OP2 results file
  9. Plot Deformation and Fringe results

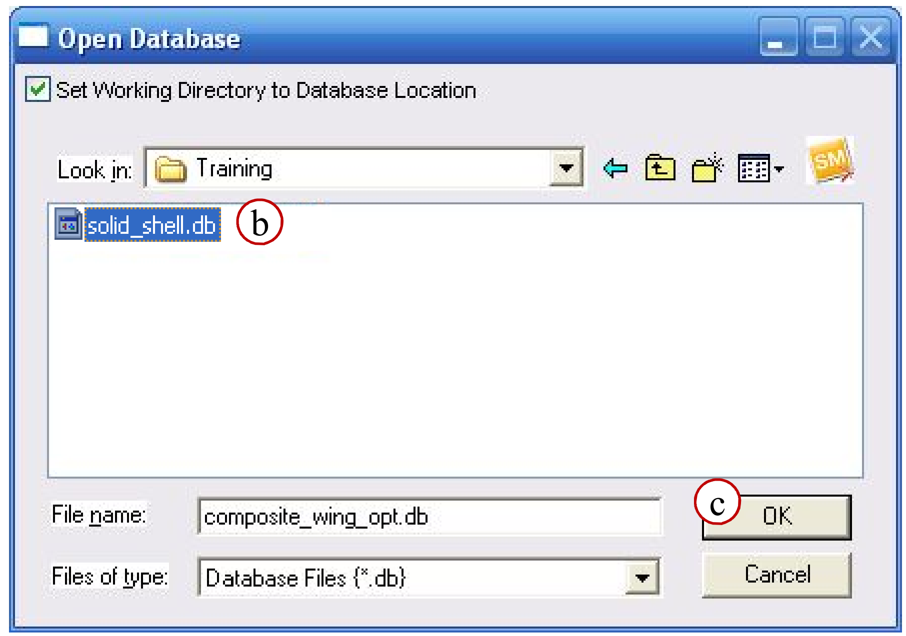
Step 1. Open an existing Patran Database

Start Patran and open a database containing the composite coupon:

a. Click **File Open** in the *Defaults* group.

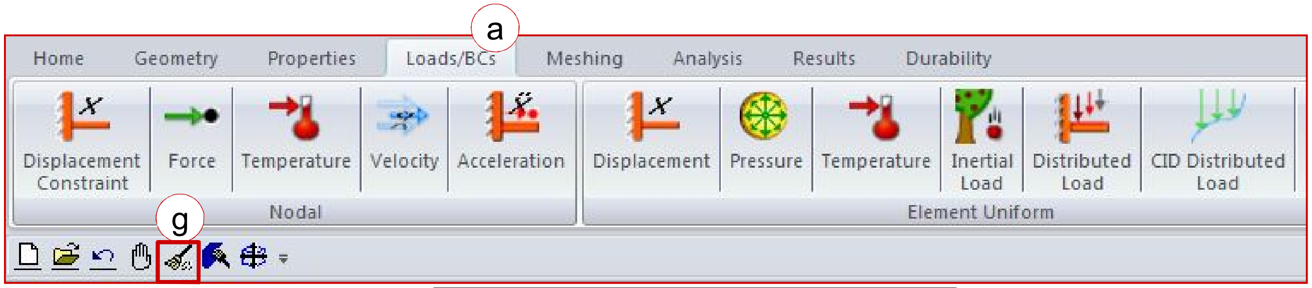
b. Select **solid_shell.db** as the *File name.*

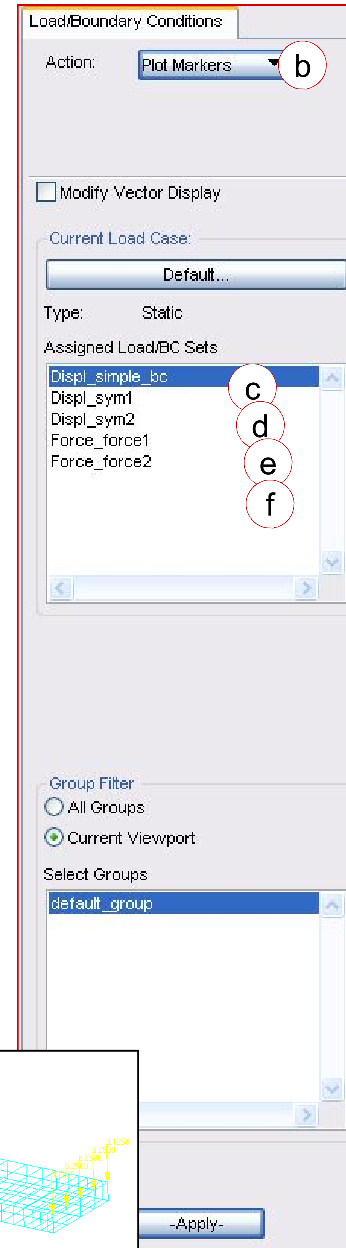
c. Click **OK**.



MSC Software

Step 2. Review Loads and Boundary Conditions





Plot loads and boundary conditions:

a. Click the **Loads/BCs** tab.

b. Set *Action* to **Plot Markers**.

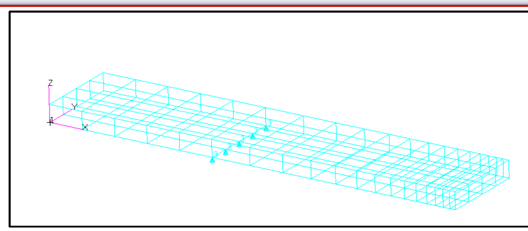
c. Select **Displ_simple_bc** and click **Apply**. This plots the simple support in the vertical direction.

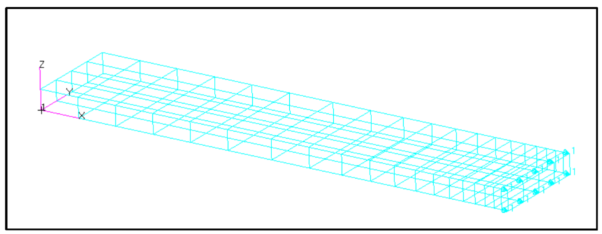
d. Select **Displ_sym1** and click **Apply**. This plots the X symmetry constraints.

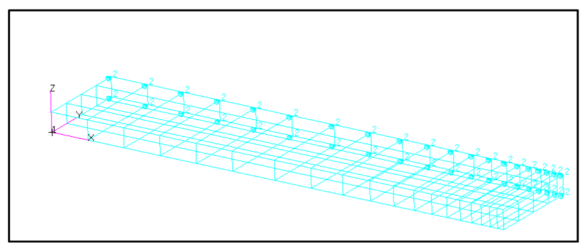
e. Select **Displ_sym2** and click **Apply**. This plots the Y symmetry constraints.

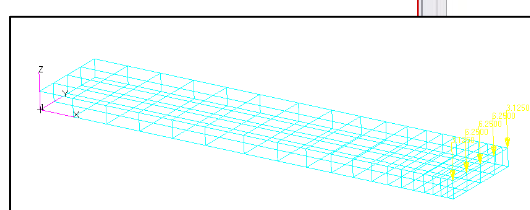
f. Select **Force_force1** and **Force_force2** and click **Apply**. This plots the applied load.

g. Click **Reset Graphics** to clear the markers.





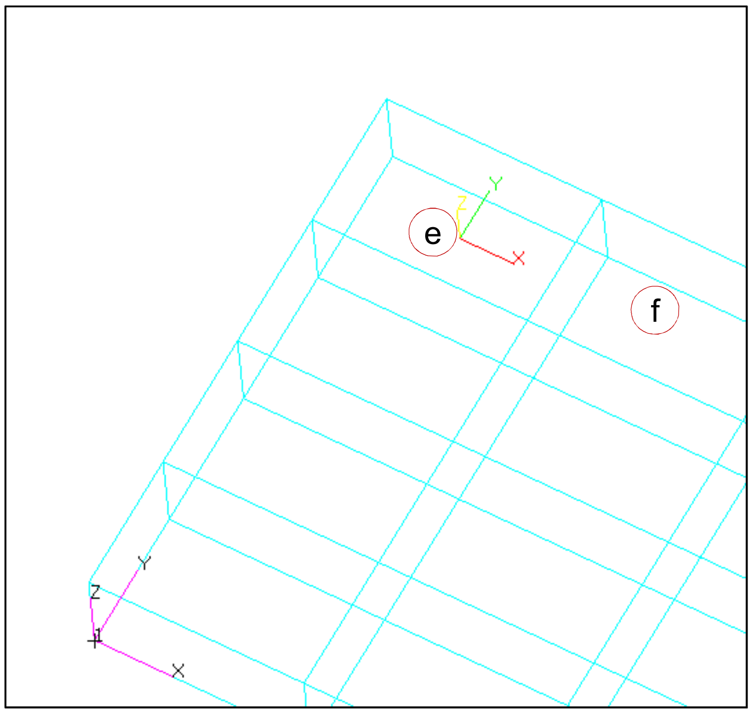


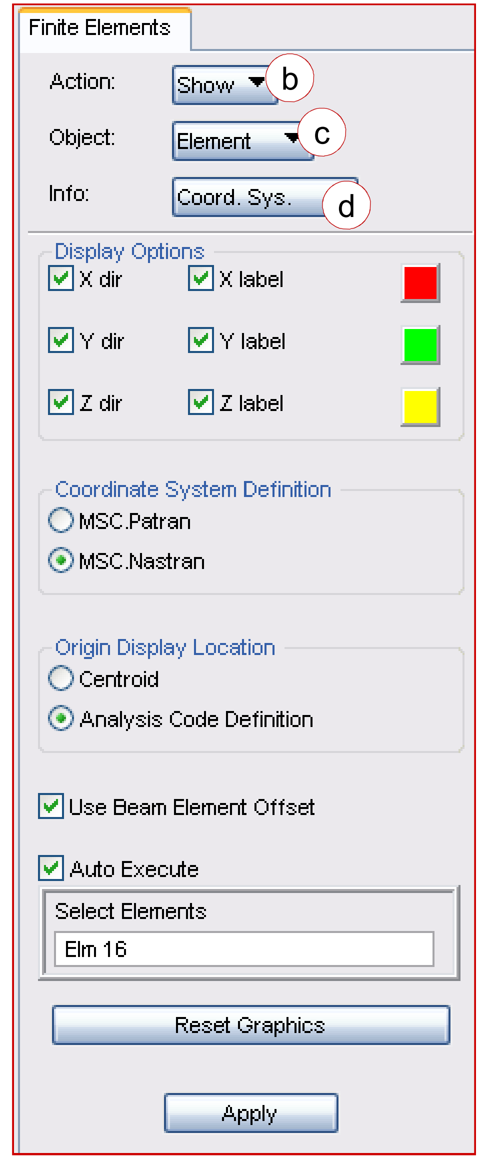


Step 3. Verify Element Coordinate System

Verify the element coordinates systems:

a. Click the **Meshing** tab.
b. Set *action* to **Show.**
c. Set *Object* to **Element**.
d. Set *Info* to **Coord. Sys**.
e. Click on any element to review the element coordinate system.
f. Click on a few more elements to review the element coordinate system. Make a mental note of the element Z direction.
g. Click **Reset Graphics**.

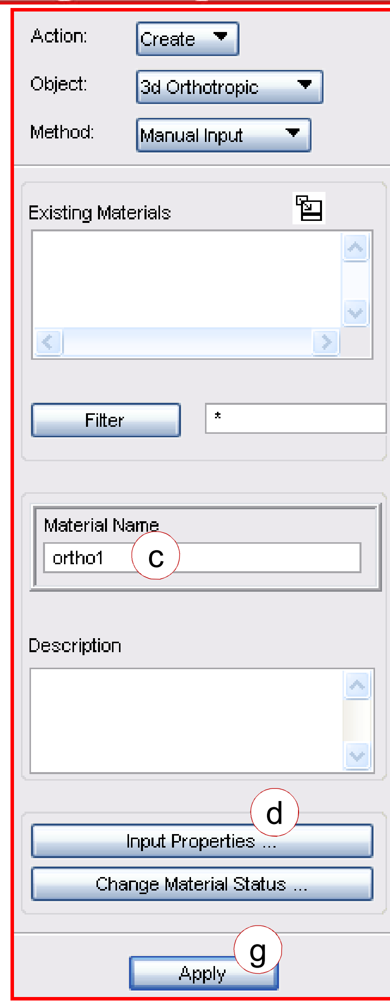


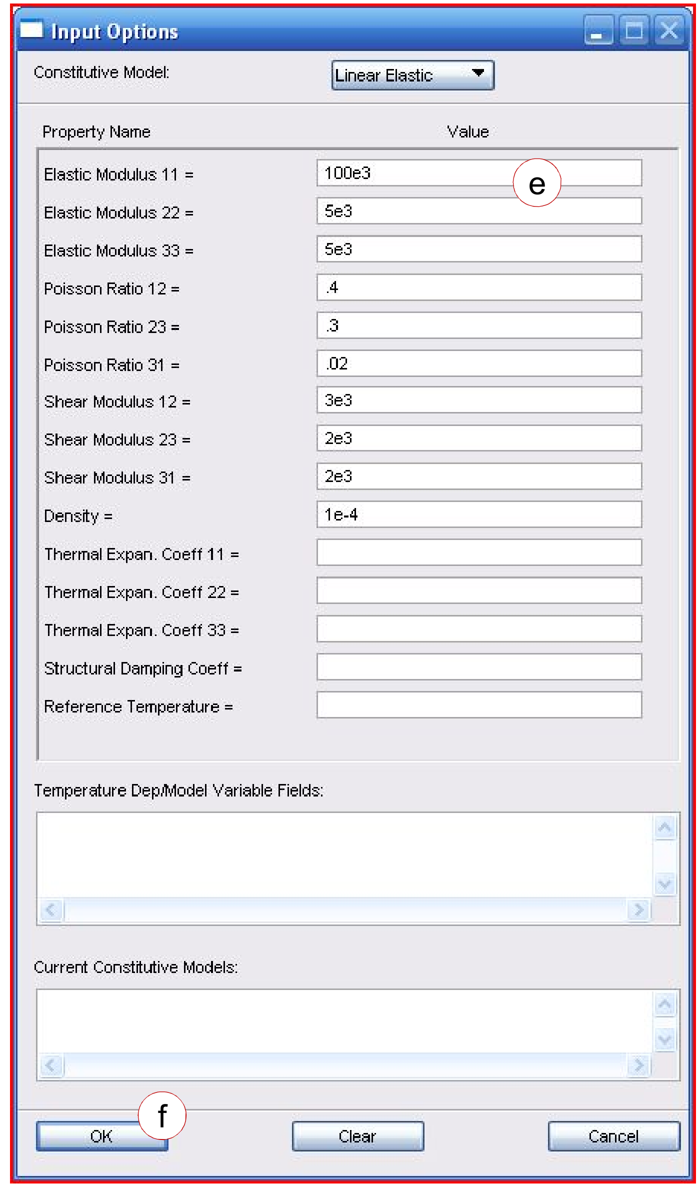


Step 4. Create Material Property

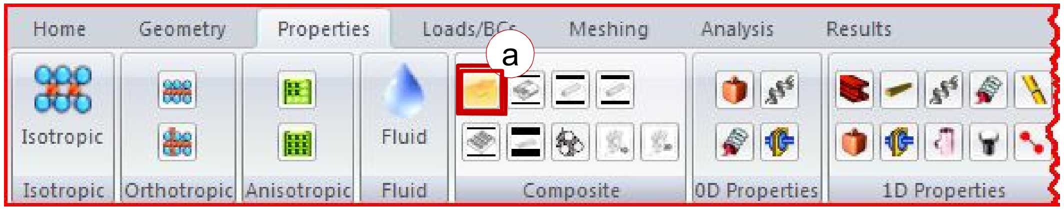
Create a material property:

a. Click the **Properties** tab.
b. Click **3D** in the Orthotropic group.
c. Name the Material **ortho1**
d. Click on **Input Properties**
e. Enter the values shown.
f. Click **OK**.
g. Click **Apply**.



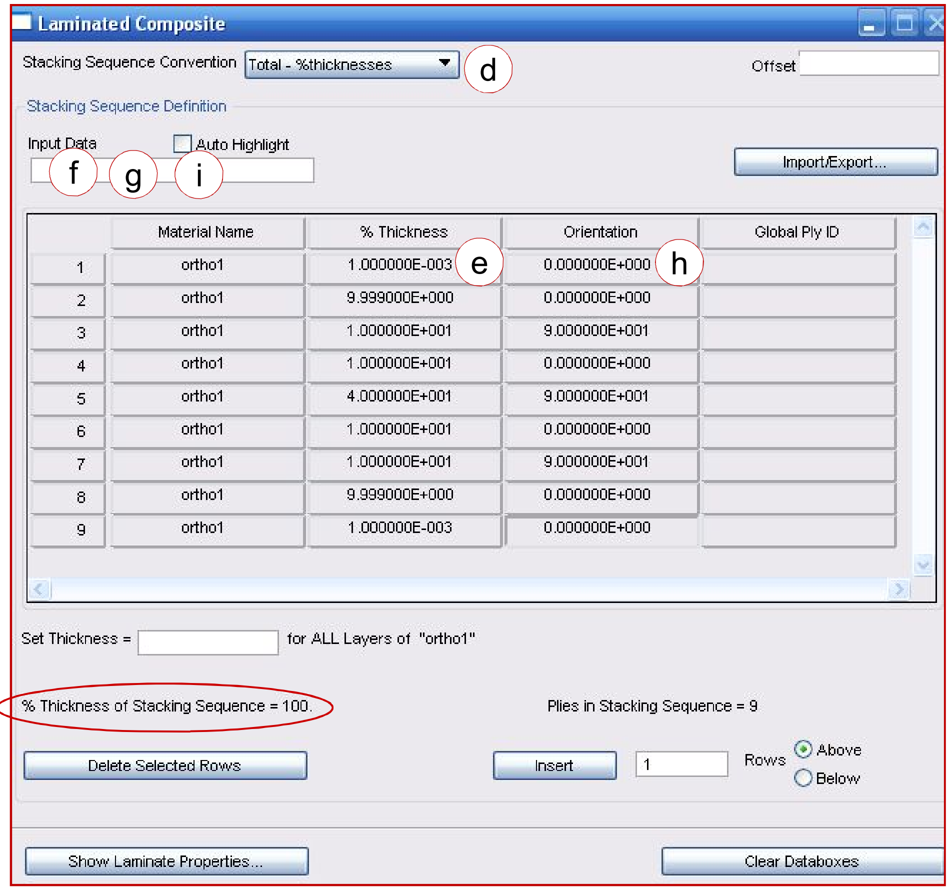


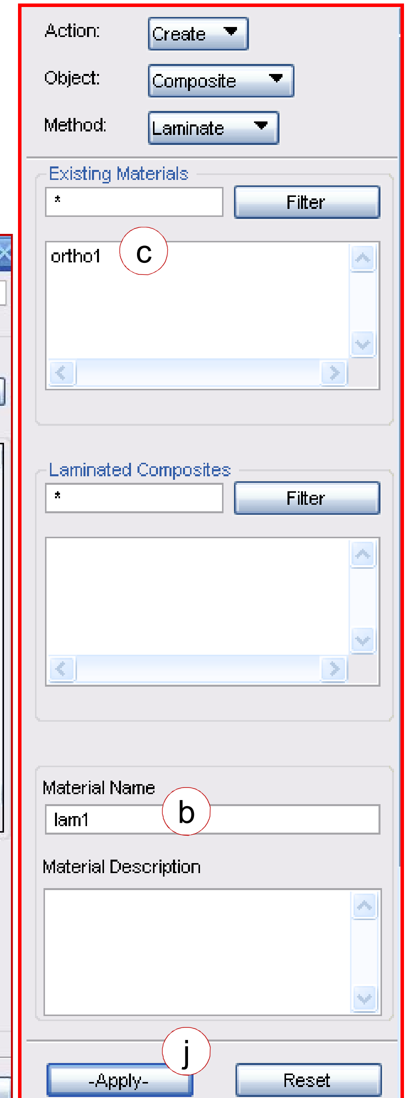
Step 5. Define the Composite Layup



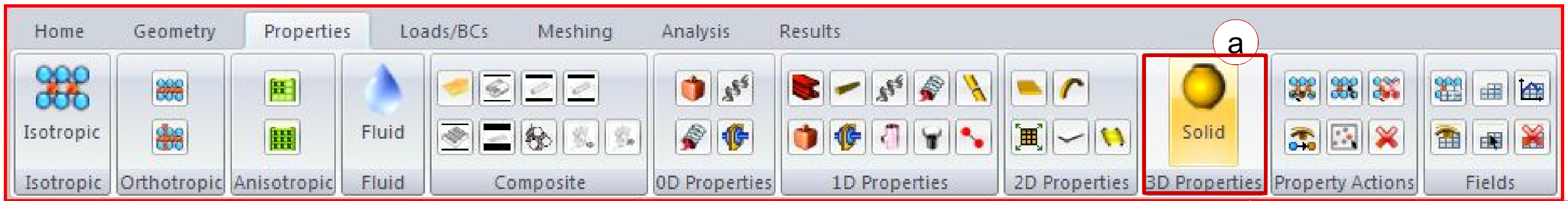
Create a Laminated Composite:

a. In the Properties tab, click **Laminate** in the Composite group.
b. Name the Material **lam1**.
c. Click on **ortho1** 9 times to create 9 rows in the table
d. Select "**Total-%thickness**" from the Stacking Sequence Convention
e. Click in the first %thickness cell
f. Click in the Input Data field and enter **0.001** and press the Enter key.
g. Repeat step **f** to enter all ply % thickness values as shown on the right. Make sure the total percentage is 100 when you are done.
h. Click in the first Orientation cell
i. Enter the ply angles one at a time as shown.
j. Click **Apply**.



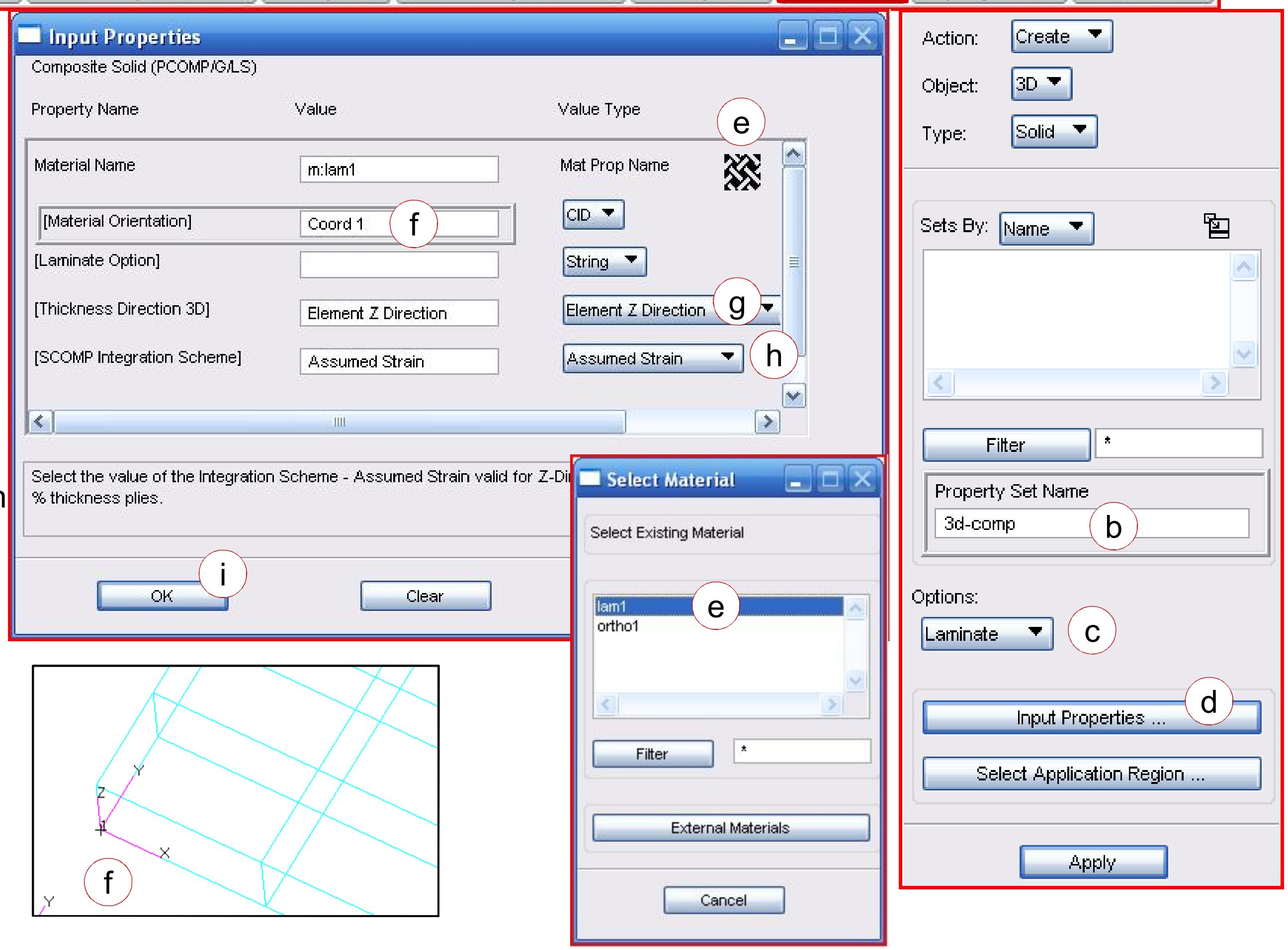


Step 6. Create Solid Property



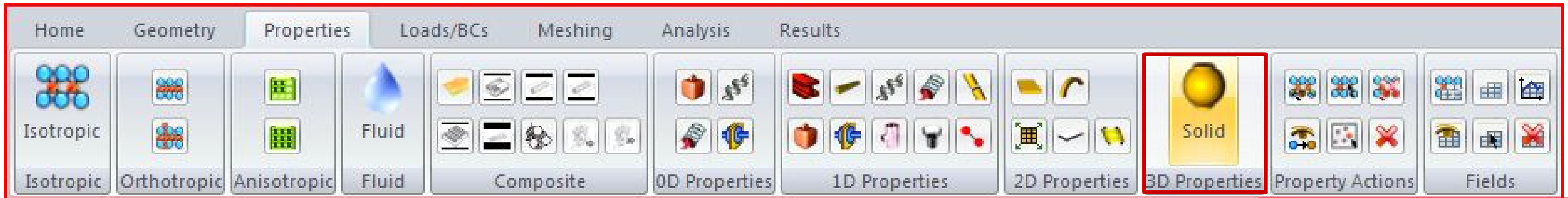
Create 3D solid property:

a. In the Properties tab, click **Solid** in the 3D Properties group. (you may have to move the Laminated Composite form out of the way).
b. For *Property Set Name,* enter **3d-comp**.
c. Change Options to **Laminate**.
d. Click **Input Properties**…
e. Click Material Prop Name icon and select **lam1**.
f. Click in the Material Orientation box, then screen select **Coord 1**.
g. For Thickness Direction 3D, select **Element Z Direction**.
h. In SCOMP Integration Scheme, select **Assumed Strain**.
i. Click **OK**.



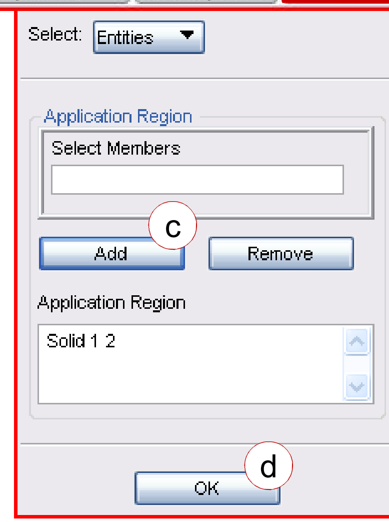
MSC Software

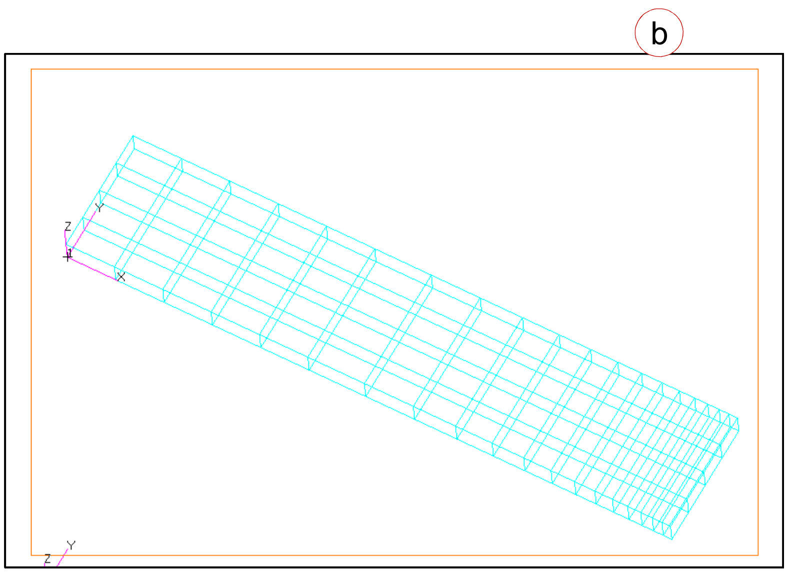
Step 6. Create Solid Property (Cont.)



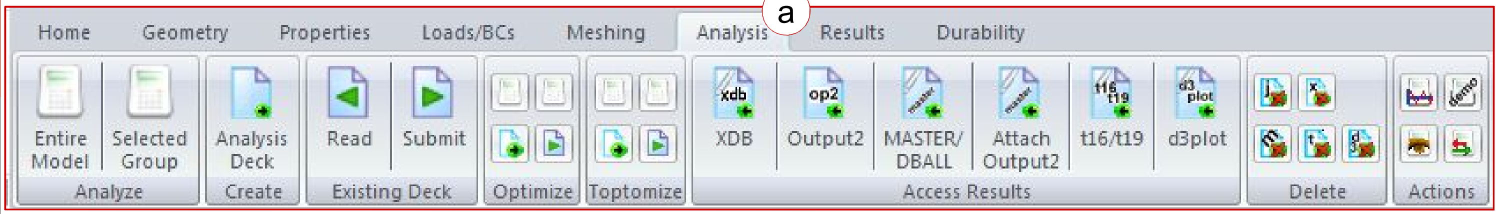
Create 3D solid property:

a. Click **Select Application Region …**
b. Rectangle select both solids.
c. Click **Add**.
d. Click **OK**.
e. Click **Apply**.



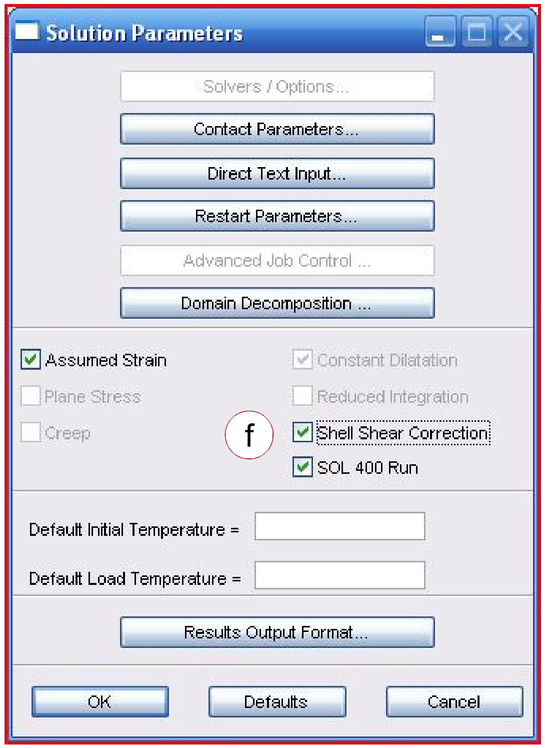


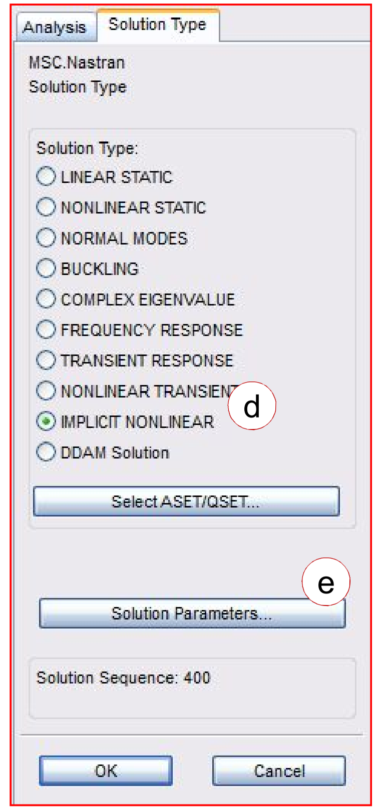
Step 7. Set up Analysis

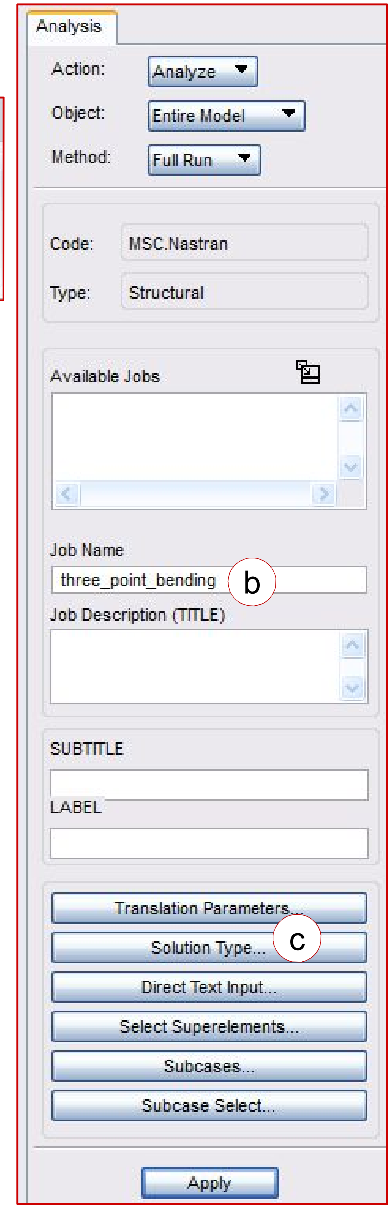


Select solution type:

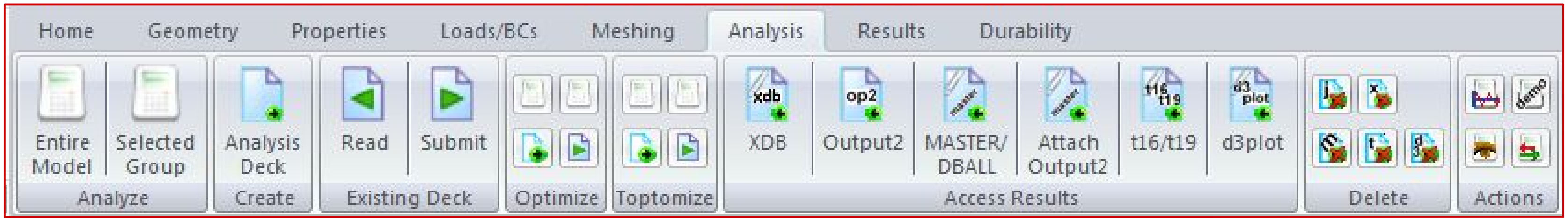
a. Click the **Analysis** tab.
b. Enter **three_point_bending** for Job Name.
c. Click on **Solution Type**.
d. Select **Implicit Nonlinear.**
e. Click **Solution Parameters**.
f. Check **Shell Shear Correction** and **SOL 400 Run**.







Step 7. Set up Analysis (Cont.)

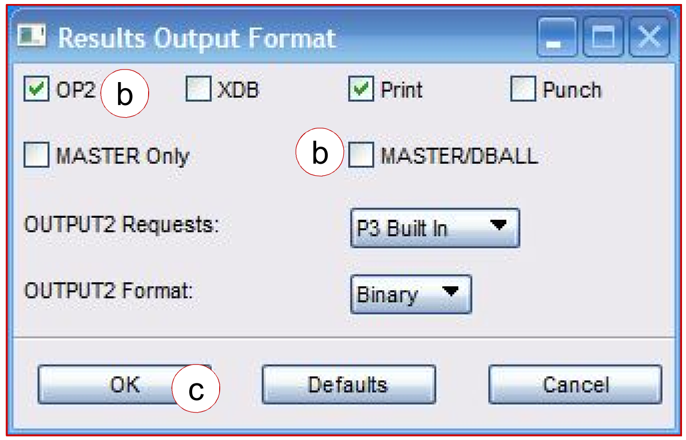


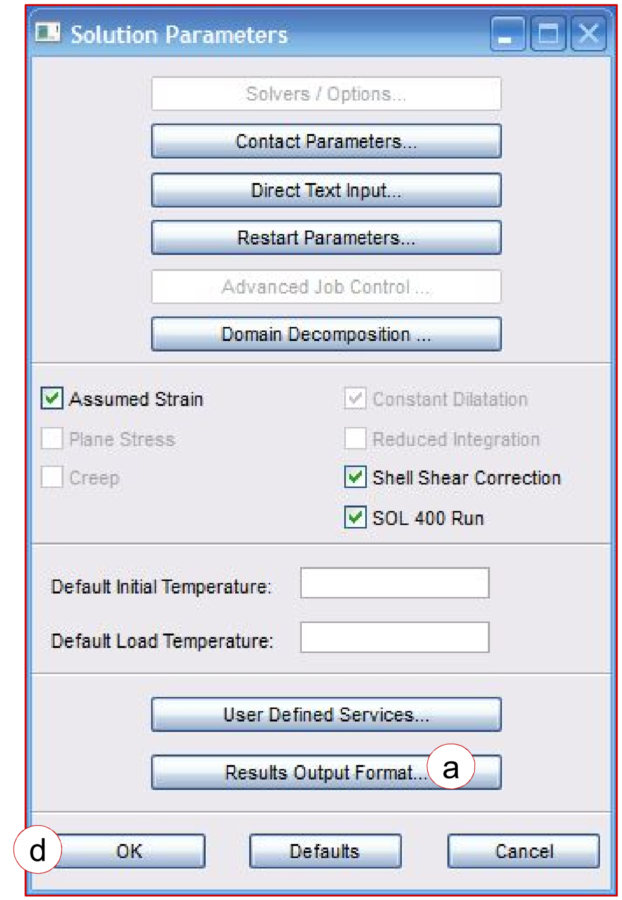
a. Select **Results Output Format….**

b. Check **OP2** and uncheck **MASTER/DBALL**.

c. Click **OK**.

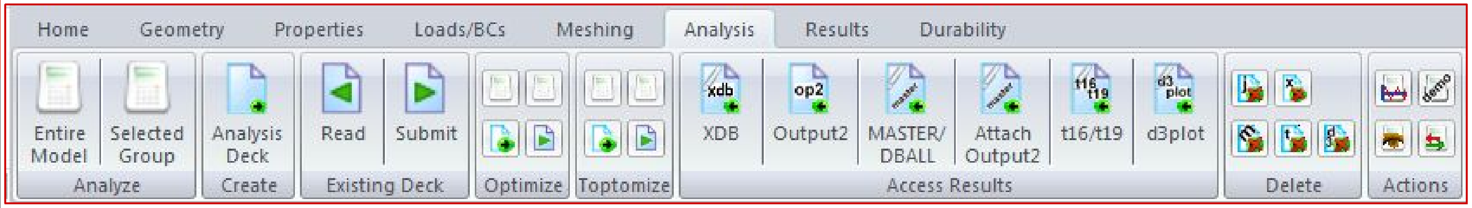
d. Click **OK**.





MSC Software

Step 7. Set up Analysis (Cont.)

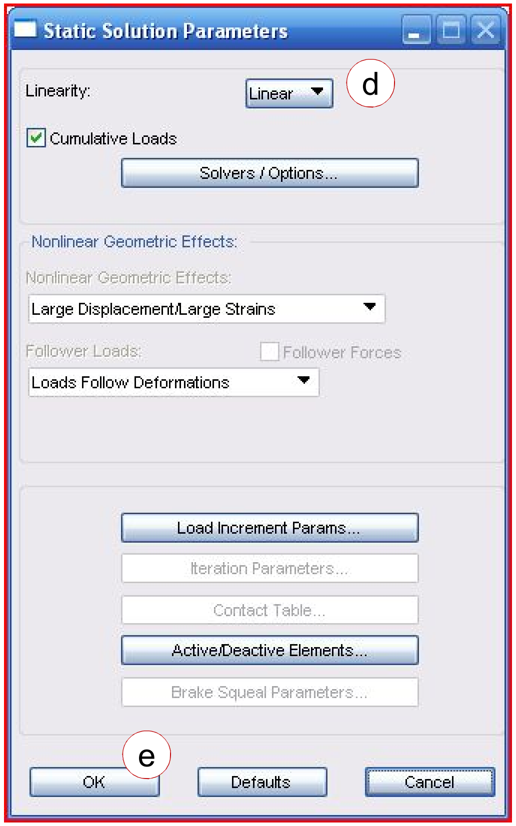


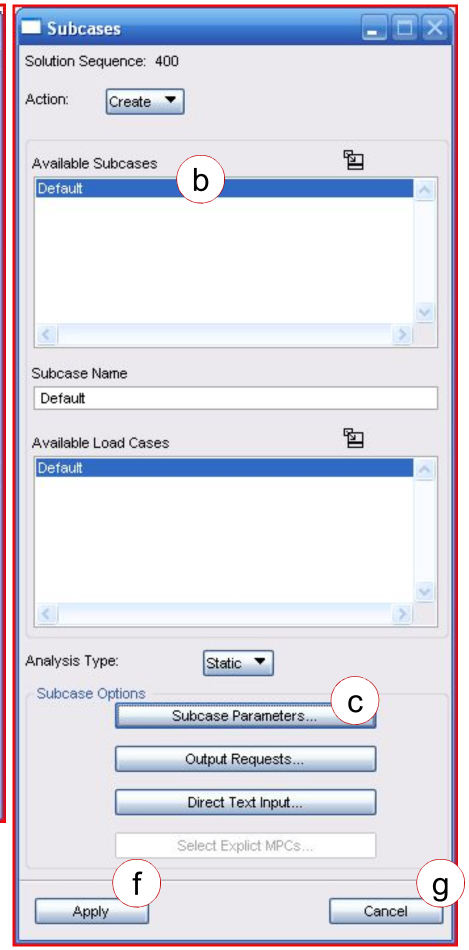
Define subcase parameters:

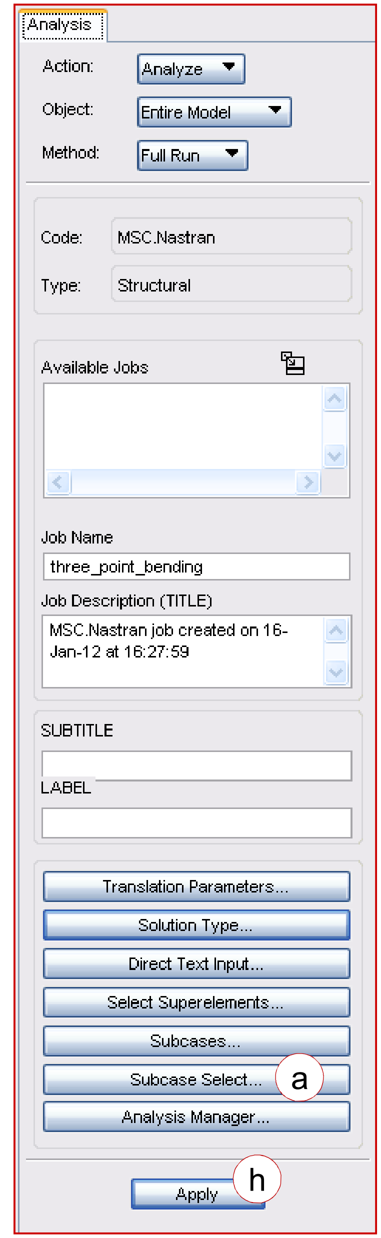
a. Click **Subcases**.
b. Select **Default**
c. Click **Subcase Parameters.**
d. Set Linearity to **Linear**.
e. Click **OK**.

f. Click **Apply**.
g. Click **Cancel**.
h. Click **Apply**.

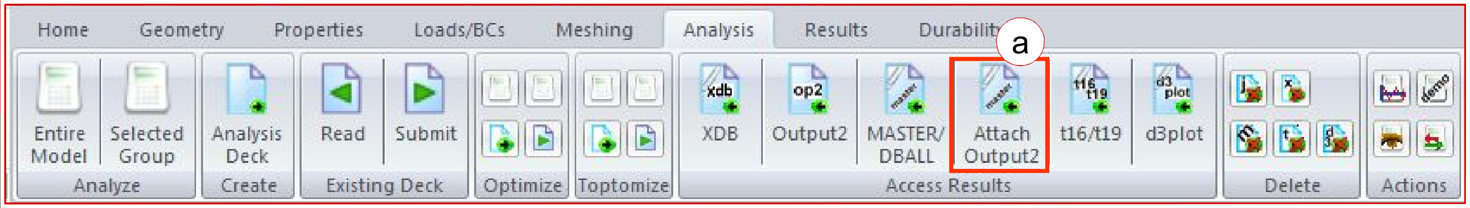
The analysis will take less than a minute to run.





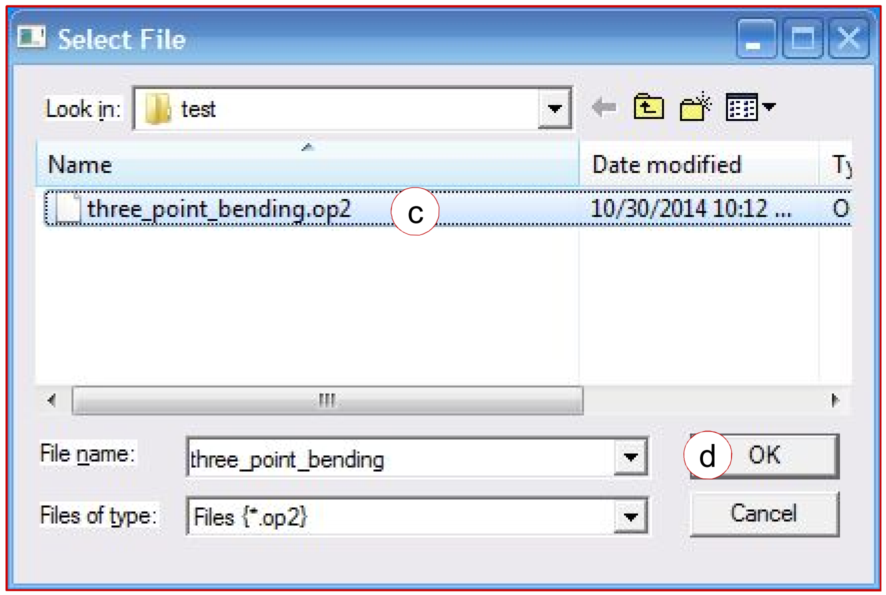


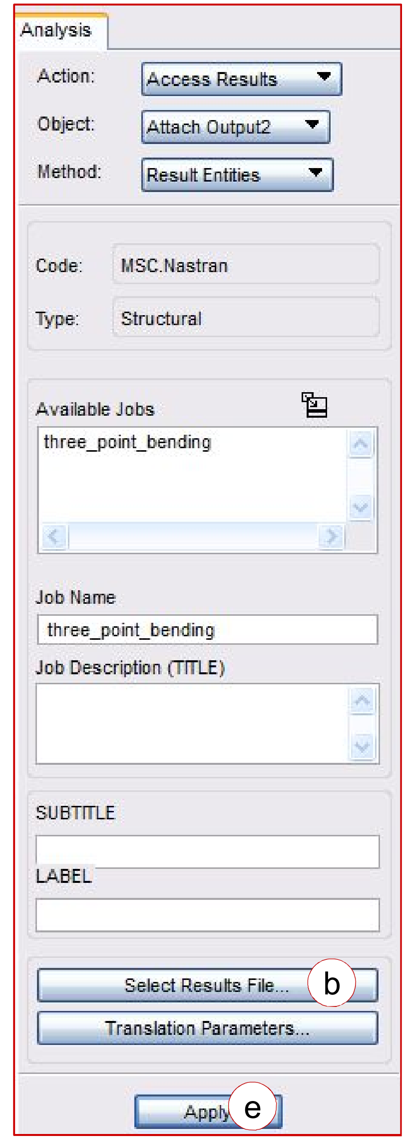
Step 8. Attach Results



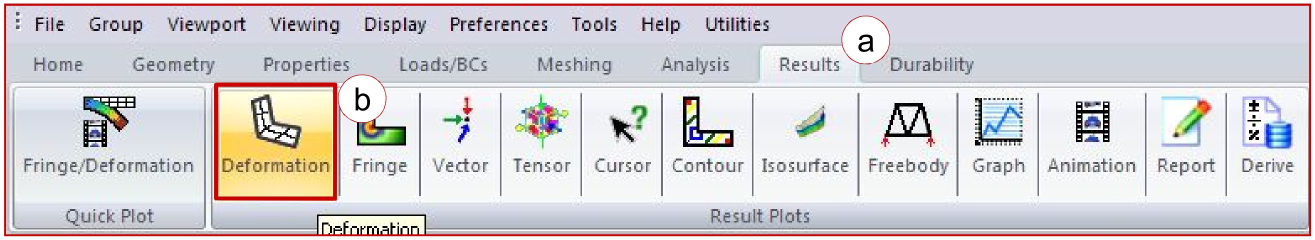
Attach the results file:

a. Click **Attach Output2** in the Access Results group.
b. Click on **Select Results File.**
c. Select the results file **three_point_bending.op2**.
d. Click **OK**.
e. Click **Apply**.





Step 9. Plot Results



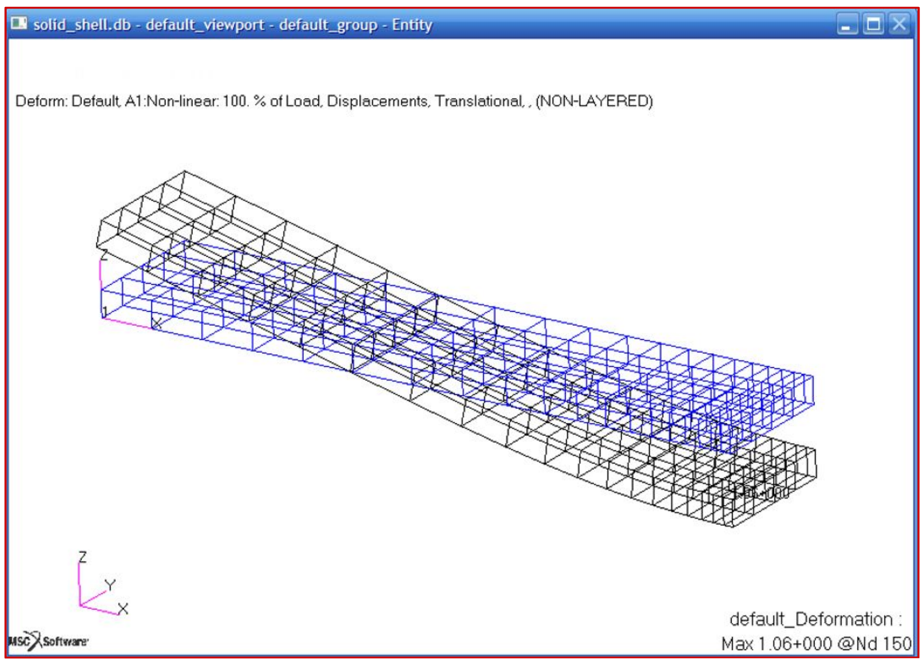
Plot deformation:

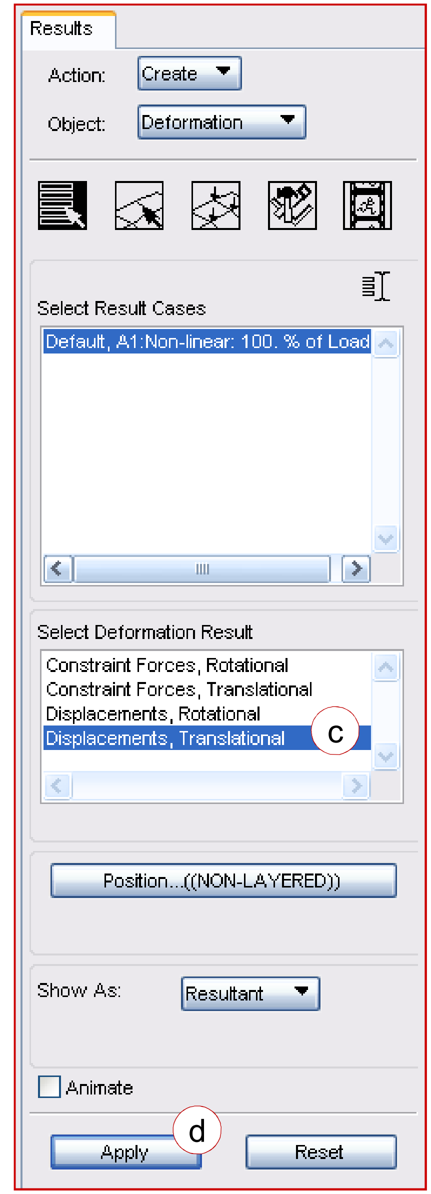
a. Click the **Results** tab.

b. Click the **Deformation** icon.

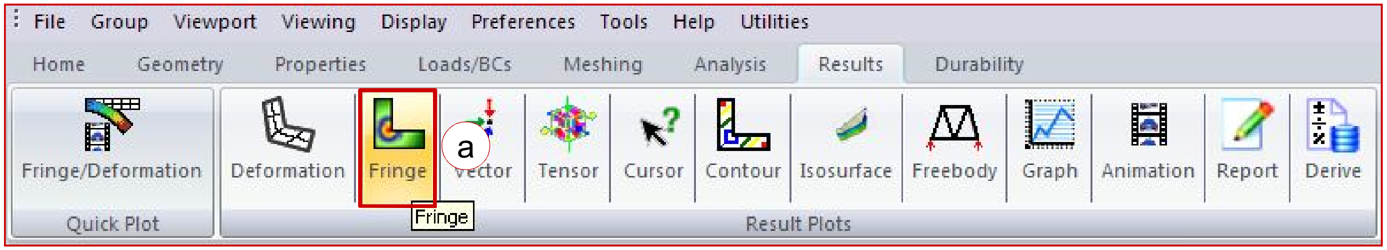
c. Select **Displacement, Translational** for *Deformation Result*.

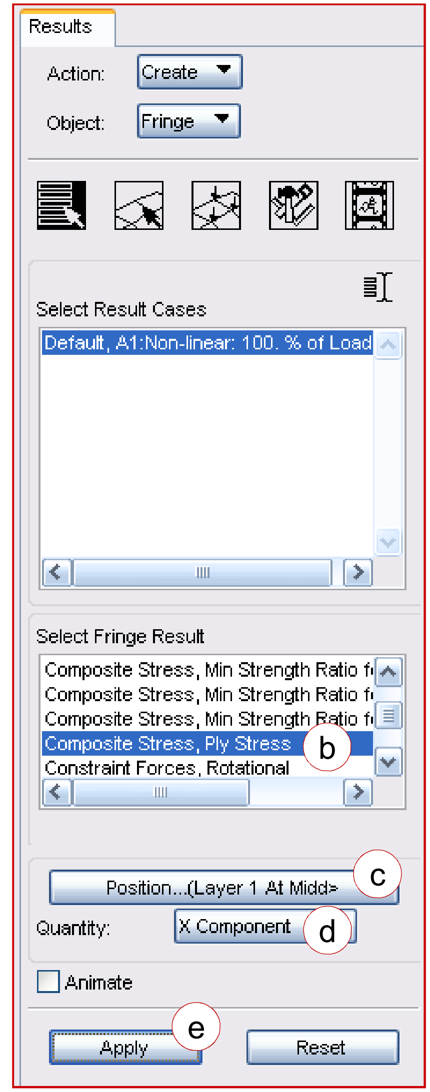
d. Click **Apply**.





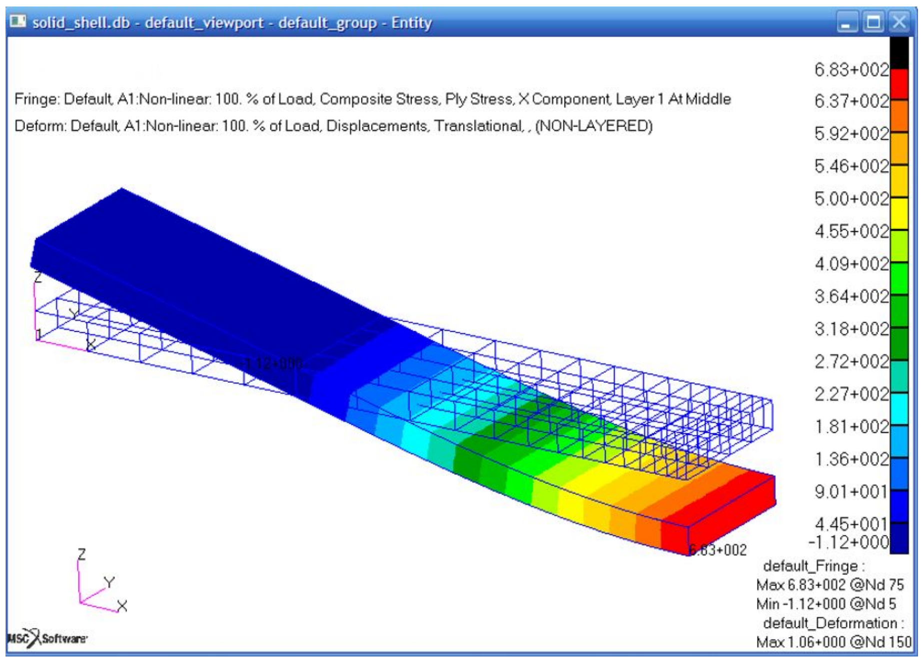
Step 9. Plot Results (cont.)



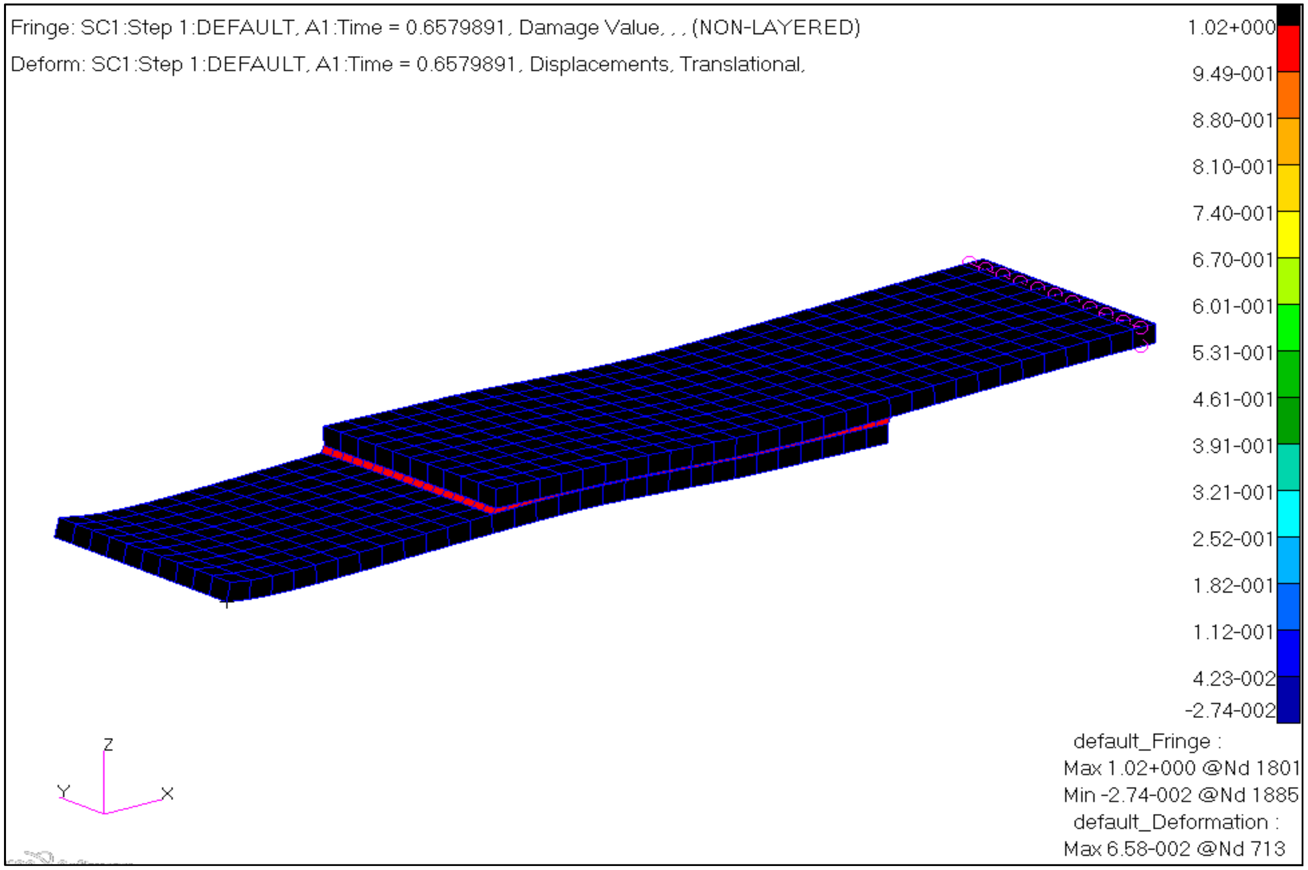


Plot stresses:

a. Click the **Fringe** icon.
b. Select **Composite Stress, Ply Stress** for *Fringe Result*.
c. Set *Position* to **Layer 1 at middle**.
d. Set *Quantity* to **X Component**.
e. Click **Apply**.



WORKSHOP 8
DELAMINATION OF A COMPOSITE SOLID SHELL BEAM



- **Workshop Objectives**
  - Perform a CZM delamination analysis.
  - Create contact pairs.
  - Create cohesive material
  - Create interface elements.

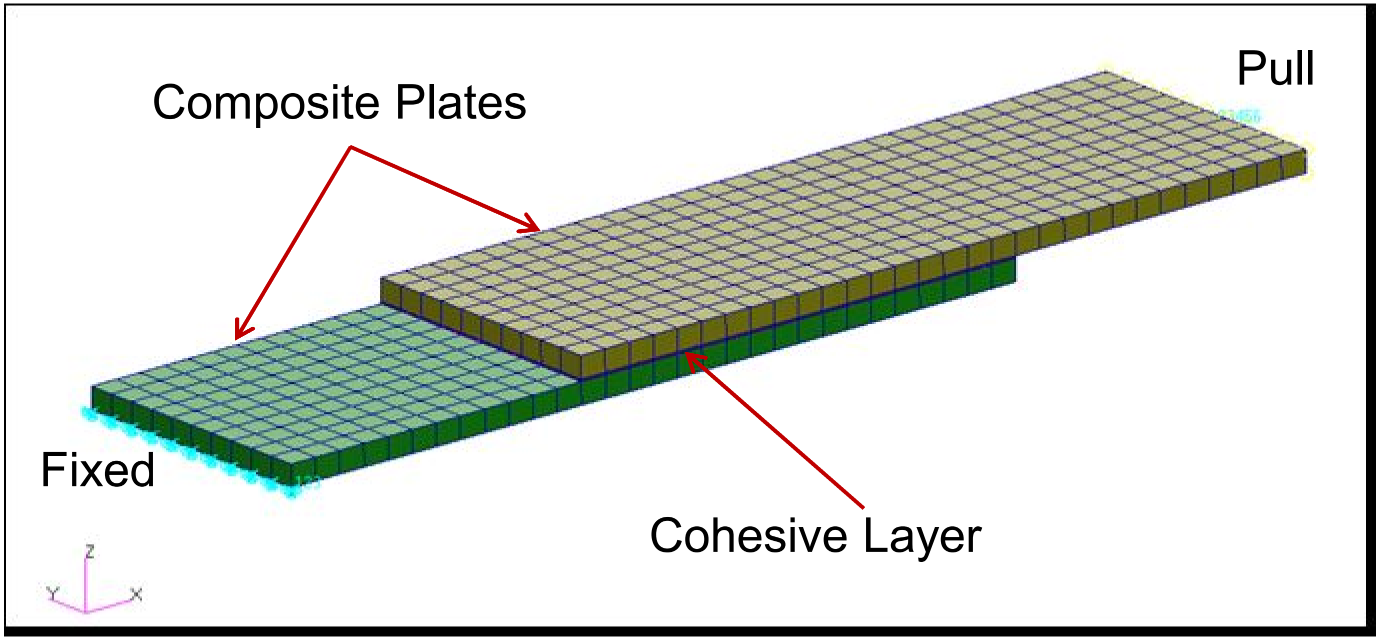
- **Software Version**
  - Patran 2013
  - MSC Nastran 2013.1

- **Required File:**
  - **build_shear.czm.ses**

- **Problem Description**
  1. Two composite plates are glued together with a adhesive layer.
  2. One end of the plates is fixed and pulling on the other end.
  3. The exponential cohesive material model is used the interface elements.

G_c = 3.0 in-lb/in^2

V_c = 0.0002

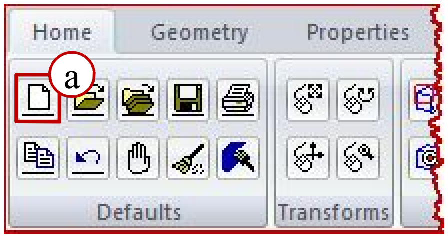


- **Suggested Exercise Steps**
  1. Create a new database.
  2. Play session file "**build_shear_czm.ses**."
  3. Review the model boundary conditions with the model tree.
  4. Review the contact bodies in the model.
  5. Create the composite material and layup.
  6. Create the contact pairs.
     - Check Glued Contact
     - Master = **plate_lower**
     - Slave = **interface**
     - Repeat for the upper plate
  7. Create cohesive material.
     - Select exponential for the Constitutive Model
     - Enter **3** for Cohesive Energy
     - Enter **0.0002** for displacement
  8. Create cohesive interface element property for all wedge elements.
  9. Set up the analysis job.
     - Implicit Nonlinear for Solution Type
     - Uncheck Permanent Gluing
     - Request op2

- **Suggested Exercise Steps (Cont.)**
    - Use an adaptive increment type with a maximum time step of **0.05**
    - Output Nonlinear Stresses

10. Review the sts file.
11. Attach op2 results file.
12. Plot interface damage.
    - Set all time increments to display Element Edges and show True Scale
    - Plot damage at Time = 0.6579891
13. Graph load vs. x-deflection curve of the independent node of the RBE2s to view the constraint forces at all time increments.

Step 1. Create a New Database



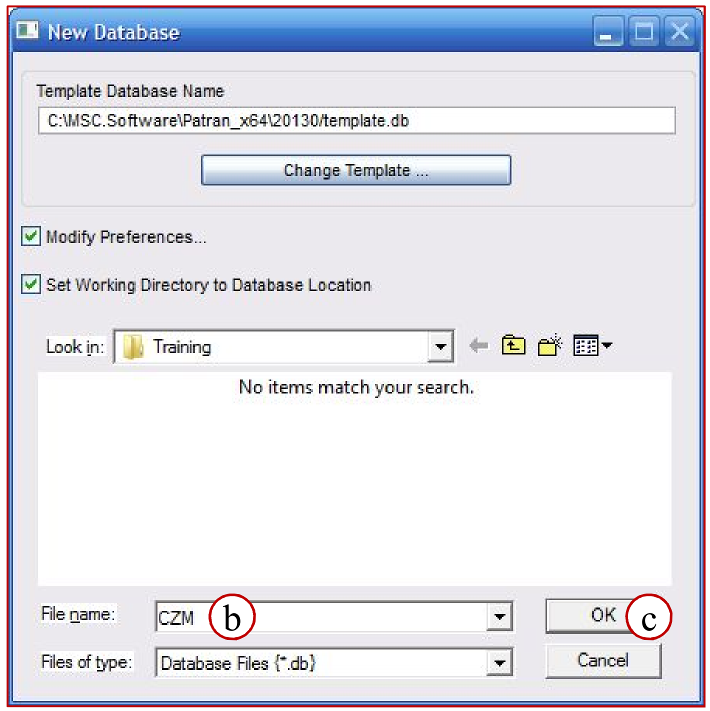
Create a new database called **CZM**:

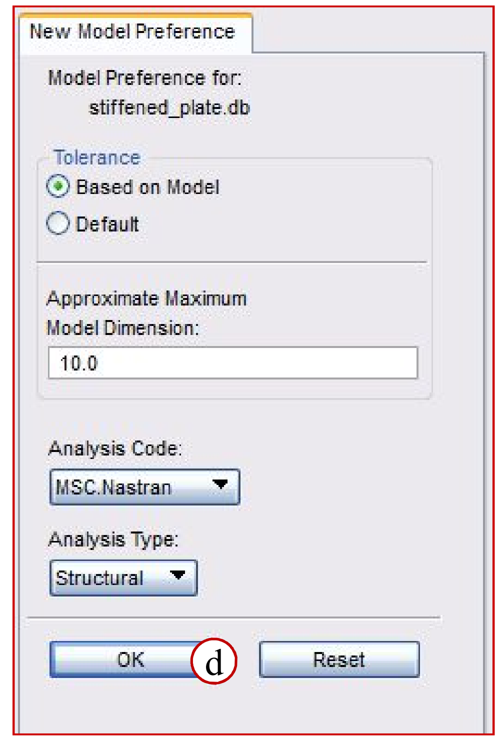
a. Under the *Home* tab, click **New** in the *Defaults* group.

b. Enter **CZM** as the *File name*.

c. Click **OK**.

d. For the *New Model Preference*, accept all defaults by clicking **OK**.





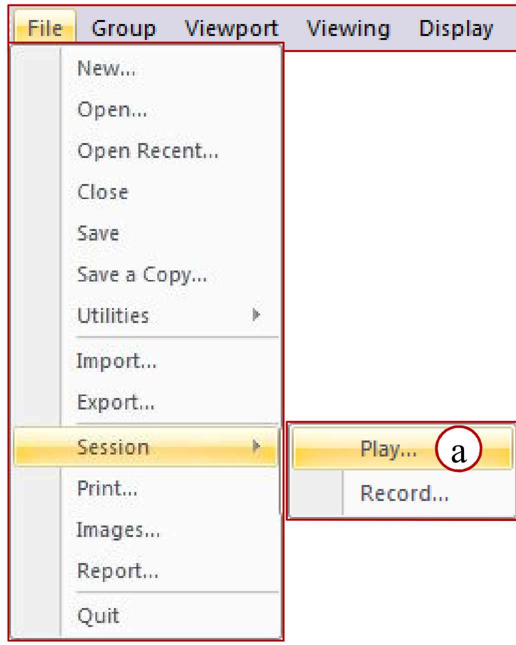
Step 2. Play Session File

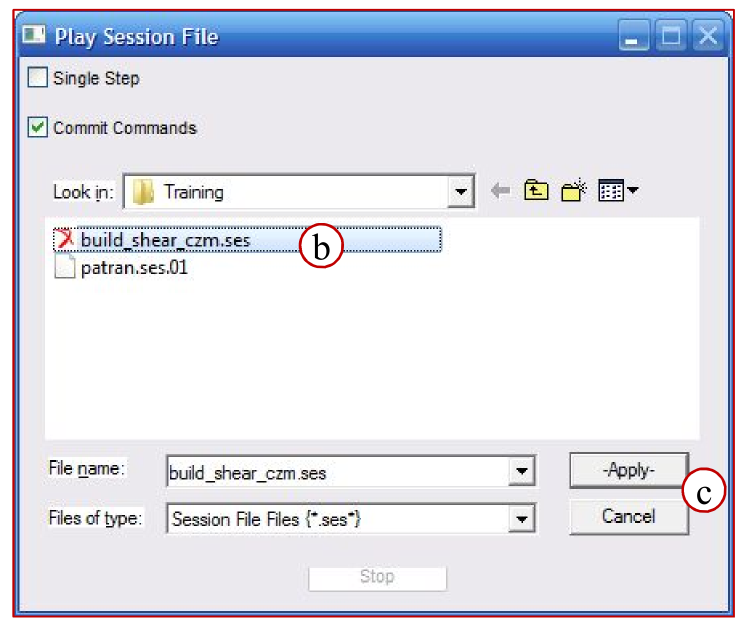
Play session file:

a. Select **File** > **Session** > **Play**.

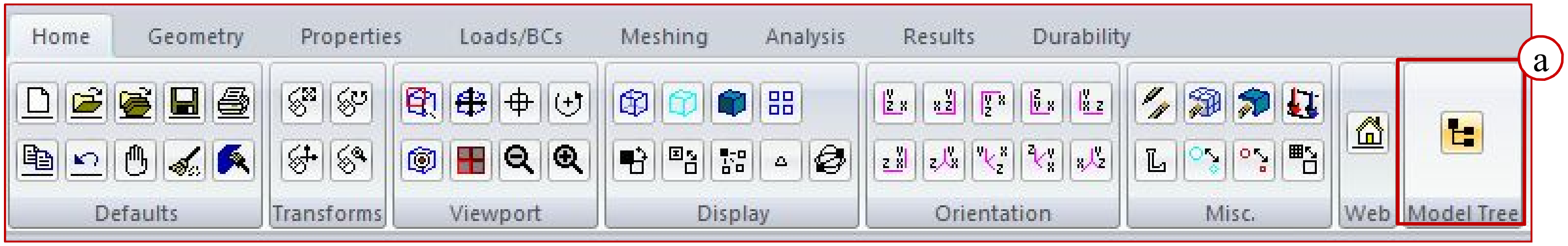
b. Select **build_shear_czm.ses**.

c. Click **Apply**.



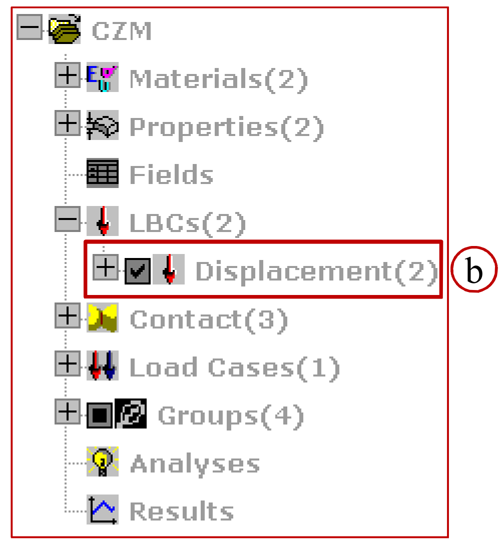


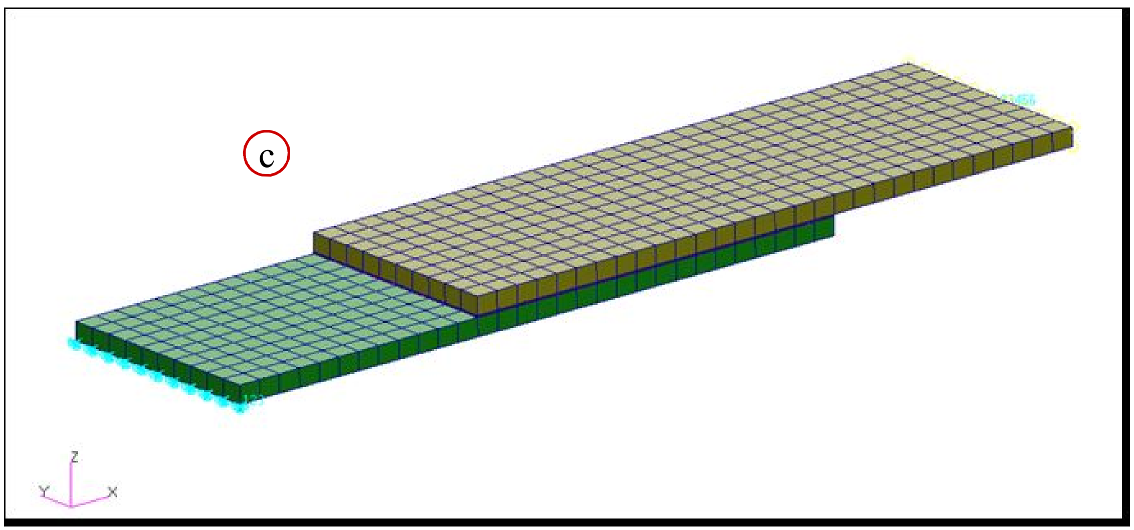
Step 3. Review the Model Boundary Conditions



Review the model boundary conditions with the model tree :

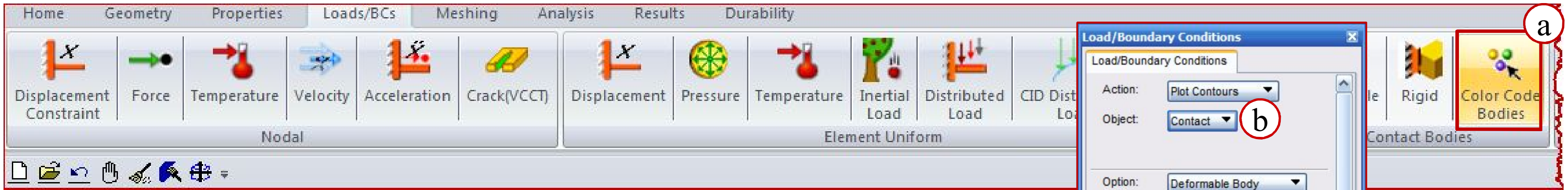
a. Under the *Home* tab, click **Show/Hide Model Tree** in the *Model Tree* group.
b. Expand *LBCs* and check the box next to *Displacements.*
c. Observe the loads and boundary conditions.
d. Uncheck when done reviewing the boundary conditions.

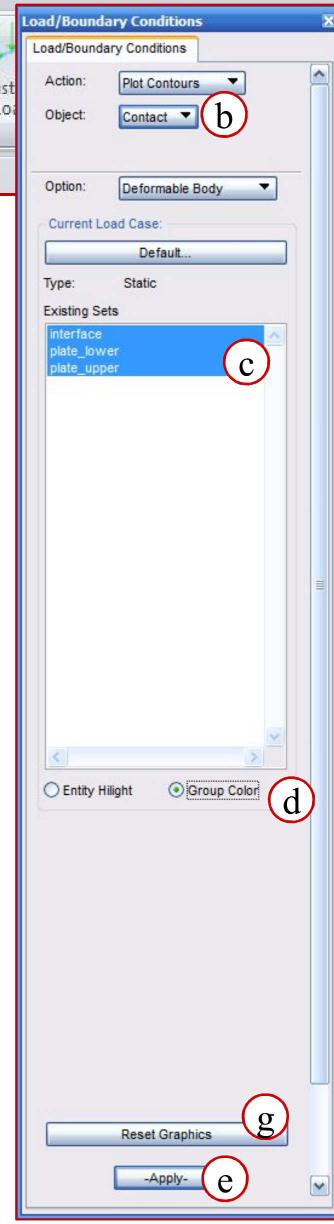




MSC Software

Step 4. Review the Model Contact Bodies





Review the contact bodies:

a. Under the *Load/BCs* tab, click **Color Code Bodies** in the *Contact Bodies* group.

b. Set *Object* to **Contact**.

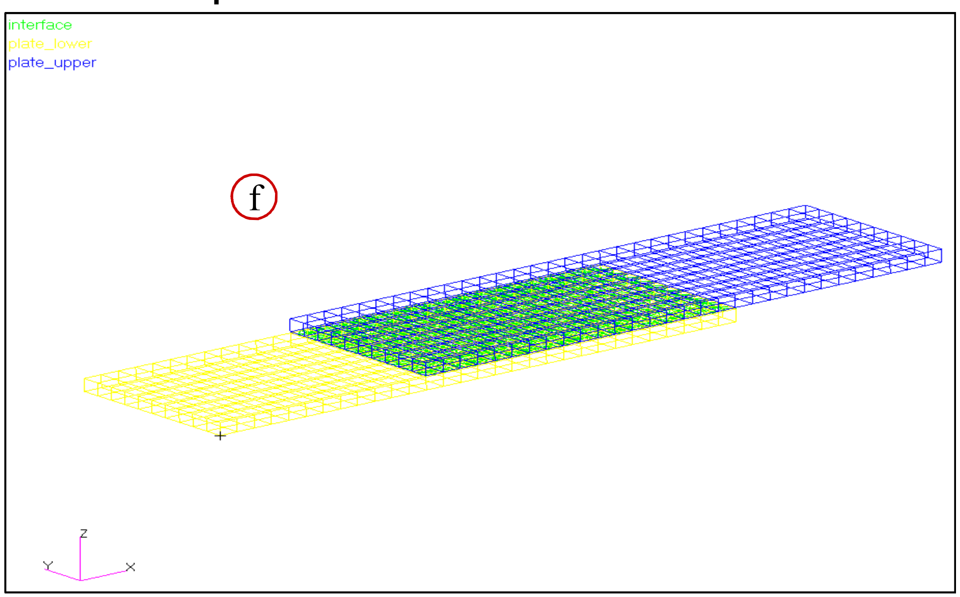
c. Select all the *Existing Sets*.

d. Select **Group Color**.

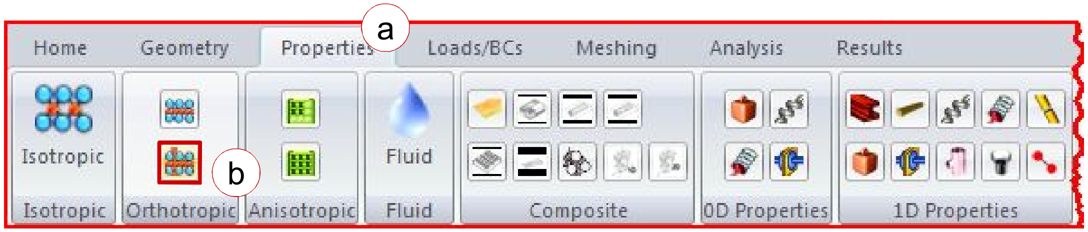
e. Click **Apply**.

f. Observe the contact bodies.

g. Click on **Reset Graphics**.

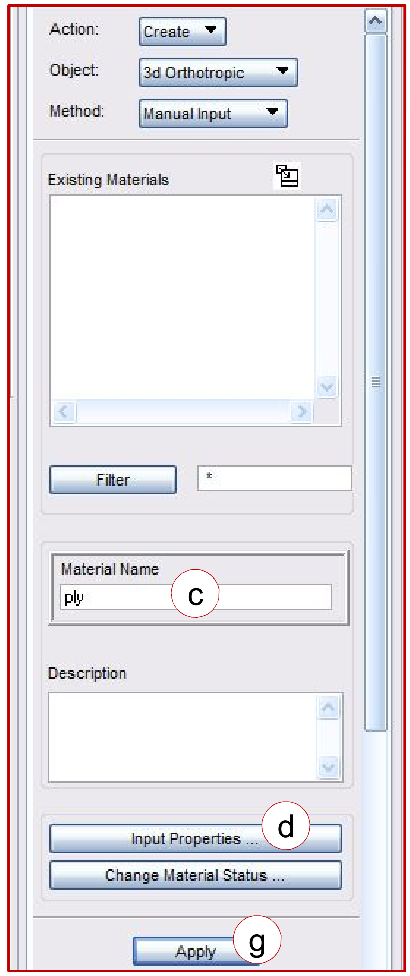


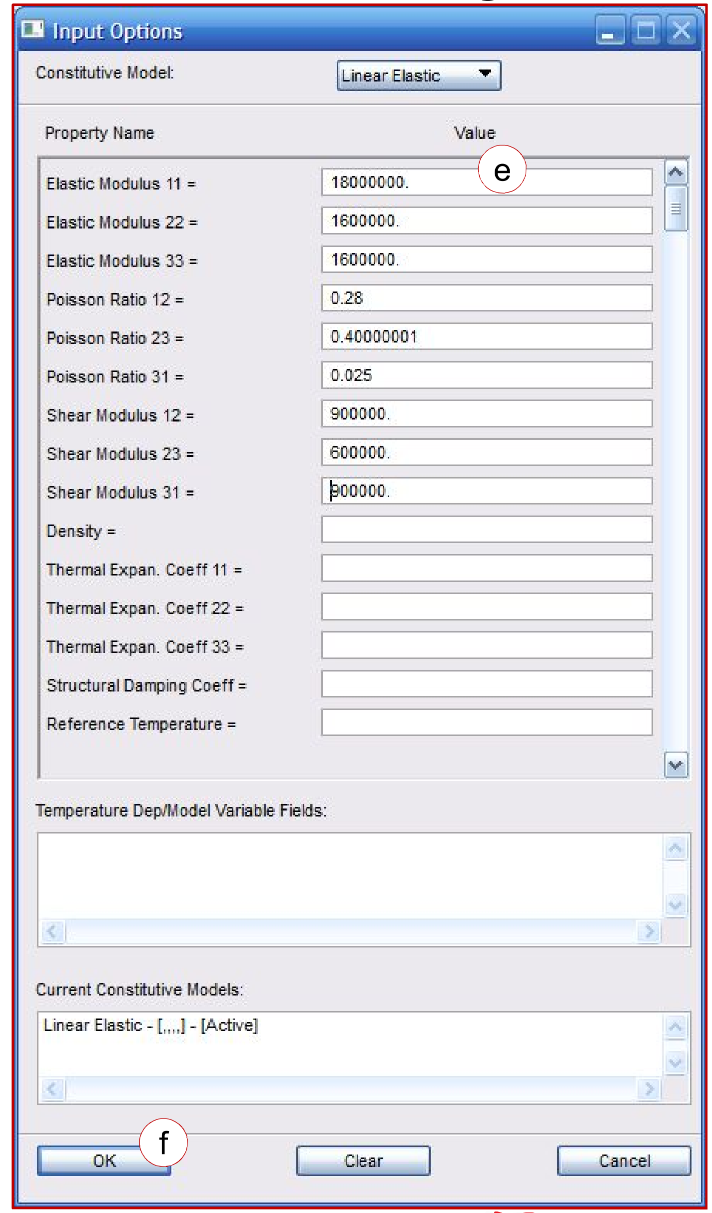
Step 5. Create the Composite Material and Layup



Create a material property:

a. Click the **Properties** tab.
b. Click **3D** in the Orthotropic group.
c. Name the Material **ply**
d. Click on **Input Properties**
e. Enter the values shown.
f. Click **OK**.
g. Click **Apply**.

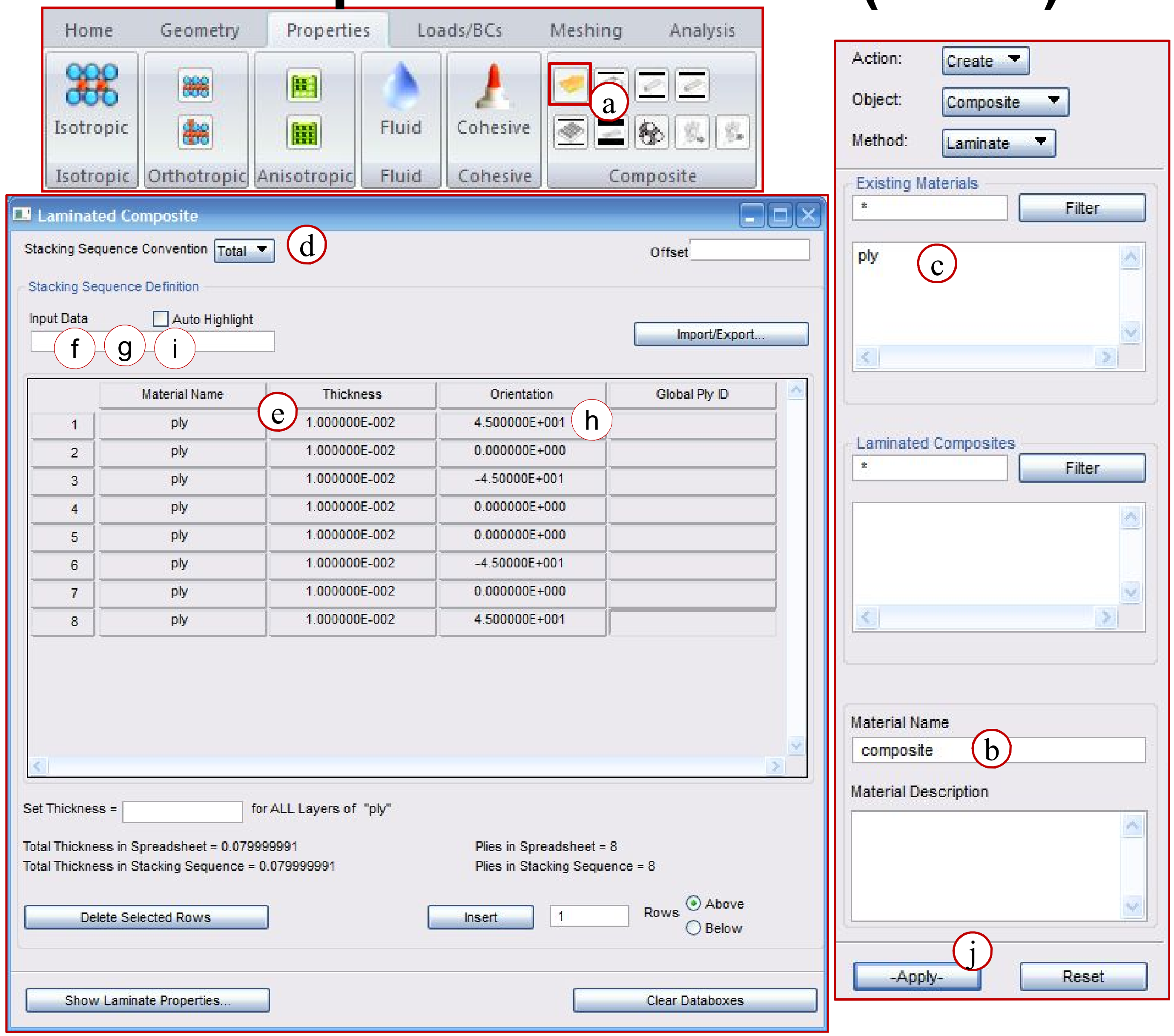




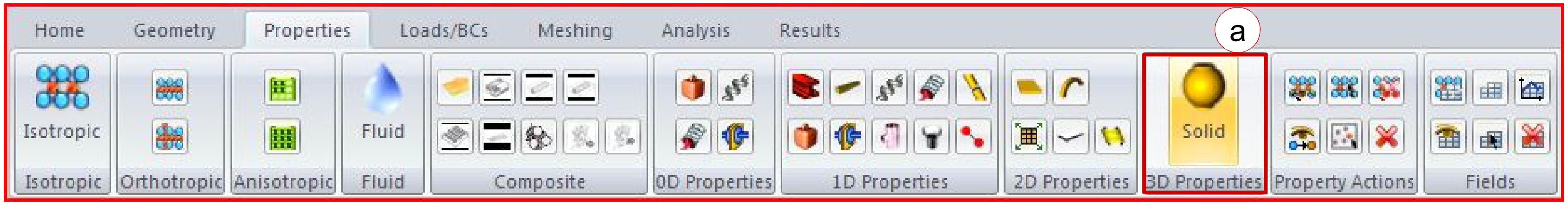
Step 5. Create the Composite Material… (Cont.)

Create a Laminated Composite:

a. In the Properties tab, click **Laminate** in the Composite group.
b. Name the Material **composite**.
c. Click on **ply** 8 times to create 8 rows in the table
d. Select “**Total**” from the Stacking Sequence Convention
e. Click in the first thickness cell
f. Click in the Input Data field and enter **0.01** and press the Enter key.
g. Repeat step **f** to enter all ply thickness values as shown on the right.
h. Click in the first Orientation cell
i. Enter the ply angles one at a time as shown.
j. Click **Apply**.

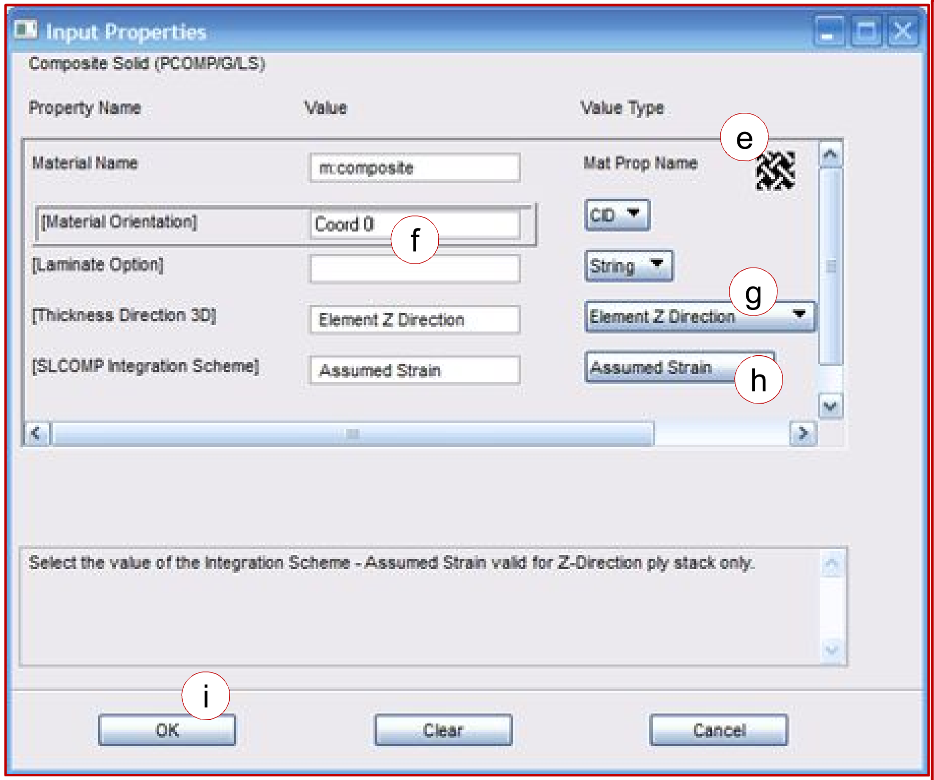


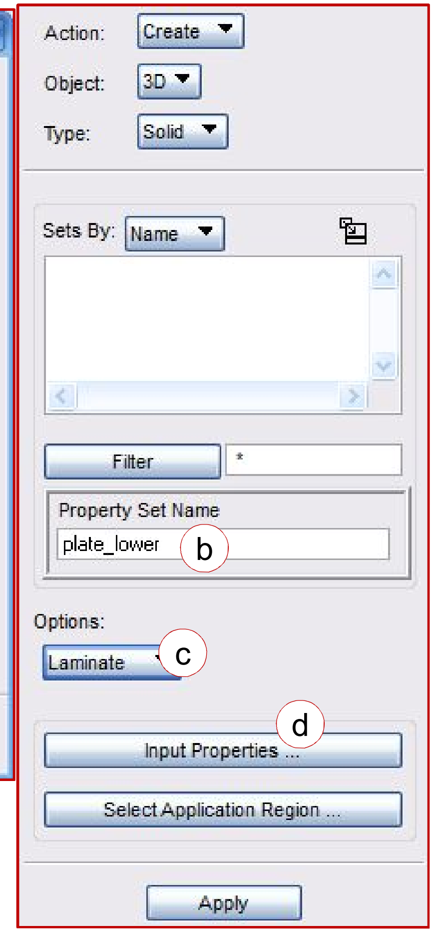
Step 5. Create the Composite Material… (Cont.)

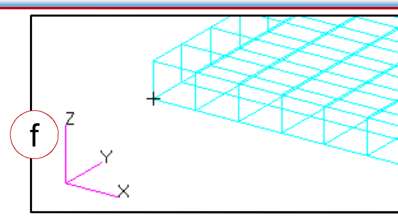


Create 3D solid property:

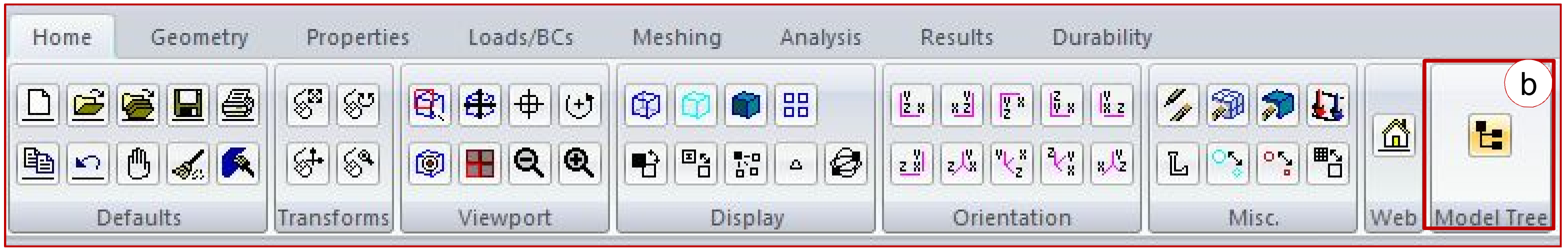
a. In the Properties tab, click **Solid** in the 3D Properties group. (you may have to move the Laminated Composite form out of the way).
b. For *Property Set Name,* enter **plate_lower**.
c. Change Options to **Laminate**.
d. Click **Input Properties**…
e. Click Material Prop Name icon and select **composite**.
f. Click in the Material Orientation box, then screen select **Coord 0**.
g. For Thickness Direction 3D, select **Element Z Direction**.
h. In SCOMP Integration Scheme, select **Assumed Strain**.
i. Click **OK**.





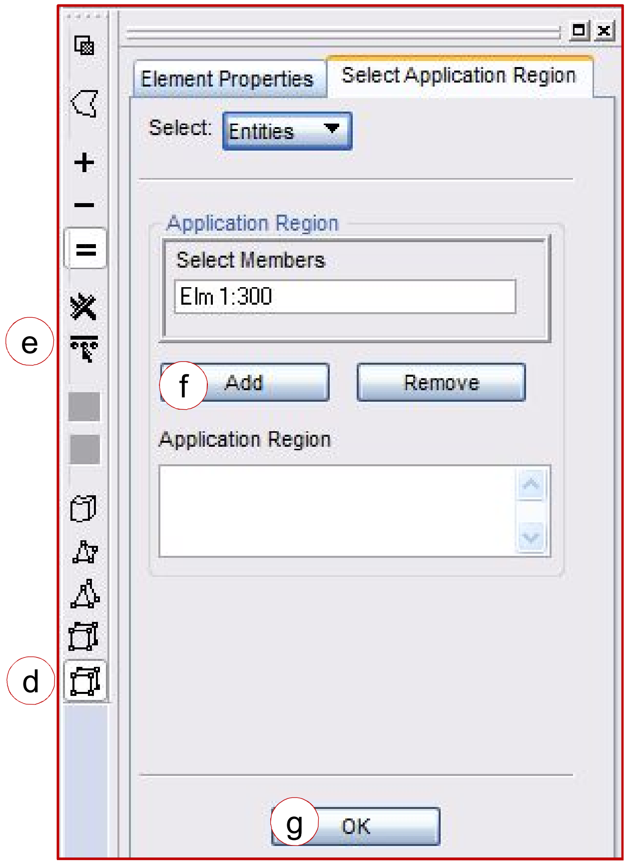


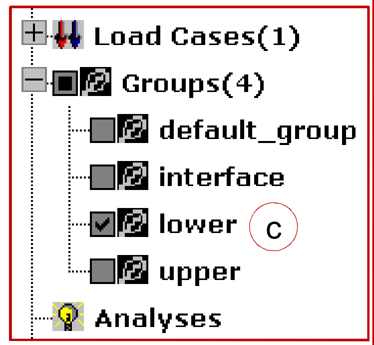
Step 5. Create the Composite Material… (Cont.)

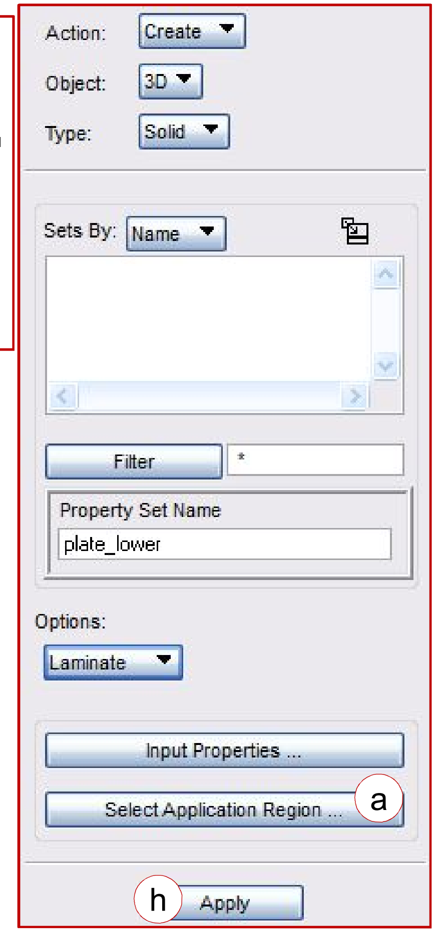


Create 3D solid property:

a. Click **Select Application Region …**
b. Under the *Home* tab, click **Show/Hide Model Tree** in the *Model Tree* group.
c. Expand *Groups* and only check the box next to ***Lower***.
d. Select **Solid element**.
e. Select **Pick All**.
f. Click **Add**.
g. Click **OK**.
h. Click **Apply**.





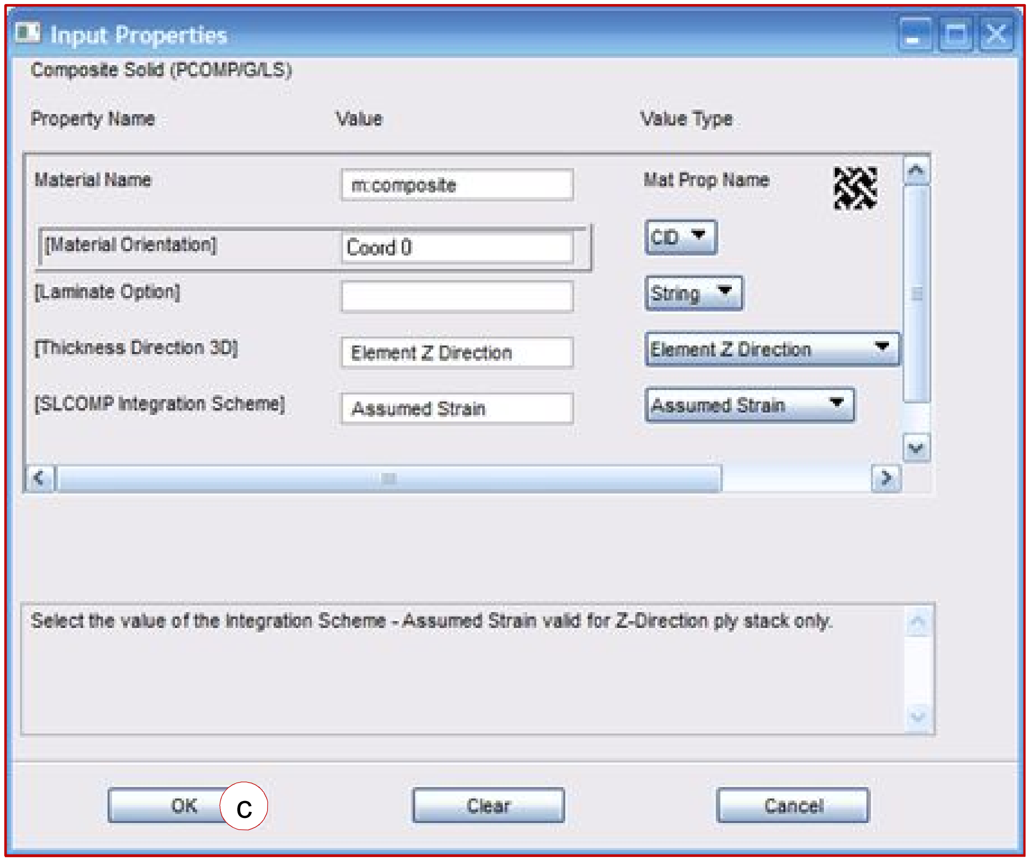


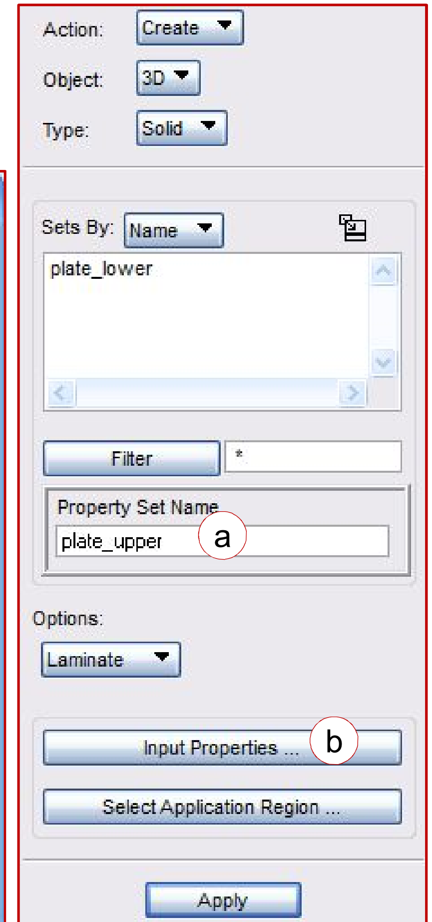
MSC Software

Step 5. Create the Composite Material… (Cont.)

Create 3D solid property:

a. In *Property Set Name,* enter **plate_upper**.
b. Click **Input Properties**…
c. Click **OK** to accept changes from the previous property.

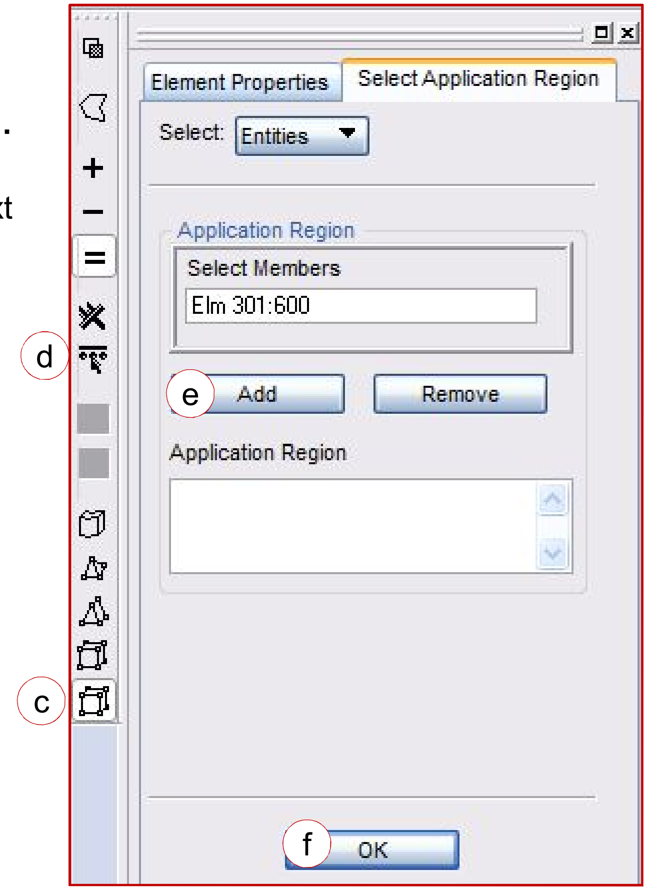


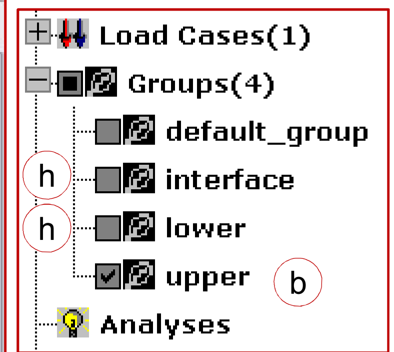


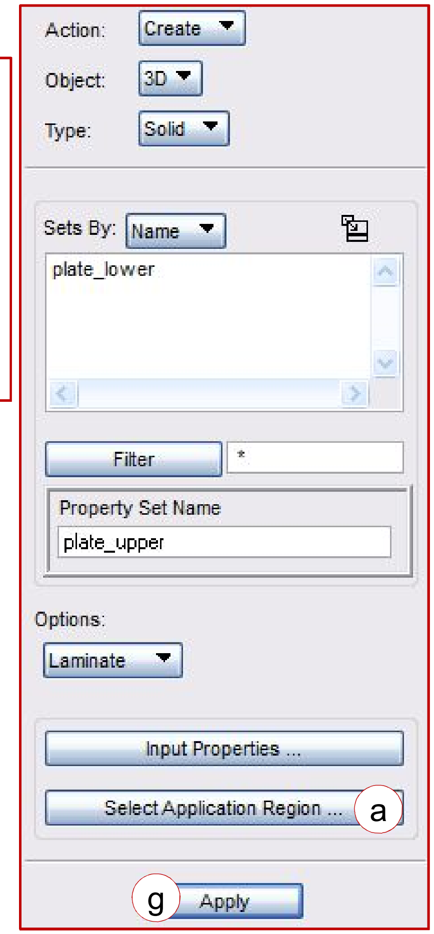
Step 5. Create the Composite Material… (Cont.)

Create 3D solid property:

a. Click **Select Application Region …**
b. Expand *Groups* and only check the box next to ***upper***.
c. Select **Solid element**.
d. Select **Pick All**.
e. Click **Add**.
f. Click **OK**.
g. Click **Apply**.
h. Check the box next to upper and interface

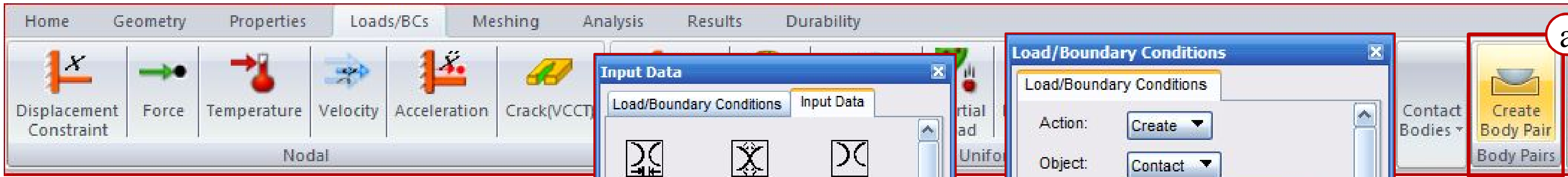


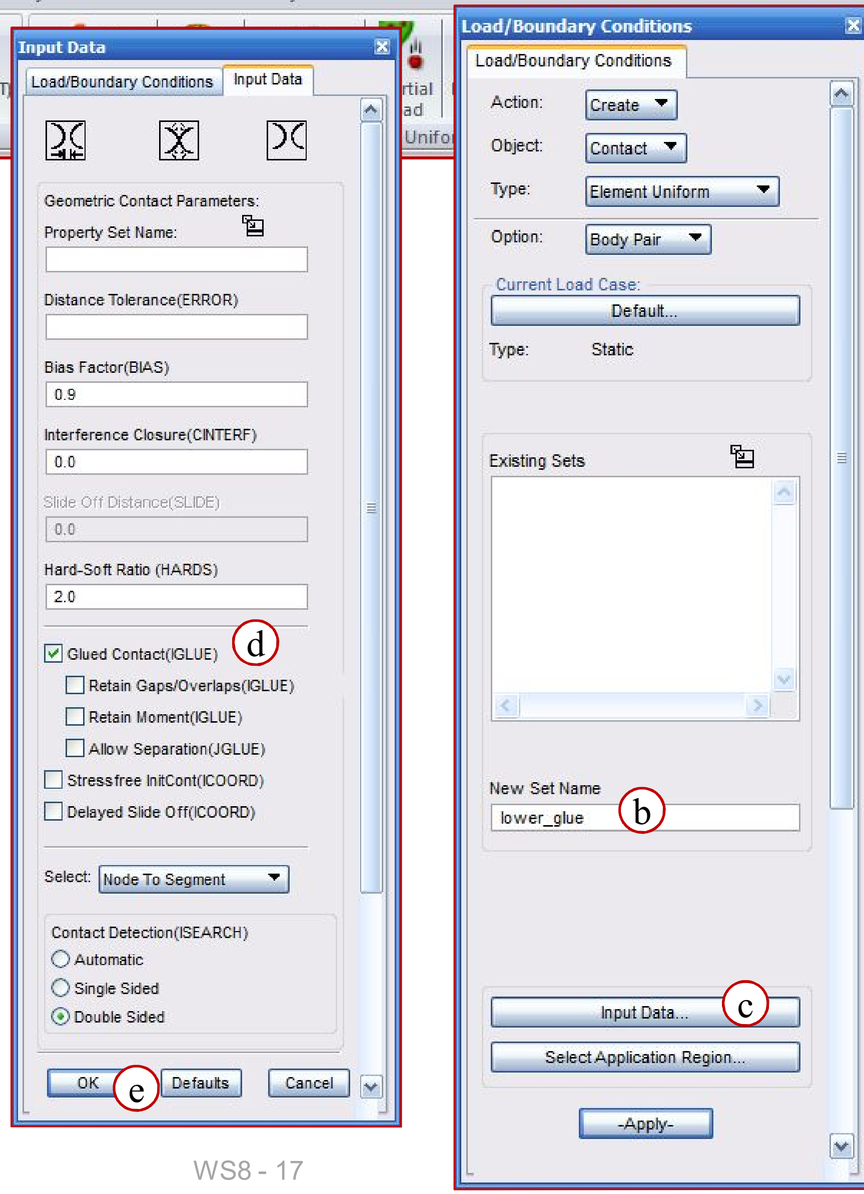




MSC Software

Step 6. Create the Contact Pairs





Create the contact pairs:

a. Under the *Loads/BCs* tab, click **Create Body Pair** in the *Body Pairs* group.
b. Enter **lower_glue** for *New Set Name*.
c. Click **Input Data**.
d. Check the box for *Glued Contact(IGLUE)*.
e. Click **OK**.

MSC Software

Step 6. Create the Contact Pairs (Cont.)

Create the contact pairs (cont.):

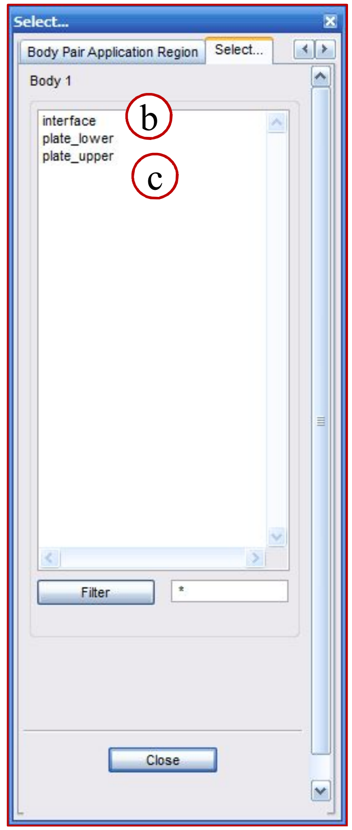
a. Click **Select Application Region**.

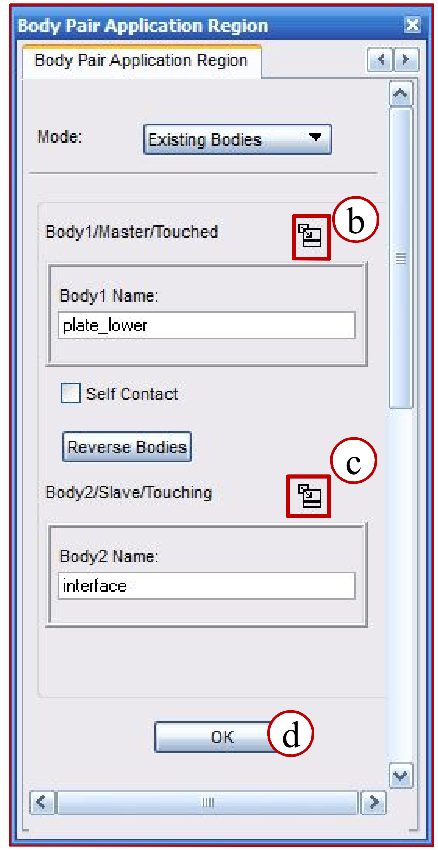
b. Click icon for *Body1/Master/Touched* and click **plate_lower**.

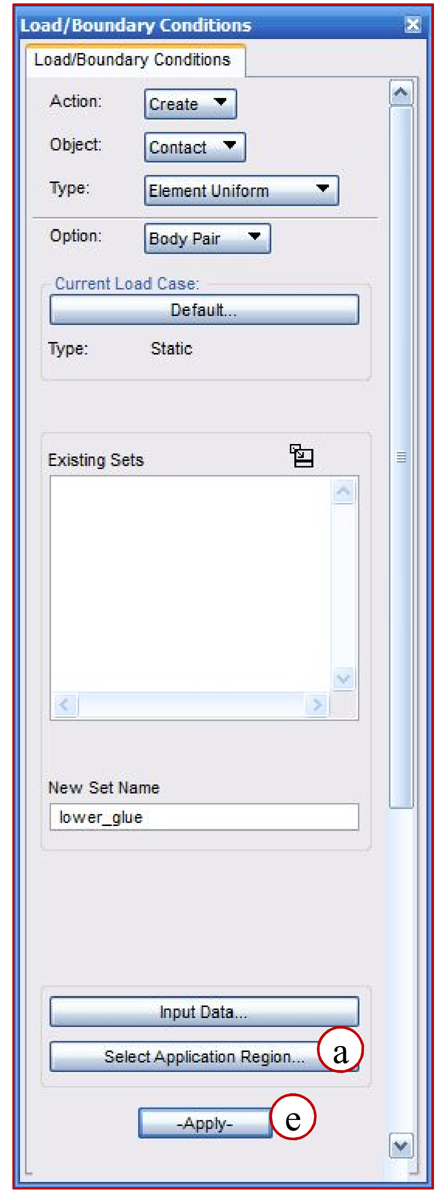
c. Click icon for *Body2/Slave/Touching* and click **interface**.

d. Click **OK**.

e. Click **Apply**.



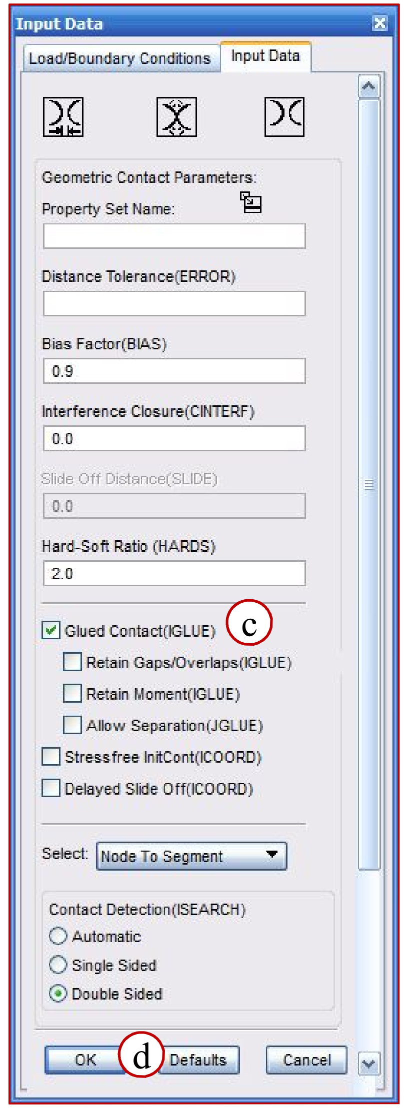


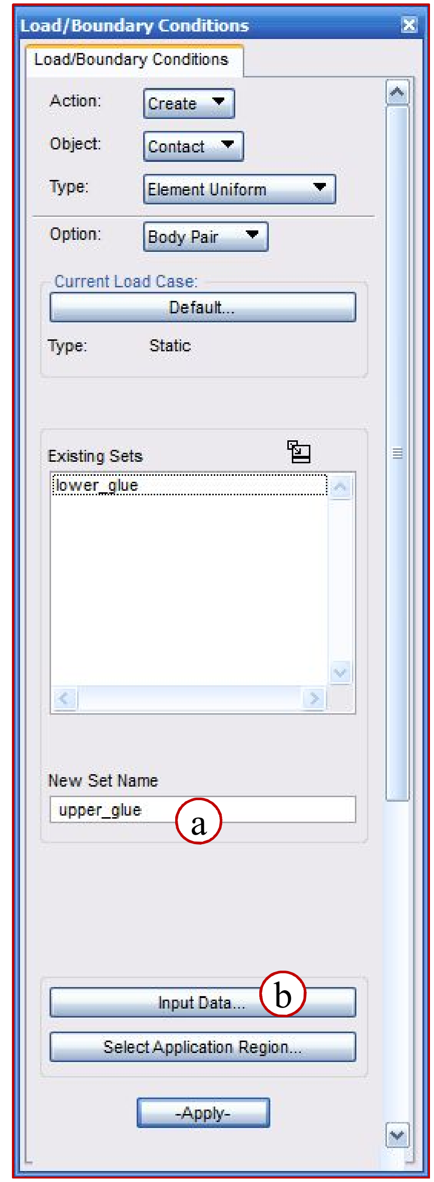


Step 6. Create the Contact Pairs (Cont.)

Create the contact pairs (cont.):

a. Enter **upper_glue** for *New Set Name*.
b. Click **Input Data**.
c. Check the box for *Glued Contact(IGLUE)*.
d. Click **OK**.





Step 6. Create the Contact Pairs (Cont.)

Create the contact pairs (cont.):

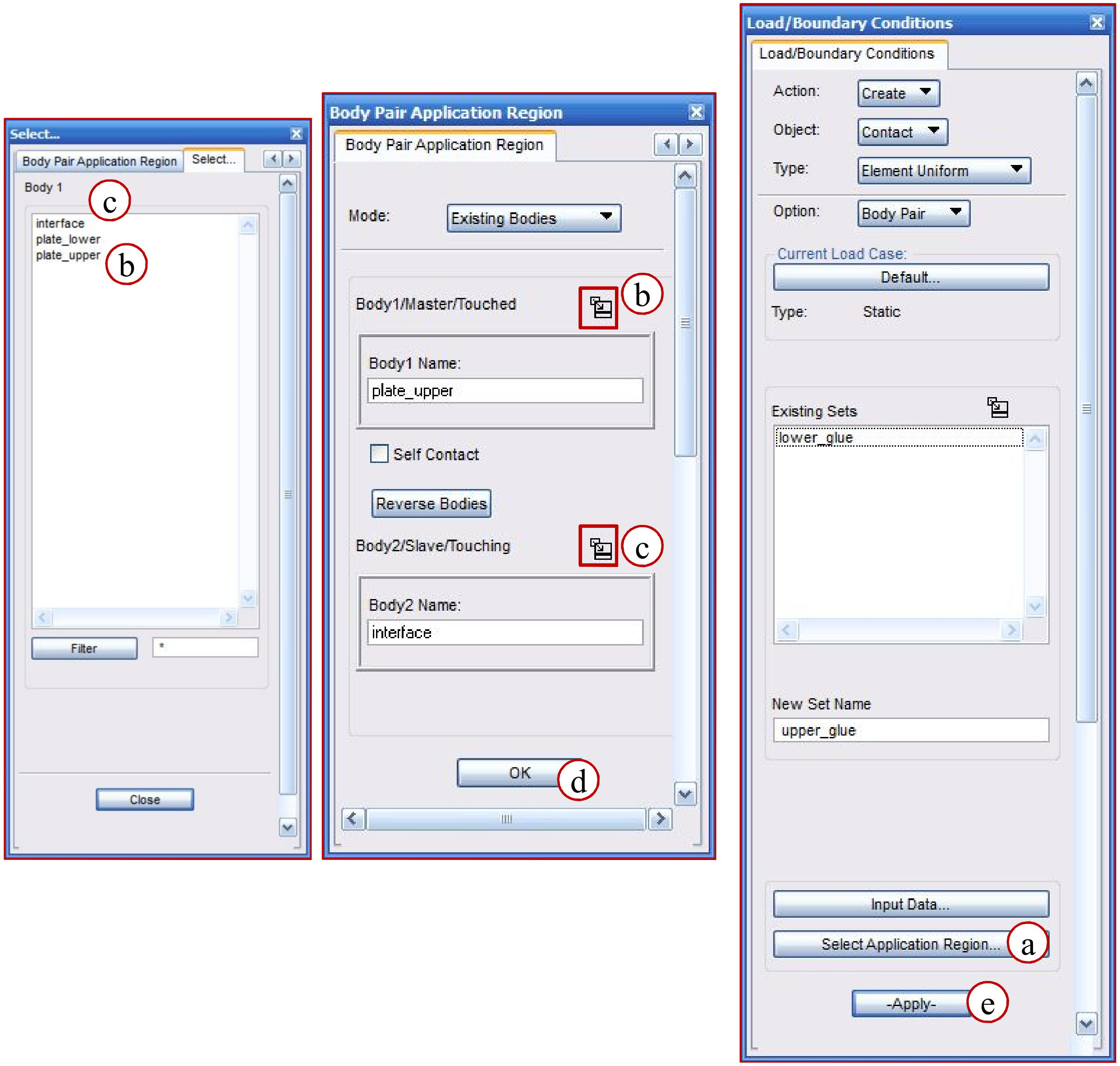
a. Click **Select Application Region**.

b. Click icon for *Body1/Master/Touched* and click **plate_upper**.

c. Click icon for *Body2/Slave/Touching* and click **interface**.

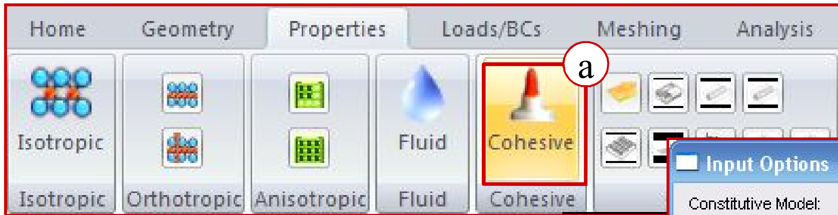
d. Click **OK**.

e. Click **Apply**.



MSC Software

Step 7. Create Cohesive Material



Create cohesive material:

a. Under the *Properties* tab, click **Cohesive**.

b. Enter *Material Name* **cohesive**.

c. Click **Input Properties**.

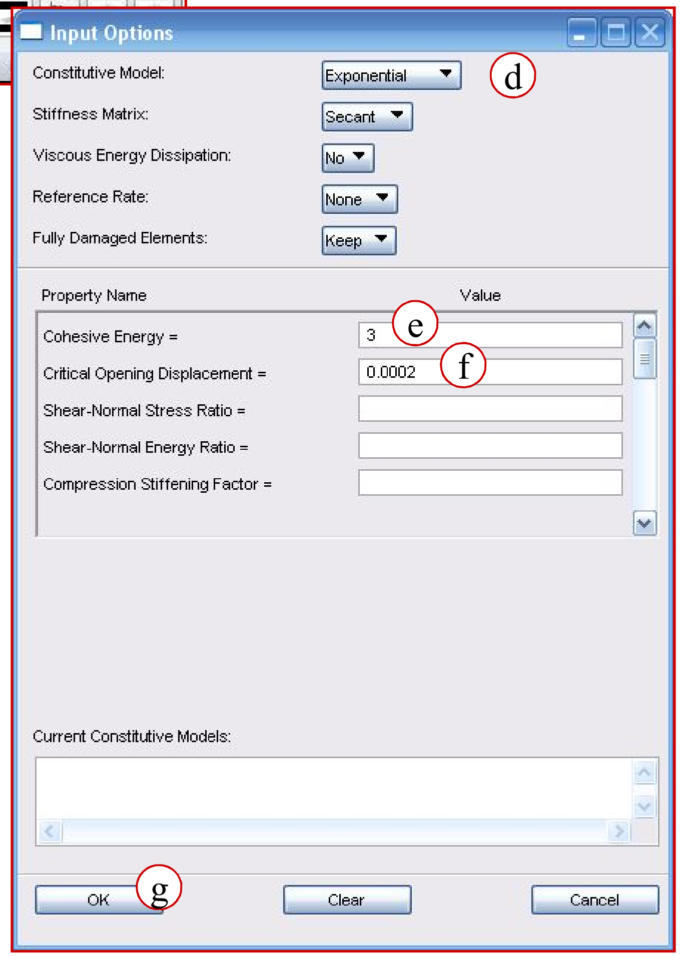
d. Select **Exponential** for *Constitutive Model*.

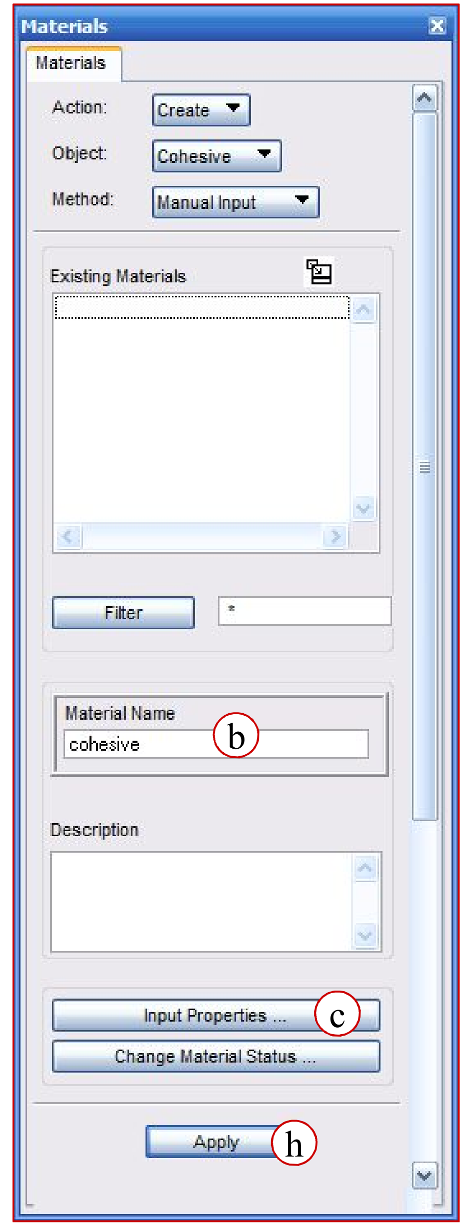
e. Enter **3** for *Cohesive Energy*.

f. Enter **0.0002** for *Critical Opening Displacement*.

g. Click **OK**.

h. Click **Apply**.



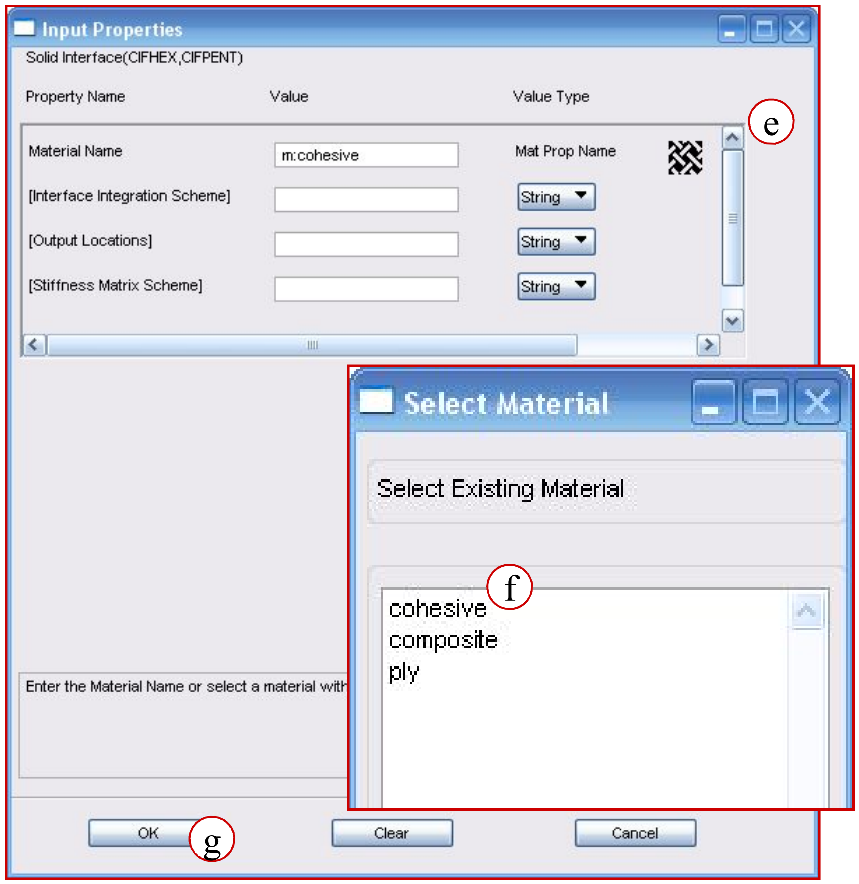


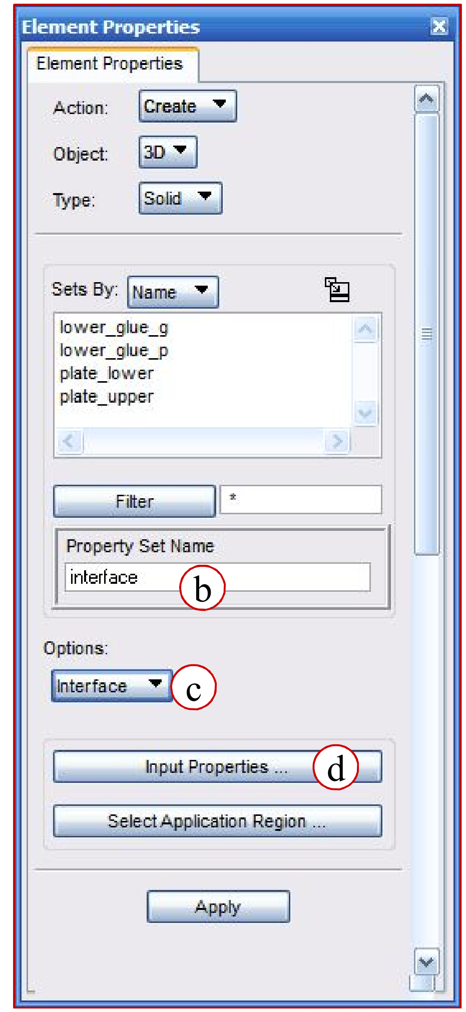
MSC Software

Step 8. Create Interface Element Property

Create cohesive interface element property:

a. Under the *Properties* tab, click **Solid**.
b. Enter *Property Set Name* **interface**.
c. Select **Interface** option.
d. Click **Input Properties**.
e. Click material icon.
f. Select **cohesive**.
g. Click **OK**.

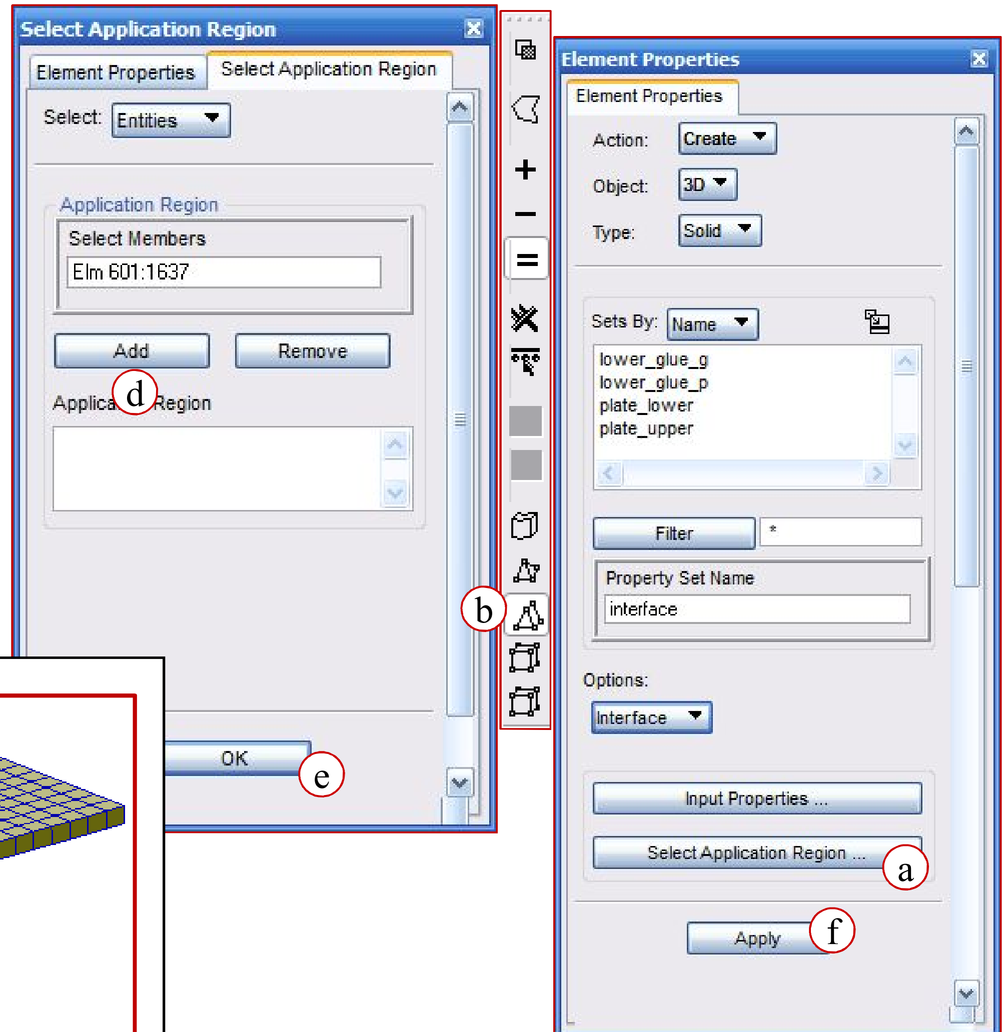


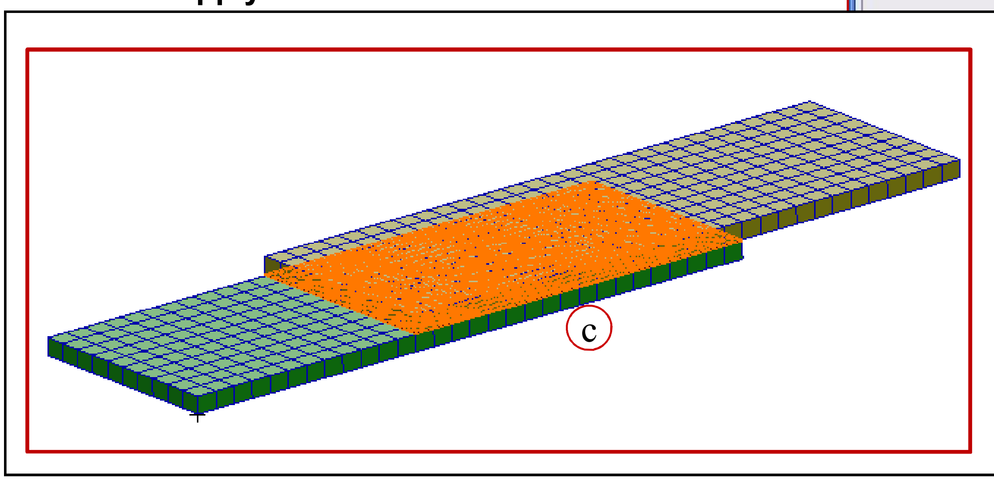


Step 8. Create Interface Element Property (Cont.)

Create cohesive interface element property (cont.):

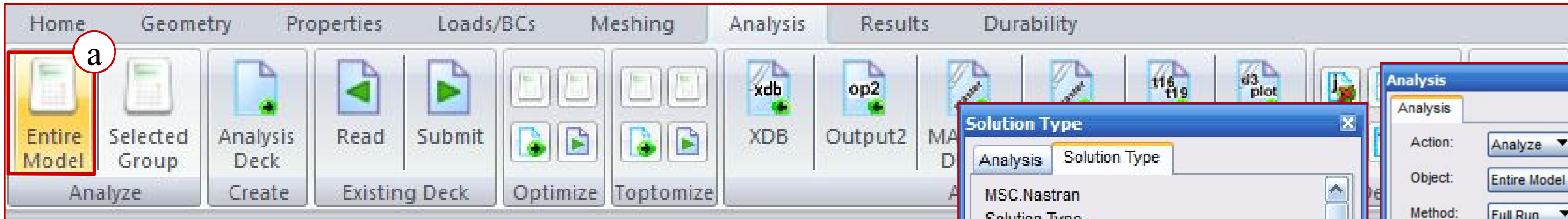
a. Click **Select Application Region**.
b. Select **Wedge Element** filter.
c. Select all wedge elements.
d. Click **Add**.
e. Click **OK**.
f. Click **Apply**.





MSC Software

Step 9. Set up the Analysis Job



Set up job parameters:

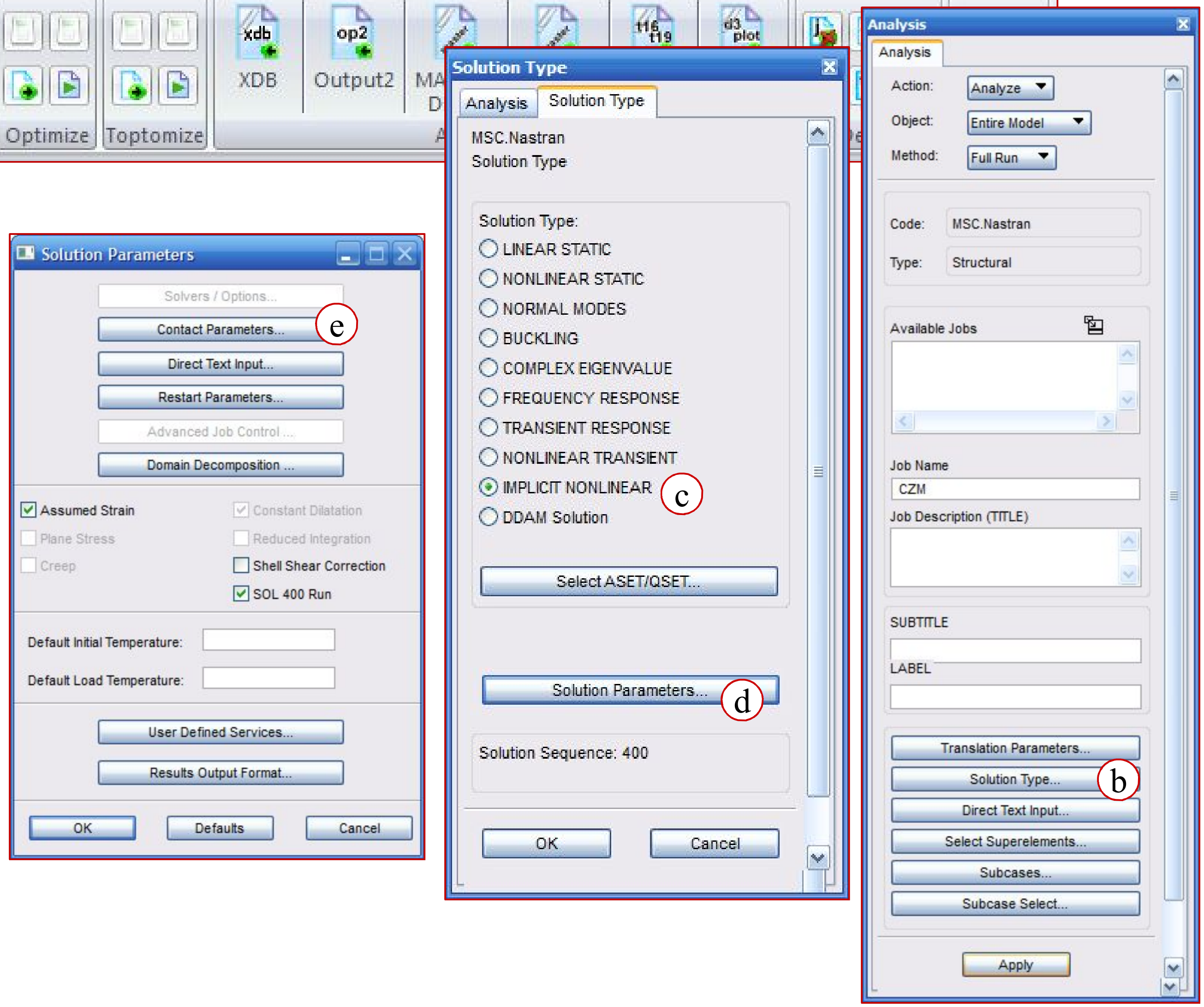
a. Under the *Analysis* tab, click **Entire Model** in the *Analyze* group.

b. Click **Solution Type**.

c. Select **Implicit Nonlinear**.

d. Click **Solution Parameters**.

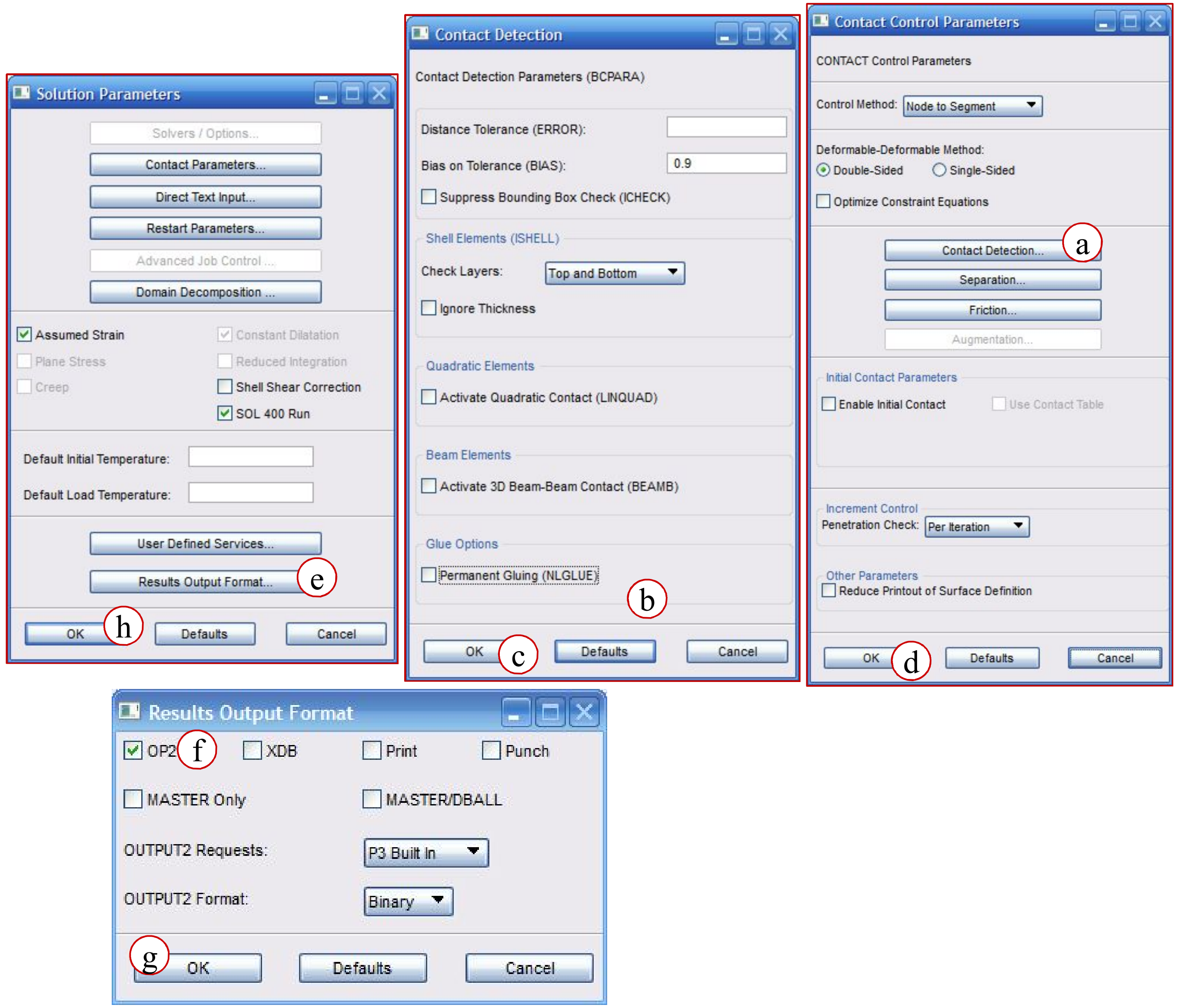
e. Click **Contact Parameters**.



Step 9. Set up the Analysis Job (Cont.)

Set up job parameters (cont.):

a. Click **Contact Detection**.
b. Uncheck *Permanent Gluing (NLGLUE)*.
c. Click **OK**.
d. Click **OK**.
e. Click **Results Output Format**.
f. Only check *OP2*.
g. Click **OK**.
h. Click **OK**.



Step 9. Set up the Analysis Job (Cont.)

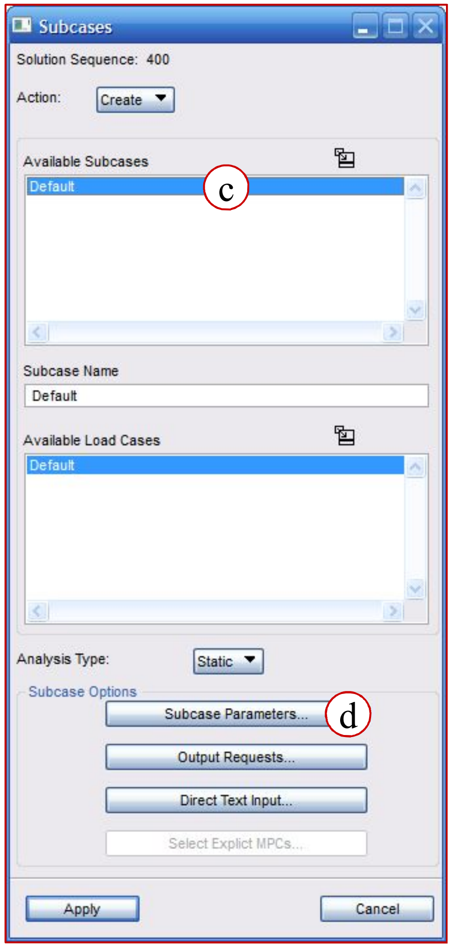
Set up job parameters (cont.):

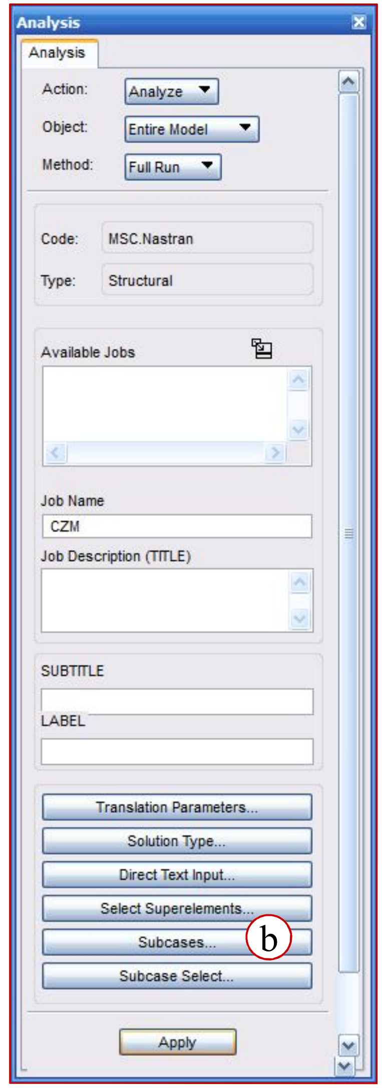
a. Click **OK**.

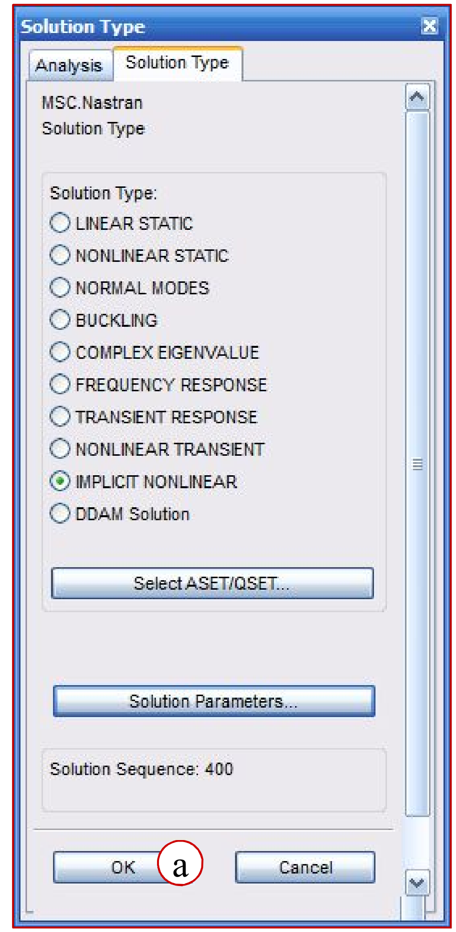
b. Click **Subcases**.

c. Select **Default** under *Available Subcases*.

d. Click **Subcase Parameters**.



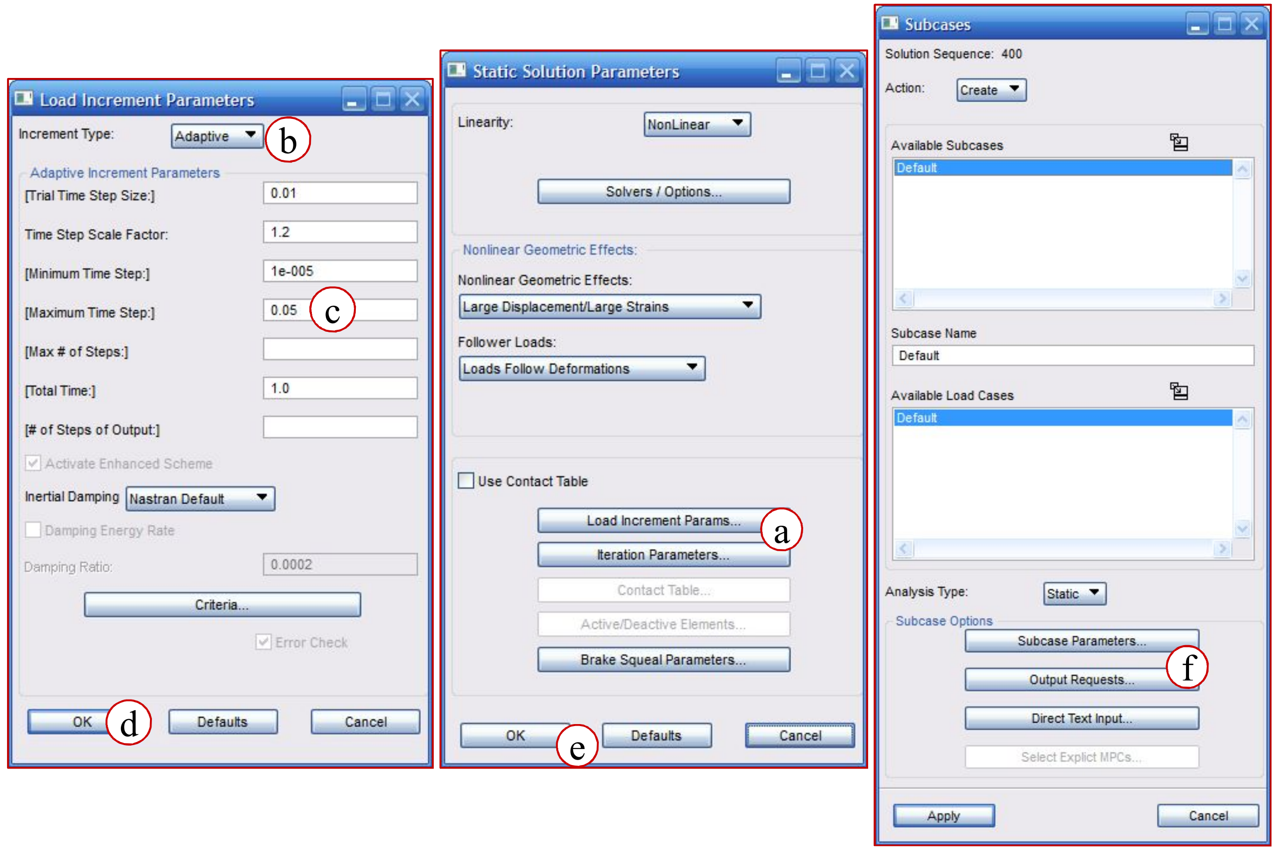




Step 9. Set up the Analysis Job (Cont.)

Set up job parameters (cont.):

a. Click **Load Increment Params**.
b. Select **Adaptive** for *Increment Type*.
c. Enter **0.05** for *Maximum Time Step*.
d. Click **OK**.
e. Click **OK**.
f. Click **Output Requests**.



Step 9. Set up the Analysis Job (Cont.)

Set up job parameters (cont.):

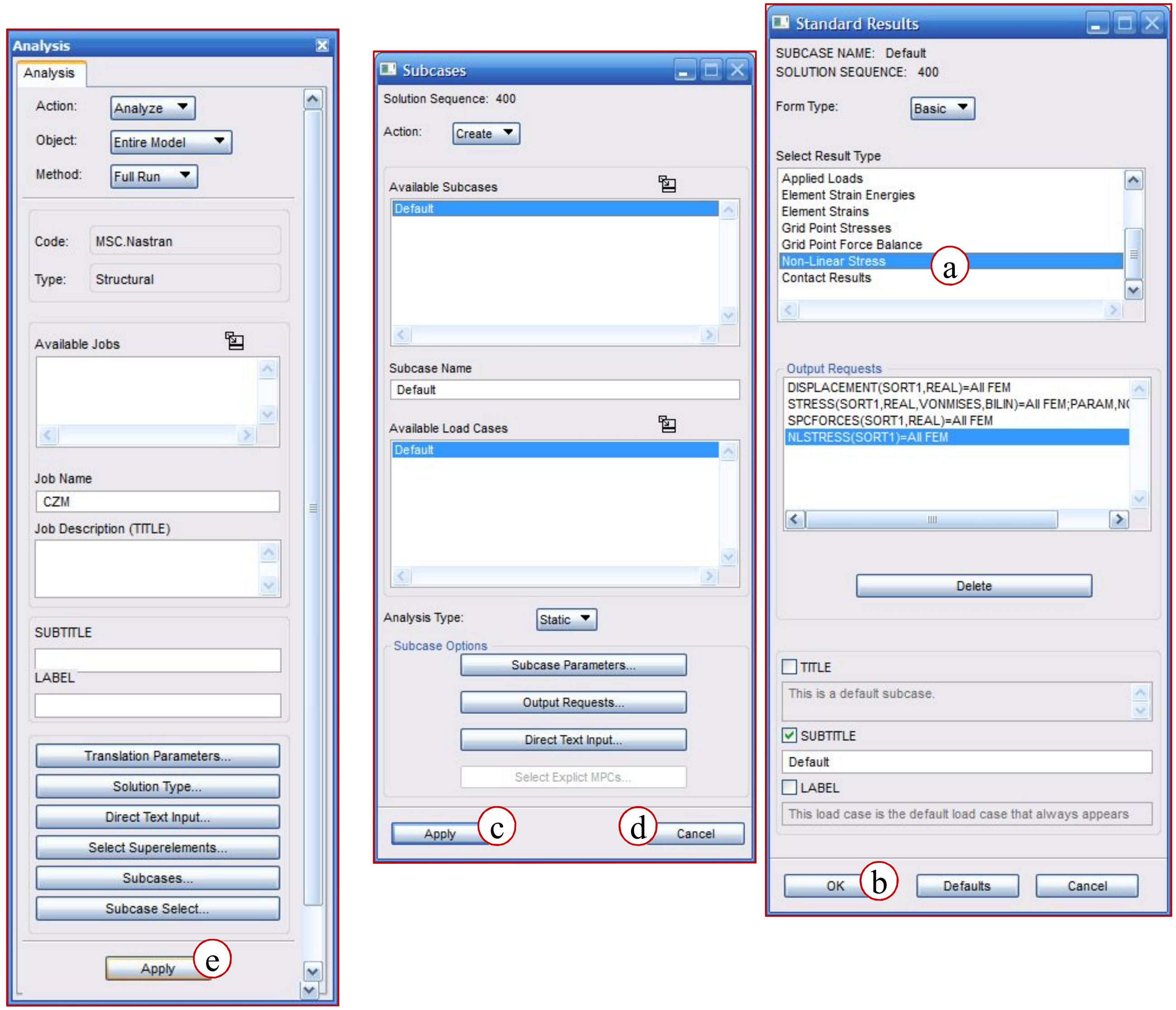
a. Select **Non-Linear Stress** under *Select Result Type*.

b. Click **OK**.

c. Click **Apply**.

d. Click **Cancel**.

e. Click **Apply**.



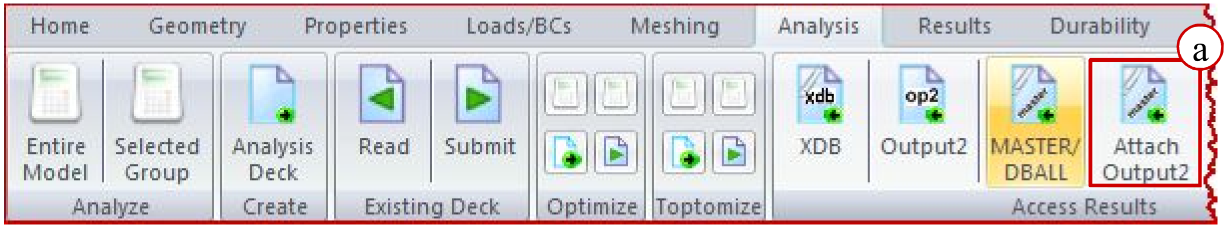
Step 10. Review the sts Files

a. Open **CZM.sts** file to check the job status – normal completion if job ends with exit number 0.

```
information summary of job:  ./czm
version: MSC Nastran 2013.1.0, Built on Nov 27, 2013


  subcase      inc  cycl  sepa cut      cycl    split    separ    cut  rmesh time step    total time  wall time    cpu time     max resp. type
  /step #        #     #     #    #        #        #        #      #      #     of          of
        1          |--of the inc--|-----------of the analysis-----------|  the inc      the job
        1        0     0     0    0        0        0        0      0      0 0.0000E+00  0.0000E+00       4.00        2.32    0.0000E+00 disp
        1        1     1     0    0        2        0        0      0      0 1.0000E-02  1.0000E-02       9.00        5.77   -4.7638E-03 disp
        1        2     1     0    0        3        0        0      0      0 1.2000E-02  2.2000E-02      11.00        7.55   -8.0557E-03 disp
        1        3     1     0    0        4        0        0      0      0 1.4400E-02  3.6400E-02      13.00        9.34   -1.1124E-02 disp
        1        4     1     0    0        5        0        0      0      0 1.7280E-02  5.3680E-02      15.00       11.14   -1.3806E-02 disp
        1        5     1     0    0        6        0        0      0      0 2.0736E-02  7.4416E-02      17.00       12.90   -1.6264E-02 disp
        1        6     1     0    0        7        0        0      0      0 2.4883E-02  9.9299E-02      18.00       14.70   -1.8665E-02 disp
        1        7     1     0    0        8        0        0      0      0 2.9860E-02  1.2916E-01      20.00       16.47   -2.0870E-02 disp
        1        8     1     0    0        9        0        0      0      0 3.5832E-02  1.6499E-01      22.00       18.25   -2.3135E-02 disp
        1        9     1     0    0       10        0        0      0      0 4.2998E-02  2.0799E-01      24.00       20.05   -2.5258E-02 disp
        1       10     1     0    0       11        0        0      0      0 5.0000E-02  2.5799E-01      26.00       21.84   -2.7115E-02 disp
        1       11     1     0    0       12        0        0      0      0 5.0000E-02  3.0799E-01      27.00       23.62    3.0799E-02 disp
        1       12     1     0    0       13        0        0      0      0 5.0000E-02  3.5799E-01      29.00       25.41    3.5799E-02 disp
        1       13     1     0    0       14        0        0      0      0 5.0000E-02  4.0799E-01      31.00       27.21    4.0799E-02 disp
        1       14     1     0    0       15        0        0      0      0 5.0000E-02  4.5799E-01      33.00       28.98    4.5799E-02 disp
        1       15     1     0    0       16        0        0      0      0 5.0000E-02  5.0799E-01      34.00       30.75    5.0799E-02 disp
        1       16     1     0    0       17        0        0      0      0 5.0000E-02  5.5799E-01      36.00       32.56    5.5799E-02 disp
        1       17     1     0    0       18        0        0      0      0 5.0000E-02  6.0799E-01      38.00       34.35    6.0799E-02 disp
        1       18     1     0    0       19        0        0      0      0 5.0000E-02  6.5799E-01      40.00       36.13    6.5799E-02 disp
        1       19     1     0    0       20        0        0      0      0 5.0000E-02  7.0799E-01      42.00       37.91    7.0799E-02 disp
        1       20     5     0    0       25        0        0      0      0 5.0000E-02  7.5799E-01      50.00       46.24    7.5815E-02 disp
        1       21     1     0    0       26        0        0      0      0 5.0000E-02  8.0799E-01      52.00       48.02    8.0815E-02 disp
        1       22     1     0    0       27        0        0      0      0 5.0000E-02  8.5799E-01      54.00       49.81    8.5816E-02 disp
        1       23     1     0    0       28        0        0      0      0 5.0000E-02  9.0799E-01      55.00       51.61    9.0816E-02 disp
        1       24     1     0    0       29        0        0      0      0 5.0000E-02  9.5799E-01      57.00       53.38    9.5817E-02 disp
        1       25     1     0    0       30        0        0      0      0 4.2011E-02  1.0000E+00      59.00       55.16    1.0002E-01 disp
 Job ends with exit number :        0
            total wall time:           60.00
            total cpu  time:           55.63

            exit DEFINITION -----
            = 0  job terminates normally
            = 1  job terminates abnormally (check Fatal Error Message in F06)
```

a

Step 11. Attach MASTER Results File



Attach Results file:

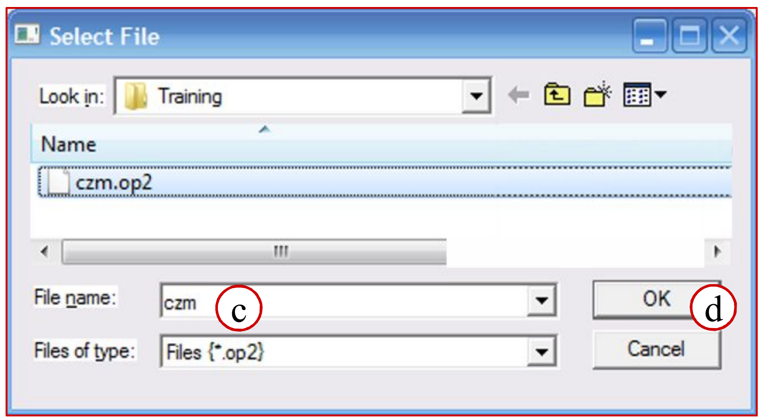
a. Under the *Analysis* tab, click the **Attach Output2** in the *Access Results* group.

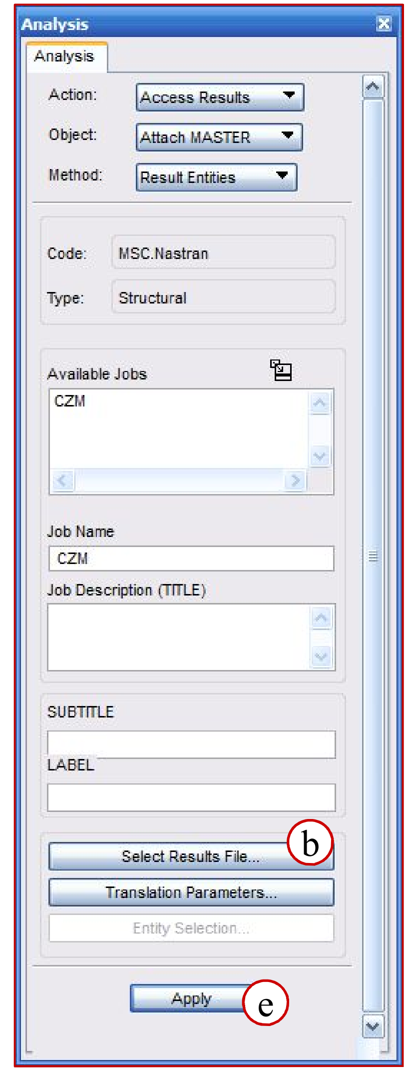
b. Click **Select Results File**.

c. Browse to the working directory and select **czm.op2**

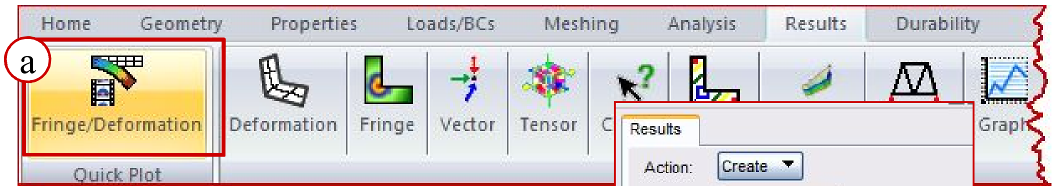
d. Click **OK**.

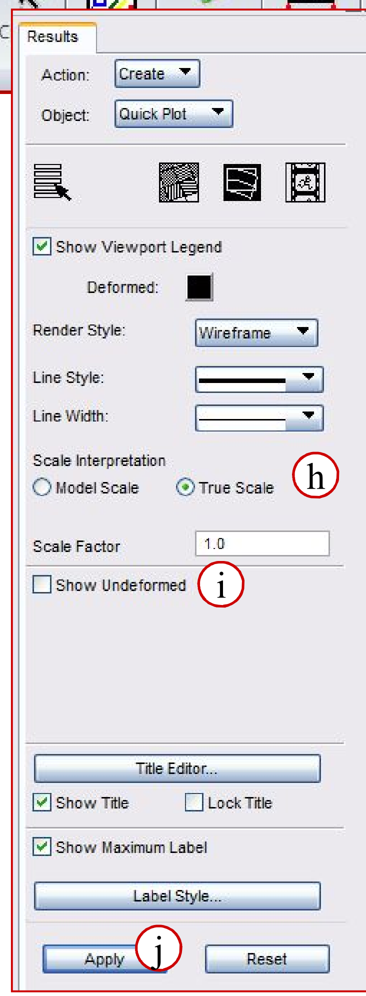
e. Click **Apply**.

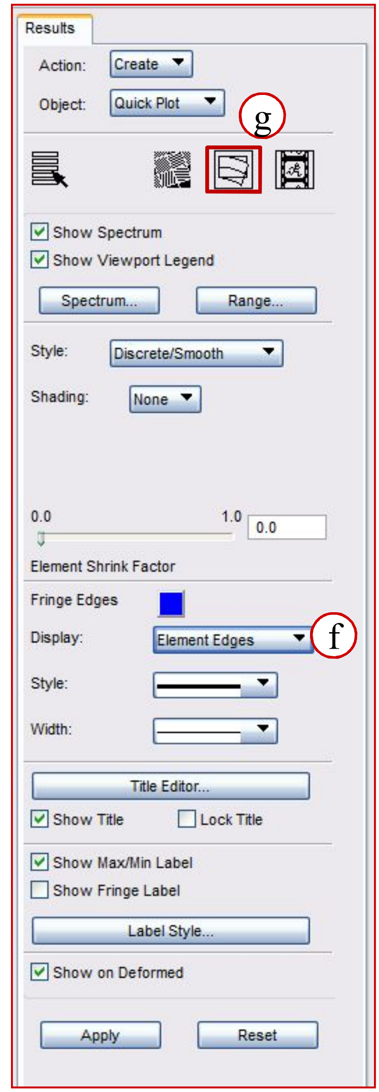


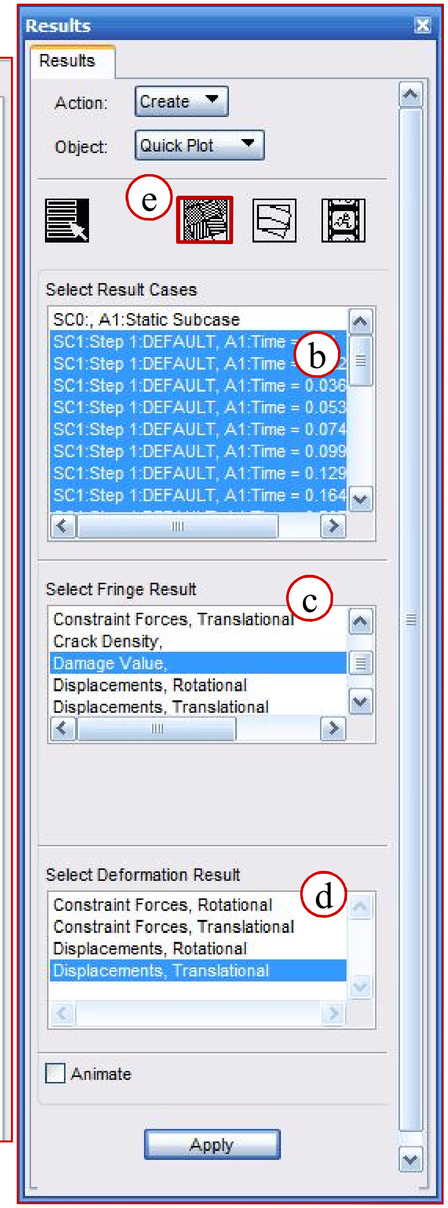


Step 12. Plot Interface Damage







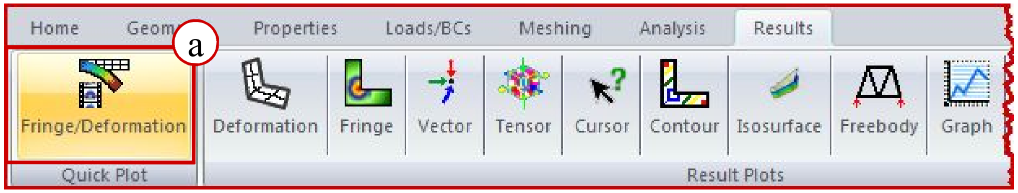


Plot interface damage:

a. Under the *Results* tab, click **Fringe/Deformation** in the *Quick Plot* group.
b. Under the *Select Result Cases*, select all increments except the first one.
c. Select **Damage Value** under *Select Fringe Result*.
d. Select **Displacements, Translational** under *Select Deformation Result*.
e. Click the **Fringe Attributes** icon.
f. Select **Element Edges** for *Display*.
g. Click the **Deform Attributes** icon.
h. Select **True Scale**.
i. Uncheck *Show Undeformed*.
j. Click **Apply**.

MSC Software

Step 12. Plot Interface Damage (Cont.)



Plot interface damage at specified time:

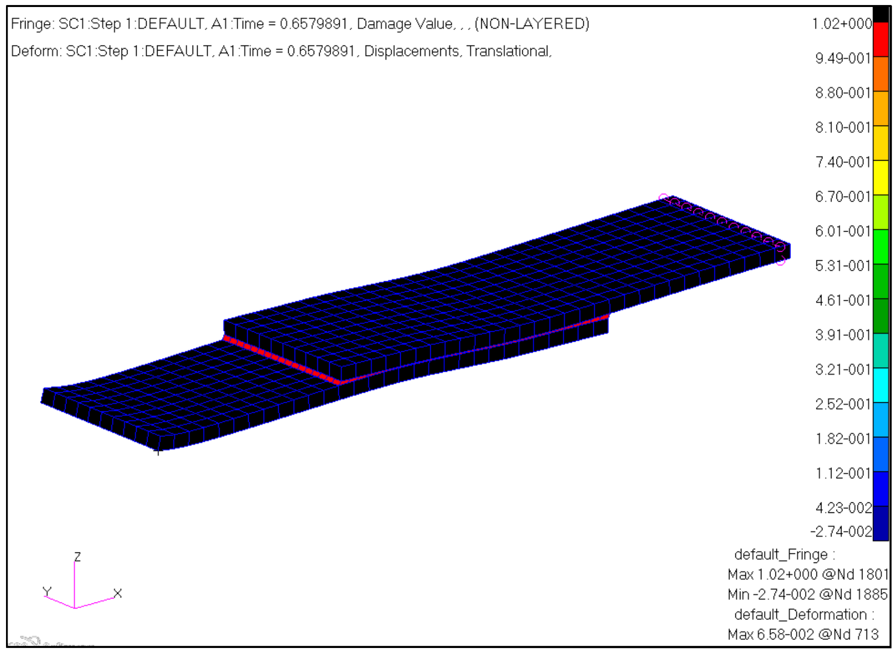
a. Under the *Results* tab, click **Fringe/Deformation** in the *Quick Plot* group.

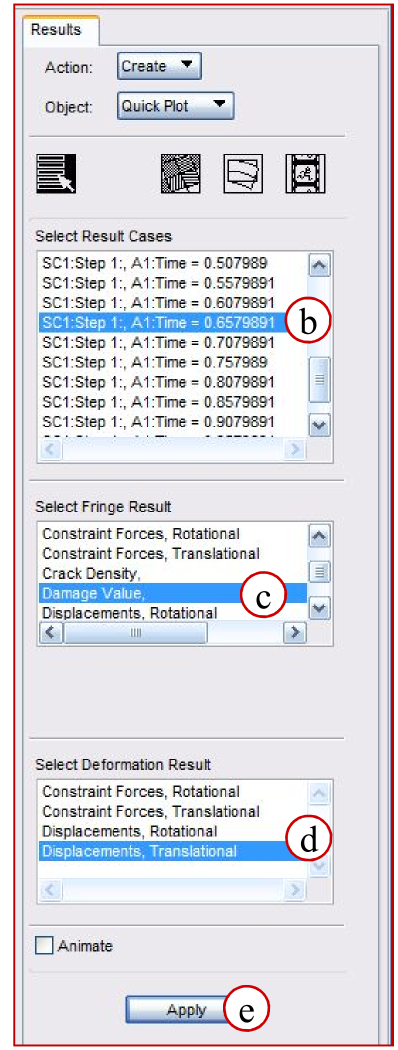
b. Select results case at **Time = 0.6579891** under *Select Result Cases*.

c. Select **Damage Value** under *Select Fringe Result*.

d. Select **Displacements, Translational** under *Select Deformation Result*.

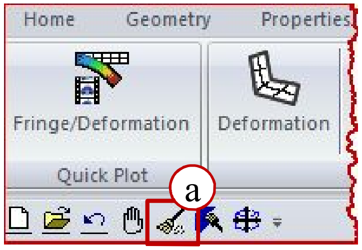
e. Click **Apply**.

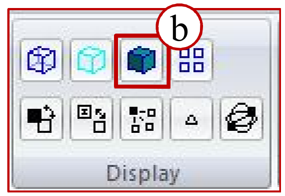




MSC Software

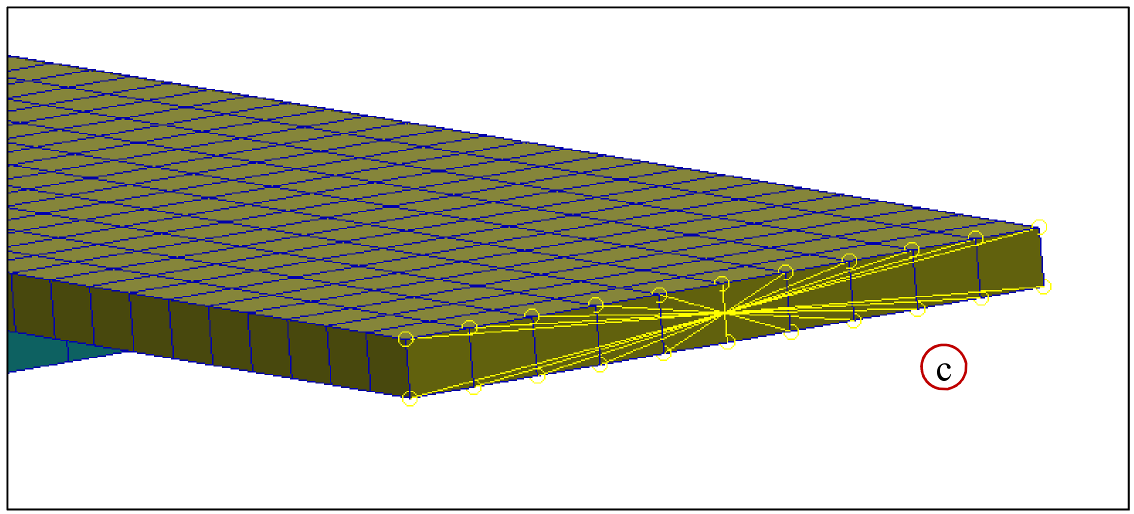
Step 13. Graph Load-Deflection Curve



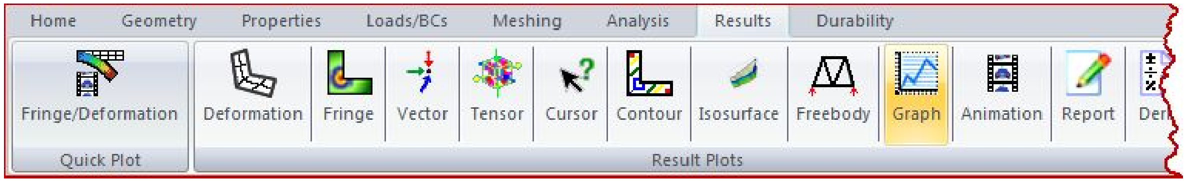


Graph load-deflection curve:

a. Click **Reset Graphics**.
b. Under the *Home* tab, click **Smooth shaded** in the *Display* group.
c. Zoom in to the right edge of the top plate.

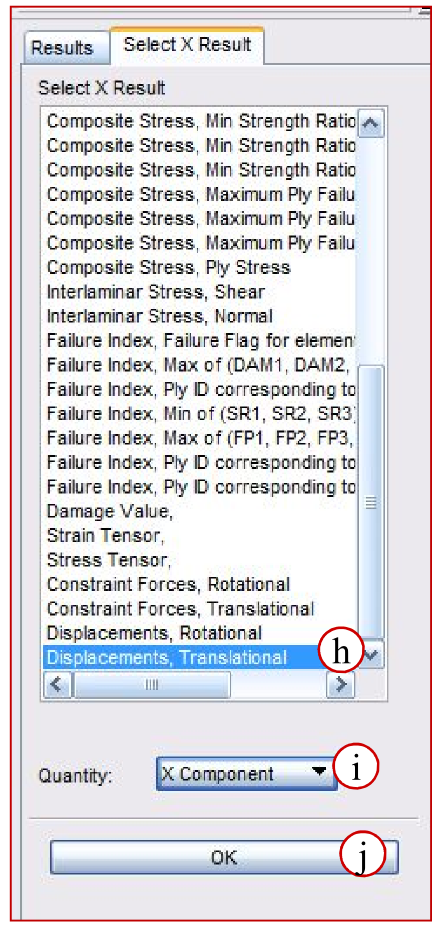


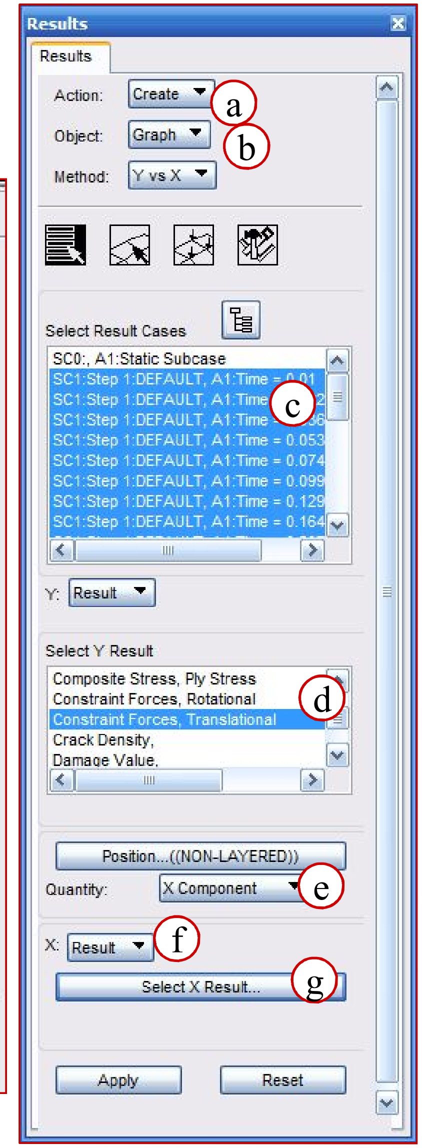
Step 13. Graph Load-Deflection Curve (Cont.)



Graph load-deflection curve of the independent node of the RBE2s:

a. Change the *Object* to **Graph**.
b. Change the *Method* the **Y vs X**.
c. Under the *Select Result Cases*, select all increments except the first one.
d. Select **Constraint Forces Translational** under *Select Y Result.*
e. Pull down *Quantity* to **X Component**.
f. Pull down *X* to **Result**.
g. Click **Select X Result**.
h. Select **Displacements, Translational** under *Select X Result.*
i. Pull down *Quantity* to **X Component**.
j. Click **OK**.





Step 13. Graph Load-Deflection Curve (Cont.)

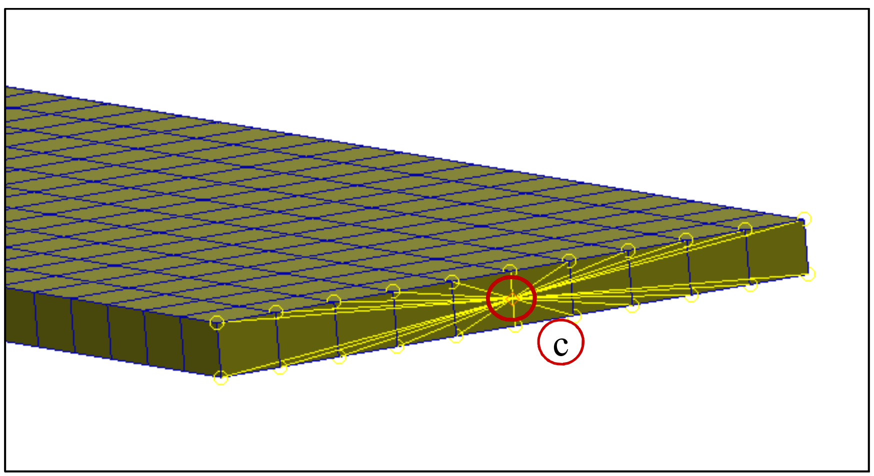
Graph load-deflection curve (cont.):

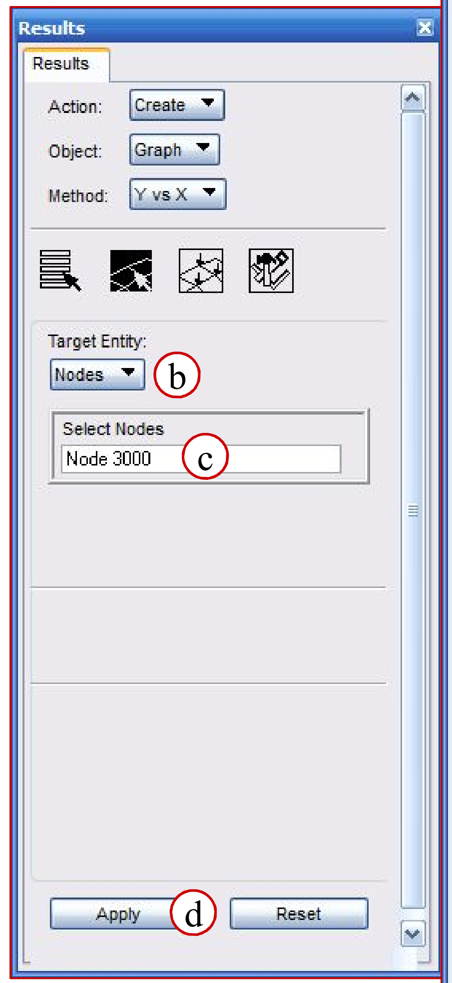
a. Click the **Target Entities** icon.

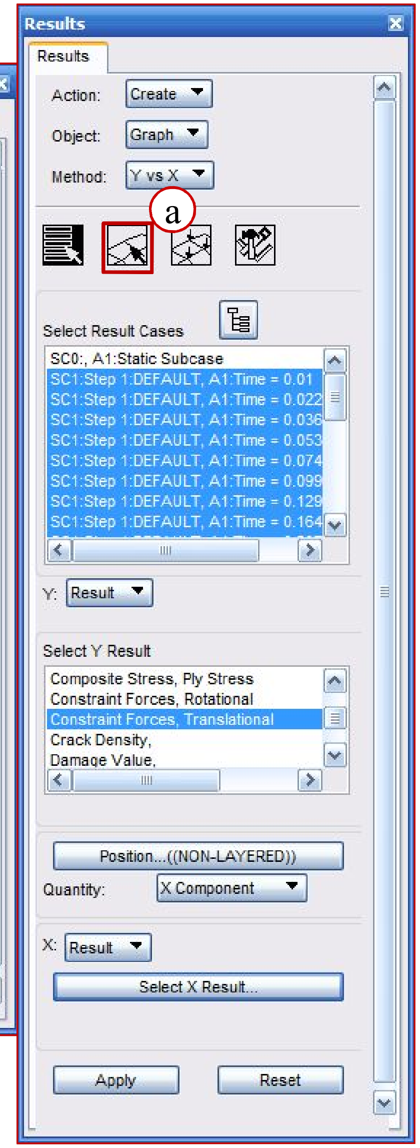
b. For the *Target Entity* select **Nodes**.

c. Select the Node highlighted in the image or enter **Node 3000** in the *Select Nodes* box.

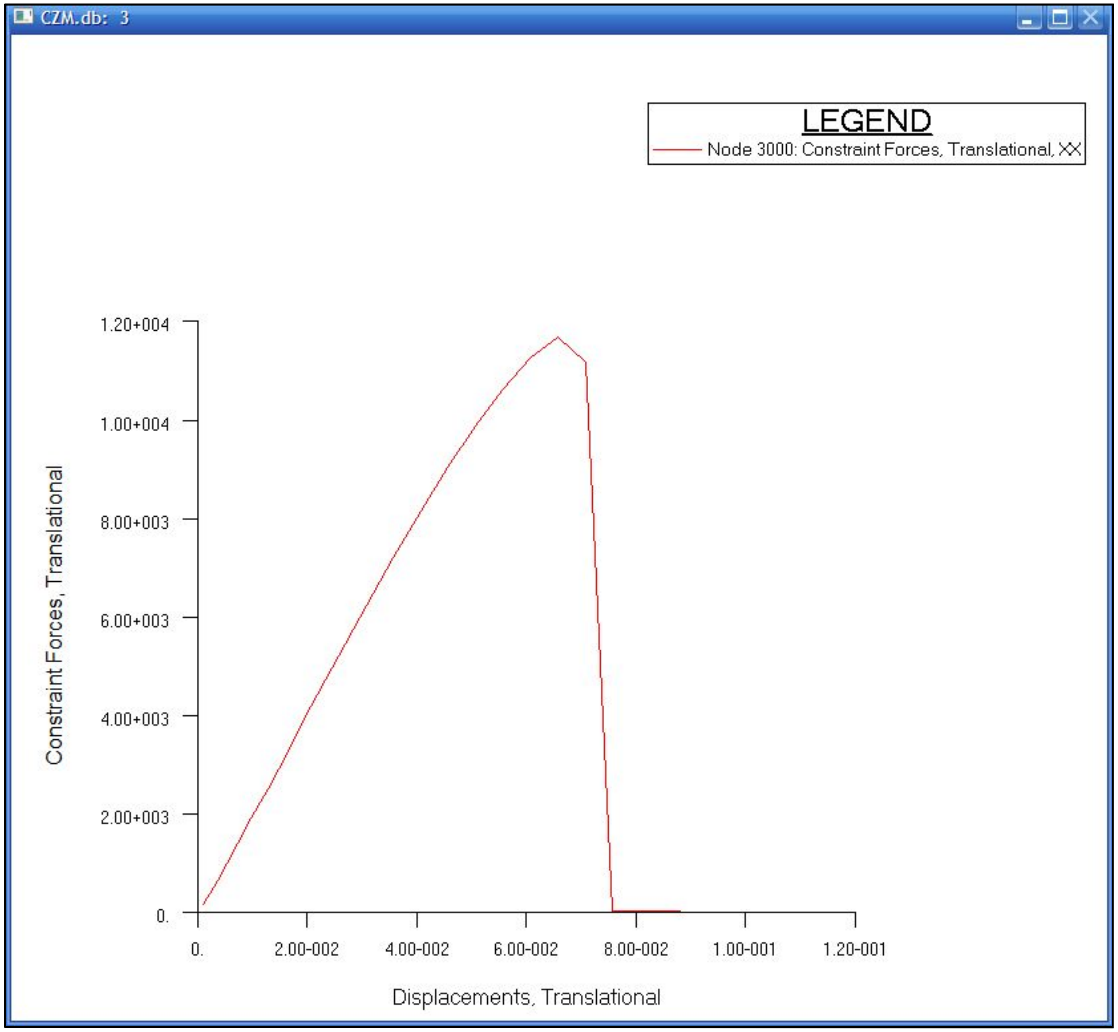
d. Click **Apply**.



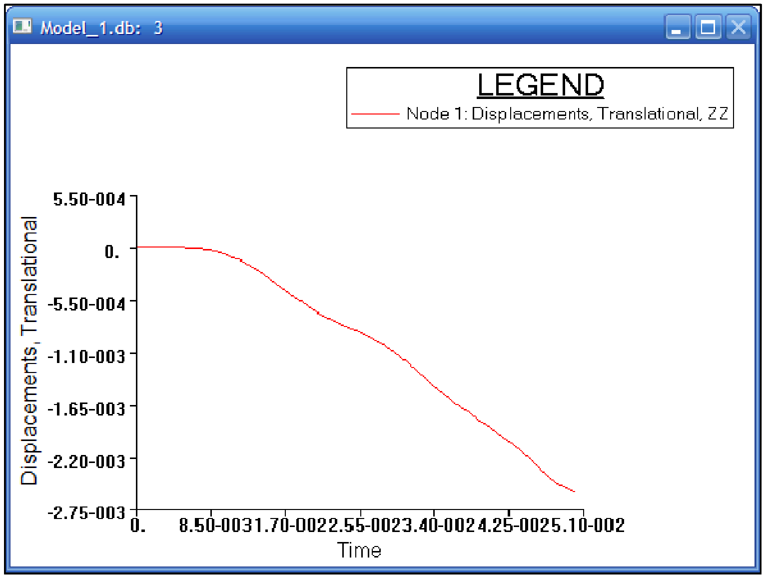


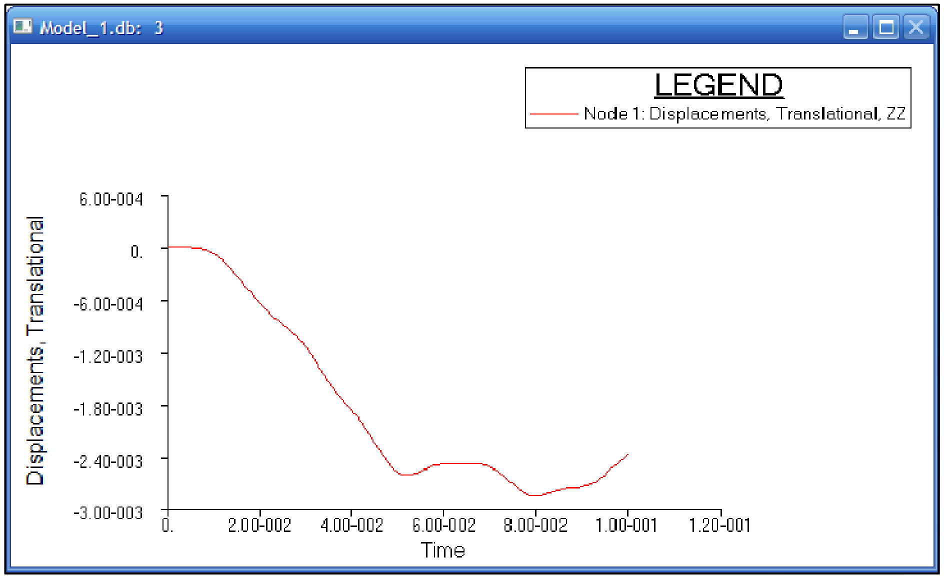


Step 13. Graph Load-Deflection Curve (Cont.)



WORKSHOP 9
RESTART





- **Workshop Objectives**
  - Use Restart Parameter

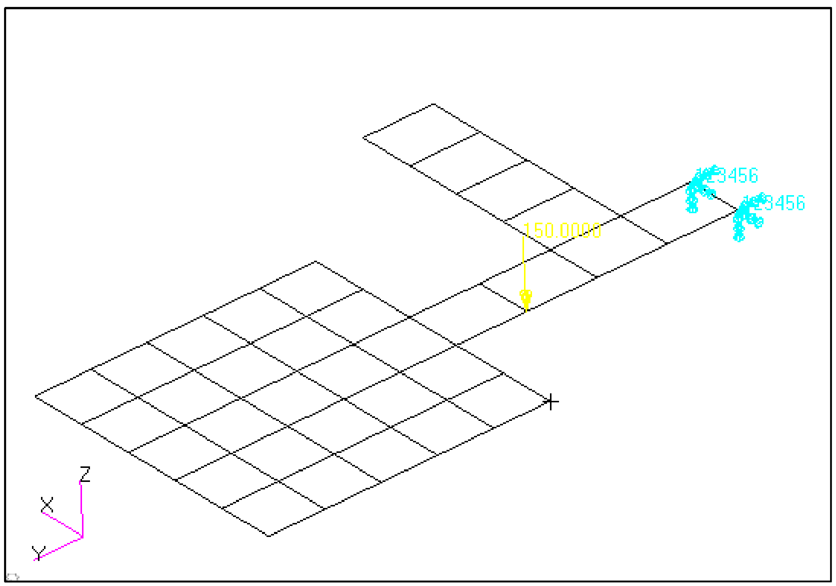
- **Software Version**
  - Patran 2013
  - MSC Nastran 2013.1

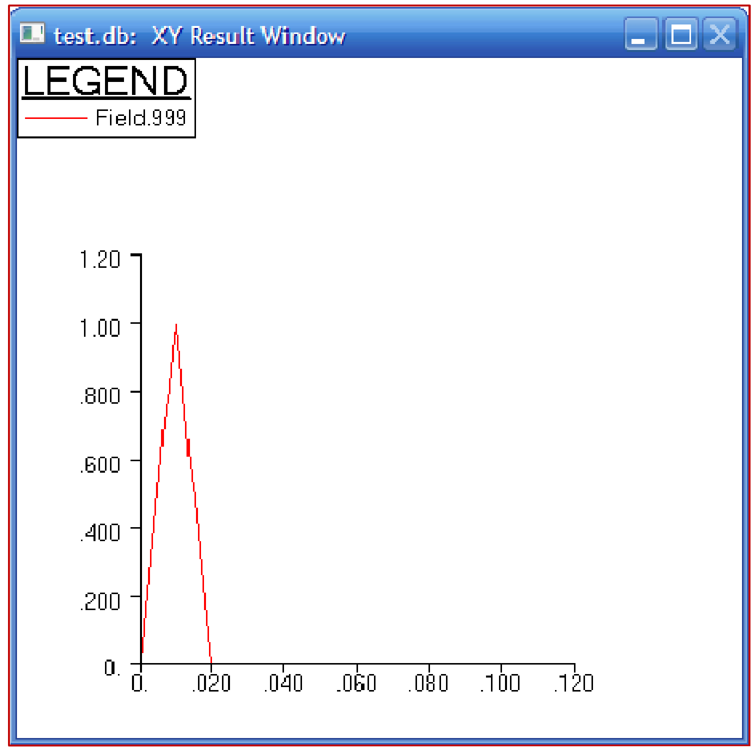
- **Required File:**
  - **Model_run1.bdf**

- **Problem Description**

  Steel Plate fixed on one side, force applied in the middle for 0.01 seconds and the removed.

  1. This is Dynamic Transient analysis and in the first run, the load is being applied up to 0.05 seconds.
  2. In the second run, the user restart the previous analysis, and runs the analysis for another 0.05 seconds





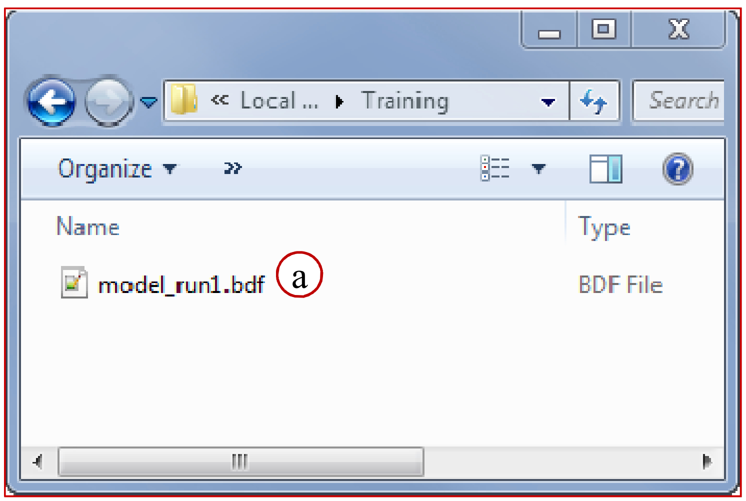
- **Suggested Exercise Steps**
  1. Review model_run1.bdf deck.
  2. Run the analysis using MSC Nastran.
  3. View the result in Patran
  4. Copy the Master/DBALL result file and rename it.
  5. Open the model_run1.bdf and modify it accordingly.
  6. Save it as model_run2.bdf
  7. Attach the result to the same Patran Database.
  8. Plot displacement vs. time in Patran.

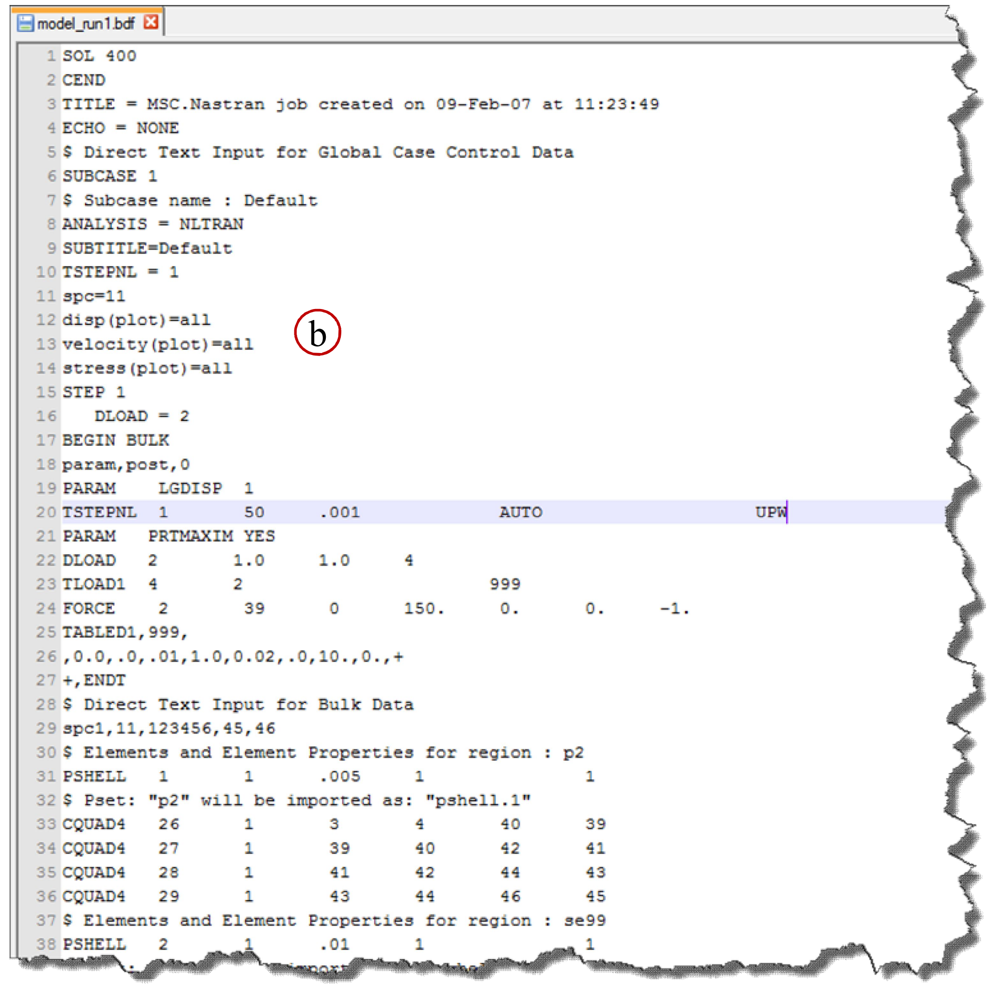
Step 1. Review BDF file.

Open the model_run1.BDF file.

a. Click **model_run.bdf**

b. Review the file, make note its SOL 400.



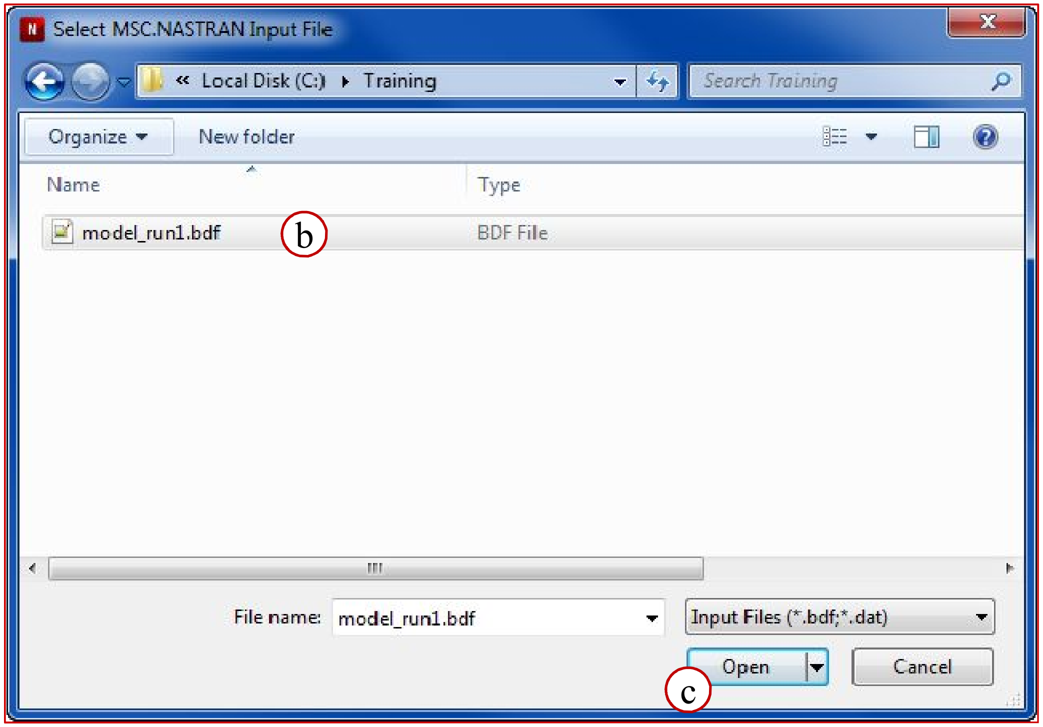


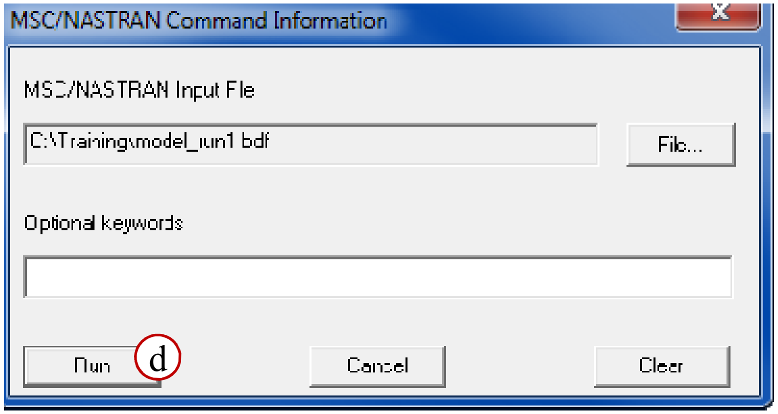

```
SOL 400
CEND
TITLE = MSC.Nastran job created on 09-Feb-07 at 11:23:49
ECHO = NONE
$ Direct Text Input for Global Case Control Data
SUBCASE 1
$ Subcase name : Default
ANALYSIS = NLTRAN
SUBTITLE=Default
TSTEPNL = 1
spc=11
disp(plot)=all
velocity(plot)=all
stress(plot)=all
STEP 1
   DLOAD = 2
BEGIN BULK
param,post,0
PARAM    LGDISP  1
TSTEPNL  1       50      .001            AUTO                    UPW
PARAM   PRTMAXIM YES
DLOAD    2       1.0     1.0     4
TLOAD1   4       2                               999
FORCE    2       39      0       150.    0.      0.      -1.
TABLED1,999,
,0.0,.0,.01,1.0,0.02,.0,10.,0.,+
+,ENDT
$ Direct Text Input for Bulk Data
spc1,11,123456,45,46
$ Elements and Element Properties for region : p2
PSHELL   1       1       .005    1               1
$ Pset: "p2" will be imported as: "pshell.1"
CQUAD4   26      1       3       4       40      39
CQUAD4   27      1       39      40      42      41
CQUAD4   28      1       41      42      44      43
CQUAD4   29      1       43      44      46      45
$ Elements and Element Properties for region : se99
PSHELL   2       1       .01     1               1
```

Step 2. Run the job

Submit the job to MSC Nastran.

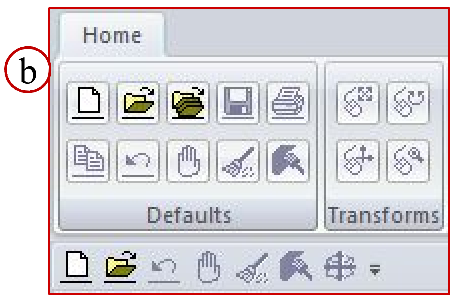
a. Open **MSC Nastran**
b. Select **model_run1.bdf**
c. Click **Open**
d. Click **Run**

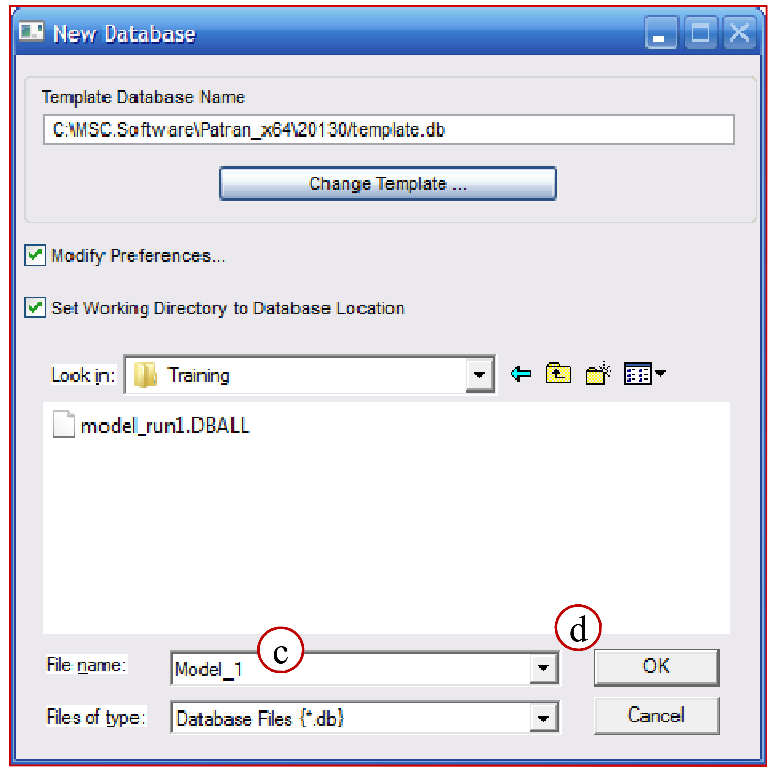




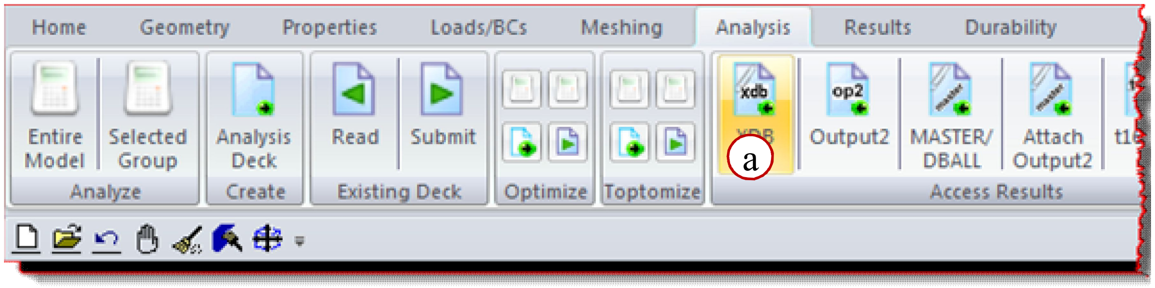
Step 3. View the result in Patran

a. Open **Patran.**
b. Click **New**
c. Go to Working directory and create a database called: **Model_1**
d. Click **OK**



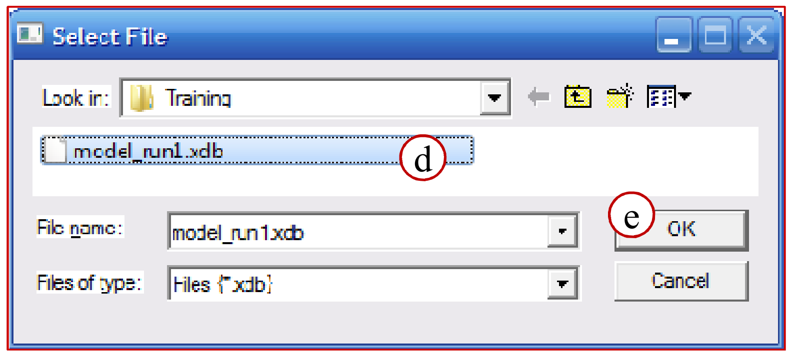


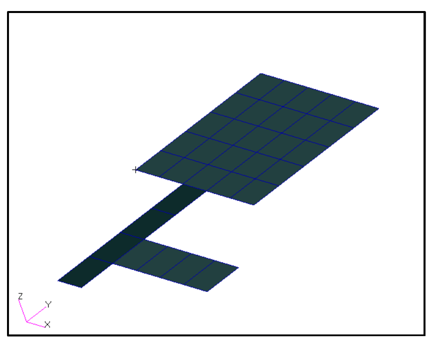
Step 3. View the result in Patran (Cont.)

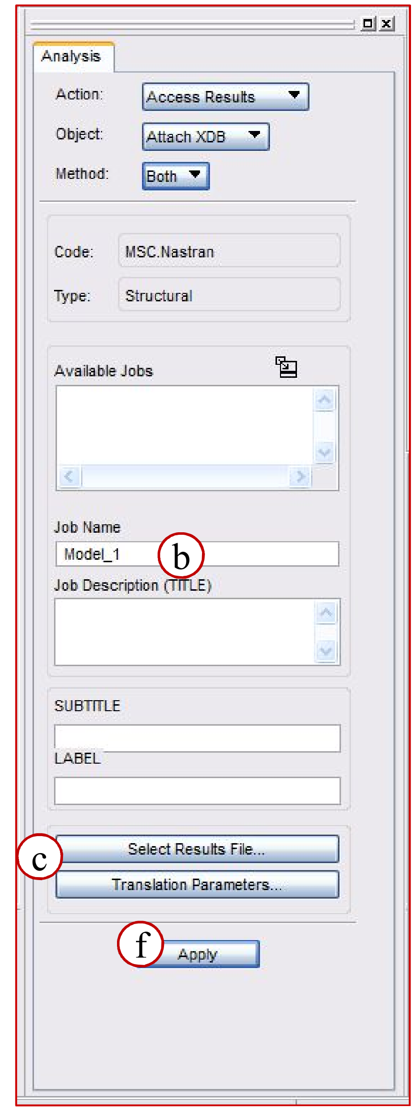


Open the *.xdb result file in Patran for post processing

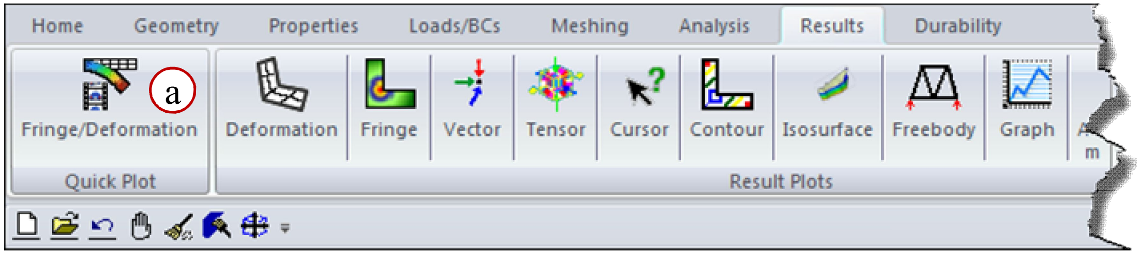
a. Under the *Analysis* tab, Click **XDB**, in the group Access Results
b. Pull down Method > **Both**
c. Click **Select Result File**
d. Select **model_run1.op2**
e. Click **OK**
f. Click **Apply**







Step 3. View the result in Patran (Cont.)



View the deformed shape using quick plot in Patran.

a. Under the *Result* tab, click **Quick Plot.**

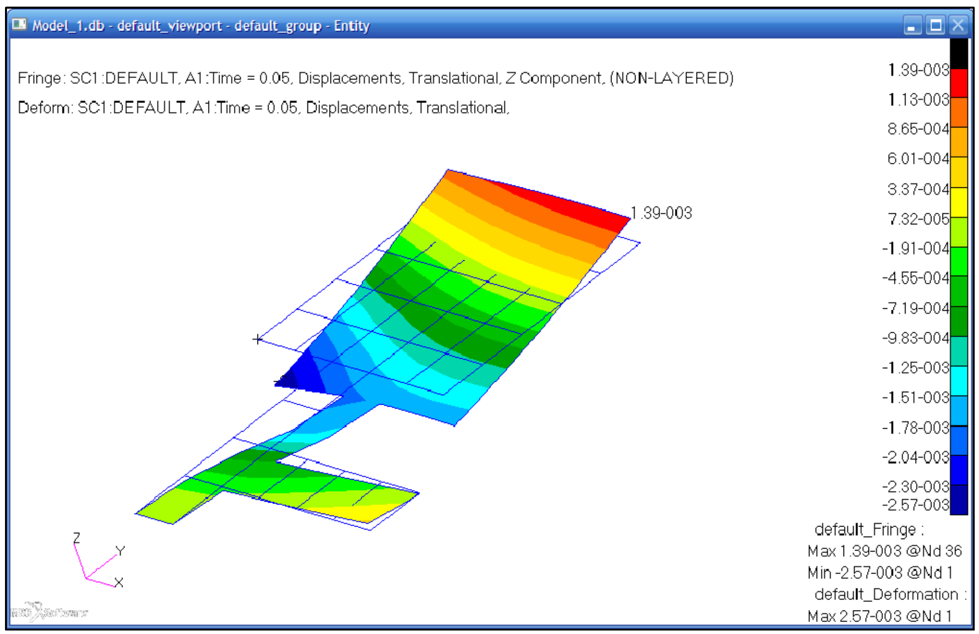
b. Select the last Result case, *Make note its at time 0.05*

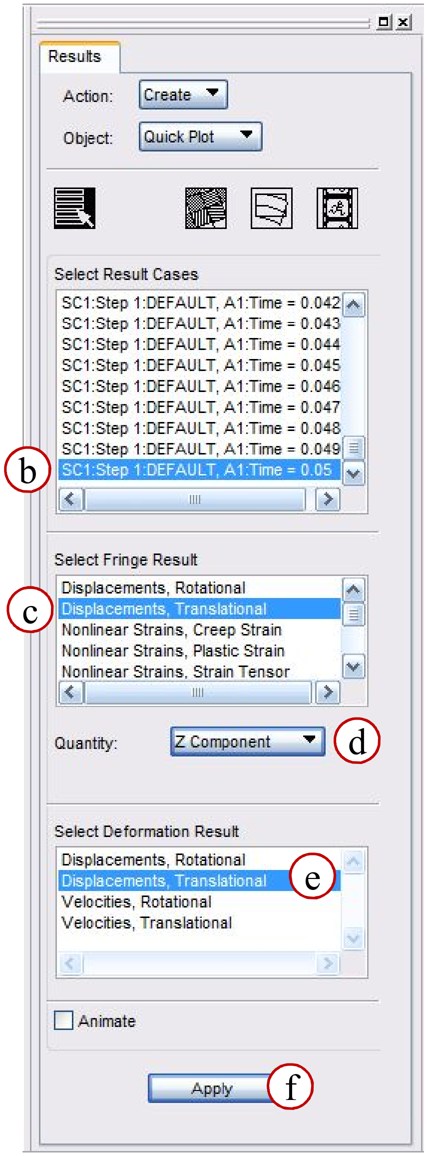
c. Select **Displacement, Translational** in *Select Fringe Result*

d. Pull down **Quantity > Z Component**

e. Select **Displacements, Translational** in *Select Deformation Result*

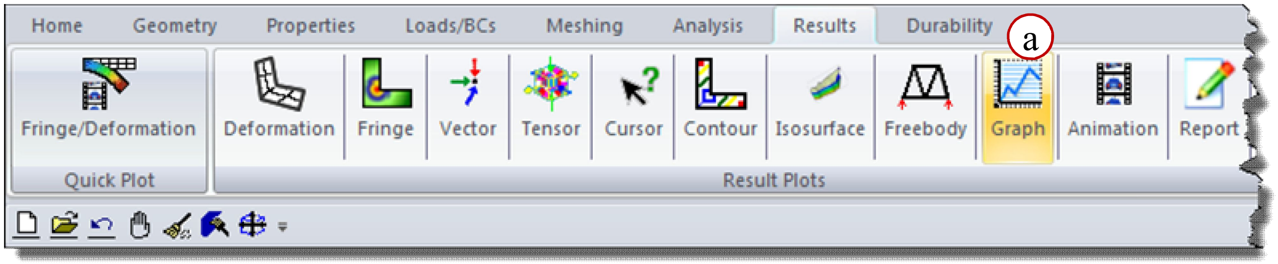
f. Click **Apply**





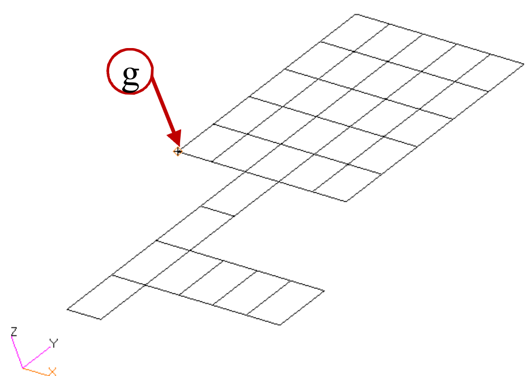
MSC Software

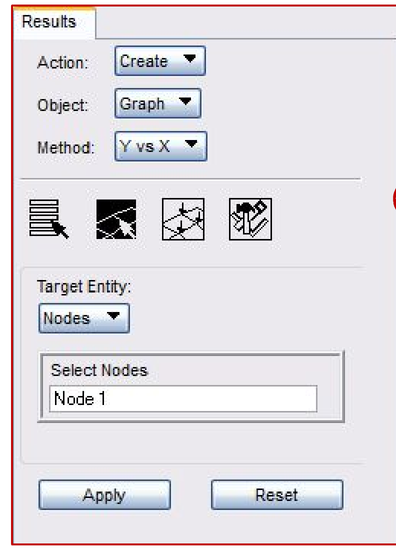
Step 3. View the result in Patran (Cont.)

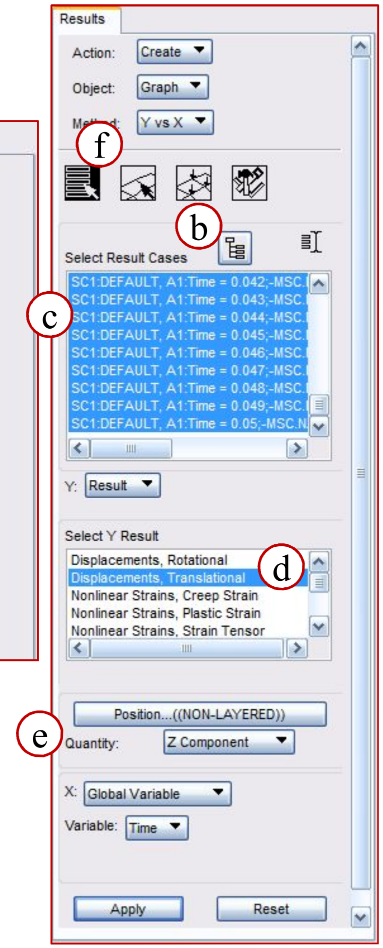


Plot Graph of Displacement vs. Time

a. Under the *Result* tab, click **Graph**
b. Select **View all result Cases**
c. Select **All result cases**
d. Select **Displacement Translational**
e. Pull down **Quantity > Z Component**
f. Select **Target Entity** tab
g. Select **Node 1**

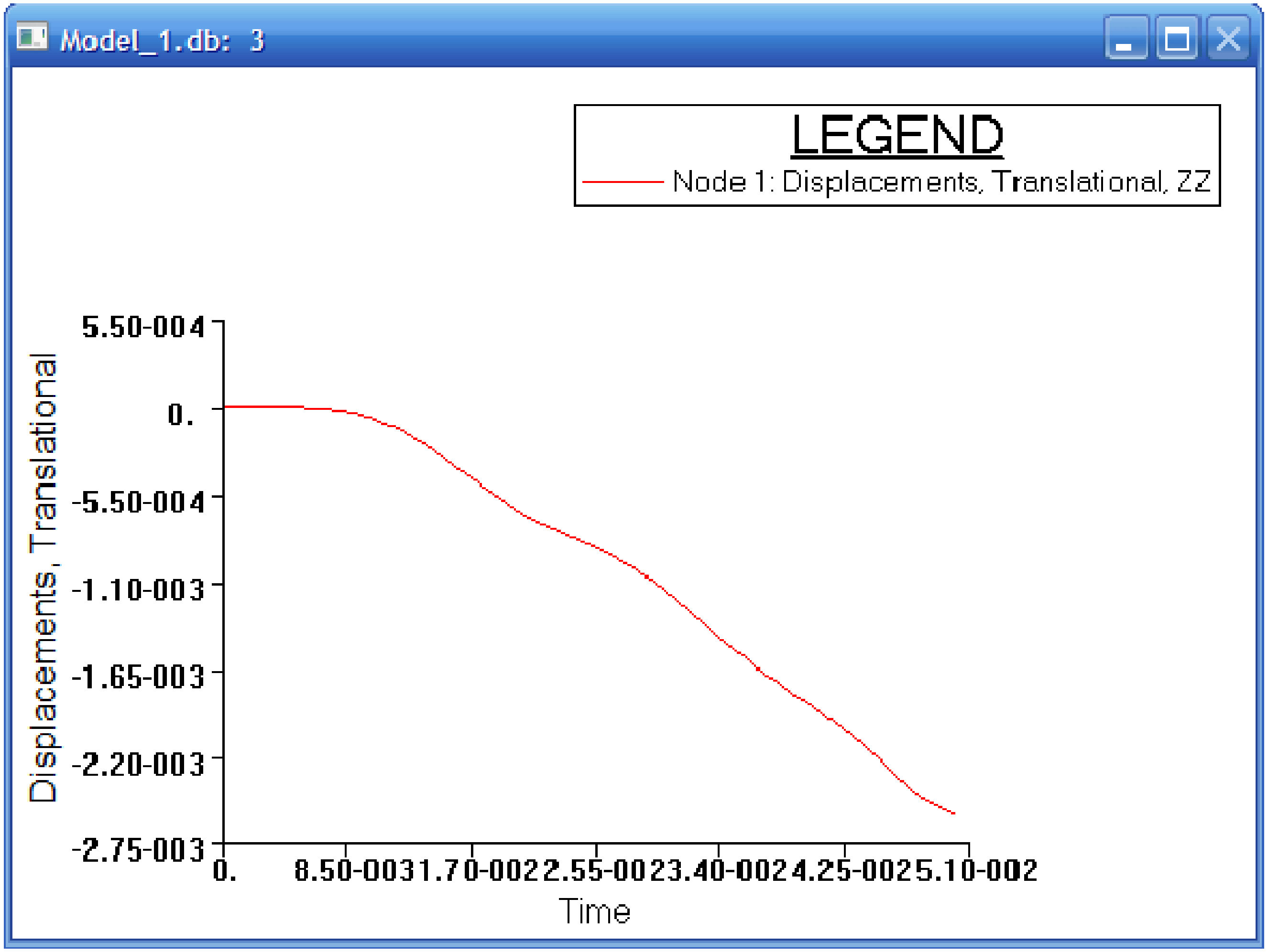






MSC Software

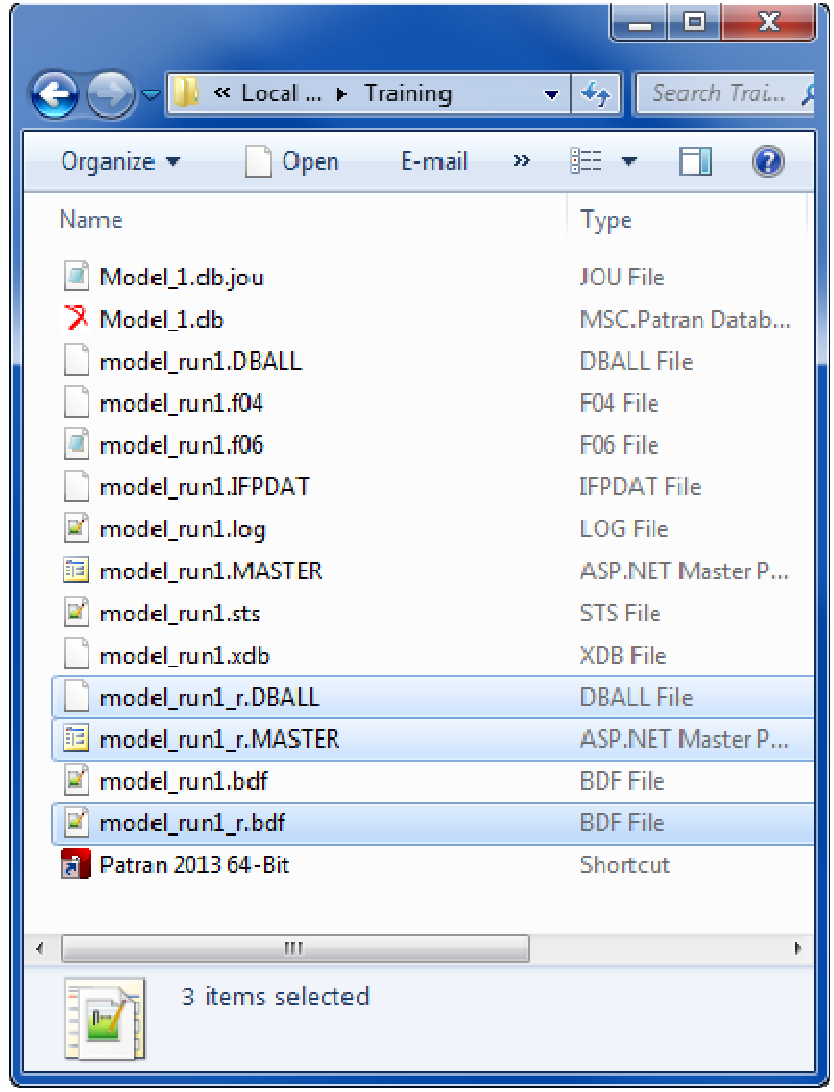
Time vs. Displacement in First Run
0 to 0.05 sec



Step 4. Copy the Result file for Restart

Make a copy of *.DBALL file for Restart run.

a. Copy **model_run1.DBALL, model_run1.MASTER** and **model_run1.bdf** paste it in same folder.
b. Rename them to **model_run1_r.DBALL, model_run1_r.MASTER** and **model_run1_r.bdf**
c. Open **model_run1_r.bdf**



Step 5. Modify Restart file

a. Add the following lines:

```
ASSIGN MASTER='model_run1_r.MASTER'
RESTART
SOL 400
CEND
NLRESTART
TITLE = MSC.Nastran job
ECHO = NONE
$ Direct Text Input for Global Case Control Data
SUBCASE 1
$ Subcase name : Default
ANALYSIS = NLTRAN
SUBTITLE=Default
TSTEPNL = 1
spc=11
disp(plot)=all
velocity(plot)=all
stress(plot)=all
STEP 1
  DLOAD = 2
STEP 2
  TSTEPNL = 20
  DLOAD = 2
BEGIN BULK
param,post,1
PARAM    LGDISP  1
TSTEPNL  1      50     .001          AUTO                UPW
TSTEPNL  20     50     .001
```

Note that the original Step should not be removed from the analysis.

Nonlinear Restart is not supported in SOL 400 if advanced elements are used.

MSC Software

Step 6. Run the Restart job.

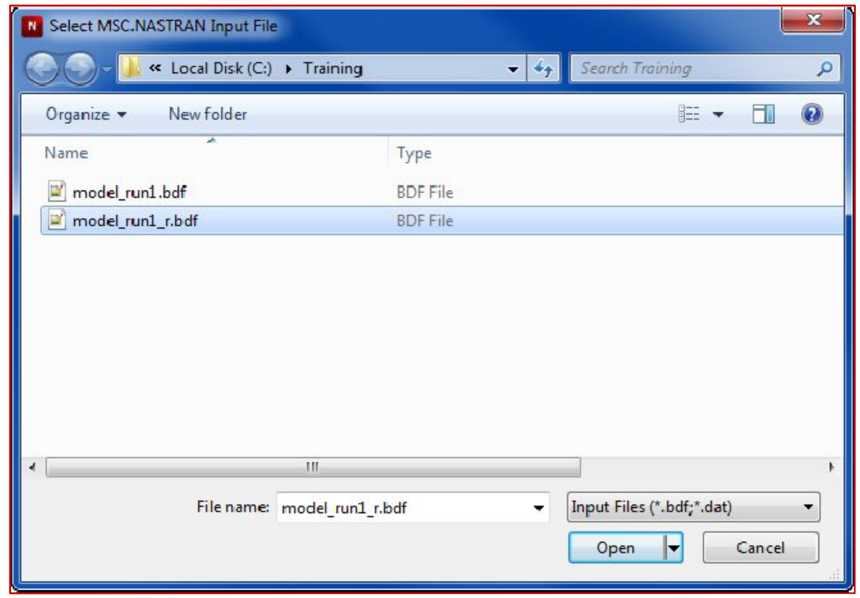
Submit the job to MSC Nastran.

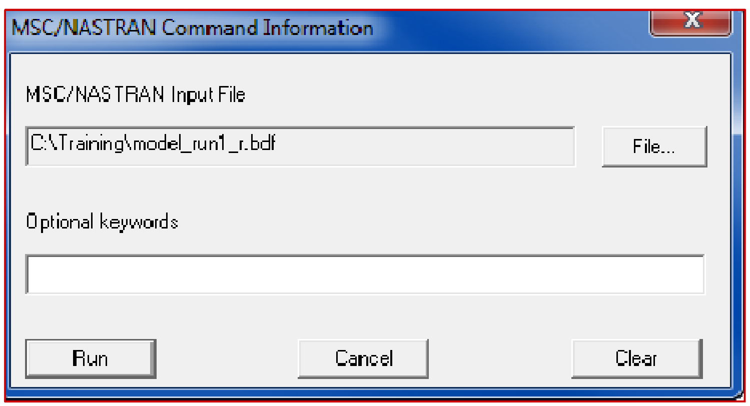
a. Open **MSC Nastran**

b. Select **model_run1_r.bdf**

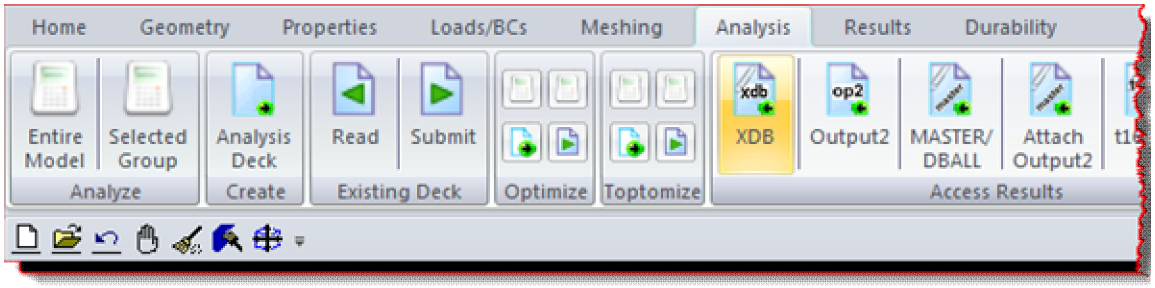
c. Click **Open**

d. Click **Run**



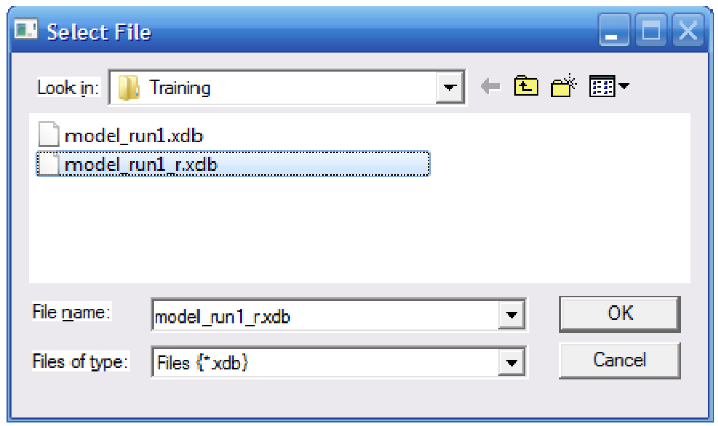


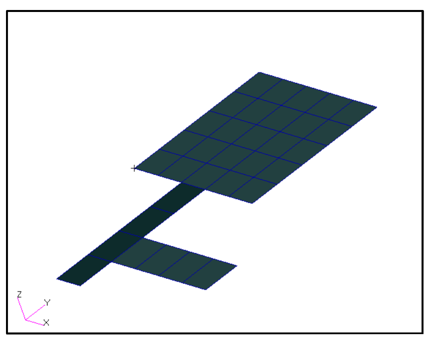
Step 7. View the Result in Patran

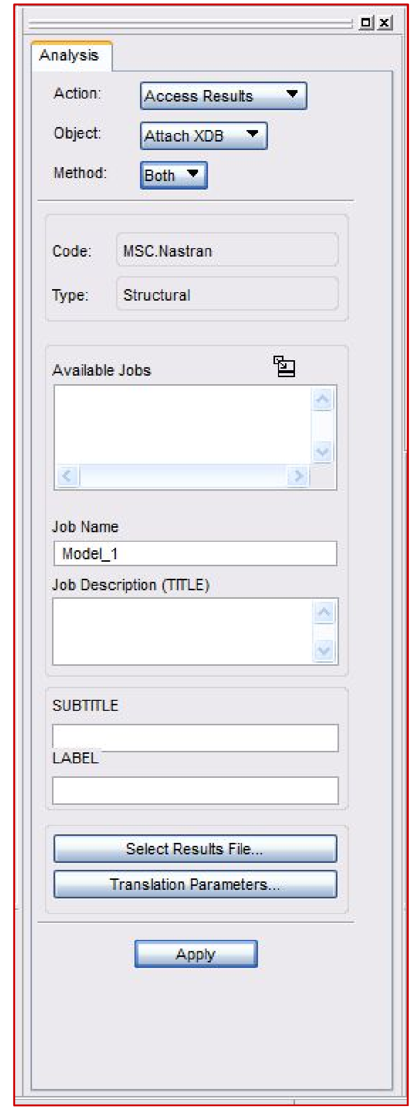


Open the *.xdb result file in Patran for post processing

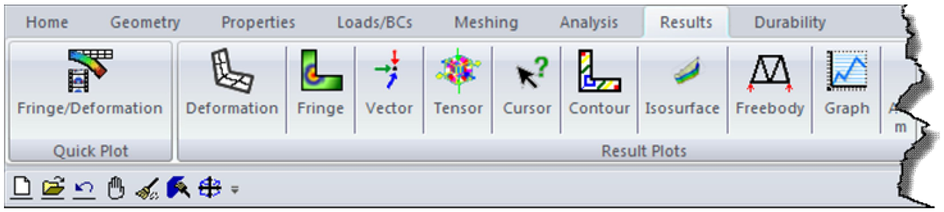
a. Under the *Analysis* tab, Click **XDB**, in the group Access Results
b. Pull down Method > **Both**
c. Click **Select Result File**
d. Select **model_run1_r1.xdb**
e. Click **OK**
f. Click **Apply**





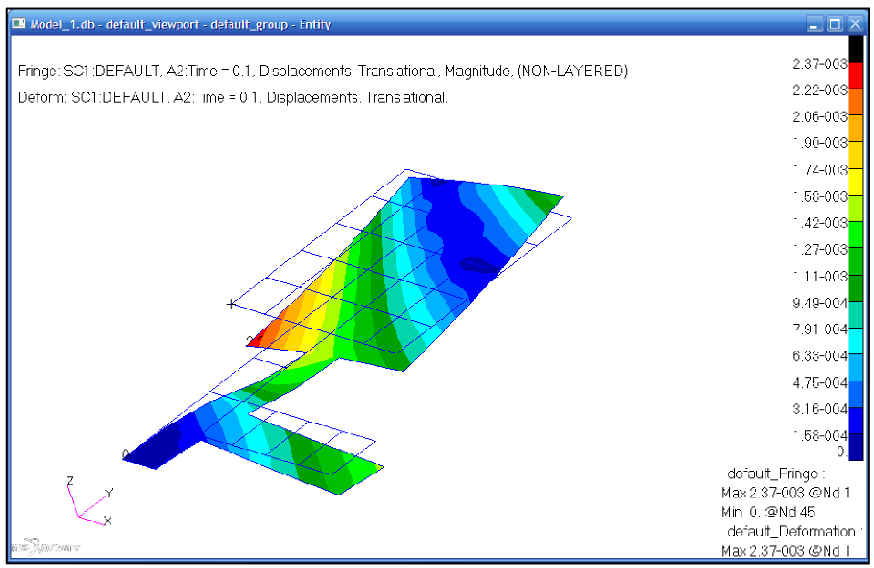


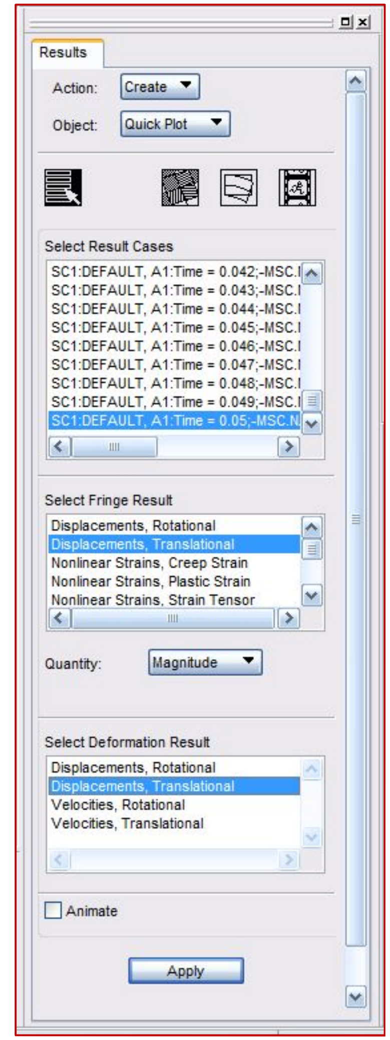
Step 7. View the Result in Patran (Cont.)



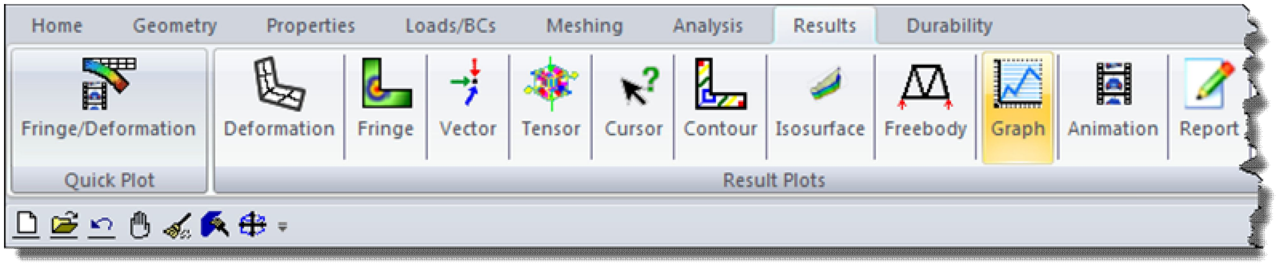
View the deforemed shape using quick plot in Patran.

a. Under the *Result* tab, click **Quick Plot.**
b. Select the last Result case, *Make note its at time = 0.1*
c. Select **Displacement**, **Translational** in *Select Fringe Result*
d. Select **Displacements, Translational** in *Select Deformation Result*
e. Click **Apply**





Step 7. View the result in Patran (Cont.)



Plot Graph of Displacement vs. Time

a. Under the *Result* tab, click **Graph**

b. Select **View all result Cases**

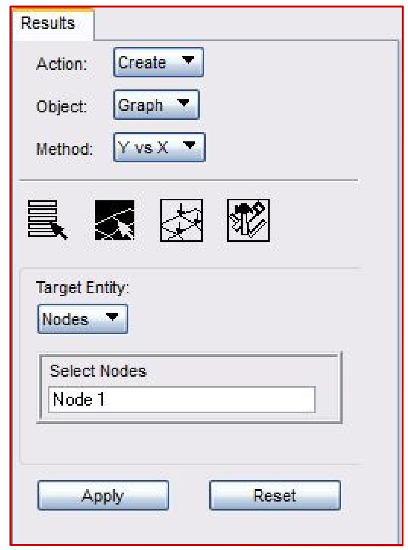
c. Select All result cases

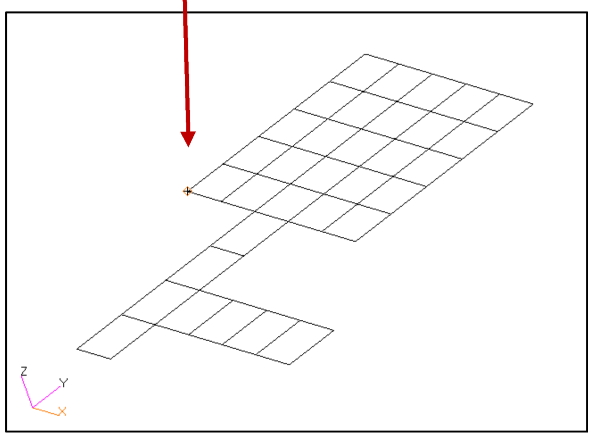
d. Select Displacement Translational

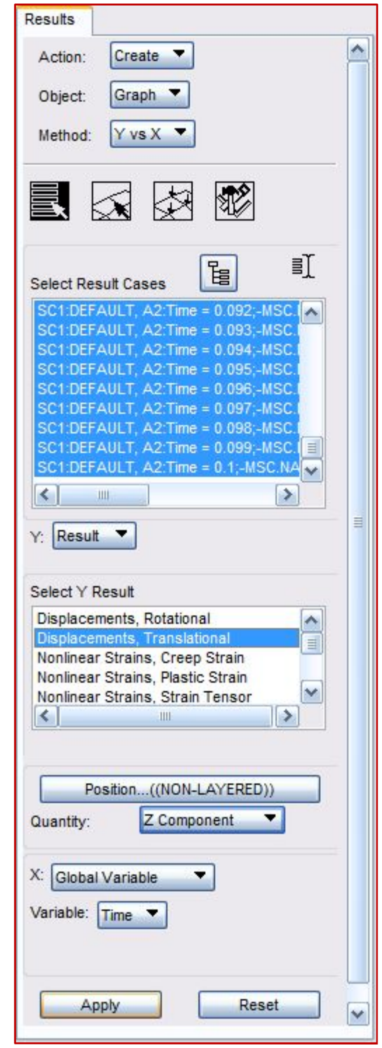
e. Pull down **Quantity > Z Component**

f. Select **Target Entity** tab

g. Select **Node 1**



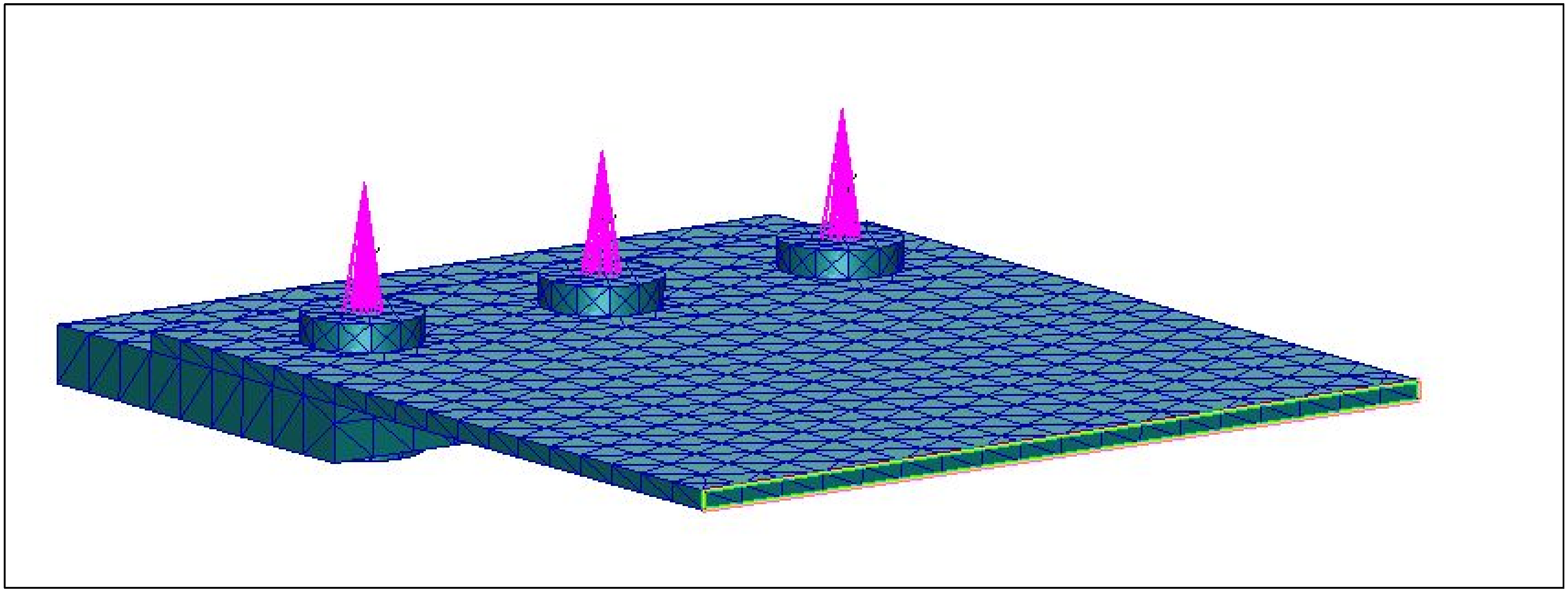




Plot Displacement vs. Time

MSC Software

WORKSHOP 10
BOLT MODELING



Workshop Objectives

- **Software Version**
  - MSC Nastran 2013.1
  - Patran 2013

- **Files Required**
  - bolted_plates.xmt

- **Problem Description**
  - Modeling Bolt preload using Patran and apply external load and analysis using SOL400.

- **Suggested Steps**
  1. Create a New database, call it bolt_modeling
  2. Import bolted_plates.xmt
  3. Define groups for plates and bolts to ease the work for further steps.
  4. Mesh the geometry one by one using TET10 elements.
  5. Modify the groups by Group Reorganize to assign mesh to its corresponding group.
  6. Define Material Properties: Steel. Elastic Modulus 30,000,000 psi, Poisson ratio 0.3
  7. Define Geometric Property: Solid Homogenous with Standard Formulation
  8. Define new Loadcase bolt_preload_step1
  9. Define 5 Deformable Contact Bodies based on groups.
  10. Define contact Pairs. Between plates, and between plates and bolts. Use Friction coefficient: 0.3
  11. Modeling Bolt Preload using Patran tools. with 450 Axial Bolt Load for each bolt.
  12. Define Boundary Condition, Clamp the lower plate
  13. Define Implicit Nonlinear job SOL400, request op2 output, activate quadratic contact, request output for all increments.
  14. Post process preloading step. plot displacement and Stresses.
  15. Define loadstep for Bolt_locking and make forces on bolts zero.
  16. Lock the bolts.
  17. Define a new loadcase for External load.
  18. Apply Total load on the tip of Upper Plate.
  19. Create a job implicit nonlinear for all three steps. preloading, locking, external loading.
  20. Attach OP2 result file and deattach the old result file.
  21. Plot Von Mises Stresses on bolts and plates after Appling the external load.
      - plot history of the reaction force on bolts

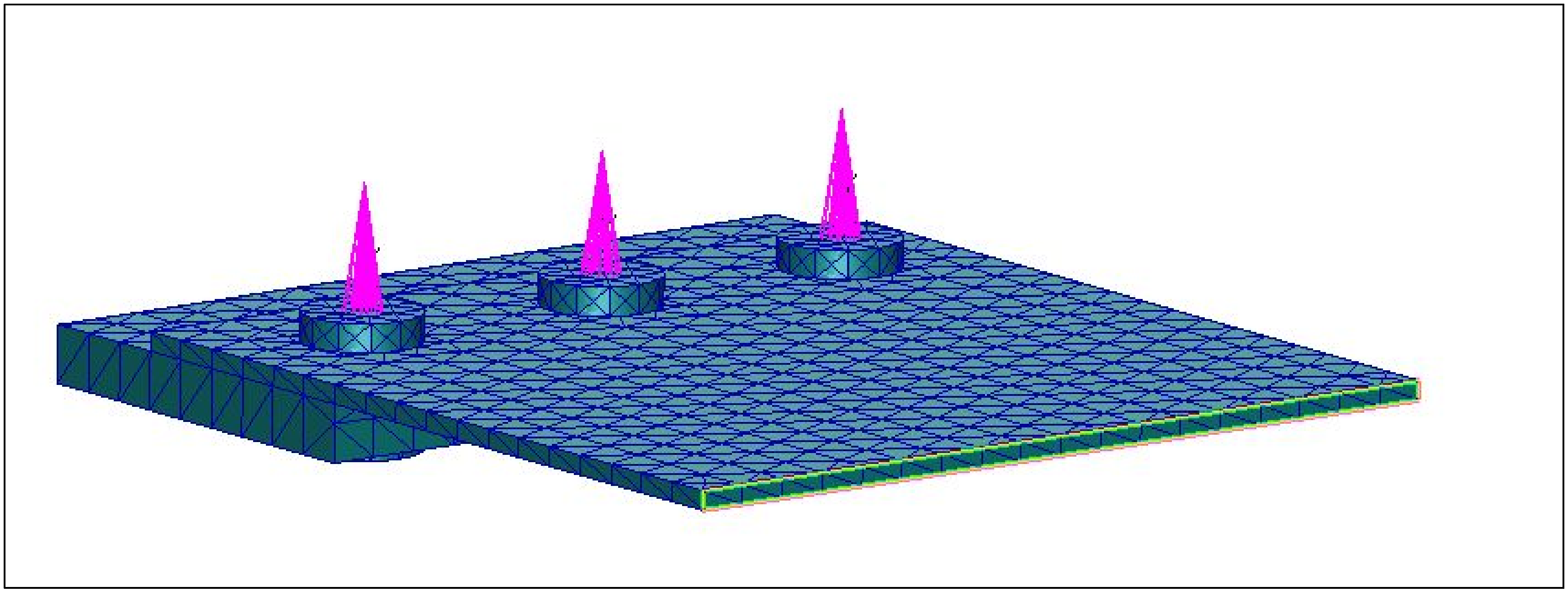
MSC Software

Bolt Modeling – 3 Steps

Step 1
Bolt Preload

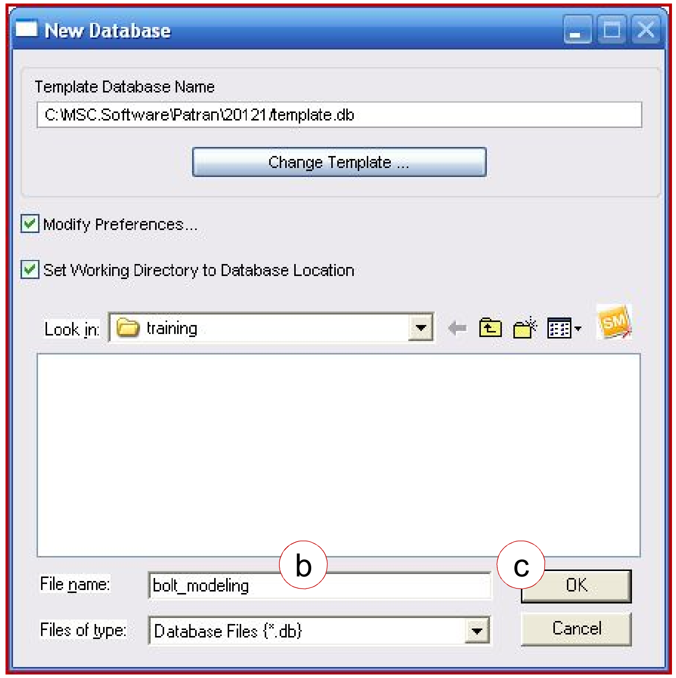
Step 2
Bolt Lock

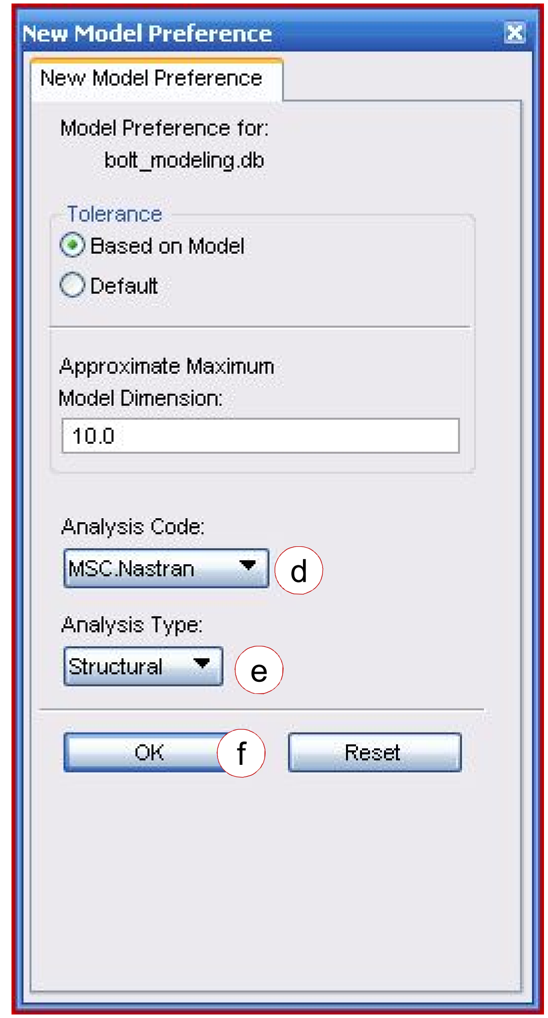
Step 3
External Load



Step 1. Create a New Database

a. Click **New** in the *Defaults* group
b. Enter **bolt_modeling** for *File_name*
c. Click **OK**
d. Pull down *Analysis Code* > **MSC.Nastran**
e. Pull down *Analysis Type* > **Structural**
f. Click **OK**





Step 2. Import the Model

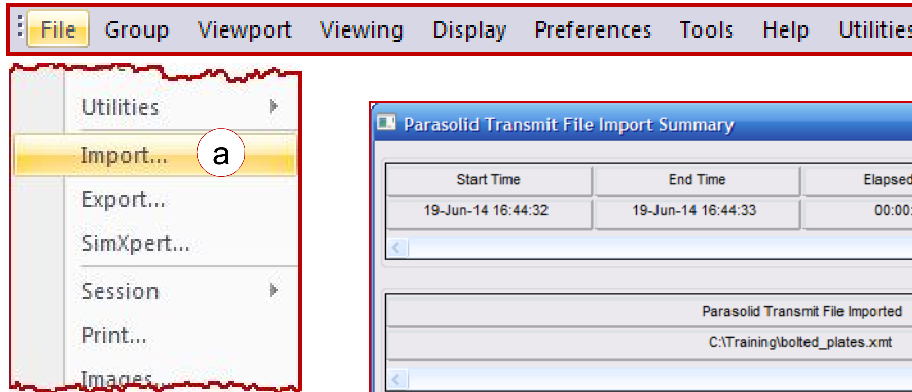
a. Pull down File > **Import**

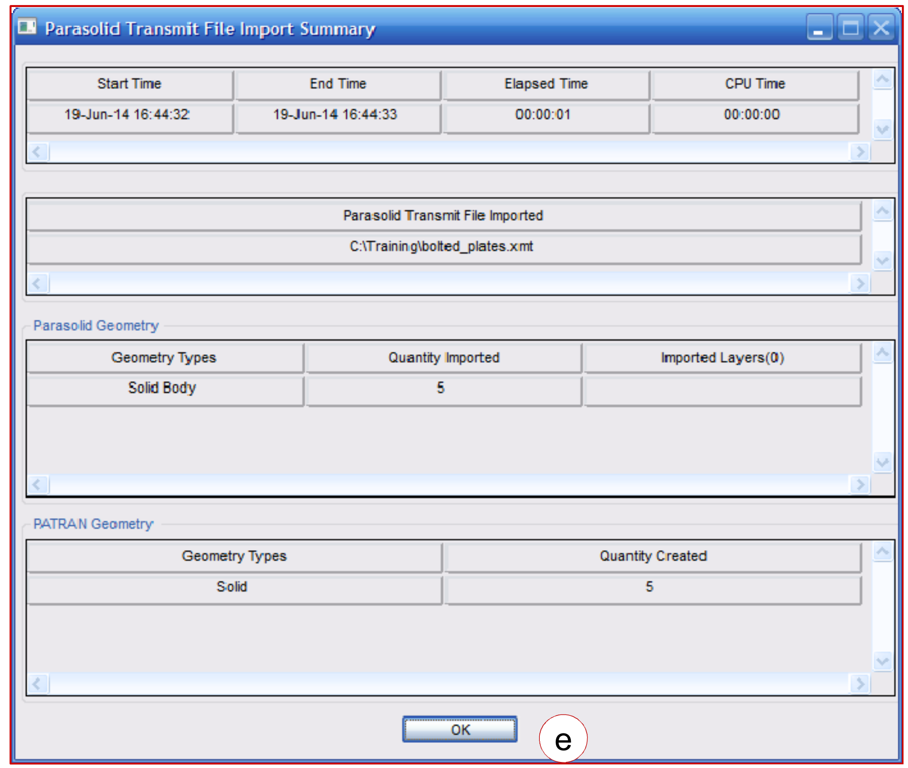
b. Pull down Source > **Parasolid xmt**

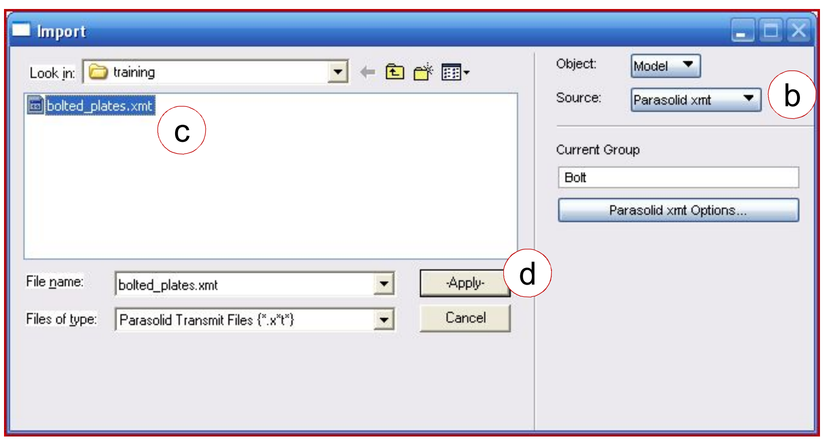
c. Select **bolted_plates.xmt**

d. Click **Apply**

e. Click **OK**







Step 3. Create Groups

Define groups for plates and bolts to ease the work for further steps.

a. Pull down Group > **Create**

b. Enter Name: **Upper**

c. Uncheck **Make Current** and check **Posted**

d. Click Entity Selection

e. Select Upper Plate (Solid 5)

f. Click **Apply.**

Do the same for the following

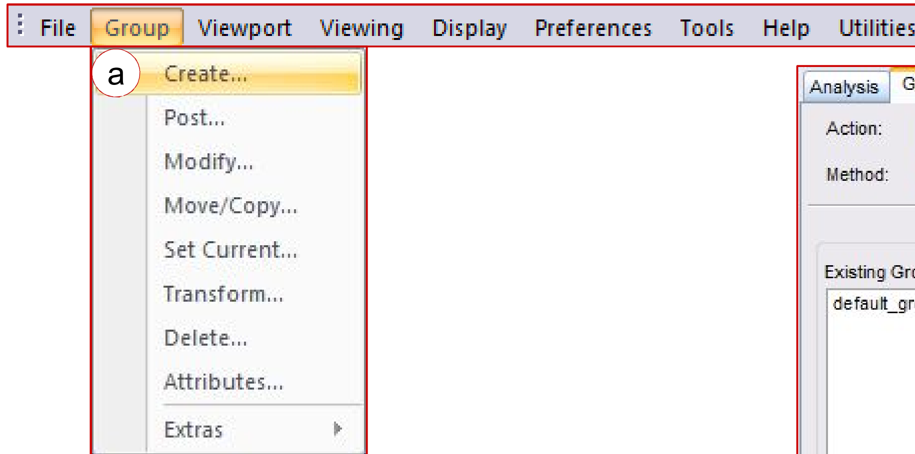
Lower: Solid 4

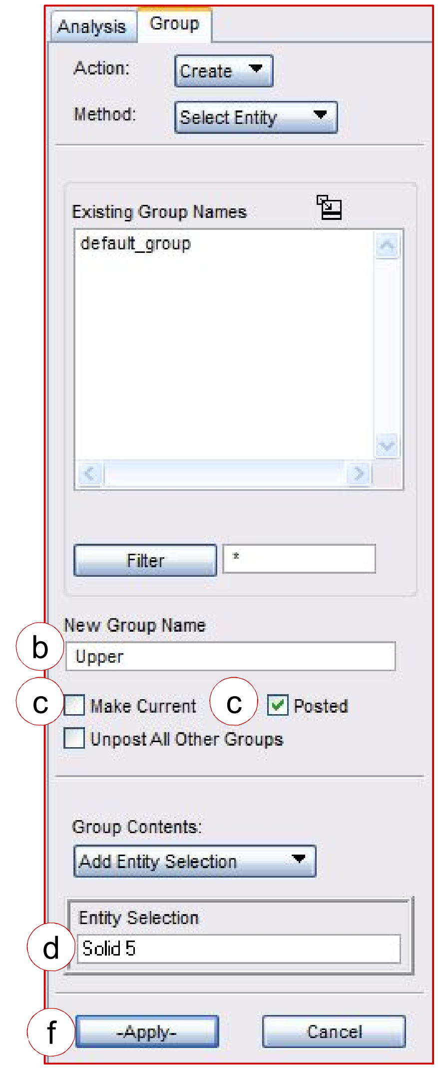
Left_bolt: Solid 2

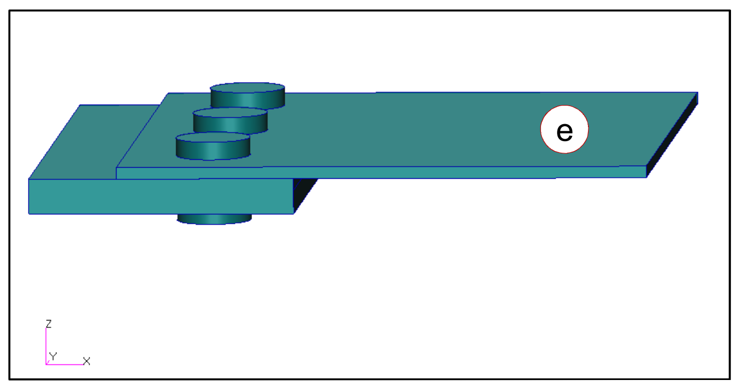
Mid_bolt: Solid 3

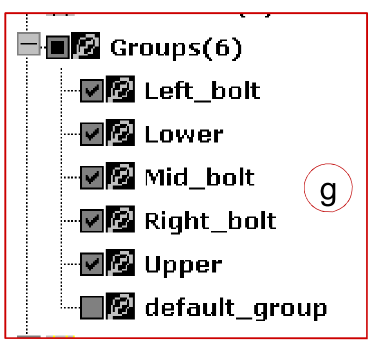
Right_bolt: Solid 1

g. In the Model Tree, there should be 6 groups.

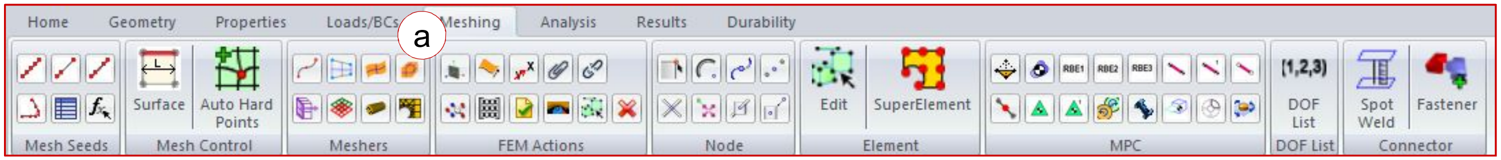






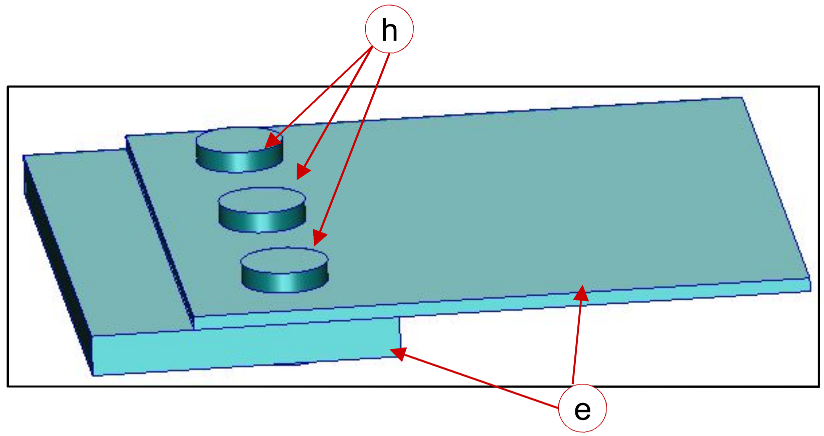


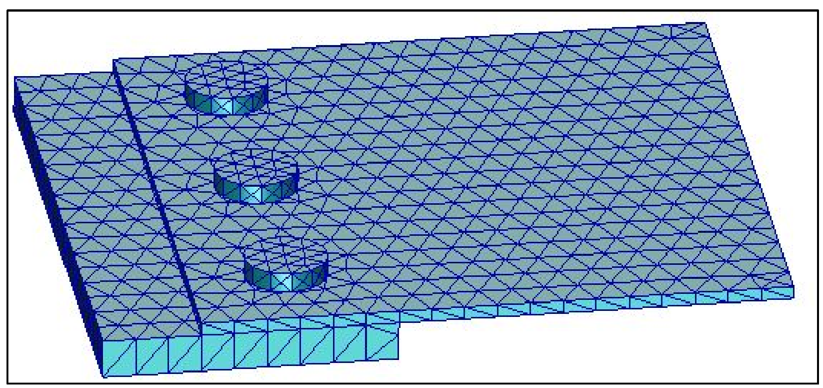
Step 4. Mesh The Geometry

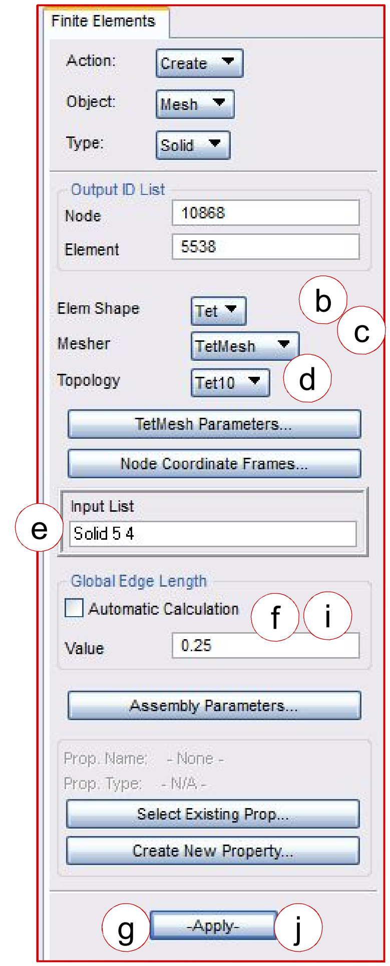


Mesh the geometry one by one using TET10 elements.

a. Under the *Meshing* tab, click **Solids**, in the *Meshers* group.
b. Pull down *Element Shape* > **Tet**
c. Pull down *Mesher* > **TetMesh**
d. Pull down *Topology* > **Tet10**
e. Select the *two plates*
f. Uncheck **Automatic Caluclation** and use Value **0.25**
g. Click **Apply**.
h. Select *Three Bolts.*
i. Enter *Global Length Value:* **0.15**
j. Click **Apply**







Step 5. Group Reorganize

a. Notice that the current group is the **default_group**.

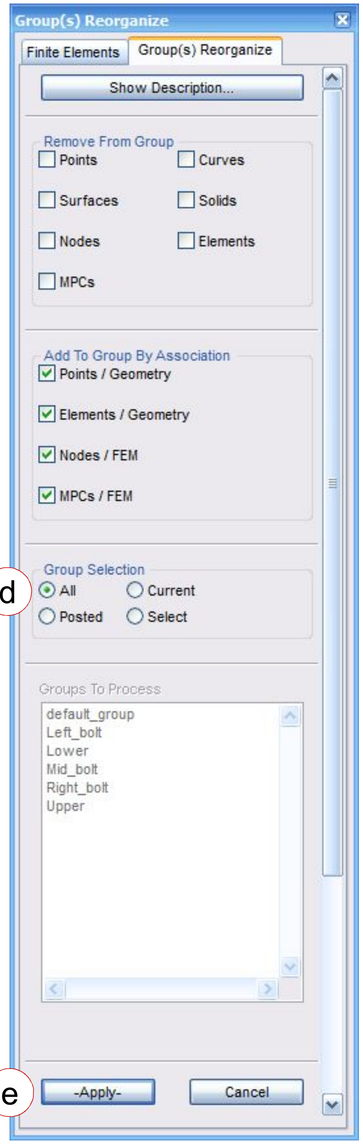
b. In the Model Tree, Uncheck Groups and then individually check each of 6 groups to post and notice that the elements are just assign to the **default_group.**

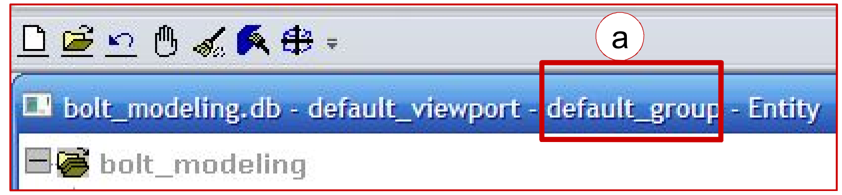
c. Pull down Utilities > Group > **Group Reorganize …**

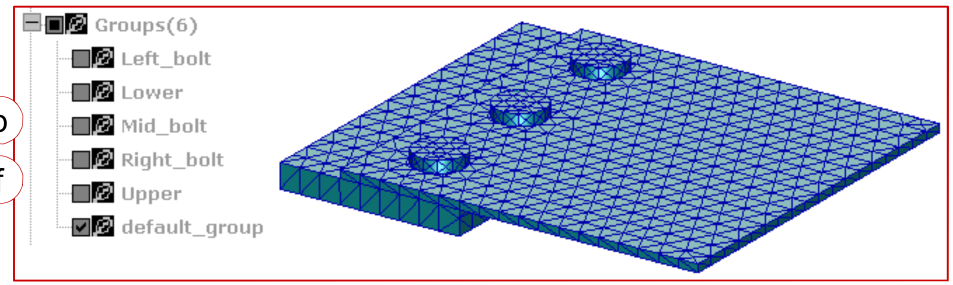
d. Select **All** in the Group Selection.

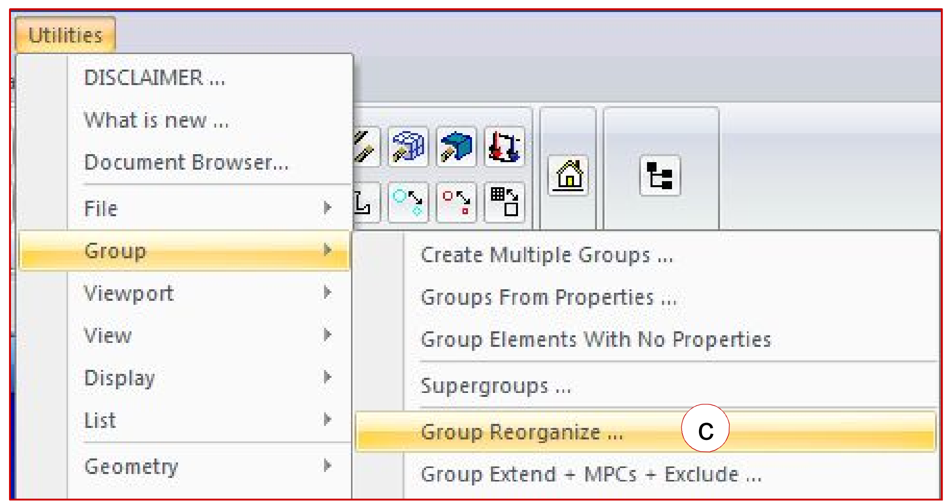
e. Click **Apply.**

f. Repeat Step **b** to check every group is associated with its own elements.

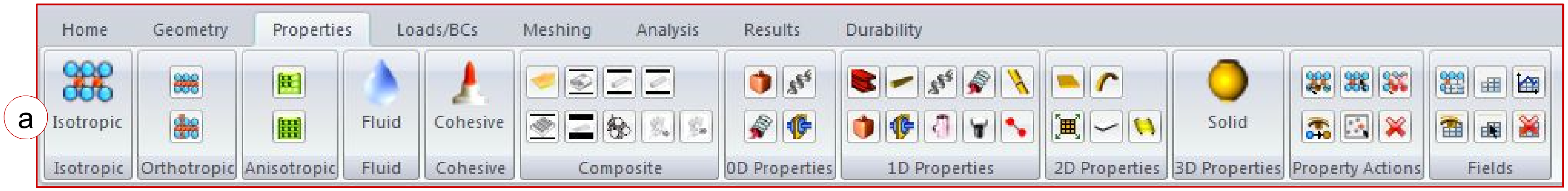






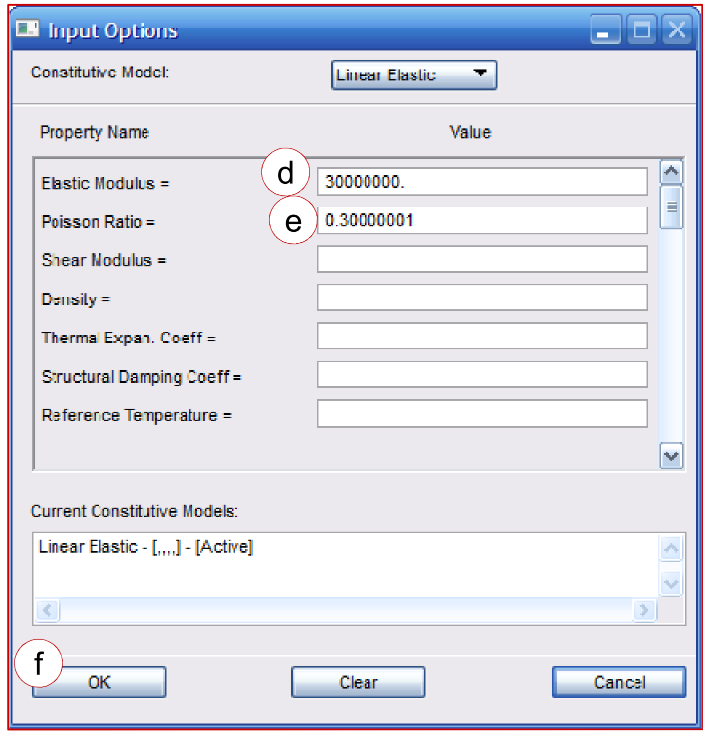


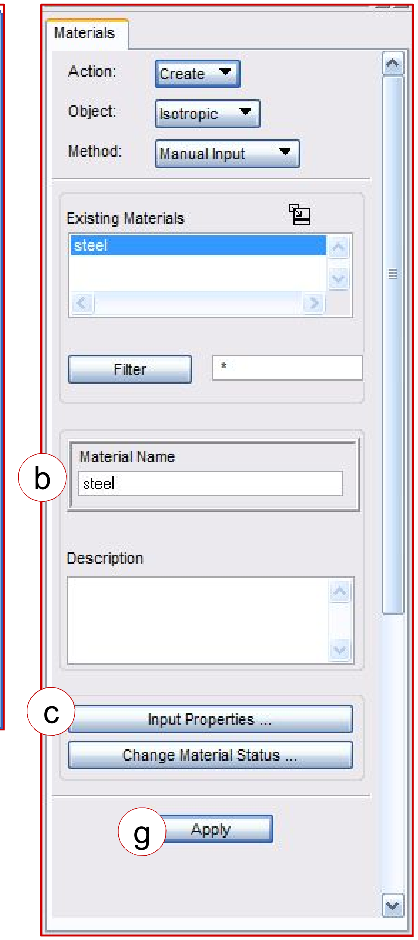
Step 6. Define Material Properties



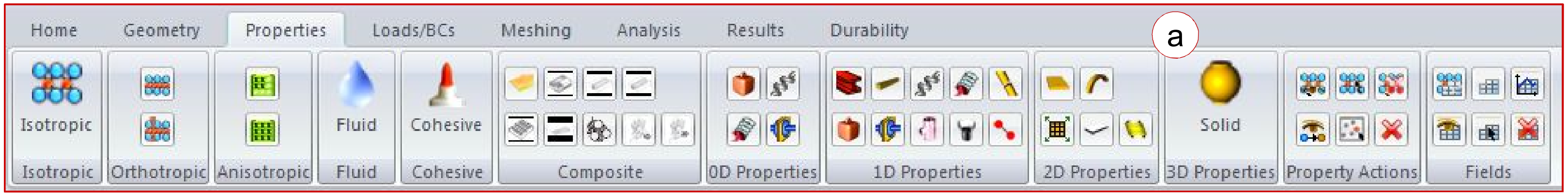
Define Material Properties

a. Under the *Properties* tab, click **Isotropic**
b. Enter Name **Steel**
c. Click **Input Data**
d. Enter **30000000** for *Elastic Modulus.*
e. Enter **0.3** for *Poisson Ratio*
f. Click **OK**
g. Click **Apply**



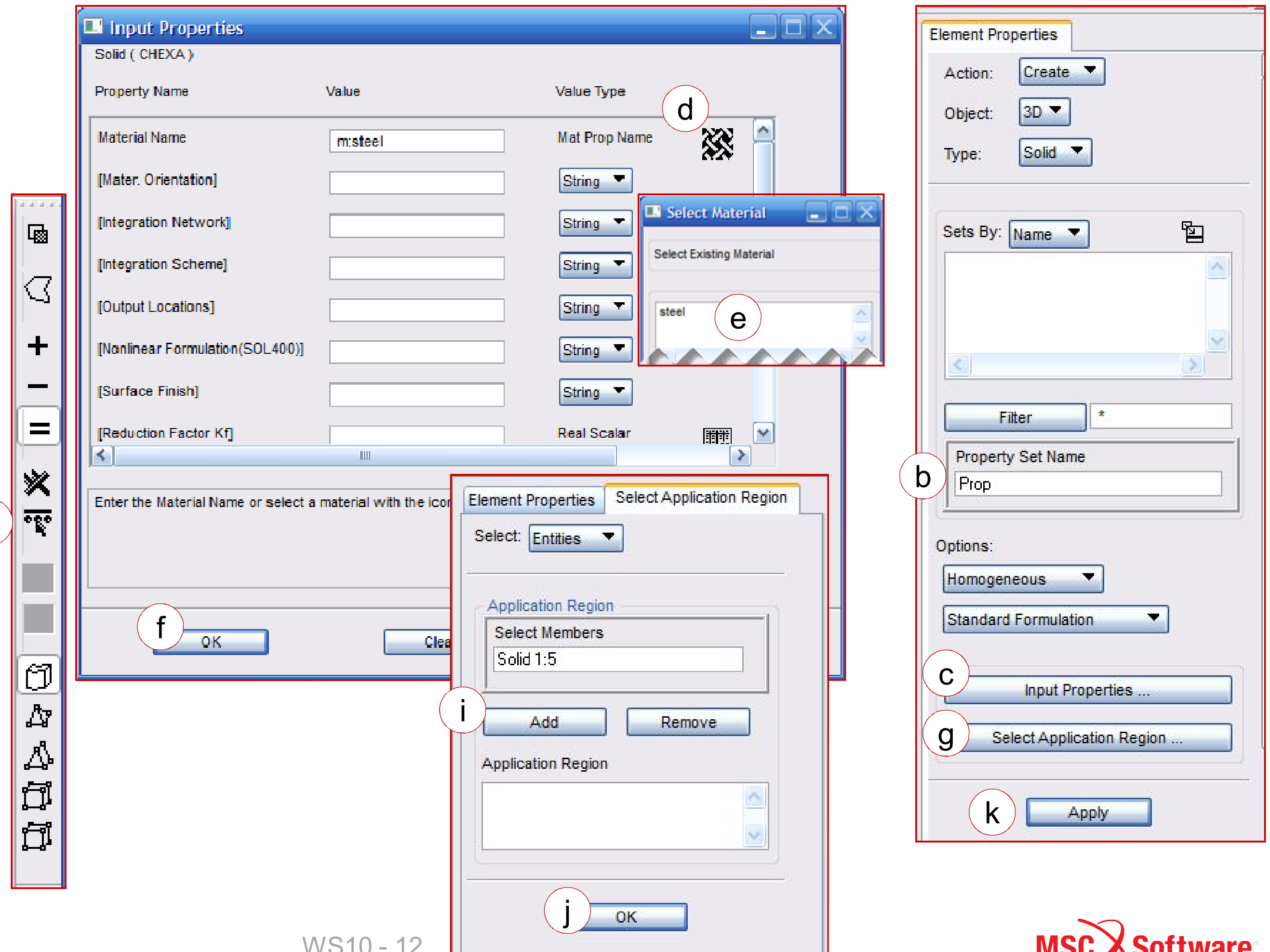


Step 7. Define Geometric Properties

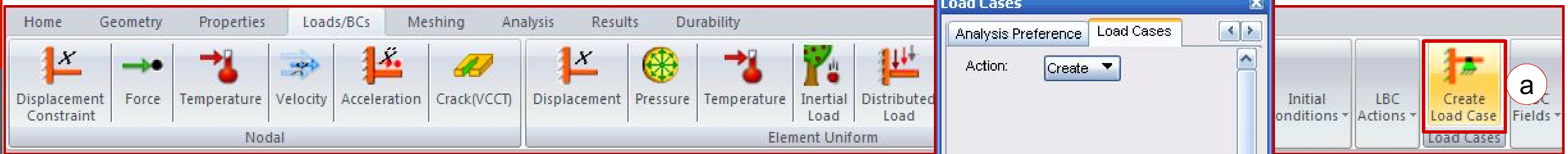


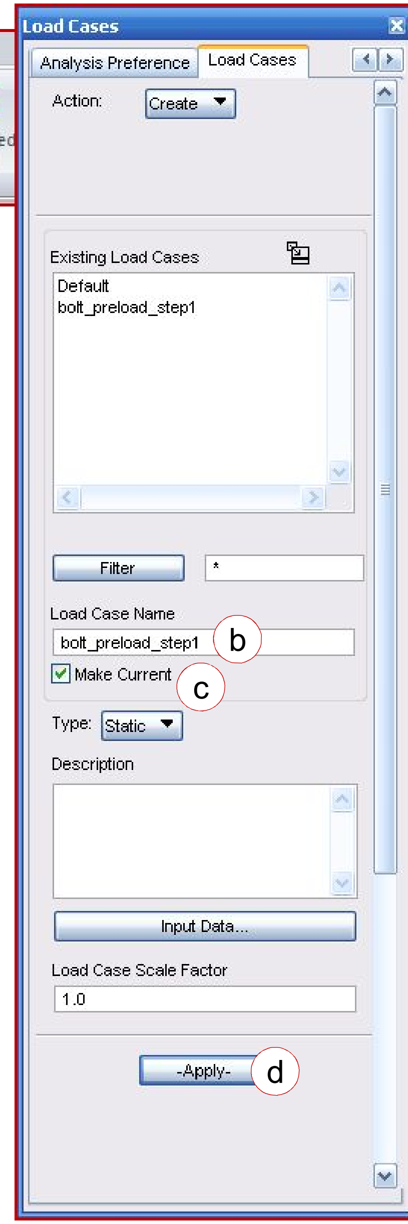
Define Solid geometric properties as Solid.

a. Click **Solid** in *3D Properties* group.
b. Enter **Prop** for *Name*
c. Click **Input Properties**
d. Click **Material** Icon
e. Select **Steel**
f. Click **OK**
g. Click **Select Application Region**
h. Select *Pick All*
i. Click **Add**
j. Click **OK**
k. Click **Apply**



Step 8. Create a New Loadcase





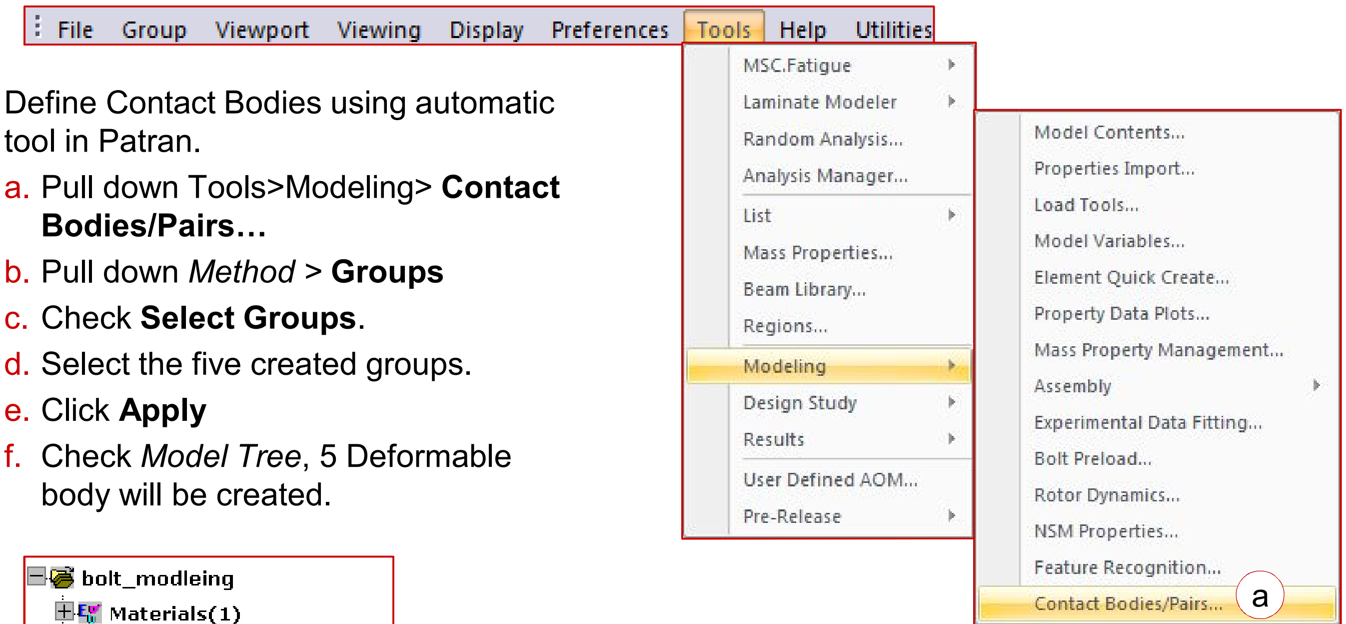
a. Under the Loads/BCs tab, click **Create Load Case** in the *Load Cases* group
b. Enter **bolt_preload_step1** for *Load Case Name*
c. Check **Make Current**
d. Click **Apply**

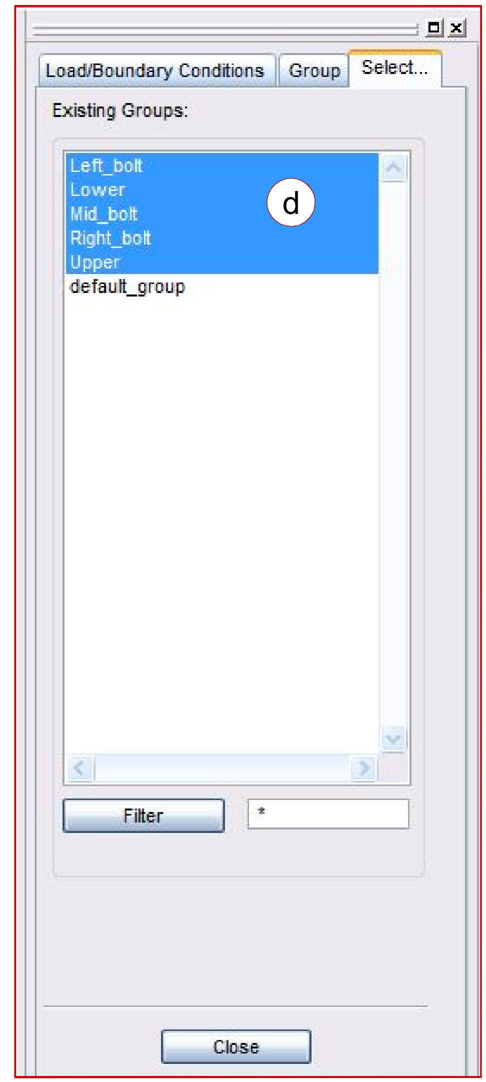
MSC Software

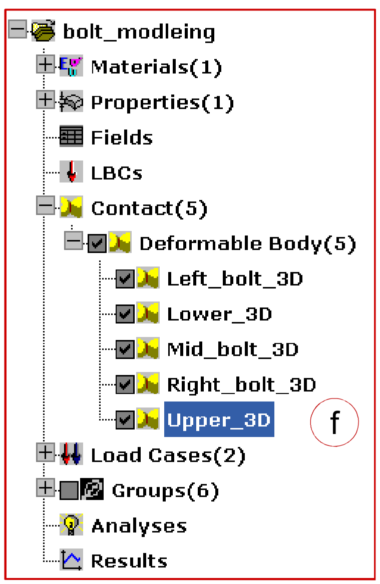
Step 9. Define Contact Bodies

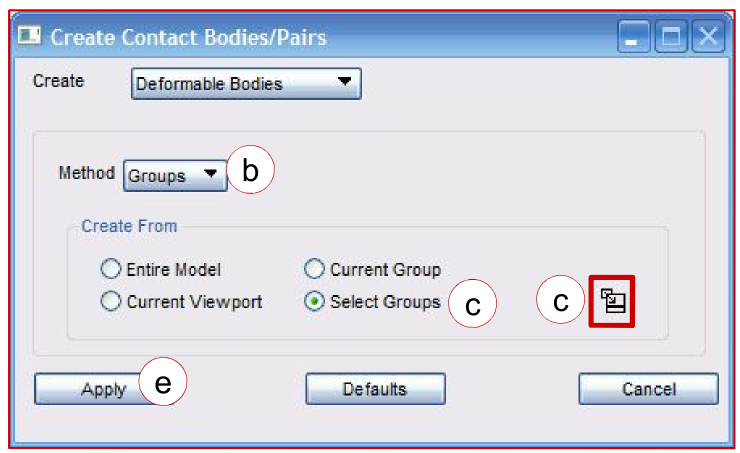
Define Contact Bodies using automatic tool in Patran.

a. Pull down Tools>Modeling> **Contact Bodies/Pairs…**
b. Pull down *Method* > **Groups**
c. Check **Select Groups**.
d. Select the five created groups.
e. Click **Apply**
f. Check *Model Tree*, 5 Deformable body will be created.





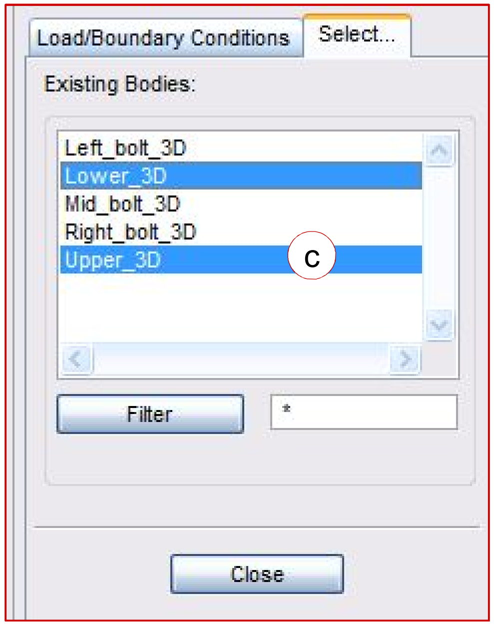


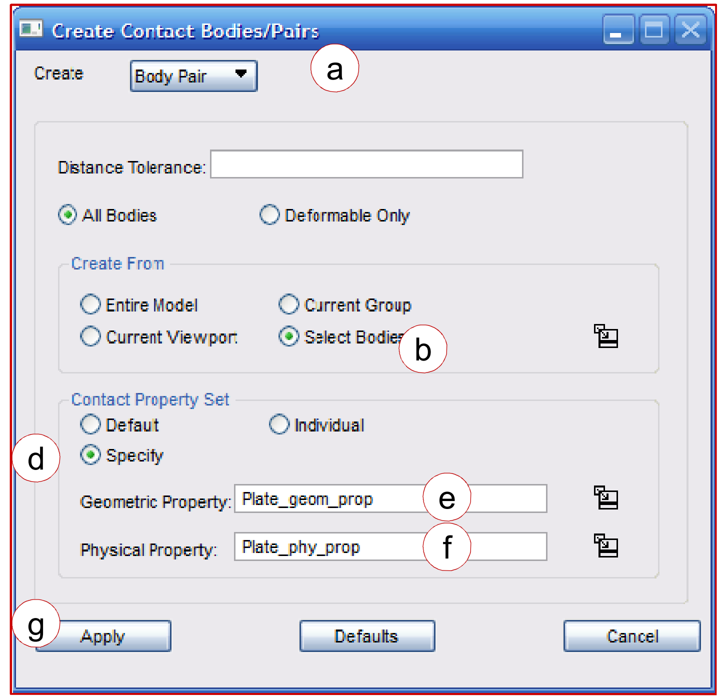


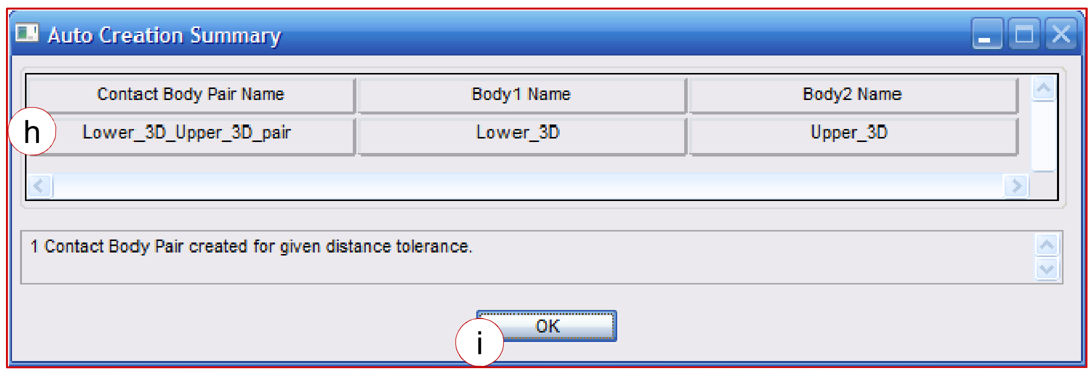
Step 10. Define Contact Pairs

Define contact pair between two plates.

a. Pull down Create > **Body Pair**
b. Select **Select Bodies**
c. Select **Lower_3D and Upper_3D** from *Existing Bodies list.*
d. Select **Specify** in *Contact Properties Set*
e. Rename Geometric Property to **Plate_geom_prop**
f. Rename Physical Property to **Plate_phy_prop**
g. Click **Apply**
h. One *Contact Pair* gets generated
i. Click **OK**





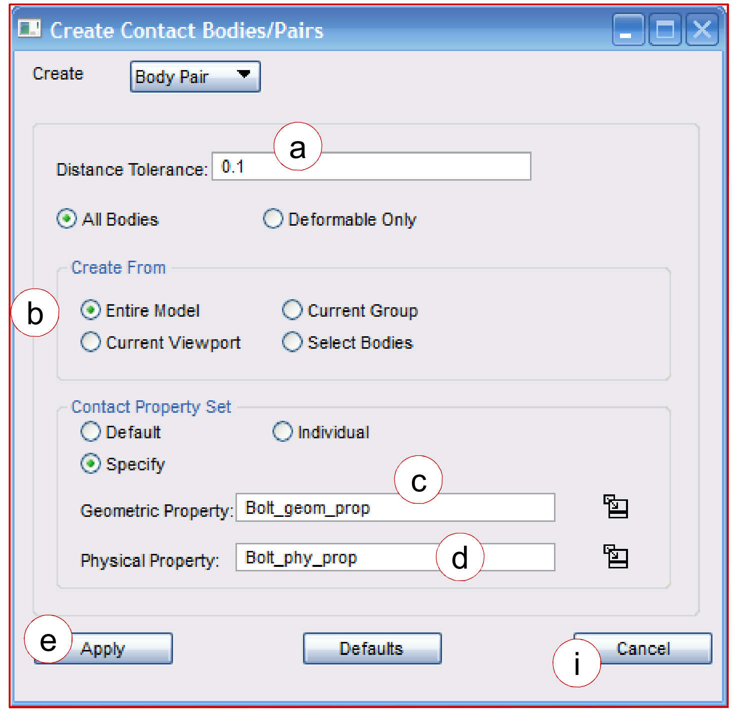


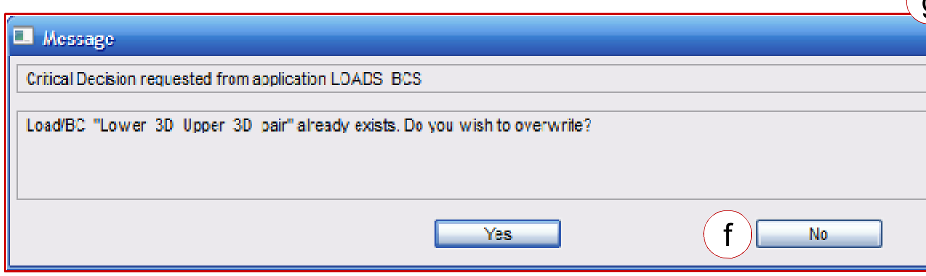
MSC Software

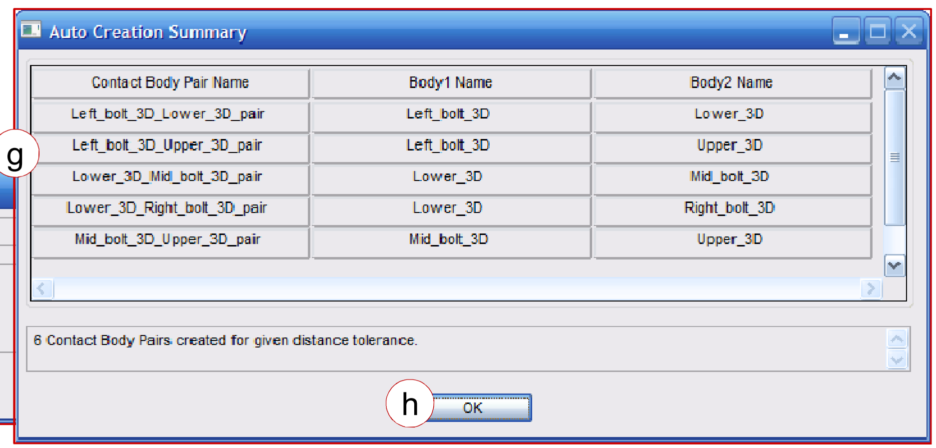
Step 10. Define Contact Pairs (Cont.)

Define contact pair between two plates.

a. Enter **0.1** for *Distance Tolerance*
b. Select **Entire Model**
c. Rename *Geometric Property* to **Bolt_geom_prop**
d. Rename *Physical Property* to **Bolt_phy_prop**
e. Click **Apply**
f. Click **No** on Message for overwriting pair.
g. 6 Pairs gets generated.
h. Click **OK**
i. Click **Cancel** to close the Create Pair window.





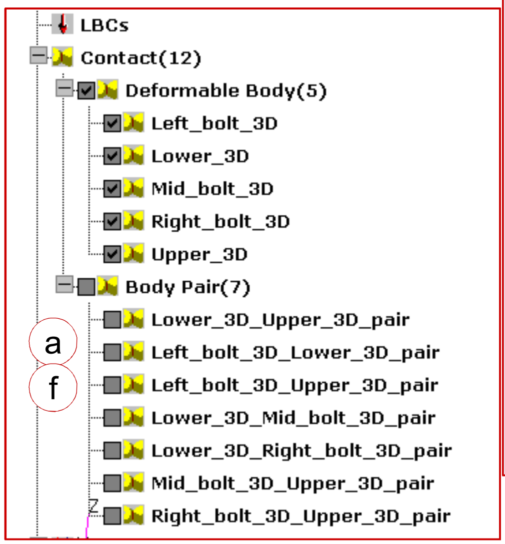


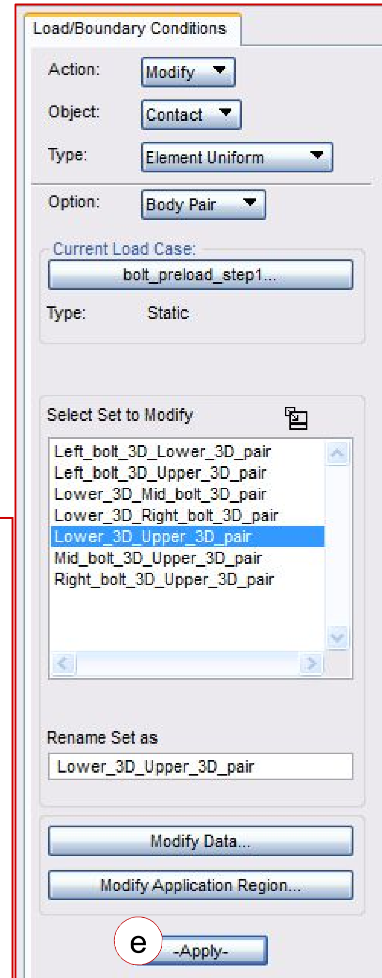
Step 10. Define Contact Pairs (Cont.)

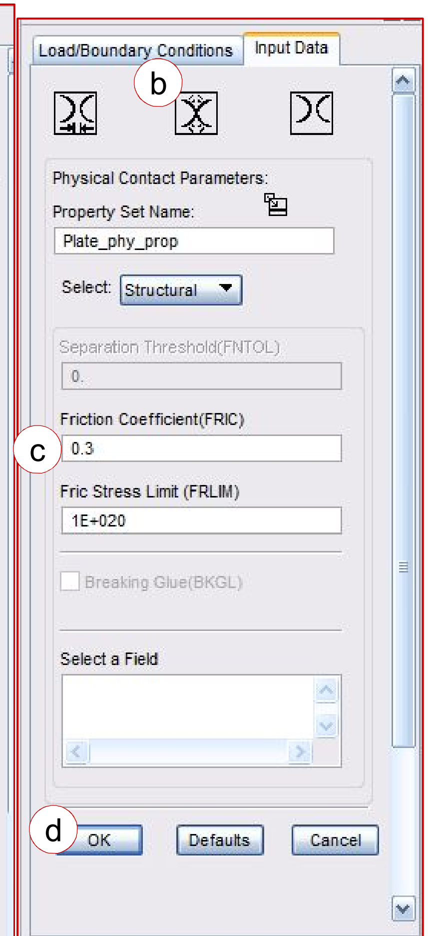
Add friction coefficient between the plates.

a. Double-click **Lower_3D_Upper_3D_pair**
b. Click **Physical Prop.**
c. Enter **0.3** Friction Coefficient
d. Click **OK**
e. Click **Apply**
f. Double-click **Left_bolt_3D_Lower_3D_pair**

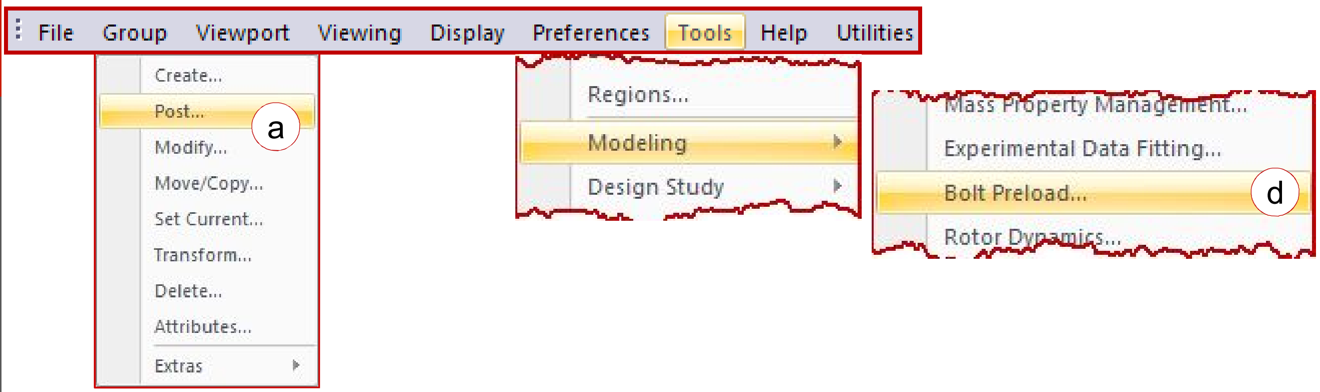
Repeat step **b, c d** and **e** to add Friction coefficient of 0.3 between bolts and the plates.



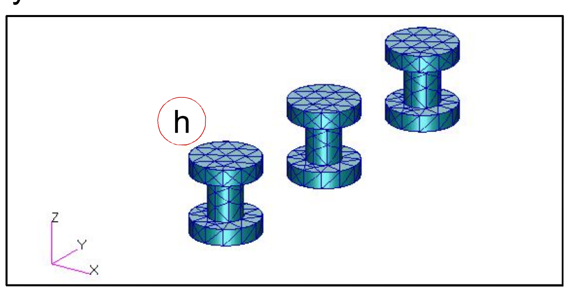


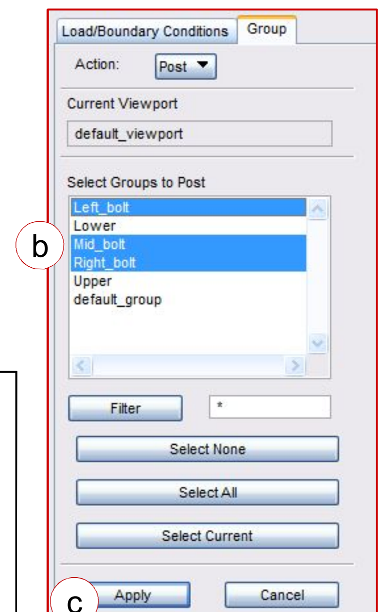


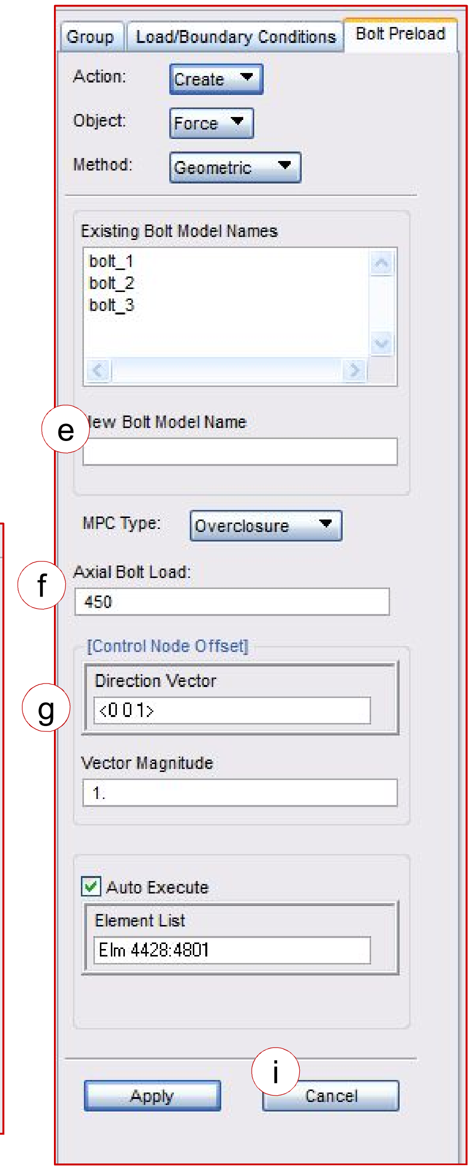
Step 11. Modeling Bolt Preload



a. Pull down Group > **Post**
b. Select **Left_bolt, Mid_Bolt, Right_bolt**.
c. Click **Apply**
d. Pull down Tools > Modeling > **Bolt Preload**
e. Enter **bolt_1** for *New Bolt Model Names*
f. Enter **450** for *Axial Bolt Load*
g. Enter **<0 0 1>** for *Direction Vector*
h. Screen select all of the elements of **one bolt**, use the CTRL to draw arbitrary selection. Do the same for every bolt.
i. Click **Cancel**

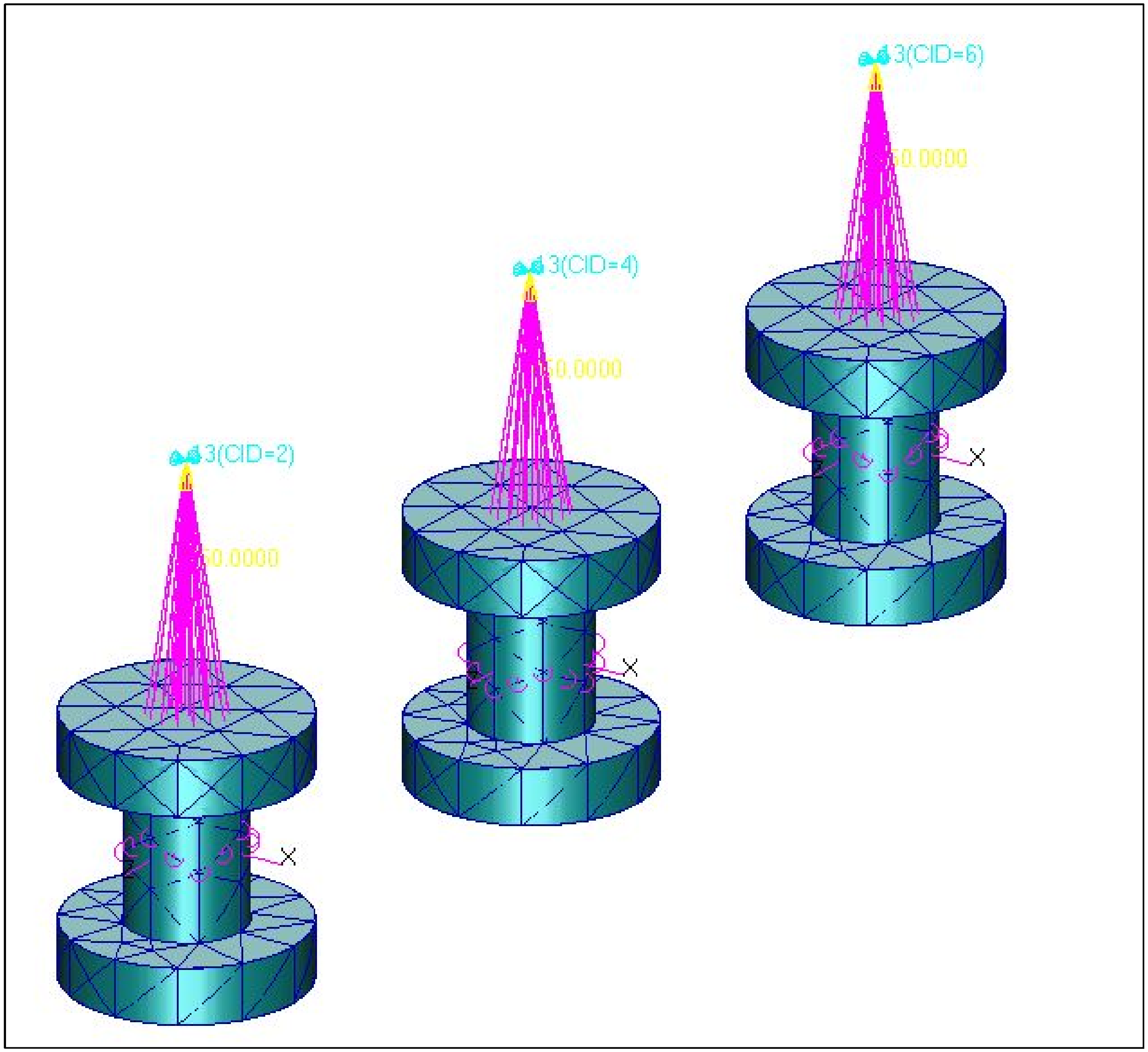




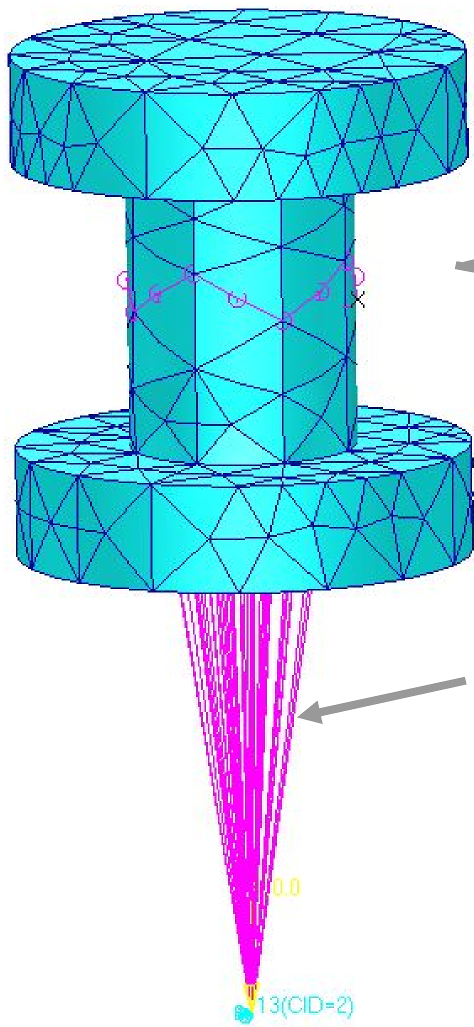


MSC Software

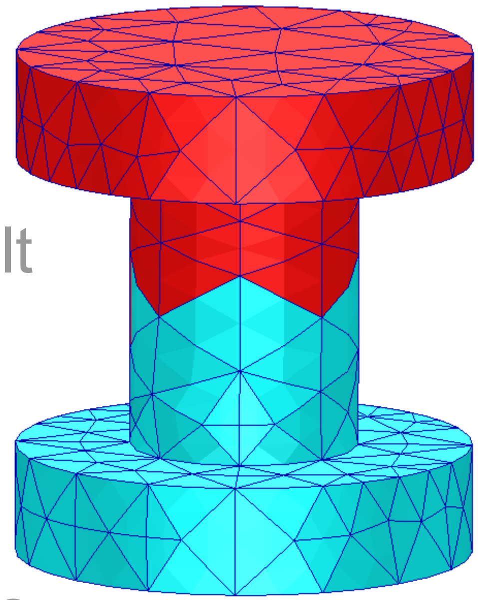
Preloaded Bolt Defined



Step 11. Modeling Bolt Preload (Cont.)

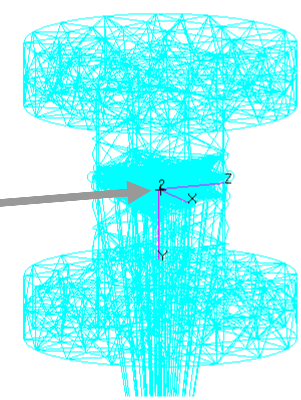


1. Splits Bolt



2. Creates MPCs tied to Control Node

3. Creates Local Coordinate Frame



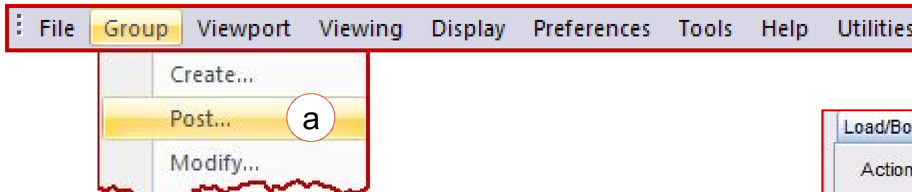
Step 12. Define Boundary Conditions

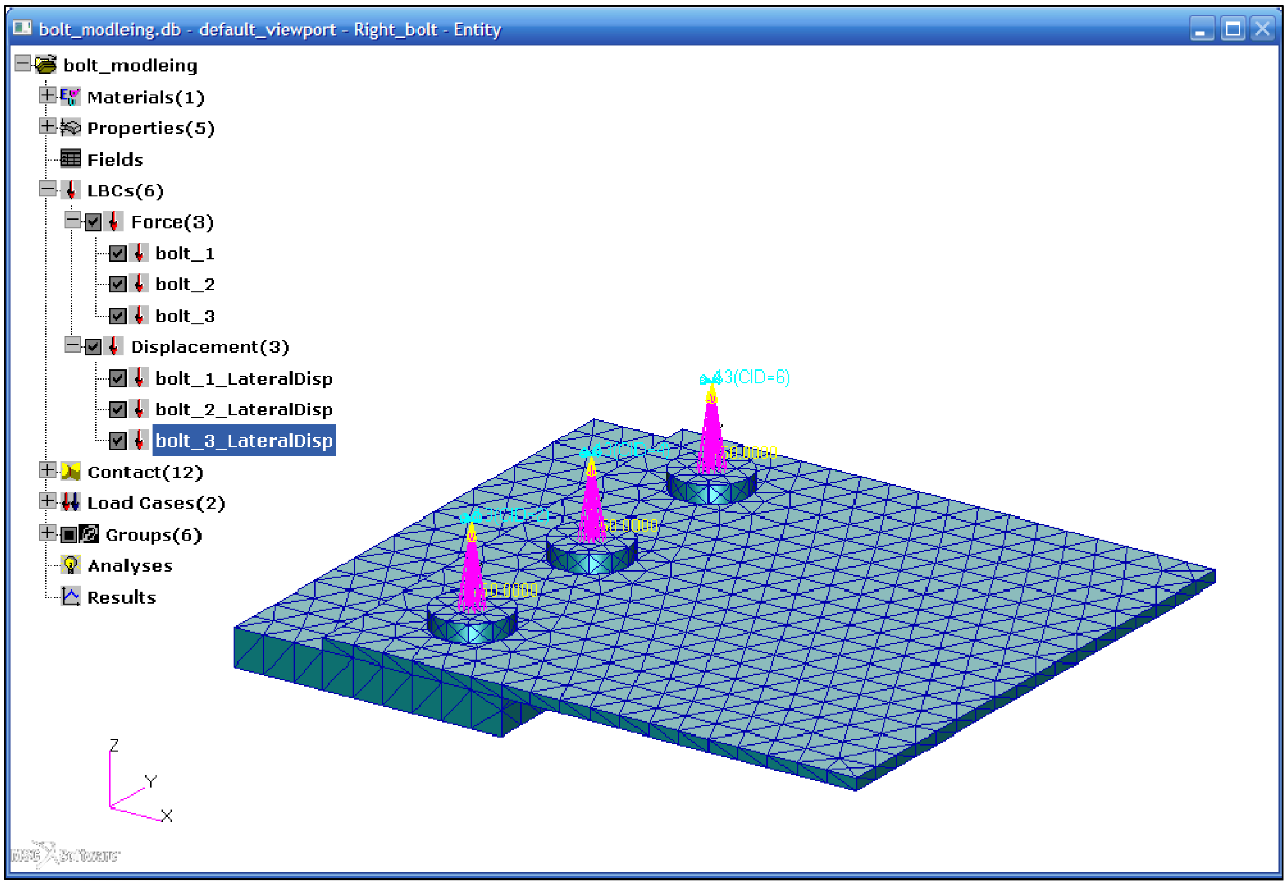
Post all of the Groups

a. Pull down Group > **Post**

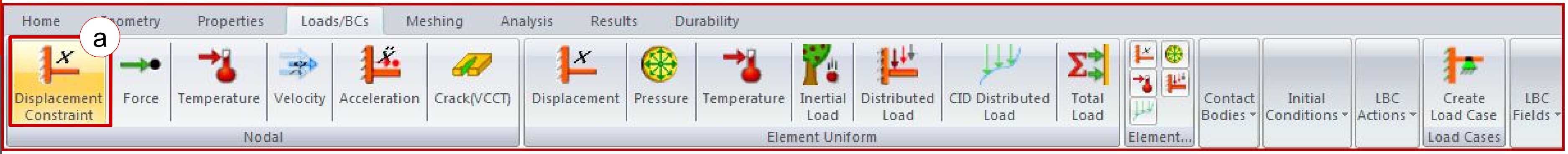
b. Select all the five groups

c. Click **Apply**

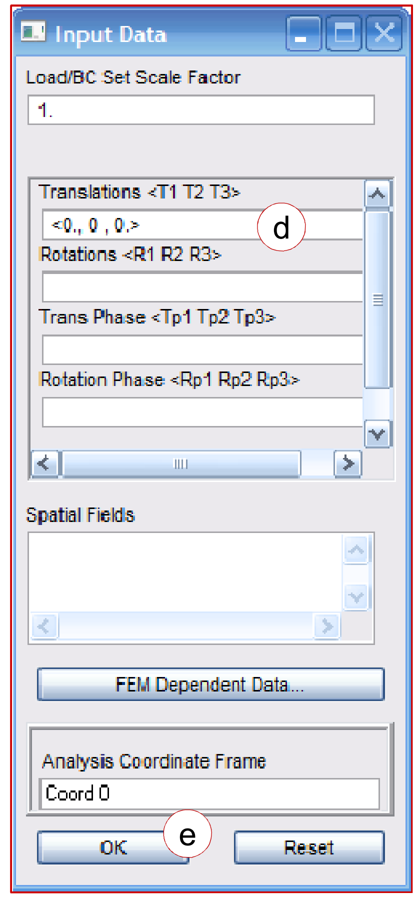


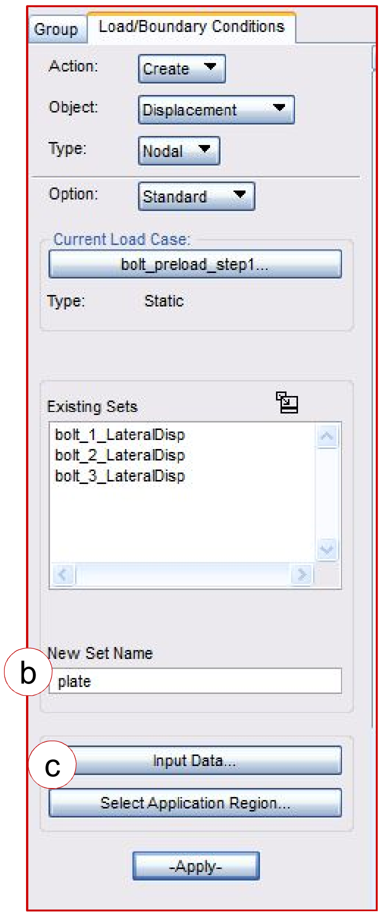


Step 12. Define Boundary Conditions (Cont.)



a. Under the *Loads/BCs* tab, click **Displacement Constraint** in the *Nodal* group
b. Enter **plate** for *New Set Name*
c. Click **Input Data…**
d. Enter **<0,0,0>** for *Translations*
e. Click **OK**

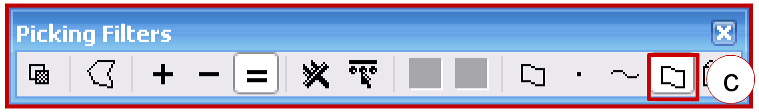


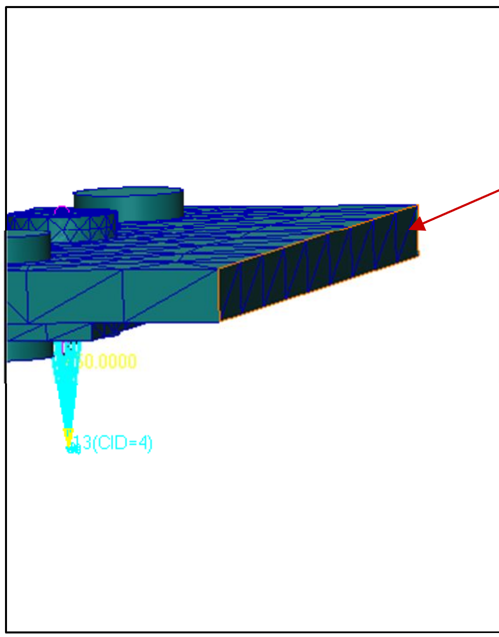


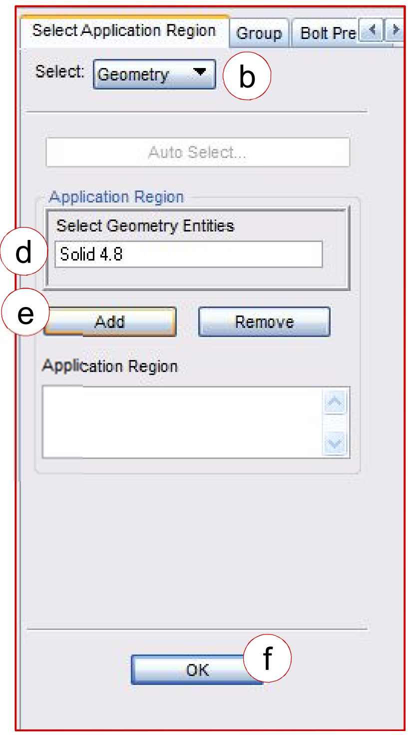
MSC Software

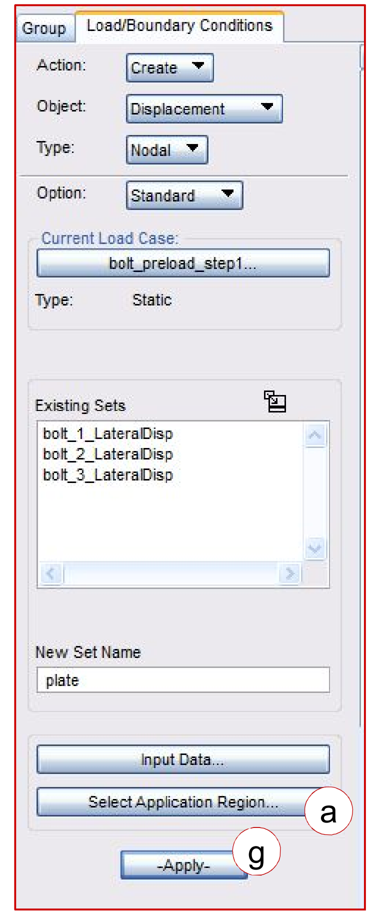
Step 12. Define Boundary Conditions (Cont.)

a. Click **Select Application Region…**
b. Pull down Select > **Geometry**
c. Click **Surface or Face** from the *Picking Filters* toolbar
d. Screen select the edges the lower plate (**Solid 4.8**)
e. Click **Add**
f. Click **OK**
g. Click **Apply**

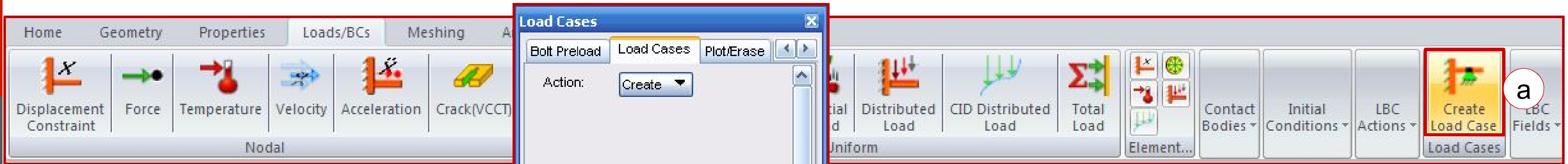


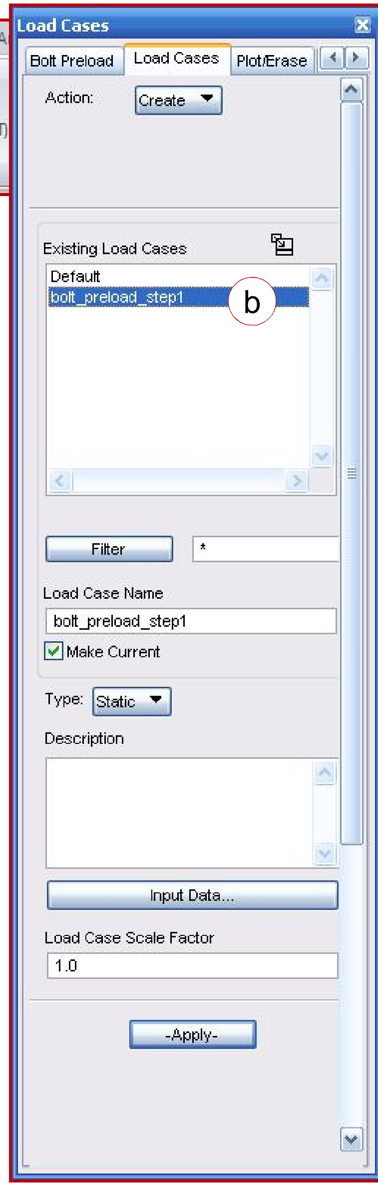


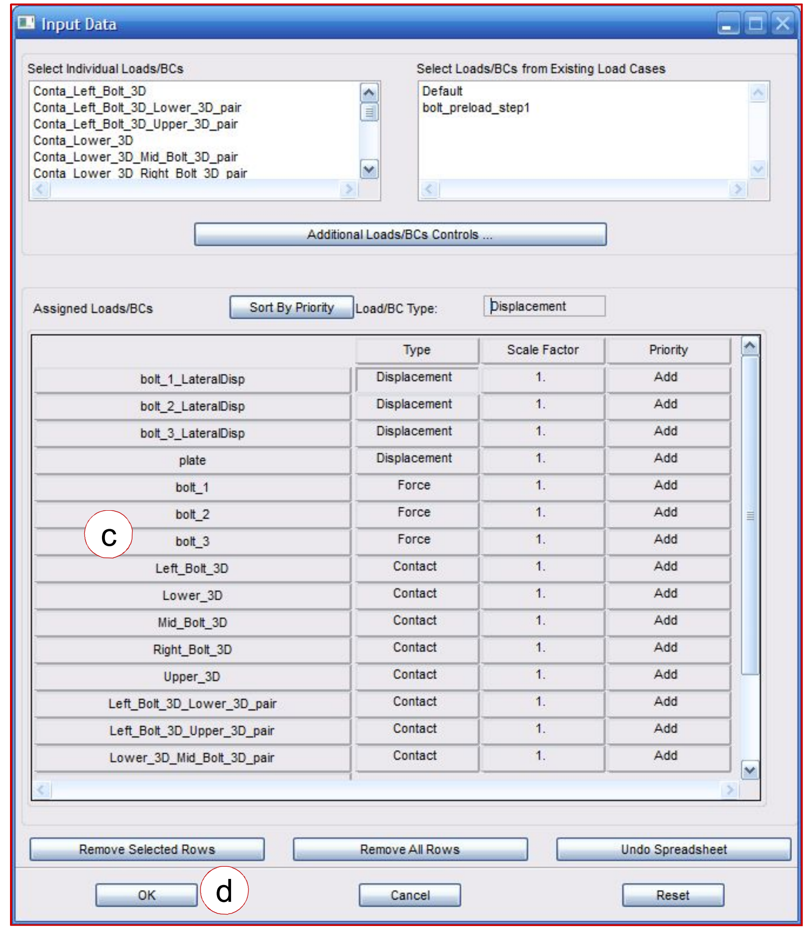




Step 12. Define Boundary Conditions (Cont.)







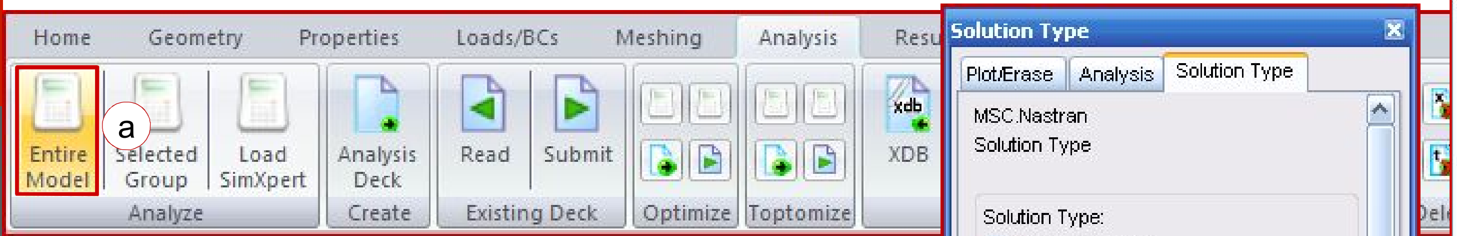
a. Click **Create Load Case** in the *Load Cases* group

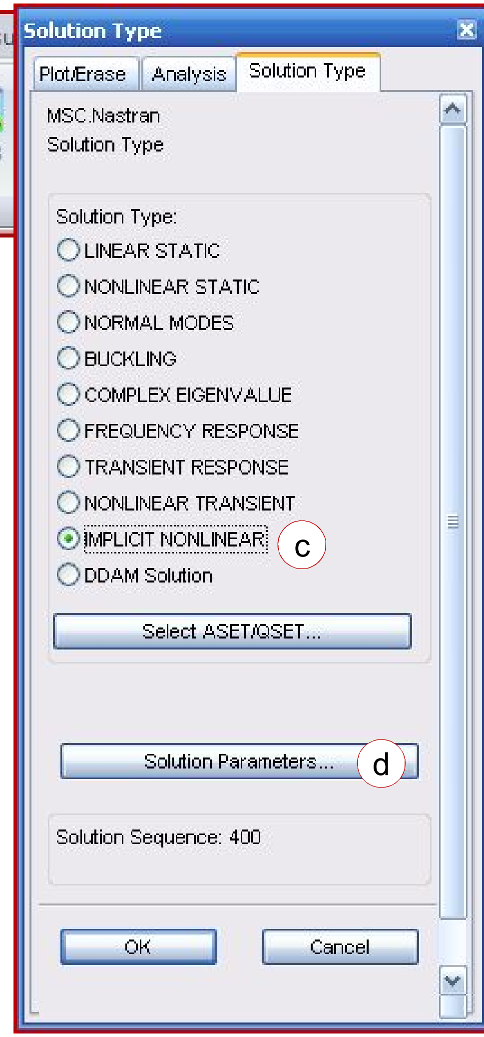
b. Click **bolt_preload_step**1 for *Existing Load Cases*

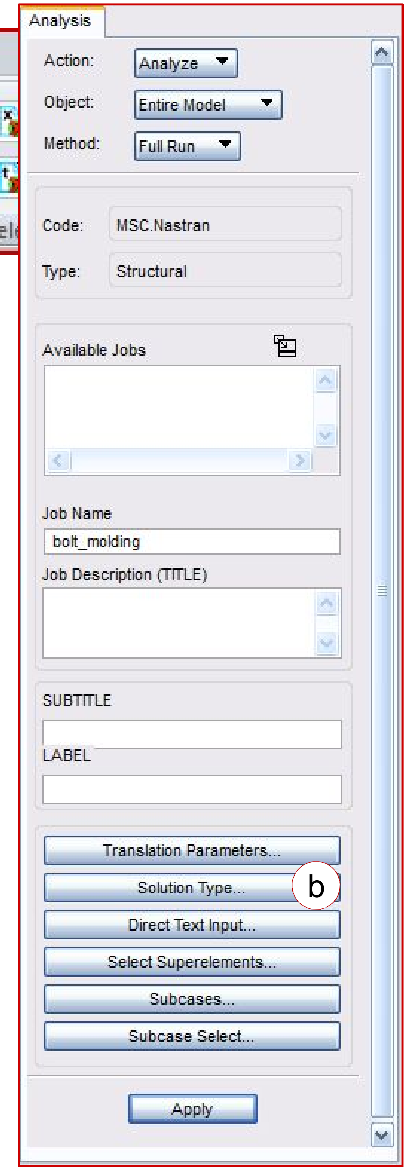
c. View the Assigned Loads/BCs there should be 5 contacts, 4 displacements, and 3 force

d. Click **OK** when done

Step 13. Define Job







Define a job named bolt, to run the Implicit Nonlinear analysis SOL400.

a. Under the *Analysis* tab, click **Entire Model** in the *Analyze* tab

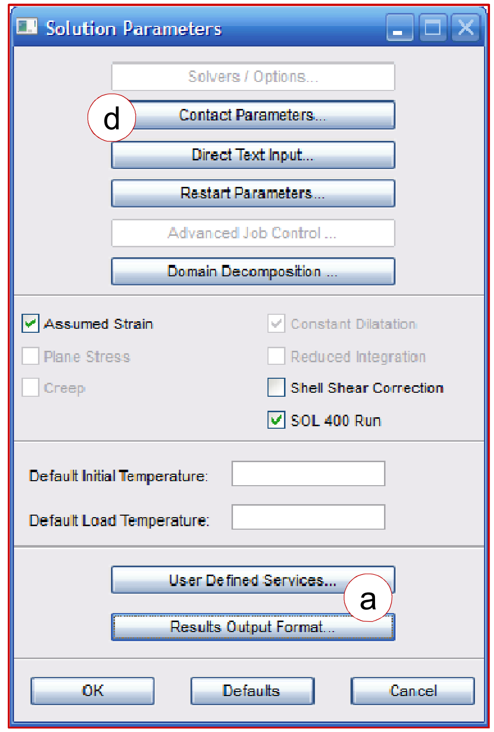
b. Click **Solution Type…**

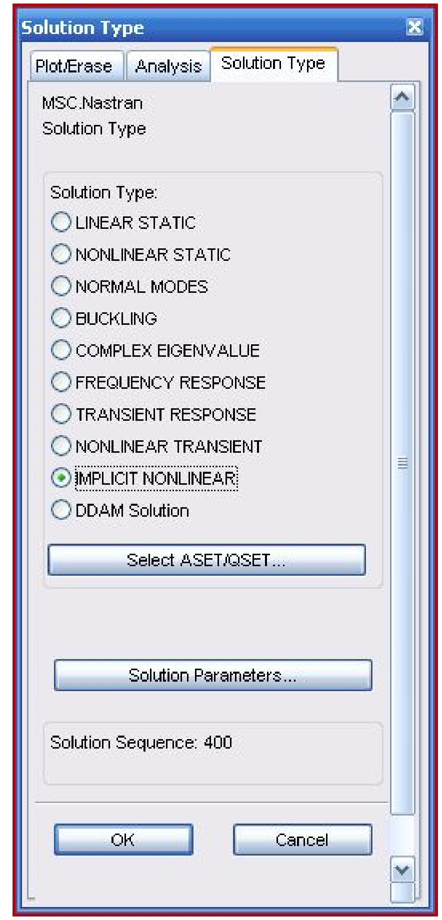
c. Select **Implicit Nonlinear**

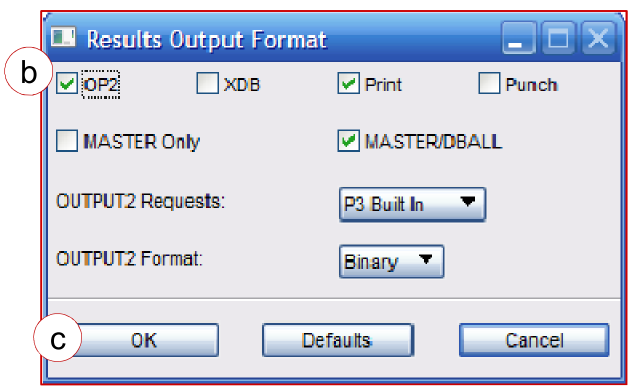
d. Click **Solution Parameters…**

Step 13. Define Job (Cont.)

a. Click **Results Output Format…**
b. Check **OP2**
c. Click **OK**
d. Click **Contact Parameters**







Step 13. Define Job (Cont.)

Activate Quadratic Contact LINQUAD.

a. Click **Contact Detection.**

b. Select **Activate Quadratic Contact (LINQUAD)**

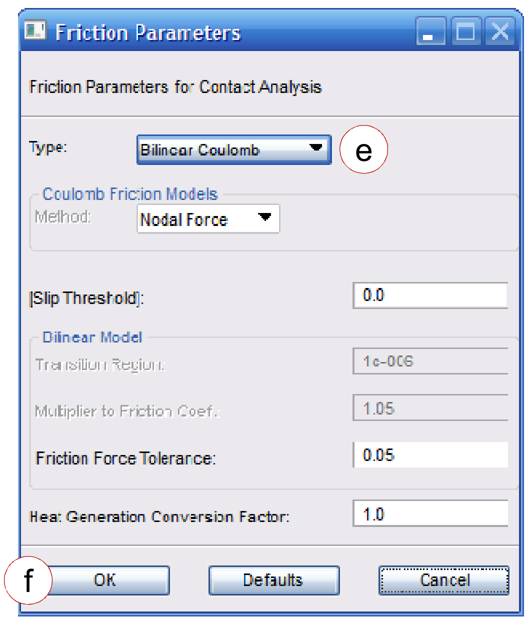
c. Click **OK.**

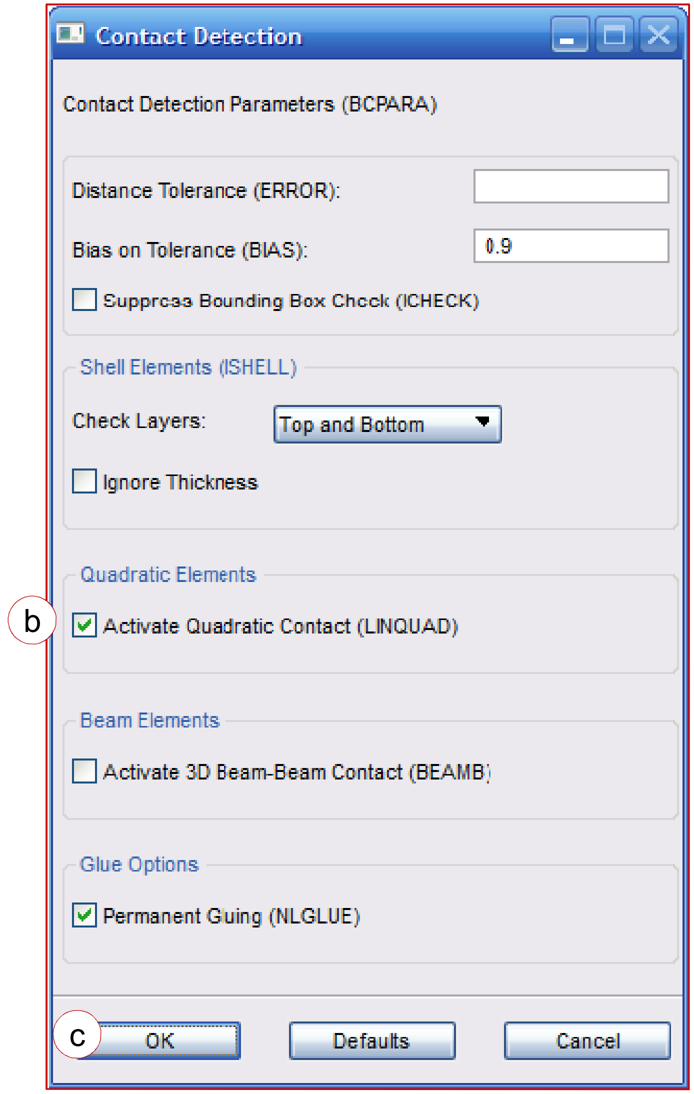
d. Click **Friction**

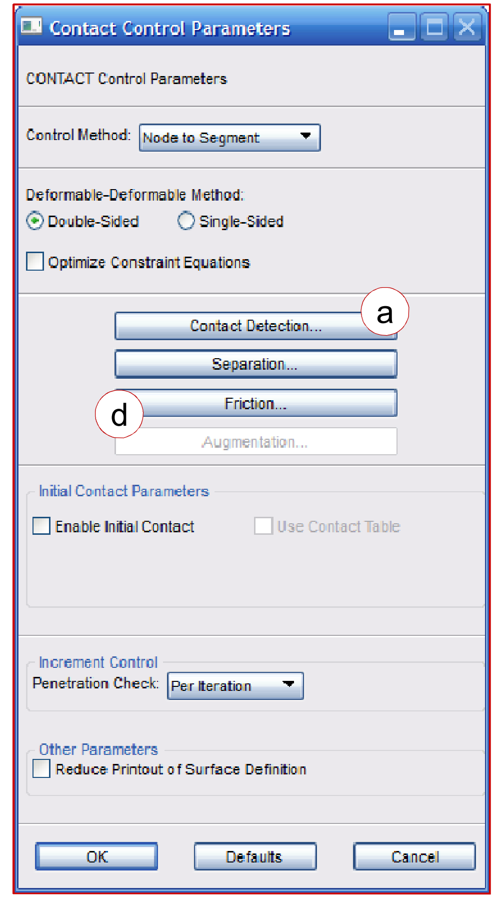
e. Pull down Type > **Bilinear Coulomb**

f. Click **OK**

Click OK on all preceding forms.







Step 13. Define Job (Cont.)

Request for outputs

a. Click **Subcases**

b. Select **botl_preload_step1**

c. Click **Output Requests…**

d. Switch Form Type > **Advanced**

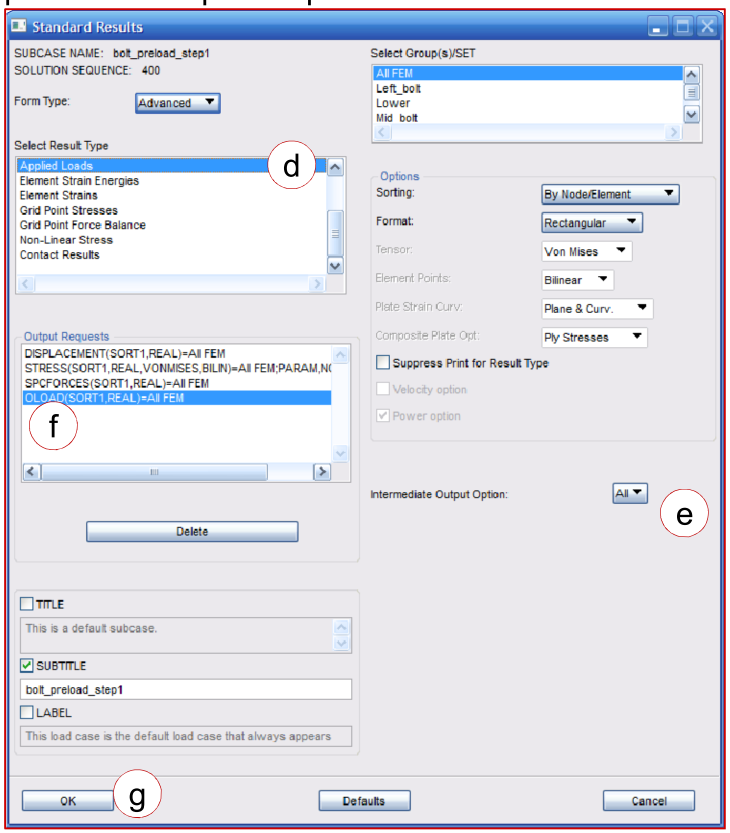
e. Pull down *Intermediate Output Option* > **All**

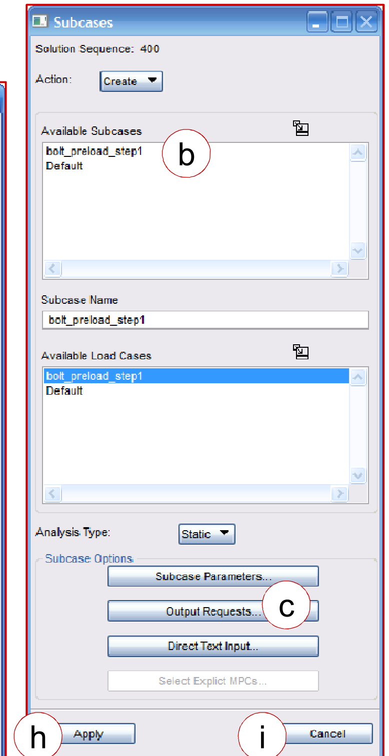
f. Check Displacement, Stress and SPCFORCES and Applied Load are requested in Output Requests.

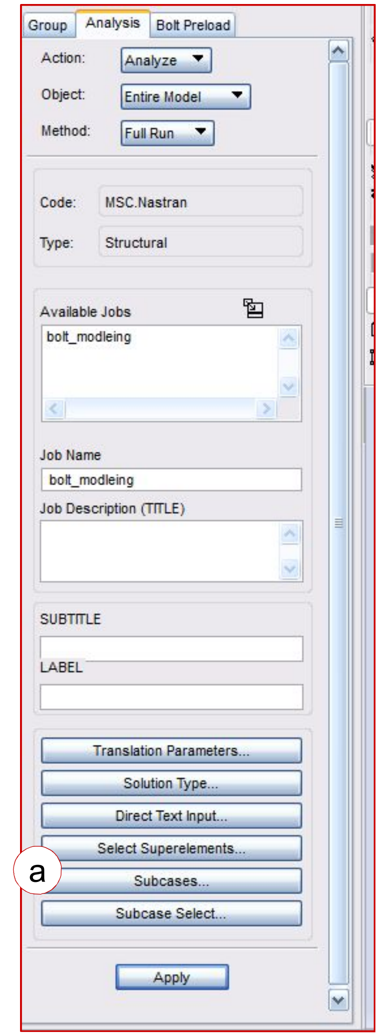
g. Click **OK**

h. Click **Apply**

i. Click **Cancel**







MSC Software

Step 13. Define Job (Cont.)

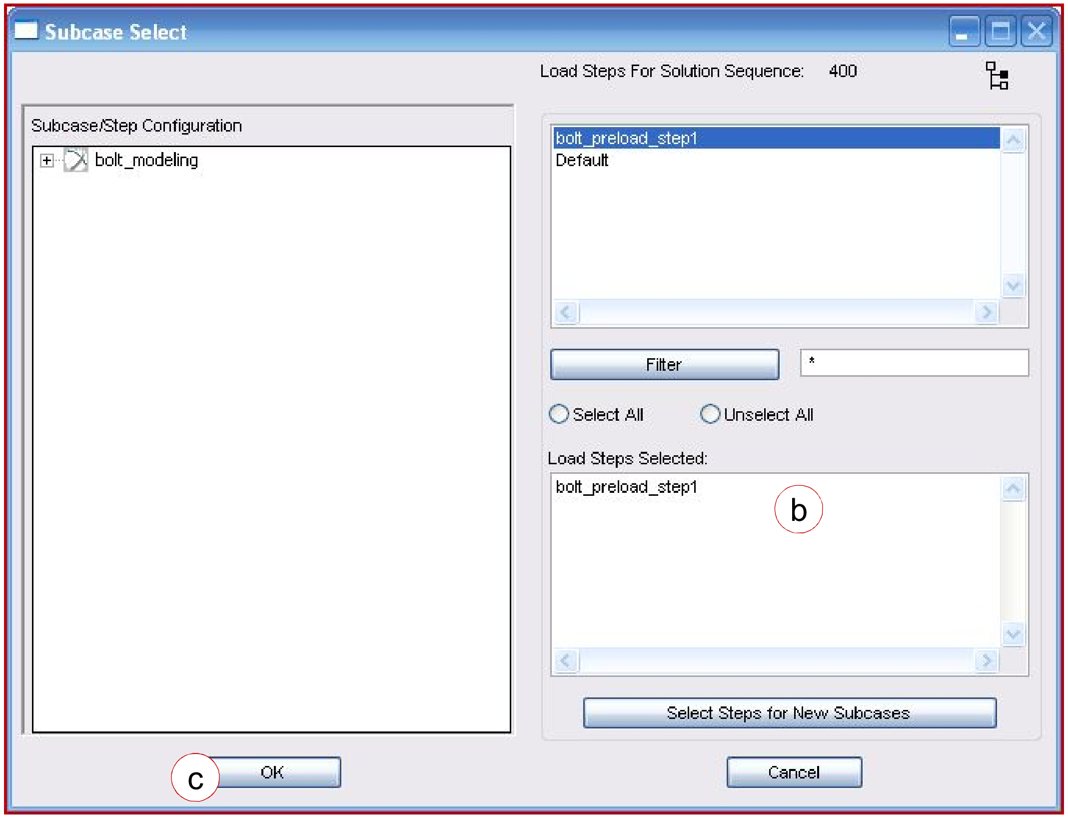
Select Correct Subcase and run the analysis.

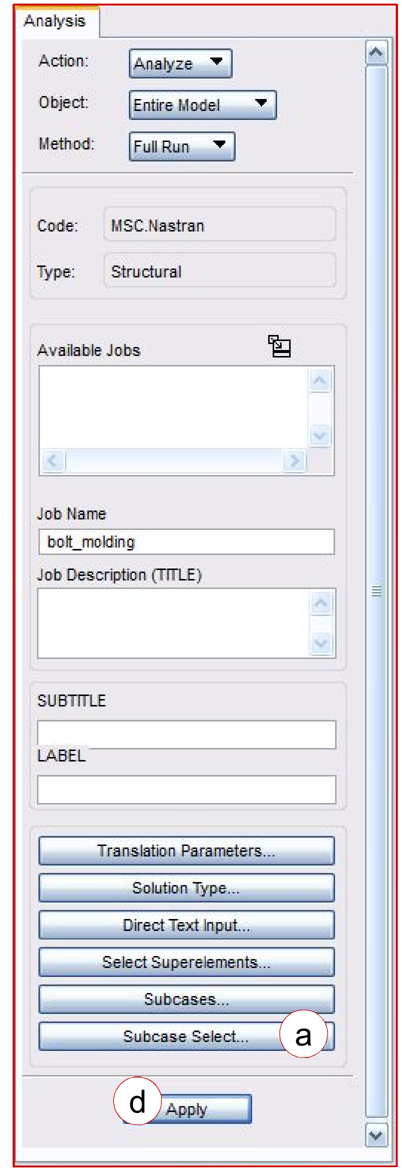
a. Click **Subcase Select…**

b. Unselect **Default** and select **bolt_preload_step1**

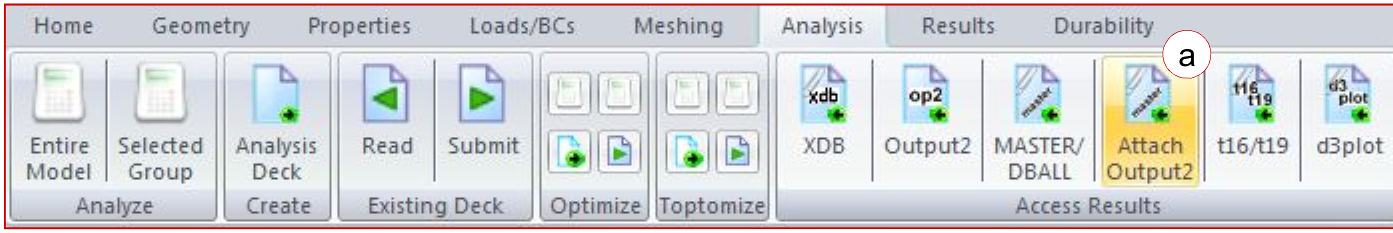
c. Click **OK**

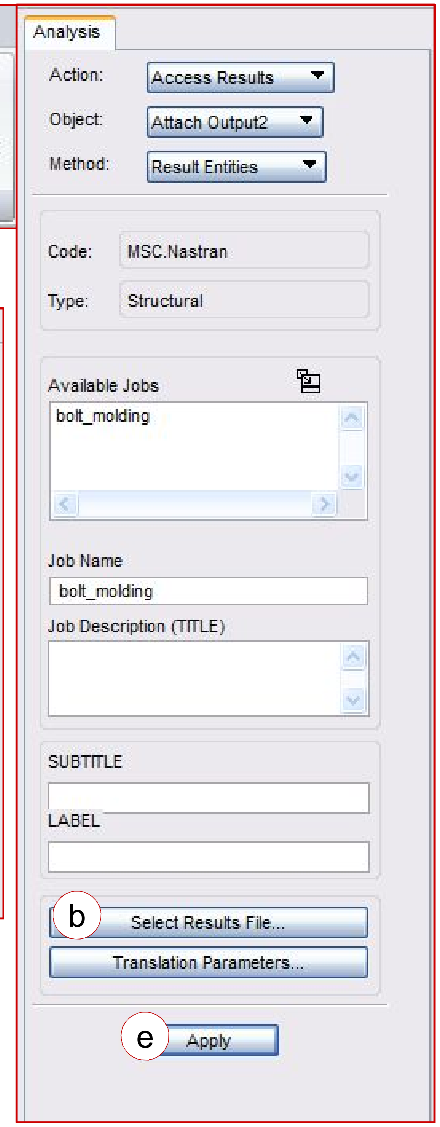
d. Click **Apply**

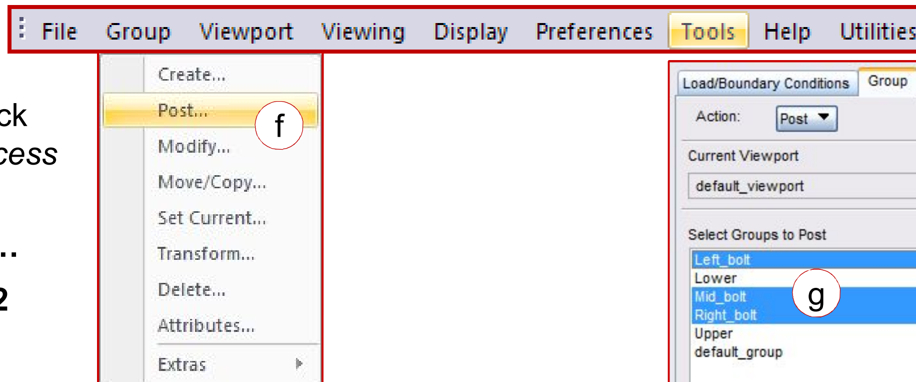




Step 14. Attach Result File

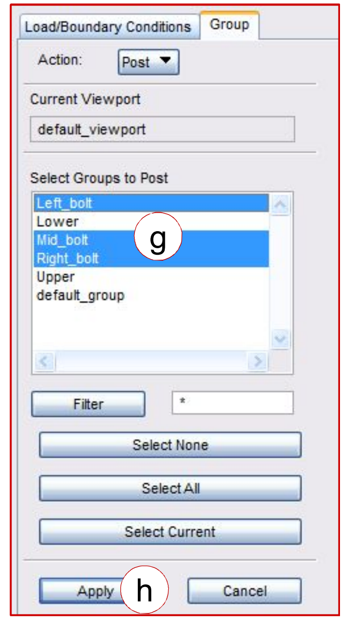


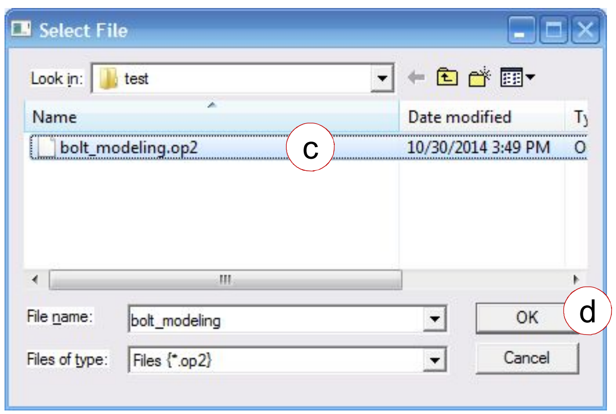




Attach OP2 result file

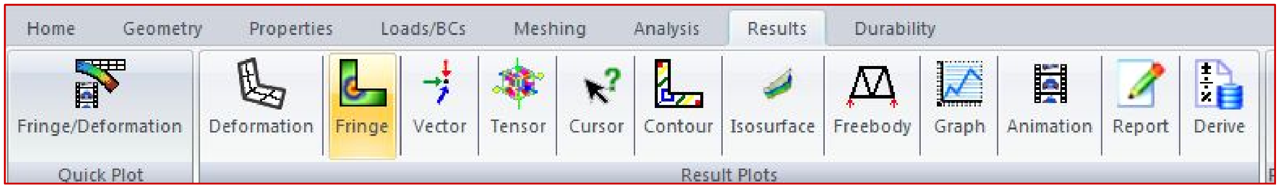
a. Under the *Analysis* tab, click **Attach Output2** in the *Access Results* group.
b. Click **Select Results File…**
c. Select **bolt_modeling.op2**
d. Click **OK**
e. Click **Apply**
f. Pull down Group > **Post**
g. Select **Left_bolt, Mid_Bolt, Right_bolt**.
h. Click **Apply**





MSC Software

Step 14. Post Processing (Cont.)



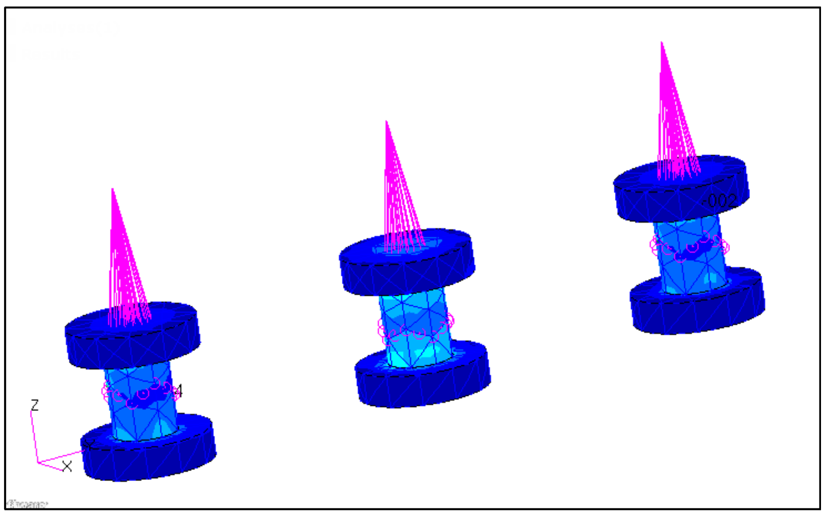
Plot fringe plot Displacement on Bolts

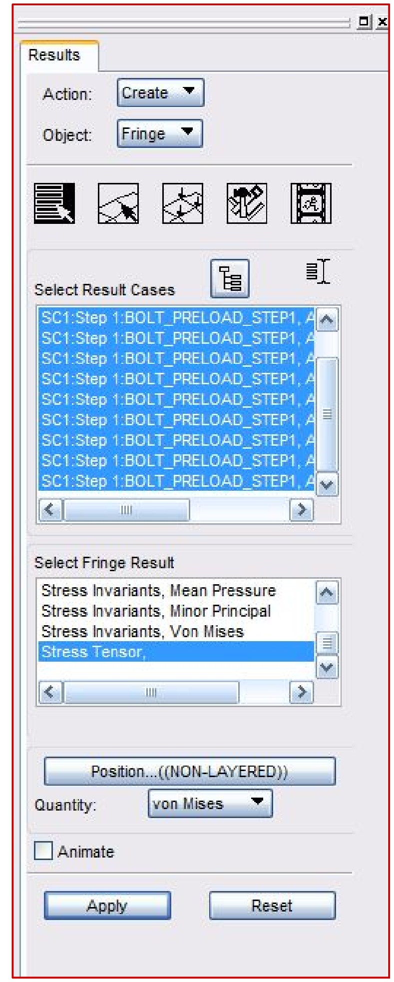
a. Under the *Results* tab, click **Fringe**, in the *Results* plots group.

b. Select all increments in **SC1**

c. Select **Stress Tensor**

d. Click **Apply**

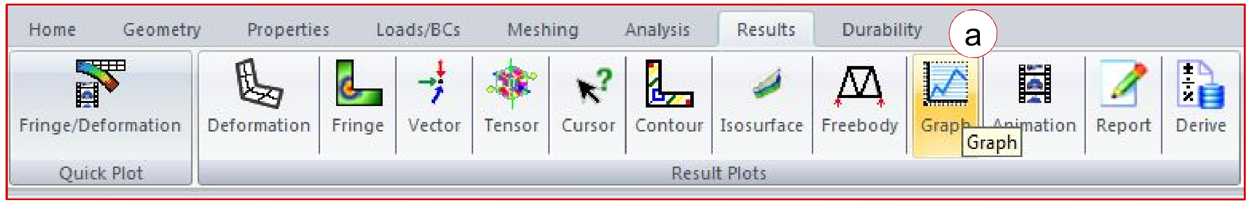


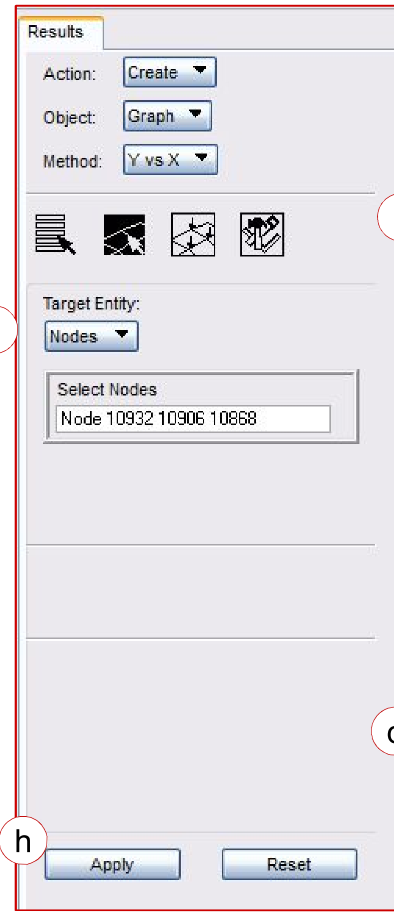


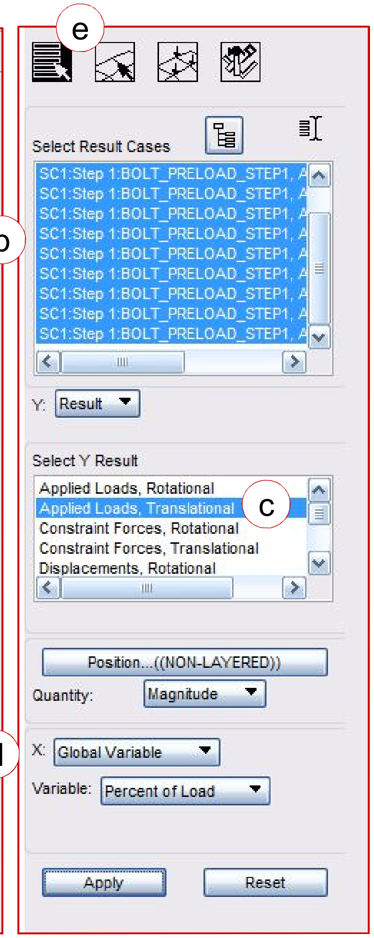
Step 14. Post Processing (Cont.)

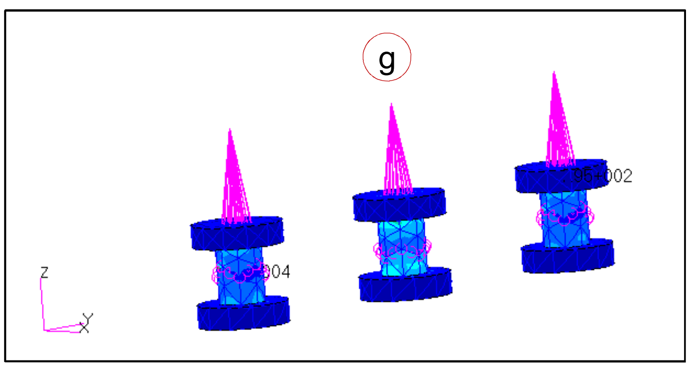
Plot Graph of applied load on bolts.

a. Under the *Results* tab, click **Graph**, in the *Results* plots group.
b. Select all increments in **SC1**
c. Select **Applied Loads, Translational**
d. Set X: to **Global Variable, Percent of Load**
e. Click **Target Entities**
f. Pull down *Target Entity* > **Nodes**
g. Select the control nodes of the bolts.
h. Click **Apply**

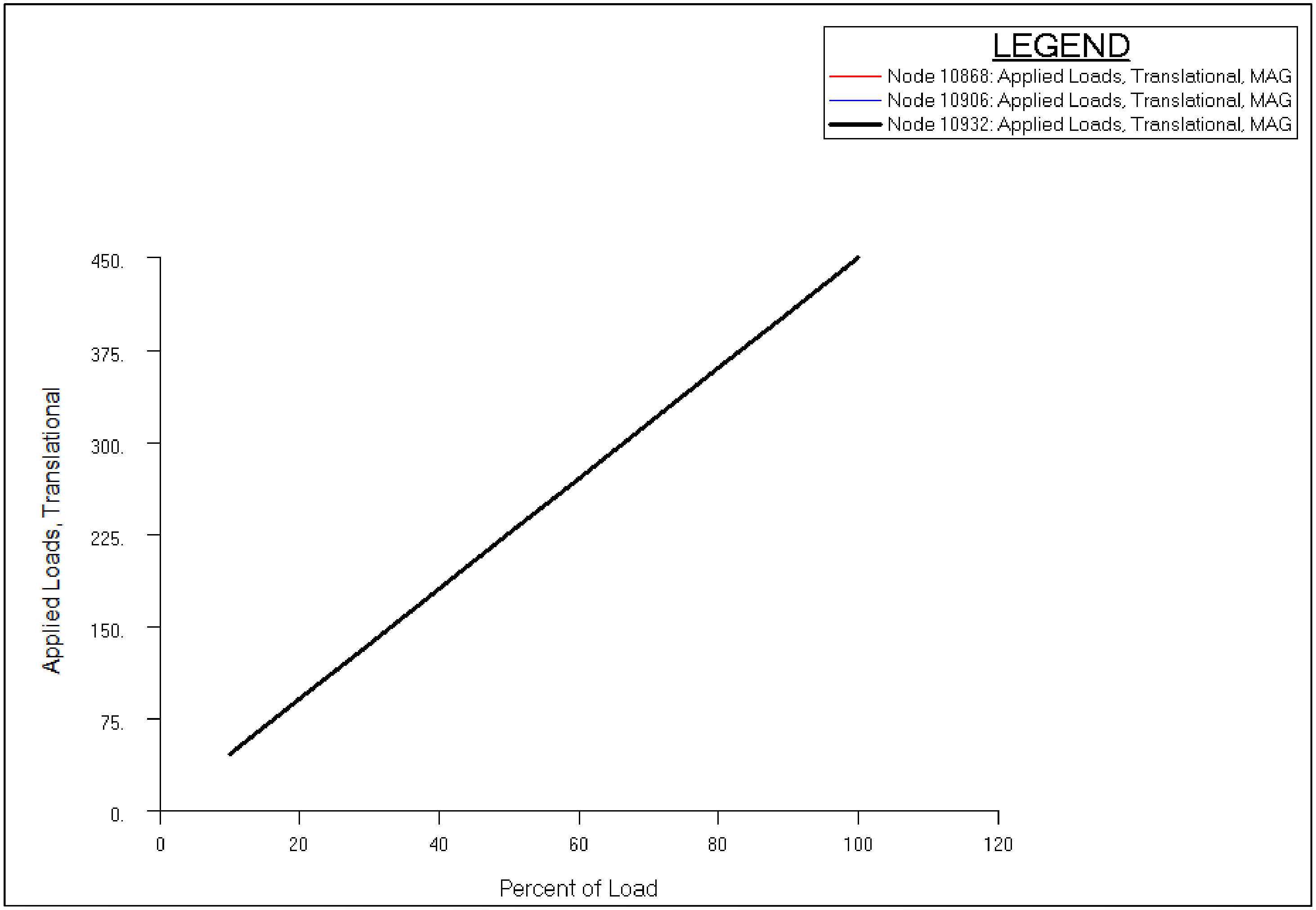








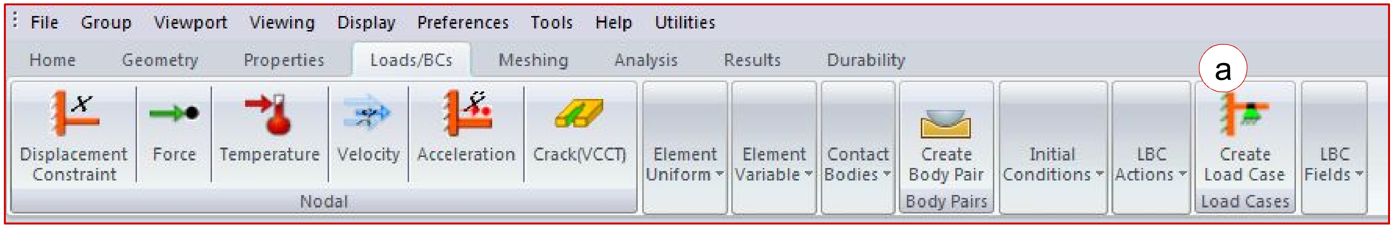
Preloading the Bolts.



Bolt Modeling – 3 Steps

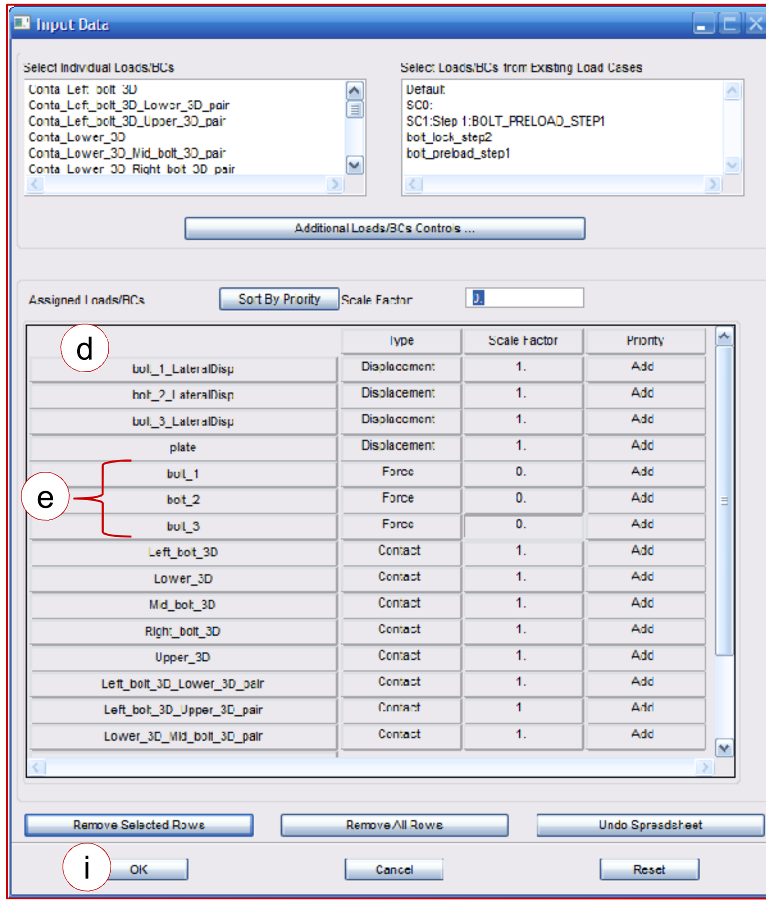
Step 1 Bolt Preload	Step 2 Bolt Lock	Step 3 External Load
- Displacement Constraints - Contact Bodies (3) - Bolt Pretension Load - Bolt Control Node Lateral Displacement Constraint (x,z)	- Displacement Constraints - Contact Bodies - Bolt Control Node Lock Displacement Constraint (x,y,z)	- Displacement Constraints - Contact Bodies (3 Interference Fit) - Bolt Control Node Lock Displacement Constraint (x,y,z) - External Load

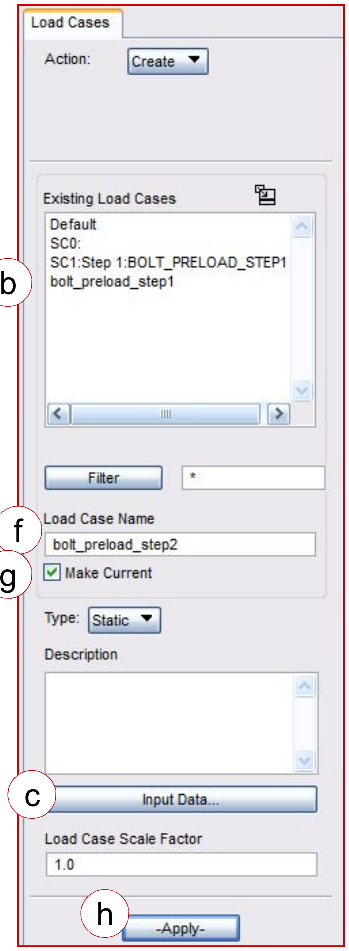
Step 15. Define New Loadcase



Define new loadcase for Bolt lock

a. Under the *Loads/BCs* tab, click **Create Load Case**
b. Click **bolt_preload_step1**
c. Click **input Data**
d. Make sure all the boundary conditions and contact bodies are listed
e. Enter **0** for *Forces* values
f. Rename Load Case to **bolt_lock_step2**
g. Make sure **Make Current** is also checked.
h. Click **Apply**
i. Click **OK**



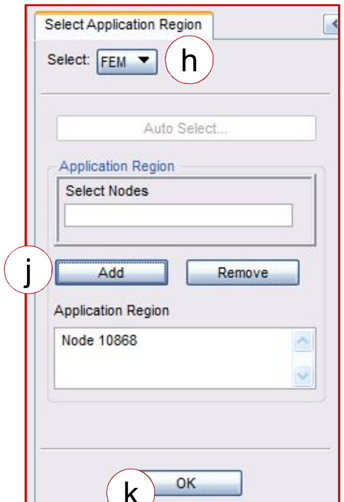


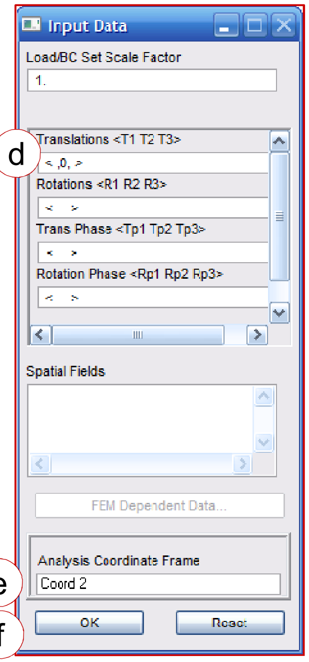
Step 16. Lock Bolts

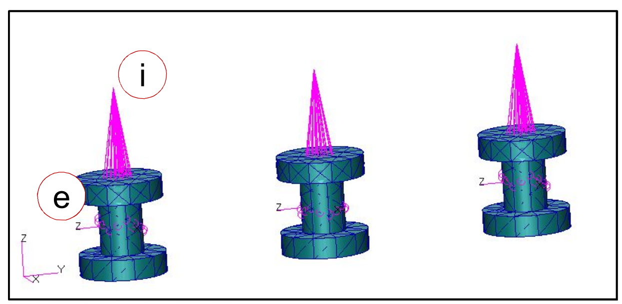
Define new Boundary condition to fix the control node in their local y direction

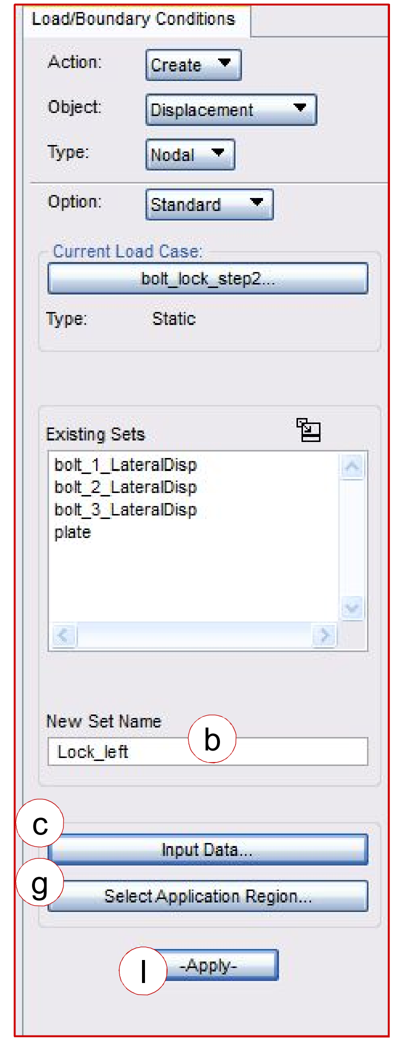
a. Under the *Loads/BCs* tab, click **Displacement Constrain** in the *Nodal* group.
b. Enter **Lock_left** for *Name*.
c. Click **Input Data**
d. Enter **< ,0, >** in *Translations*
e. Select **Coord 2** from the viewport on the left bolt
f. Click **OK**
g. Click **Select Application Region**
h. Pull down Select > **FEM**
i. Select *Control Node* of the left bolt.
j. Click **Add**
k. Click **OK**
l. Click **Apply**

Do the same for the other two bolts and fix them in their local Y axis.



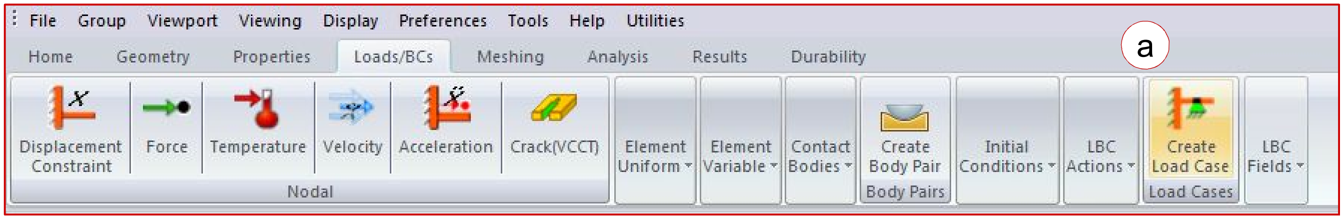






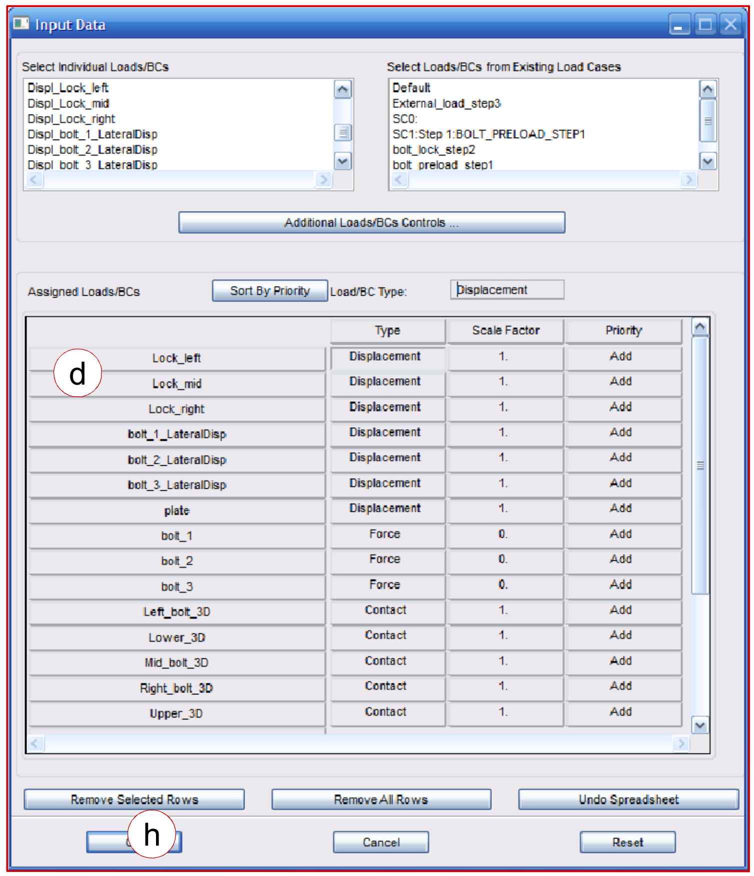
MSC Software

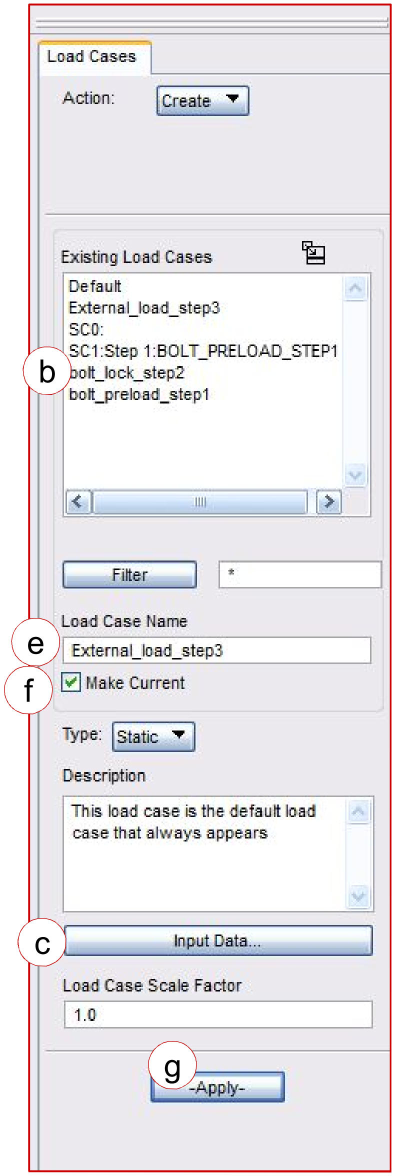
Step 17. Loadcase for External Load



Create a new Loadcase called External_Load_step3

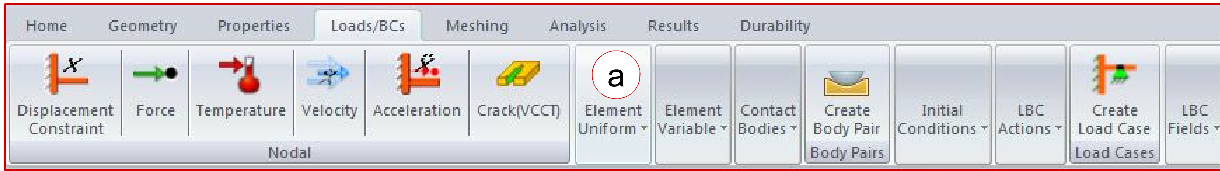
a. Under the *Loads/BCs* tab, click **Create Load Case**
b. Click **bolt_lock_step2**
c. Click **input Data**
d. Make sure all the boundary conditions and contact bodies are listed.
e. Rename Load Case to **External_load_step3**
f. Make sure **Make Current** is also checked.
g. Click **Apply**
h. Click **OK**

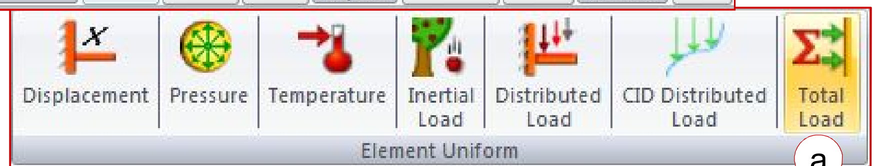


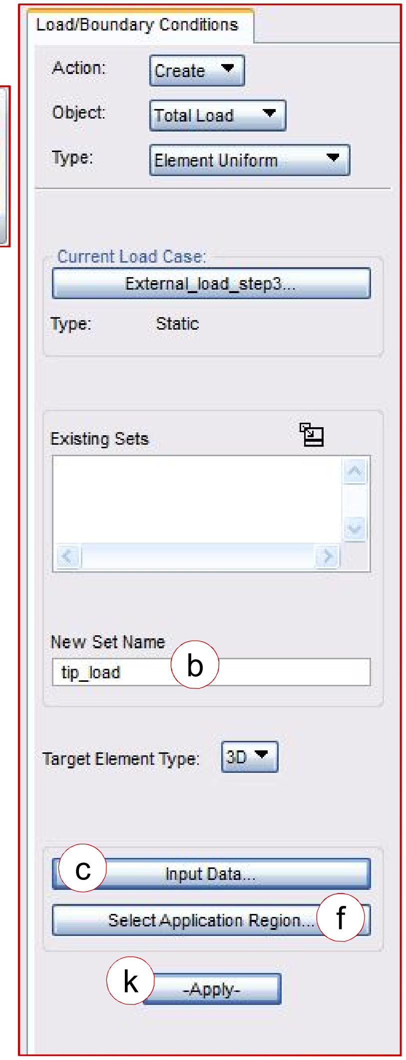


MSC Software

Step 18. Define External Load

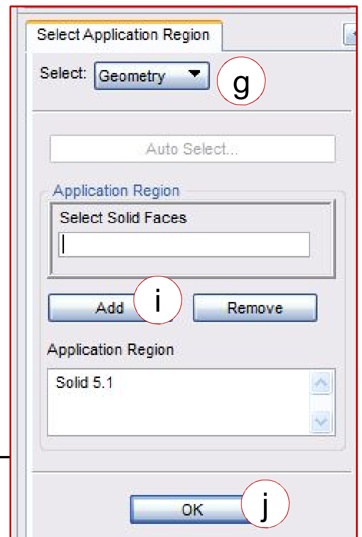


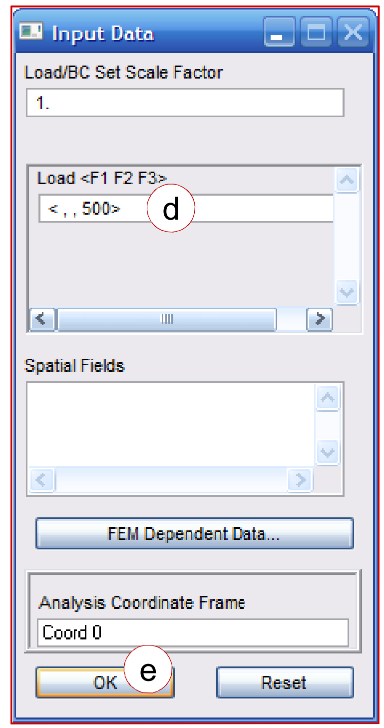


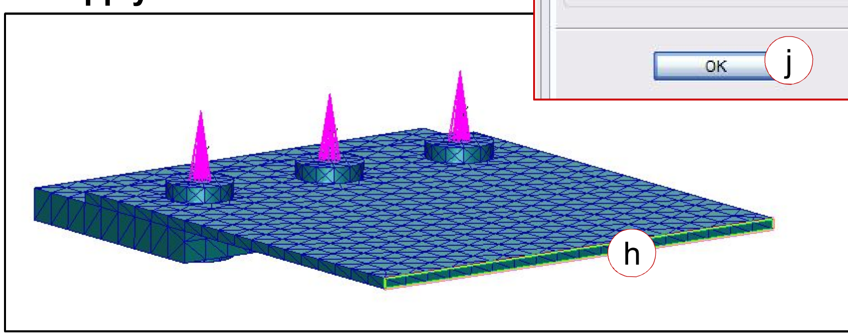


Define External load applied to the plates.

a. Click **Total Load** in the *Element Uniform* group.
b. Enter **Tip_load** for *Name*
c. Click **Input Data**
d. Enter **< , ,500>**
e. Click **OK**
f. Click **Select Application Region**
g. Pull down Select > **Geometry**
h. Select the tip of upper plate
i. Click **Add**
j. Click **OK**
k. Click **Apply**

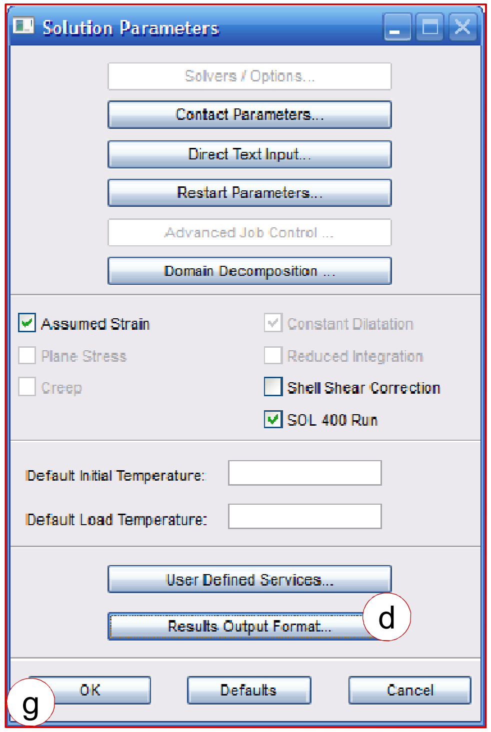


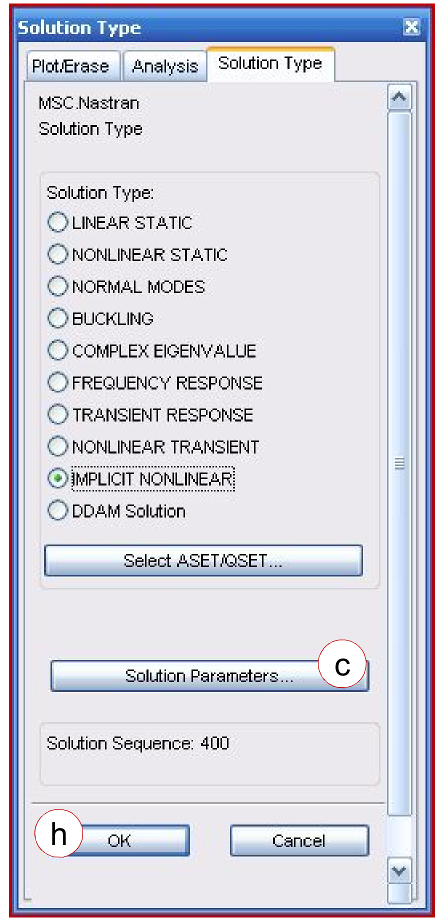


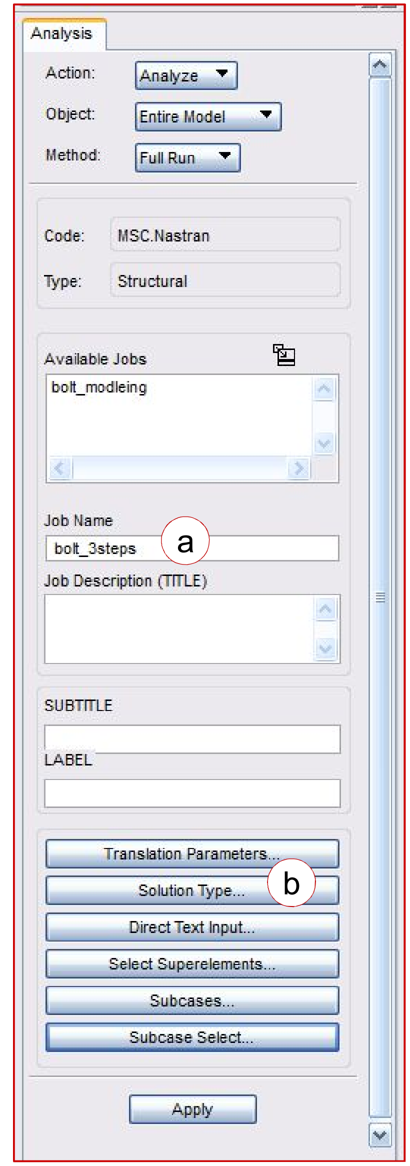


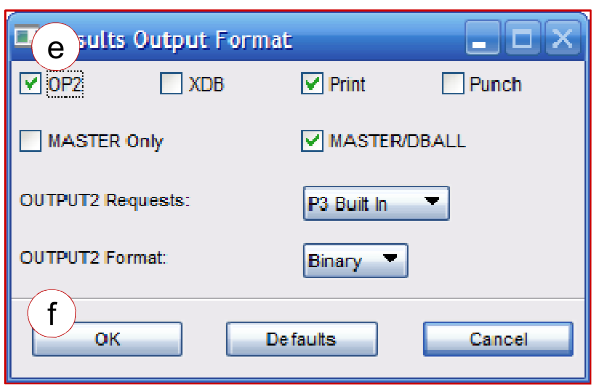
Step 19. Define Job

a. Enter **bolt_3steps** for Jon Name
b. Click **Solution Type**
c. Click **Solution Parameters.**
d. Click **Results Output Format…**
e. Check **OP2**
f. Click **OK**
g. Click **OK**
h. Click **OK**









Step 19. Define Job (Cont.)

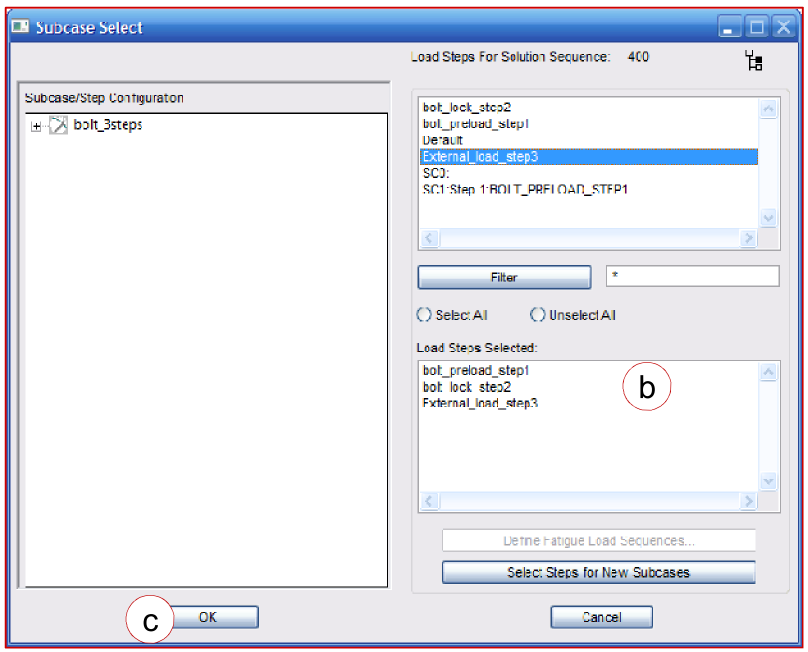
Run Analysis with all three steps of bolt modeling.

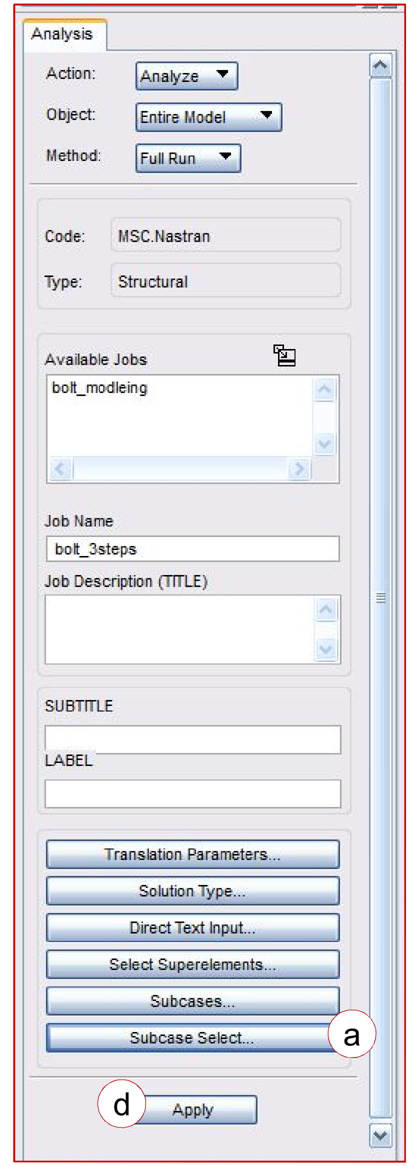
a. Click **Subcase Select…**

b. Unselect **Default** and select
bolt_preload_step1
bolt_lock_step2
External_load_step3

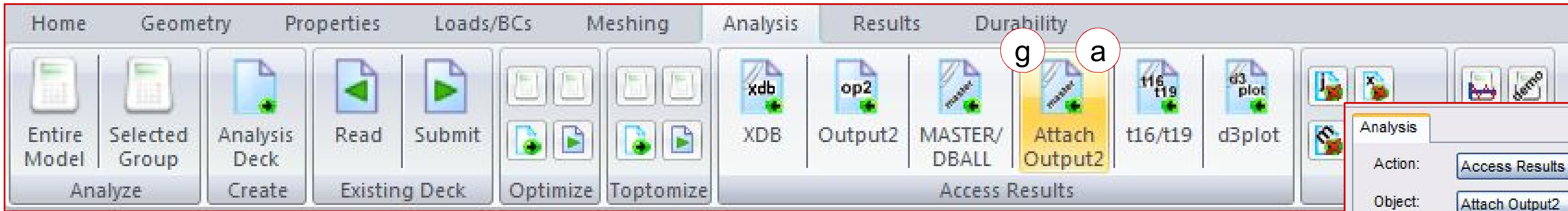
c. Click **OK**

d. Click **Apply**



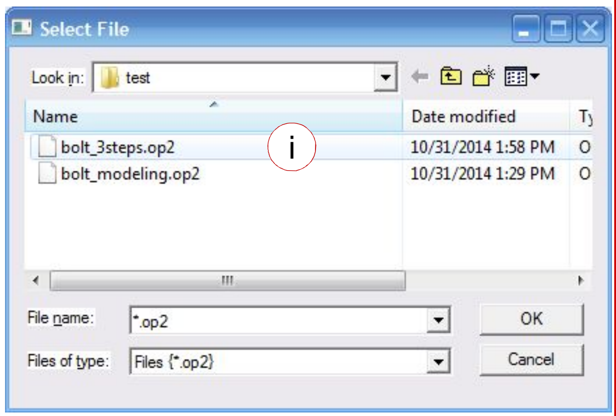


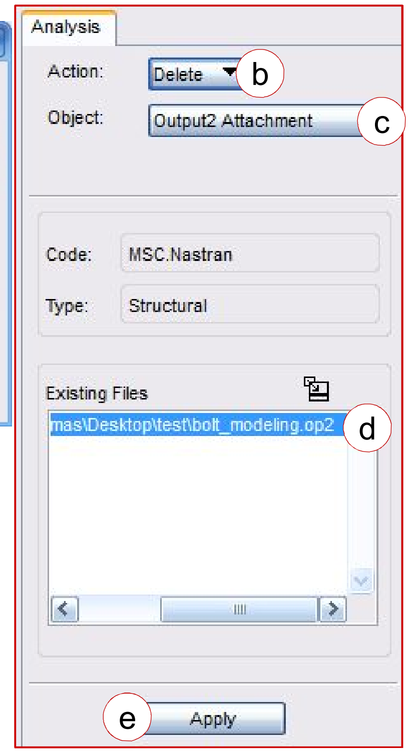
Step 20. Attach Result File

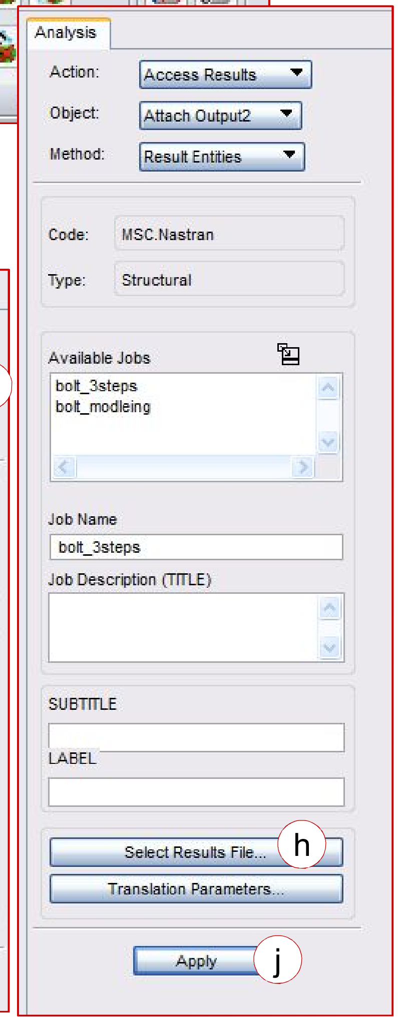


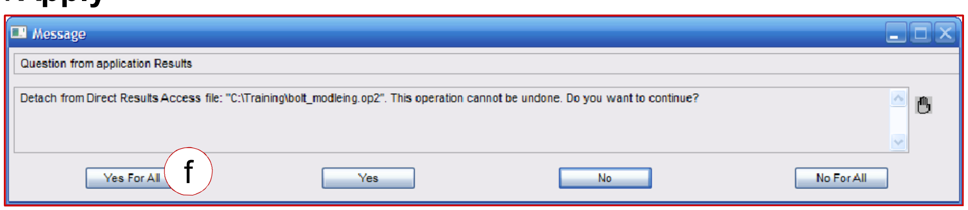
Attach OP2 result file

a. Under the *Analysis* tab, click **Attach Output2** in the *Access Results* group.
b. Pull down Action > **Delete**
c. Pull down Object > **OP2 Attachment**
d. Select *Bolt_modeling.op2*
e. Click **Apply**
f. Click **Yes For All**
g. Click **Attach Output2** in the *Access*
h. Click **Select Results File…**
i. Select **bolt_3steps**
j. Click **Apply**

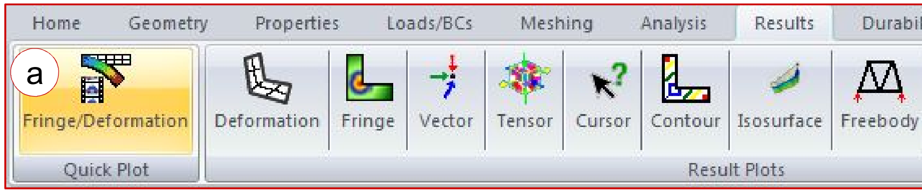


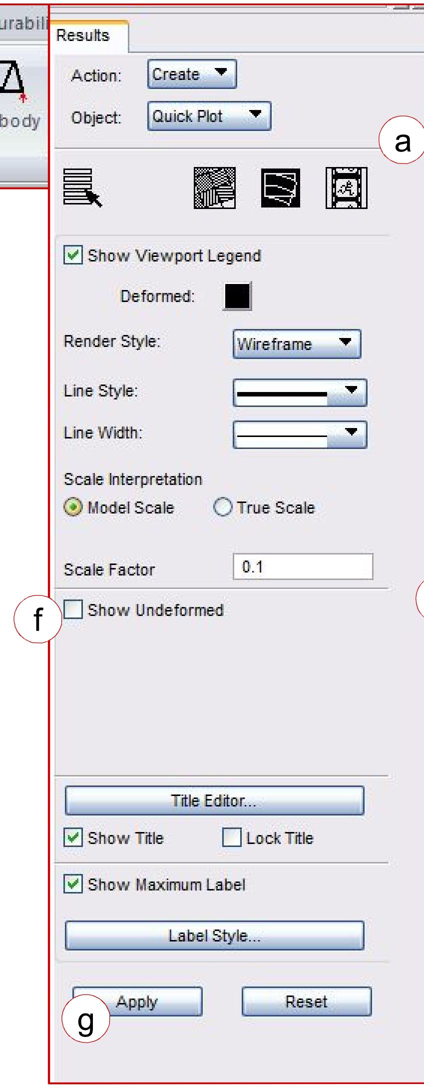


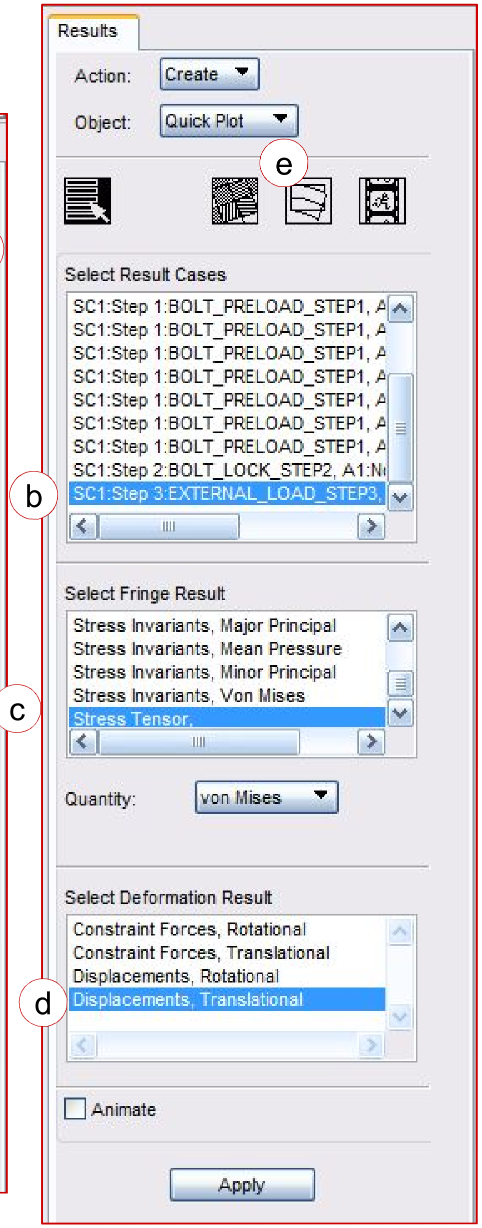




Step 21. Post Processing



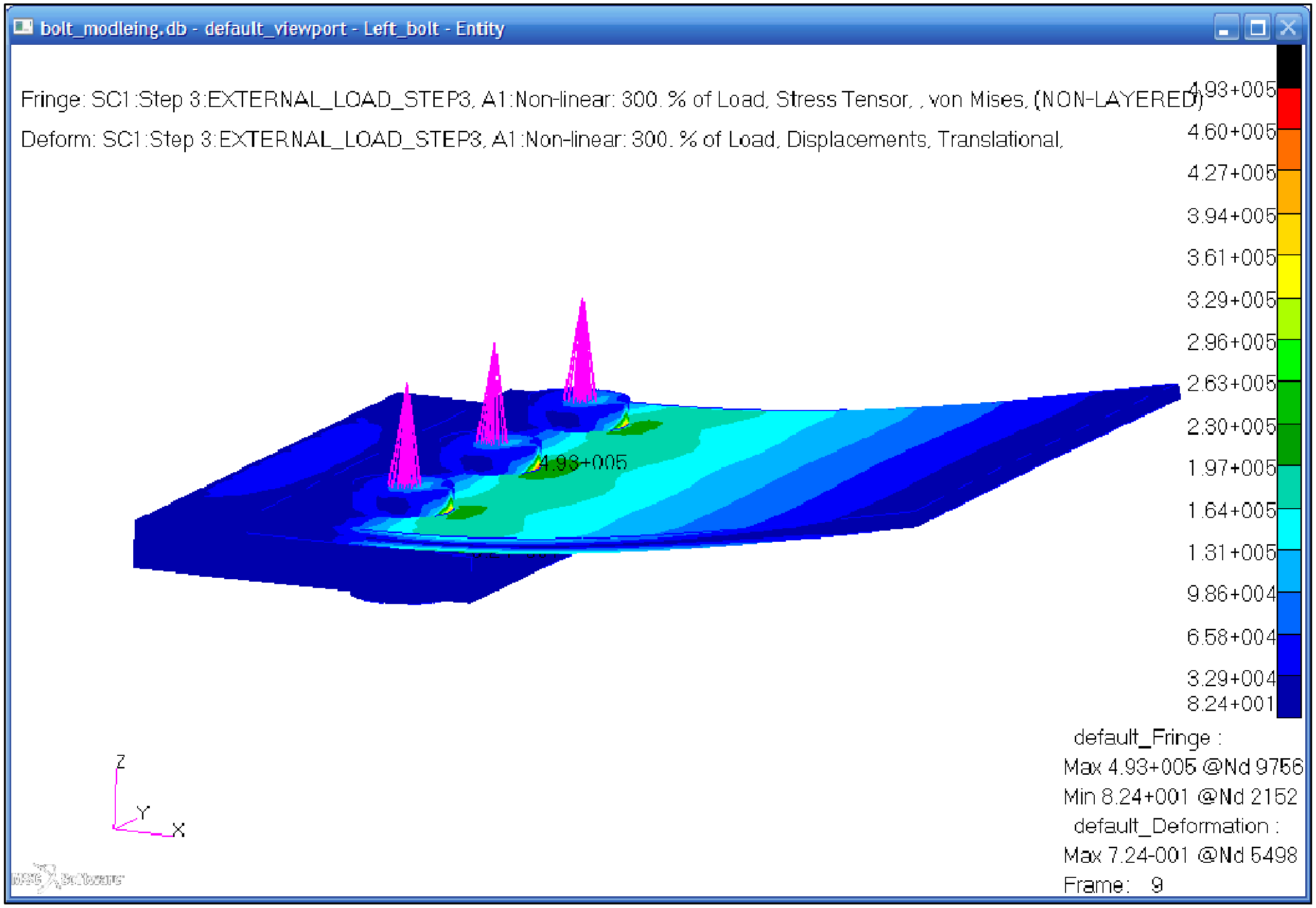




Plot fringe plot Displacement on Bolts

a. Under the *Results* tab, click **Fringe/Deformation**, in the *Results* plots group.
b. Select last increment
c. Select **Stress Tensor**
d. Select **Displacement Translational**
e. Click **Deformation Attribute**
f. Uncheck **Show deformed**
g. Click **Apply**

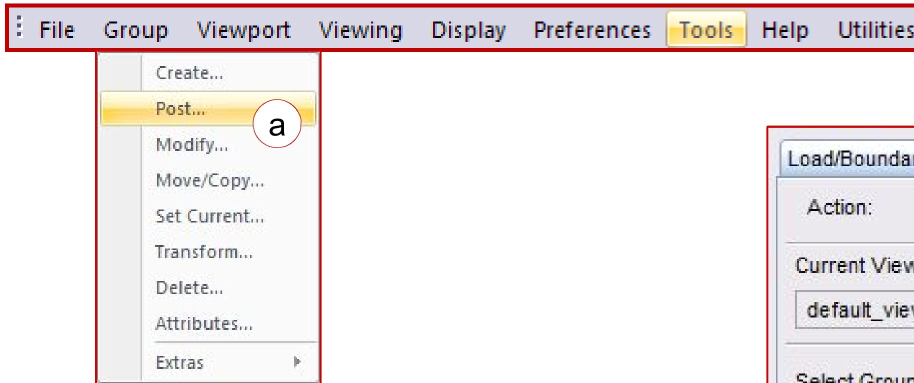
Von Mises Stress

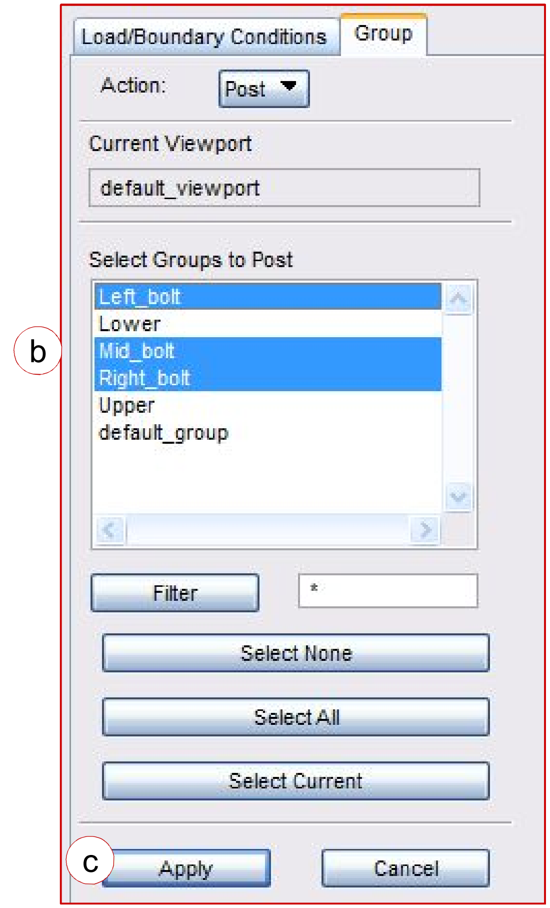


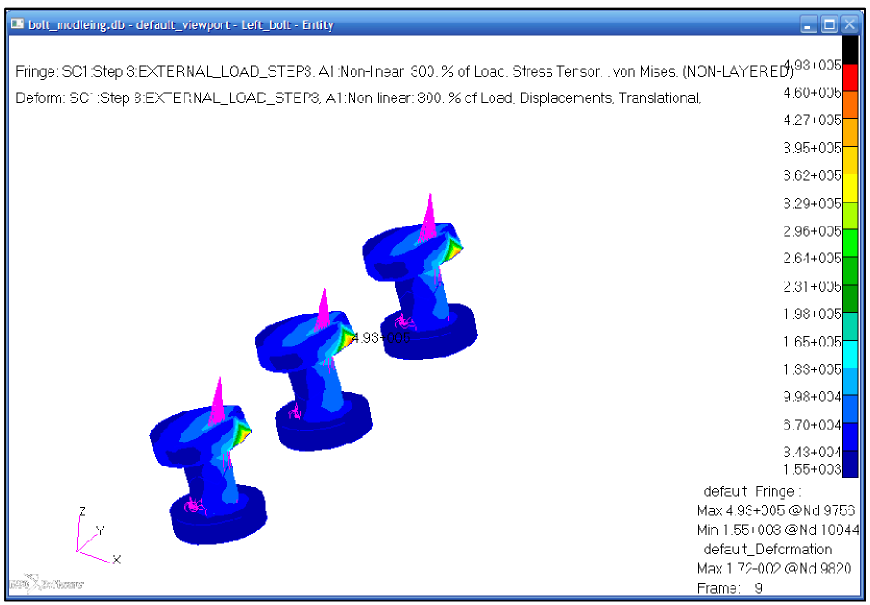
Step 21. Post Processing (Cont.)

Post Bolts for post processings

a. Pull down Group > **Post**
b. Select **Left_bolt, Mid_Bolt, Right_bolt**.
c. Click **Apply**





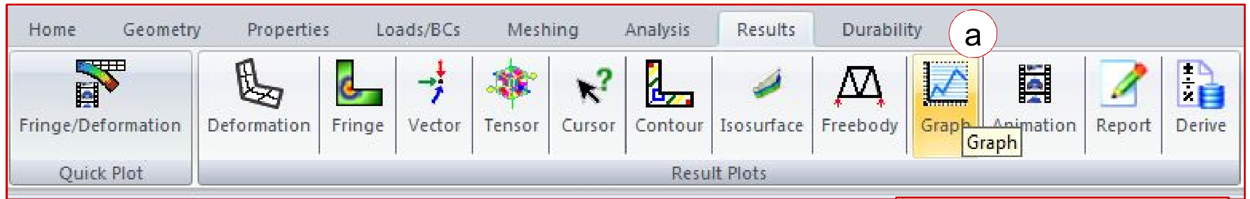


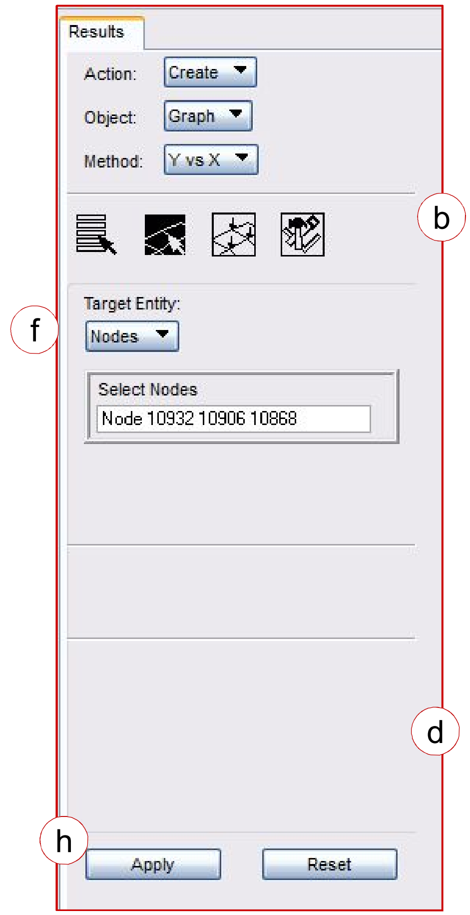
MSC Software

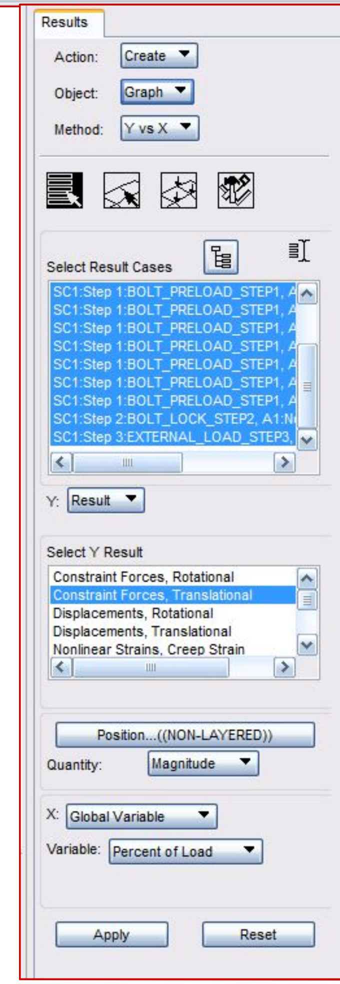
Step 21. Post Processing (Cont.)

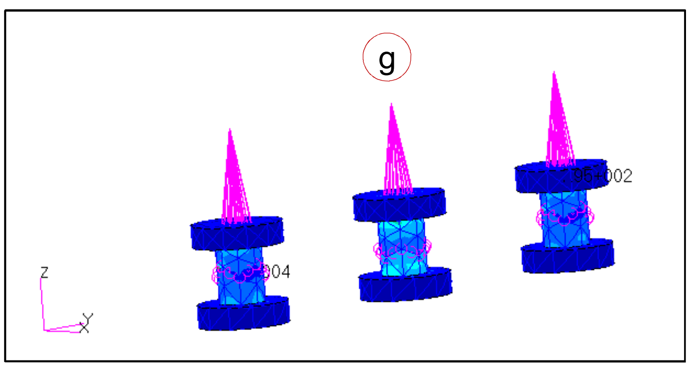
Plot Graph of applied load on bolts.

a. Under the *Results* tab, click **Graph**, in the *Results* plots group.
b. Select all increments except SC0
c. Select **Constrain Forces, Translational**
d. Set X: to **Global Variable, Percent of Load**
e. Click **Target Entities**
f. Pull down *Target Entity* > **Nodes**
g. Select the control nodes of the bolts.
h. Click **Apply**



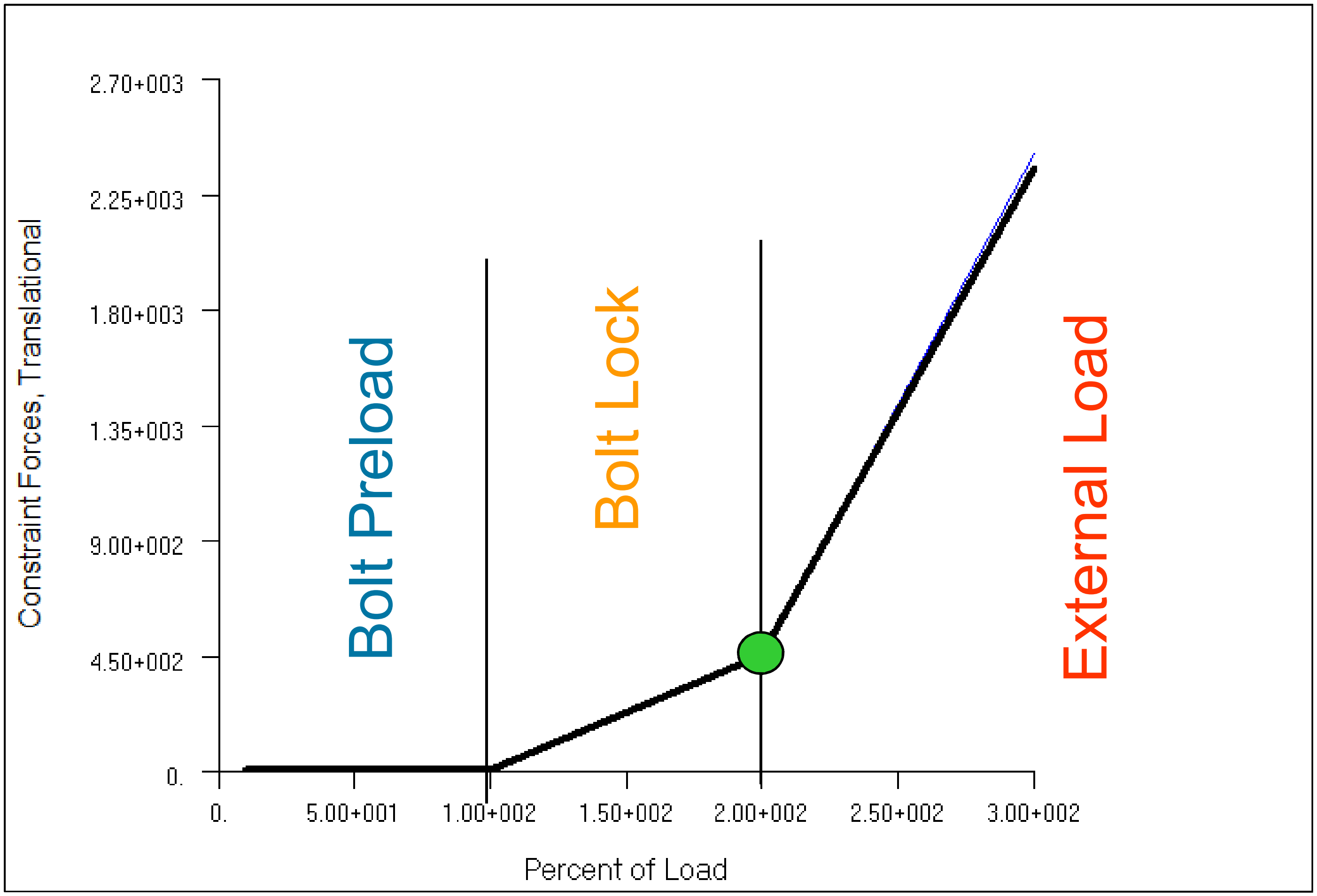






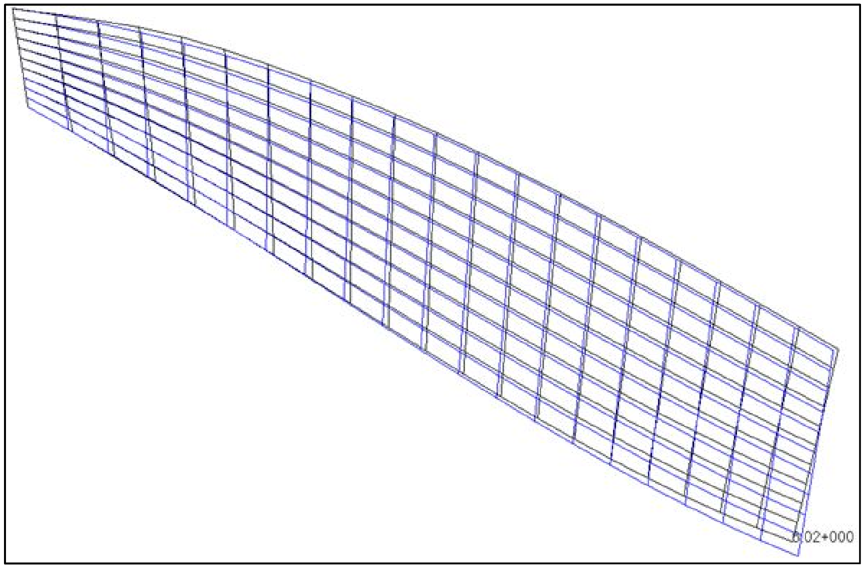
Step 21. Post Processing (Cont.)

Step 1
Bolt Pretension → **Step 2**
Bolt Lock → **Step 3**
Structural Load



MSC Software

WORKSHOP B1
NORMAL MODES ANALYSIS OF A PRESTIFFENED BLADE



- **Workshop Objectives**
  - Perform a nonlinear static analysis and a normal modes analysis. Plot von Mises stresses and normal modes.
  - Define a stress-strain curve for material plasticity.
- **Software Version**
  - Patran 2013
  - MSC Nastran 2013.1
- **Required**
  - Blade_model.bdf
- **Problem Description**
  - In this exercise you will simulate the vibration of rotating fan blades of a jet engine . The FEM nodes and elements will be imported as a MSC Nastran data file. The model length unit is a meter.
  - The analysis will be performed in two steps. In the first step, a nonlinear static analysis will be performed to compute the stresses in the blade due to a rotational speed of 133.3 revolutions per second. In the second step, the normal modes and frequencies of the pre-stressed blade will be computed.
  - Once the analysis is complete, the stresses and the normal modes will be plotted.

- **Suggested Exercise Steps**

1. Create a new database called **blade**.
2. Import model **blade_model.bdf**
3. Fix all translations and rotations at the left end of the blade.
4. Create an inertial load in the Z direction where the rotational velocity is **133.3**.
5. Create a stress vs. strain field for plasticity with the following values:

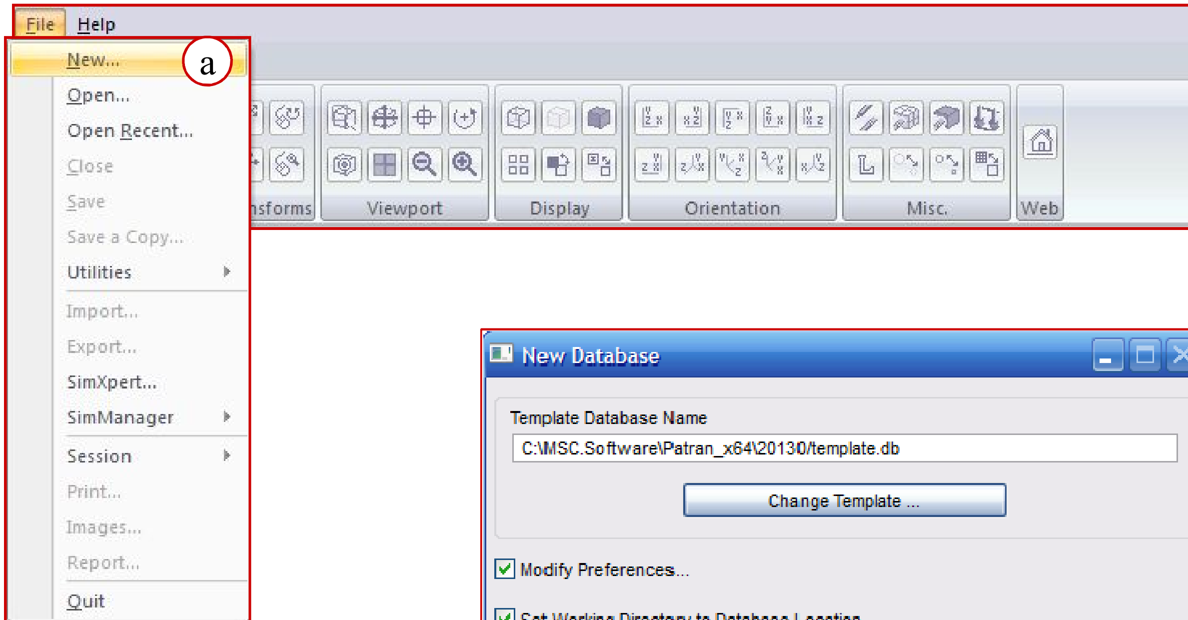
Strain	Value
0	2e8
0.691	5e8
1	6e8

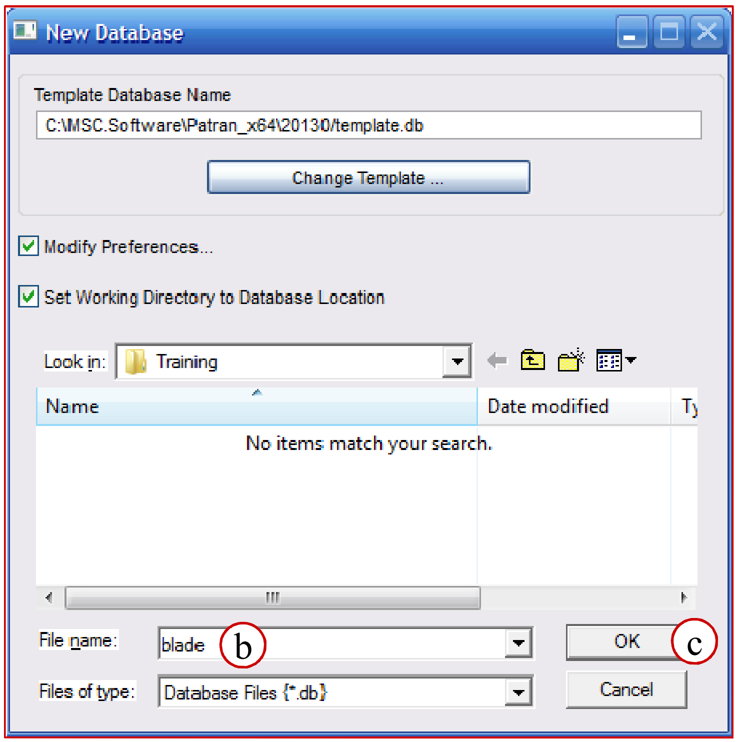
6. Plot the field.
7. Create a steel material with **2.1e11** Elastic Modulus, **0.3** Poisson Ratio, and **7800** density.
   - Define an elastoplastic constitutive model and apply the plasticity field.
8. Create a 2D shell property for the model.
   - Apply steel material.
   - Thickness = **0.0025**
   - Select **Large Strain** for Nonlinear Formulation (SOL400)
   - Apply to entire model.

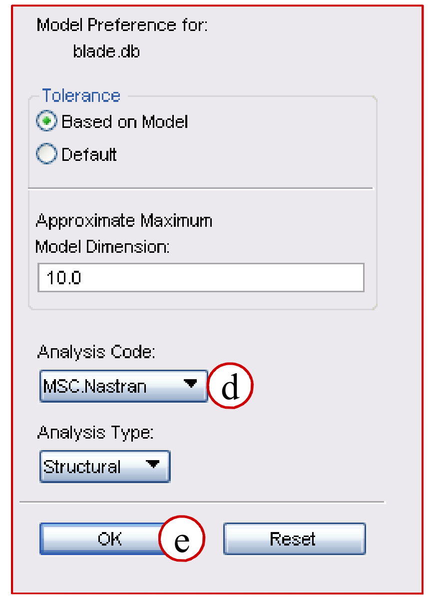
- **Suggested Exercise Steps (Cont.)**

9. Create and run an implicit nonlinear analysis.
   - In Solution Parameters select Solution Type **SOL400** and **Assumed Strain**.
   - Select .op2 as results output format.
   - Create a **NL_static** subcase.
   - In Load Increment Parameters, set the number of increments to **10**.
   - Select to output element strains.
   - Create a **modes** subcase with a **normal modes** analysis type and **5** desired roots.
   - Deselect the default subcase and select the NL_static and modes subcases.
   - Run analysis.
10. Attach .op2 result file.
11. Plot the von Mises stress superimposed on the deformed shape for the final nonlinear result case.
12. Plot the deformed shape of the first mode.
13. Quit Patran.

Step 1. Create a New Database



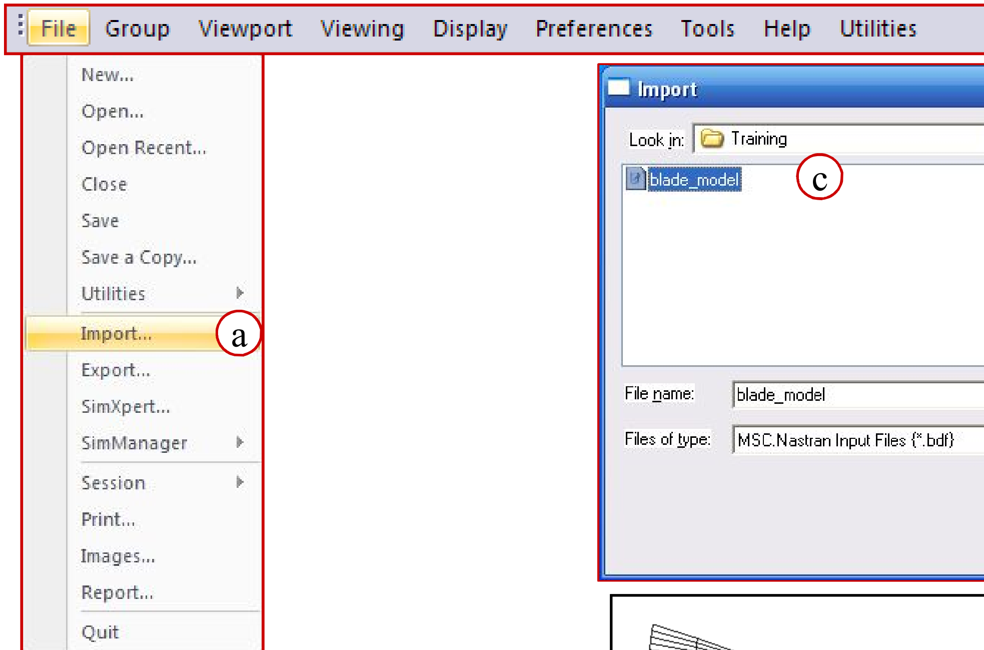


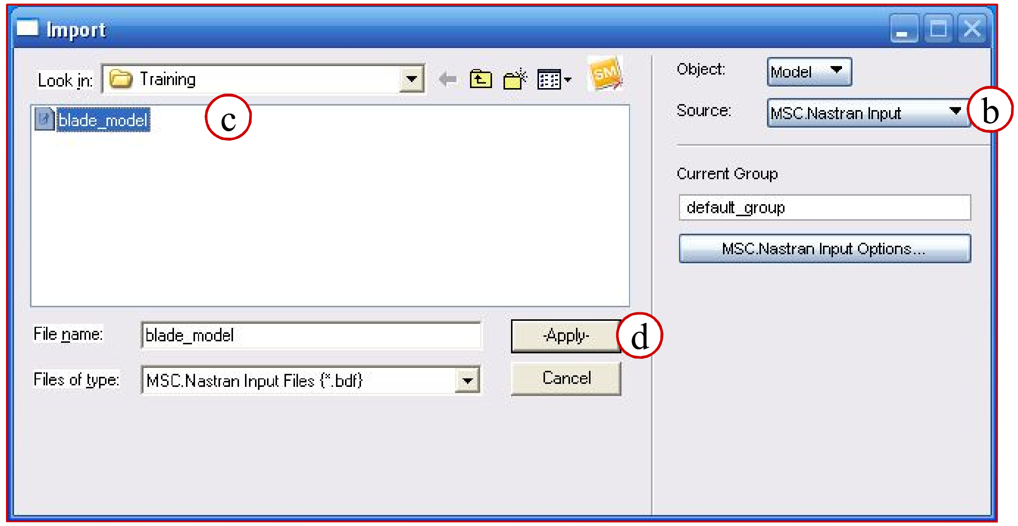


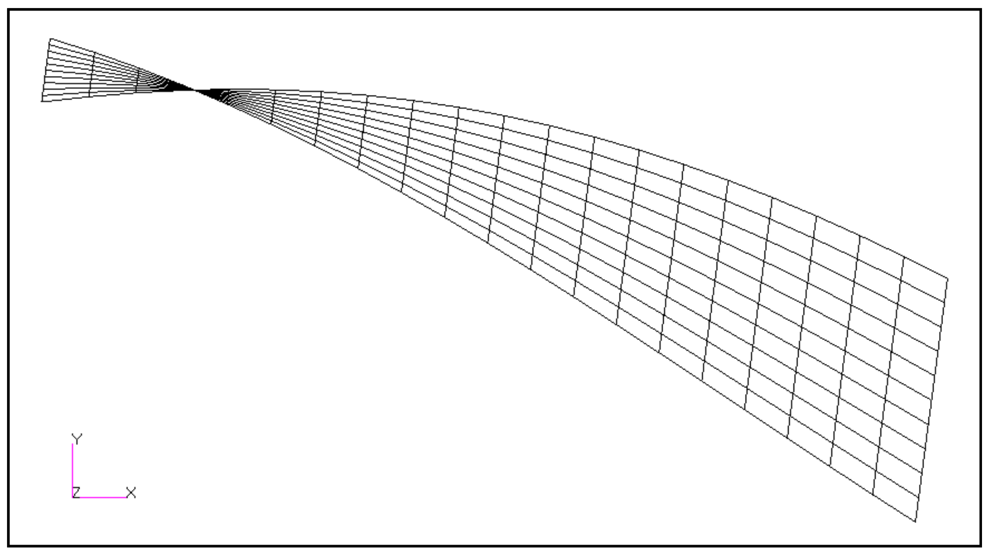
Create a new database.
Name it **blade.db.**

a. Pull down **File > New**.
b. Enter **blade** as the *file name*.
c. Click **OK**.
d. Confirm **MSC Nastran** as the *Analysis Code.*
e. Click **OK**.

Step 2. Import the FEM Mesh







Import the FEM model.

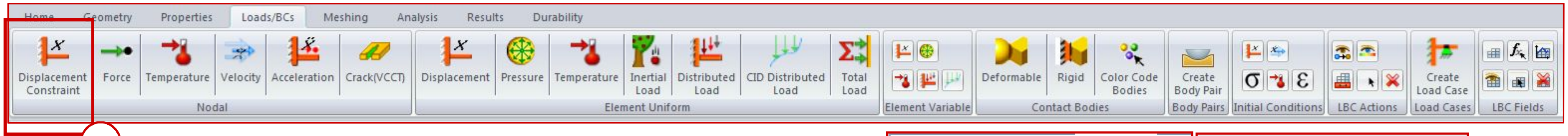
a. Pull down **File > Import**.

b. Select **MSC Nastran Input** as the *Source*.

c. Select **blade_model.bdf**.

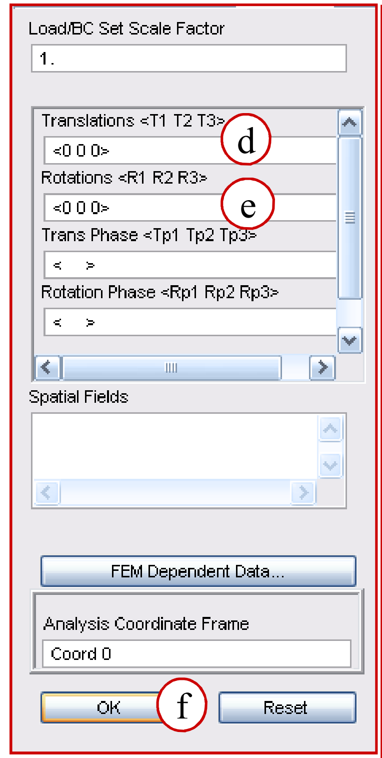
d. Click **Apply** (click OK on the File Import Summary).

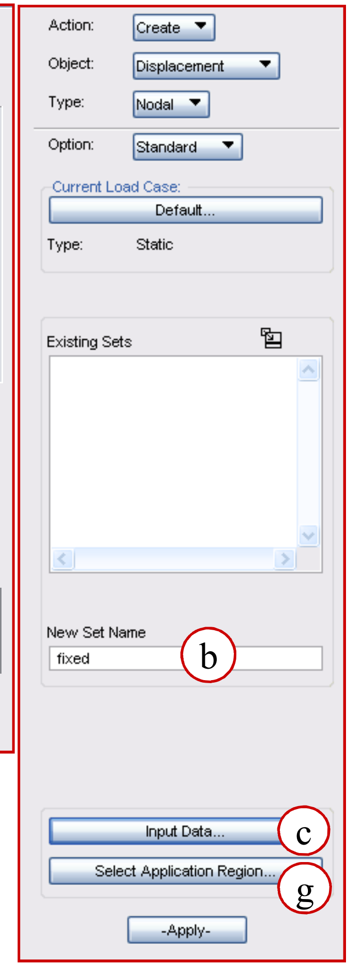
Step 3. Create the Displacement Boundary Condition



Create the fixed boundary condition at the left edge.

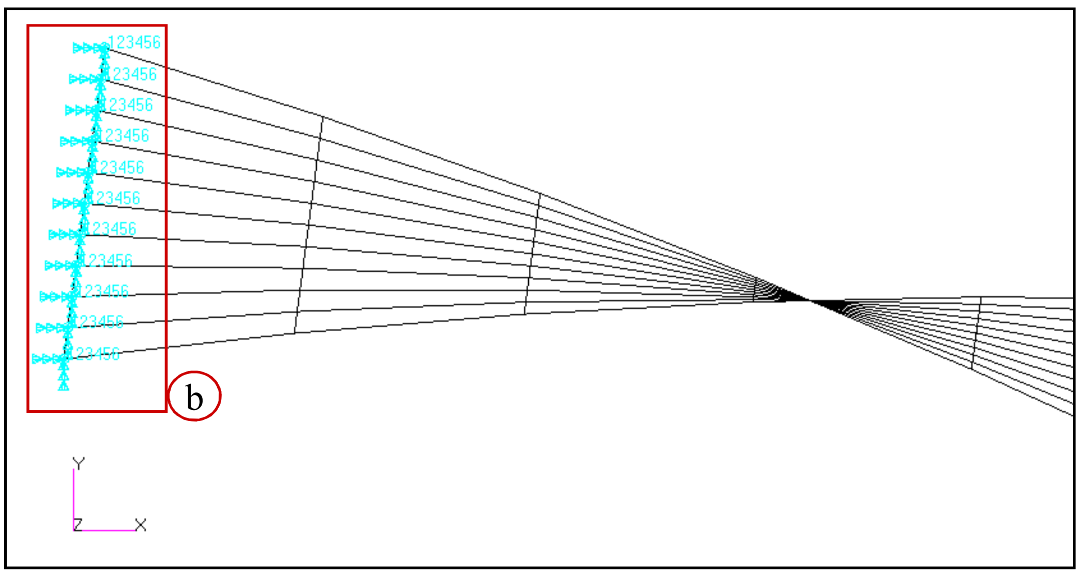
a. Under the *Loads/BCs* tab, click **Displacement Constraint** in the *Nodal* group.
b. Enter **fixed** for *New Set Name*.
c. Click **Input Data**.
d. Enter **<0 0 0>** for the *Translations.*
e. Enter **<0 0 0>** for the *Rotations*.
f. Click **OK**.
g. Click **Select Application Region**.

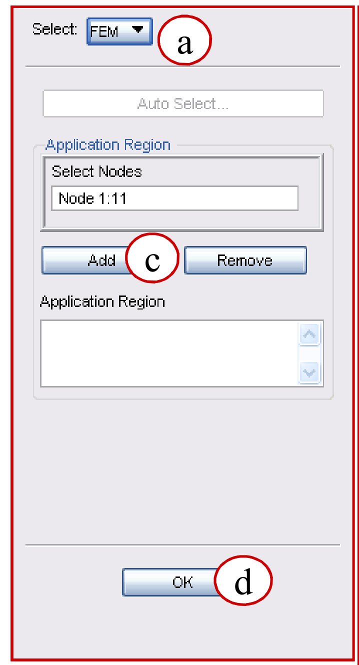


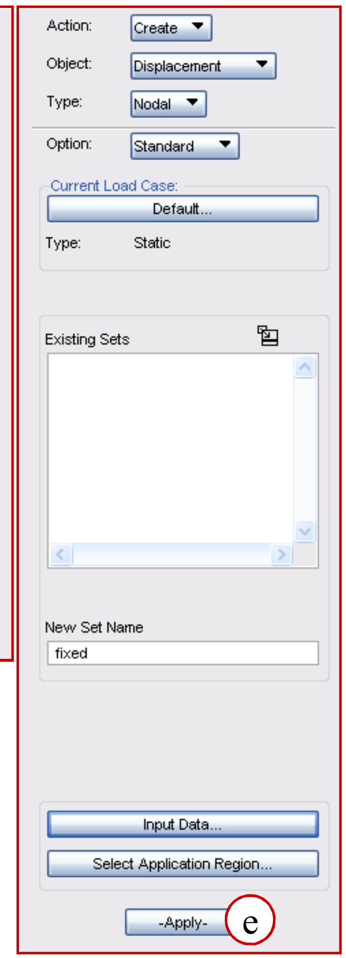


Step 3. Create the Displacement Boundary Condition (Cont.)

a. Pull down *Select* to **FEM**.
b. Drag a rectangle to select the nodes at the root of the blade.
c. Click **Add**.
d. Click **OK**.
e. Click **Apply**.

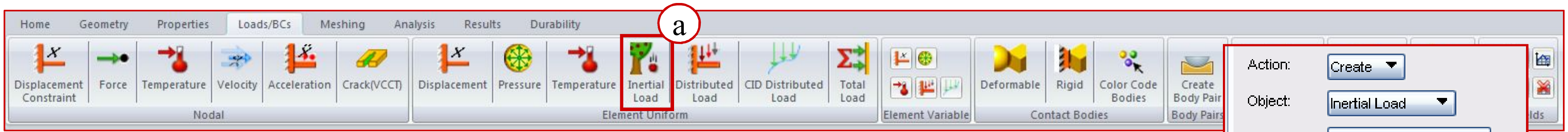






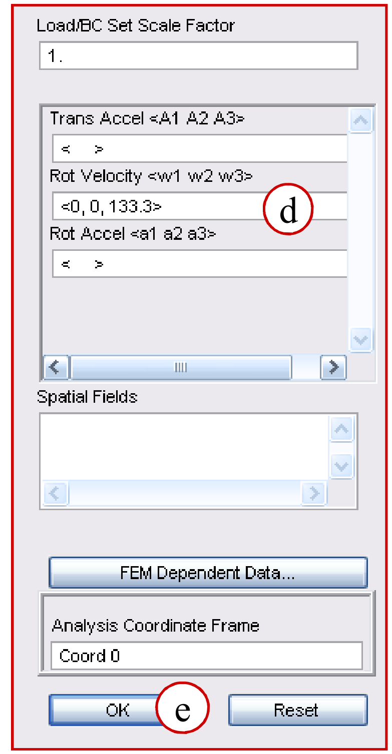
MSC Software

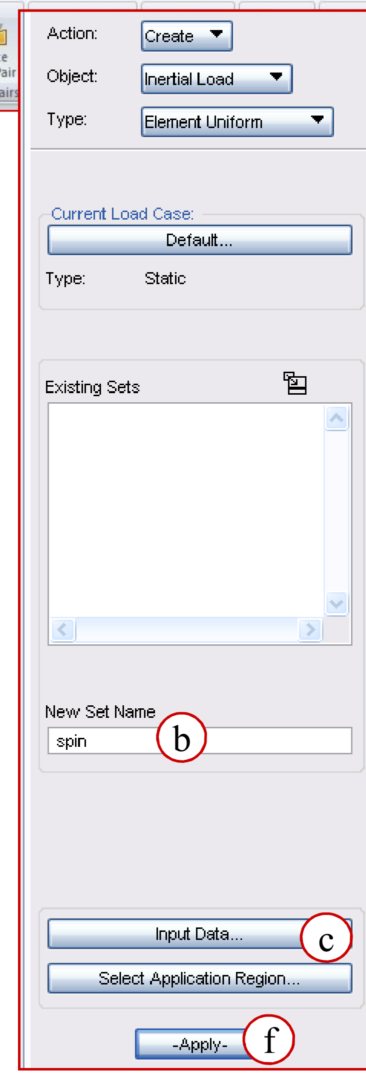
Step 4. Apply the Inertial Load

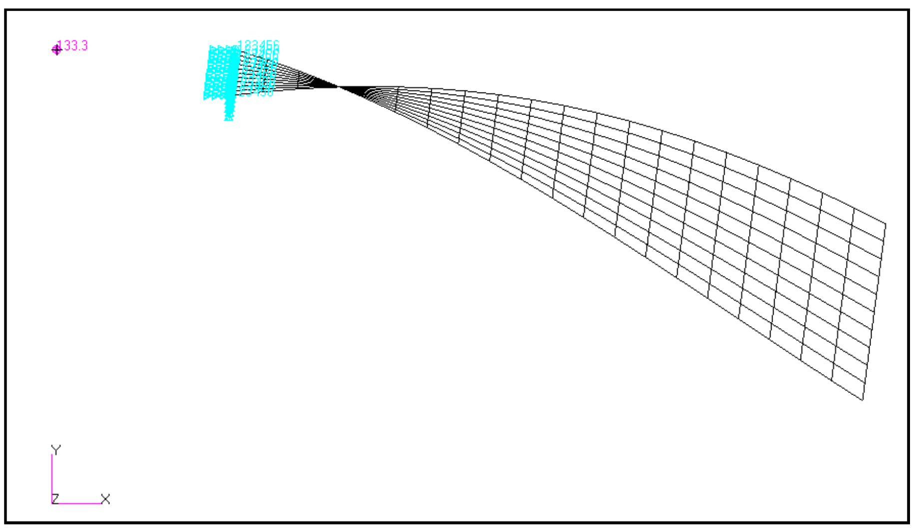


Create an inertial load in the Z direction where the rotational velocity is **133.3**.

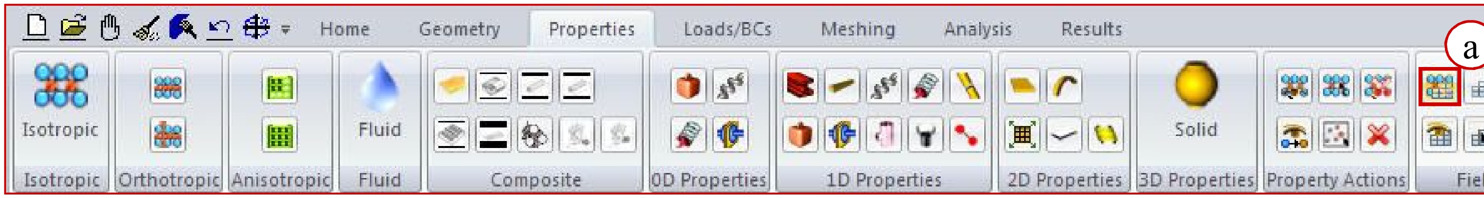
a. Click **Inertial Load** in the *Element Uniform* group.
b. Enter **spin** for *New Set Name*.
c. Click **Input Data**.
d. Enter **<0, 0, 133.3>** for the **Rot Velocity**.
e. Click **OK**.
f. Click **Apply**

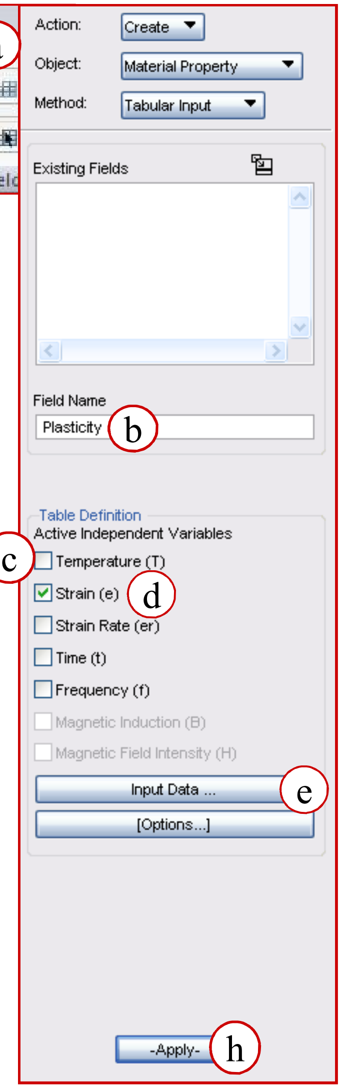






Step 5. Create a Field for Stress vs. Plastic Strain



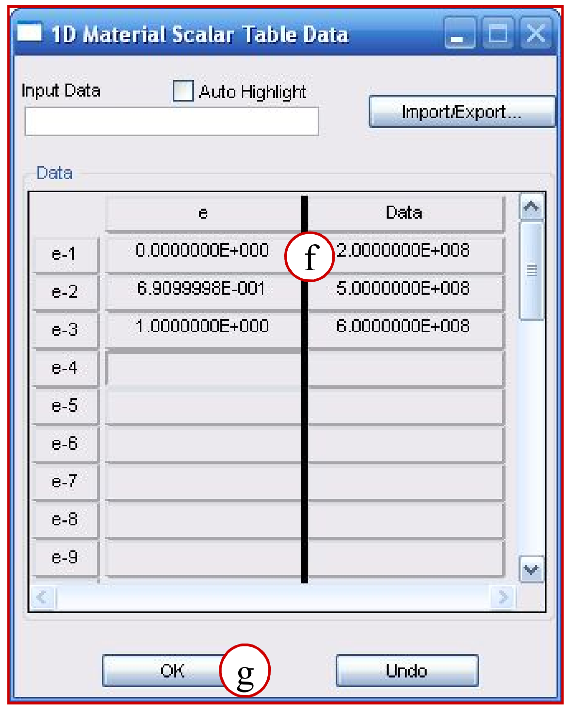


Define the Stress vs. Plastic Strain field for the steel material.

a. Under the *Properties* tab, click on **Material** in the *Fields* group.
b. Enter **Plasticity** for *Field Name*.
c. Uncheck **Temperature (T)** under *Active Independent Variables*.
d. Check **Strain (e)**.
e. Click **Input Data…**
f. Enter the following data as shown:

Strain(e)	Stress
0.0	2.0E8
0.691	5.0E8
1.0	6.0E8

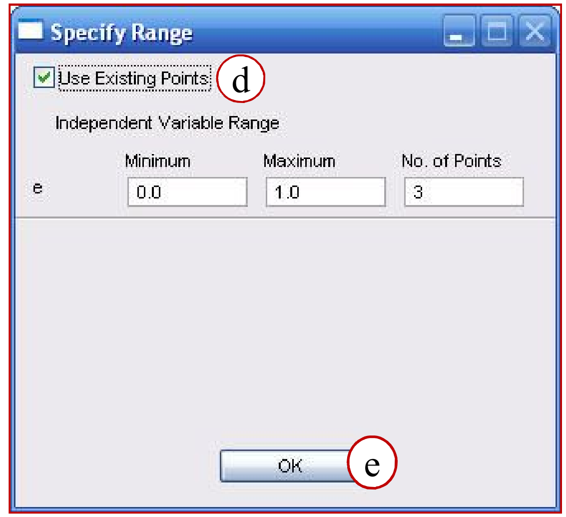
g. Click **OK**.
h. Click **Apply**.

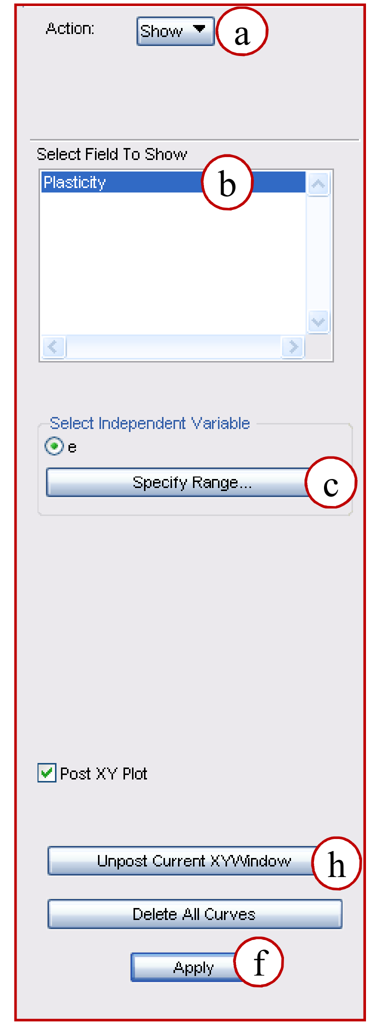


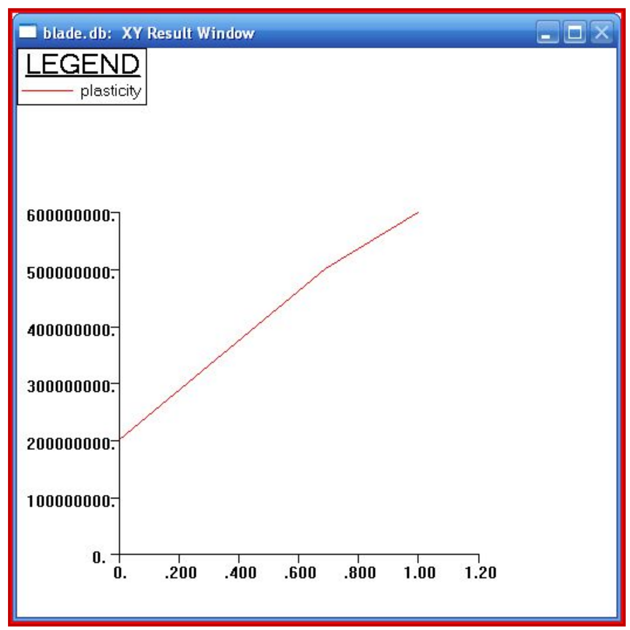
Step 6. Plot the Stress vs. Plastic Strain Data

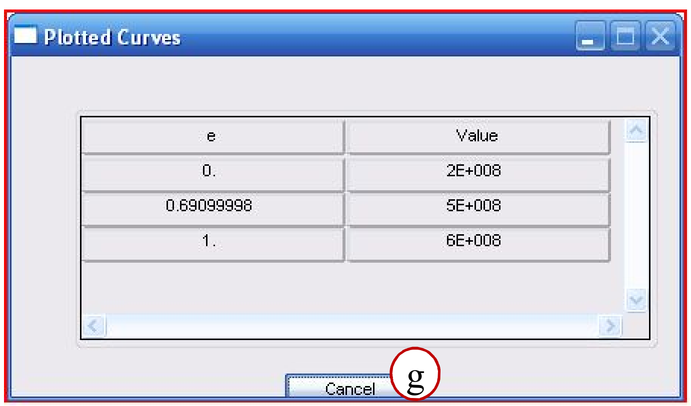
Plot the Stress vs. Plastic Strain Data.

a. Pull down *Action* to **Show**.
b. Select **Plasticity**.
c. Click **Specify Range**.
d. Check **Use Existing Points**.
e. Click **OK**.
f. Click **Apply**.
g. Click **Cancel** on the *Plotted Curves* form.
h. Click **Unpost Current XYWindow** when finished viewing the data.

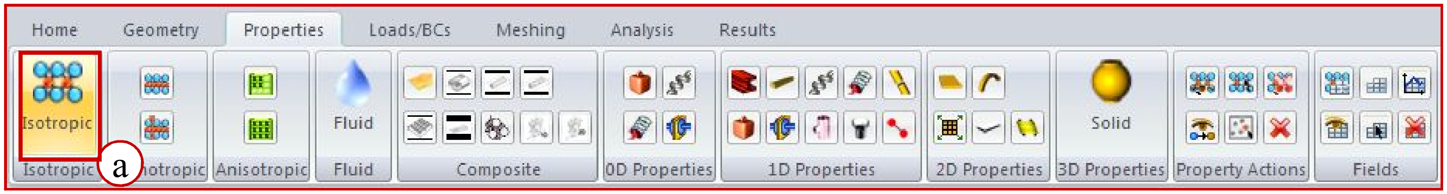






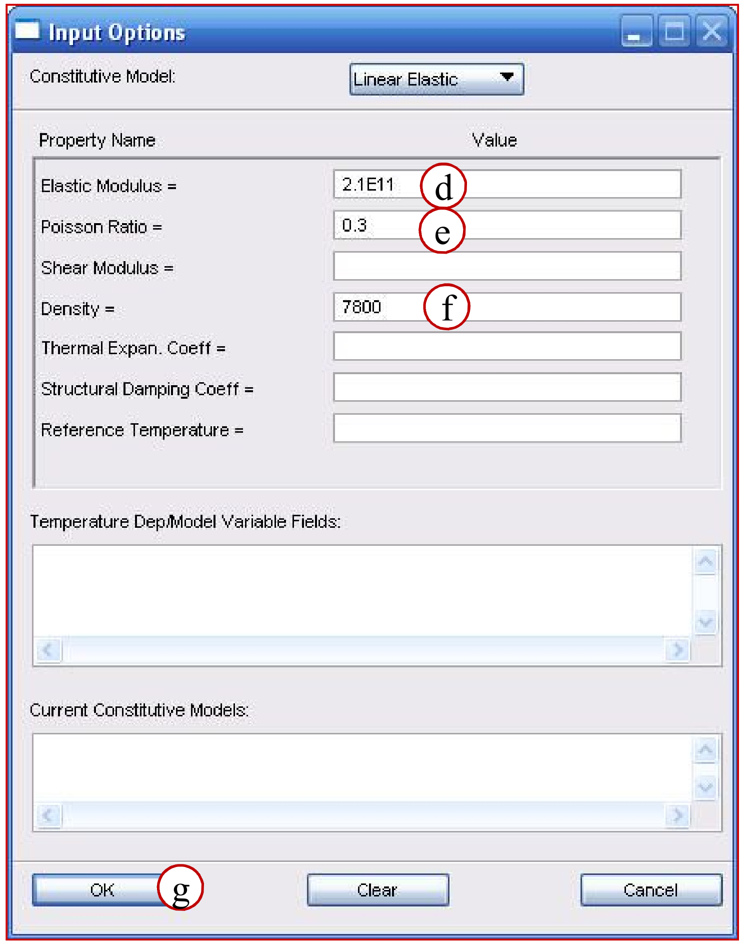


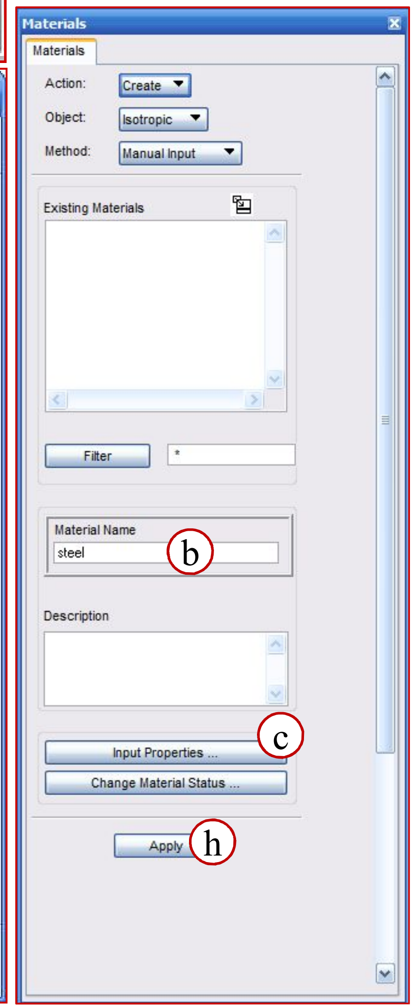
Step 7. Create the Steel Material Property



Create material properties for steel.

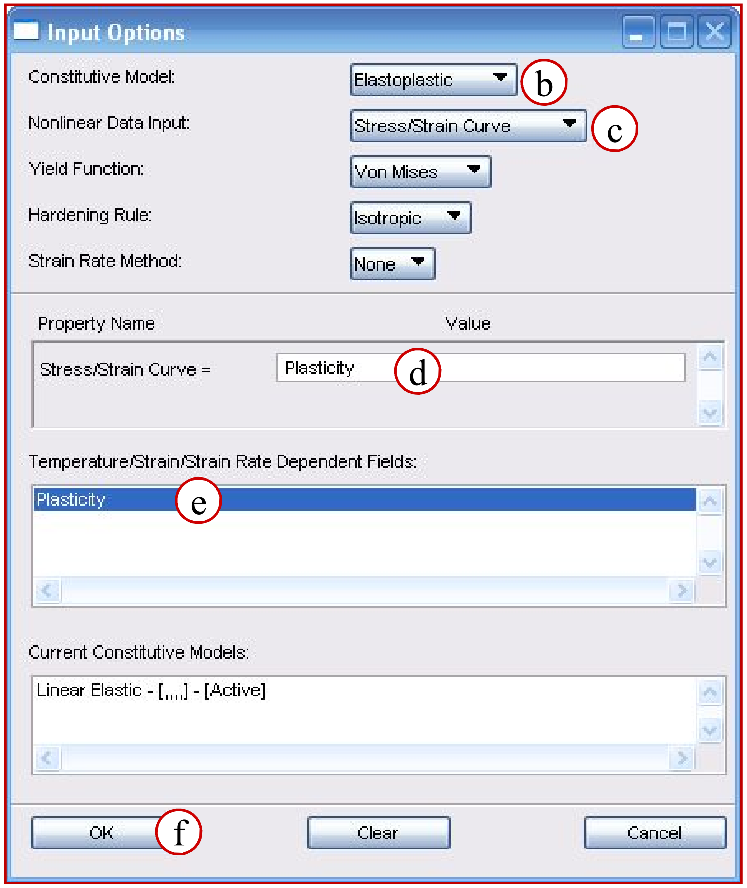
a. Click **Isotropic** in the *Isotropic* group.
b. Enter **steel** for *Material Name*.
c. Click **Input Properties**.
d. Enter **2.1E11** for the *Elastic Modulus.*
e. Enter **0.3** for the *Poisson Ratio*.
f. Enter **7800** for the *Density*.
g. Click **OK**.
h. Click **Apply**.

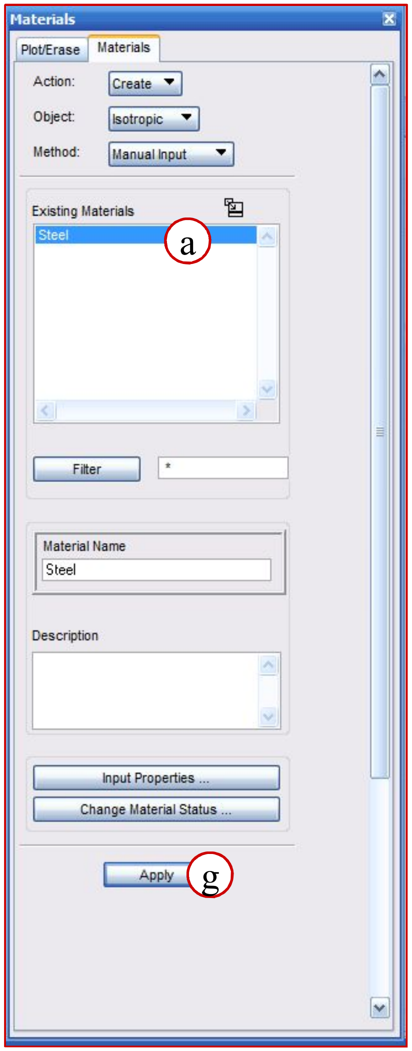




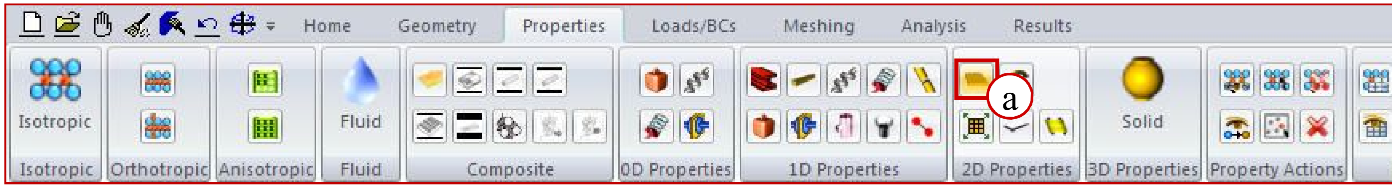
Step 7. Create the Steel Material Property (Cont.)

a. Select **steel** under the *Existing Materials* list.
b. Pull down **Elastoplastic** for *Constitutive Model*
c. Verify that **Stress/Strain Curve** is selected for *Nonlinear Data Input.*
d. Click in the **Stress/Strain Curve** box.
e. Select **Plasticity**
f. Click **OK**.
g. Click **Apply**.



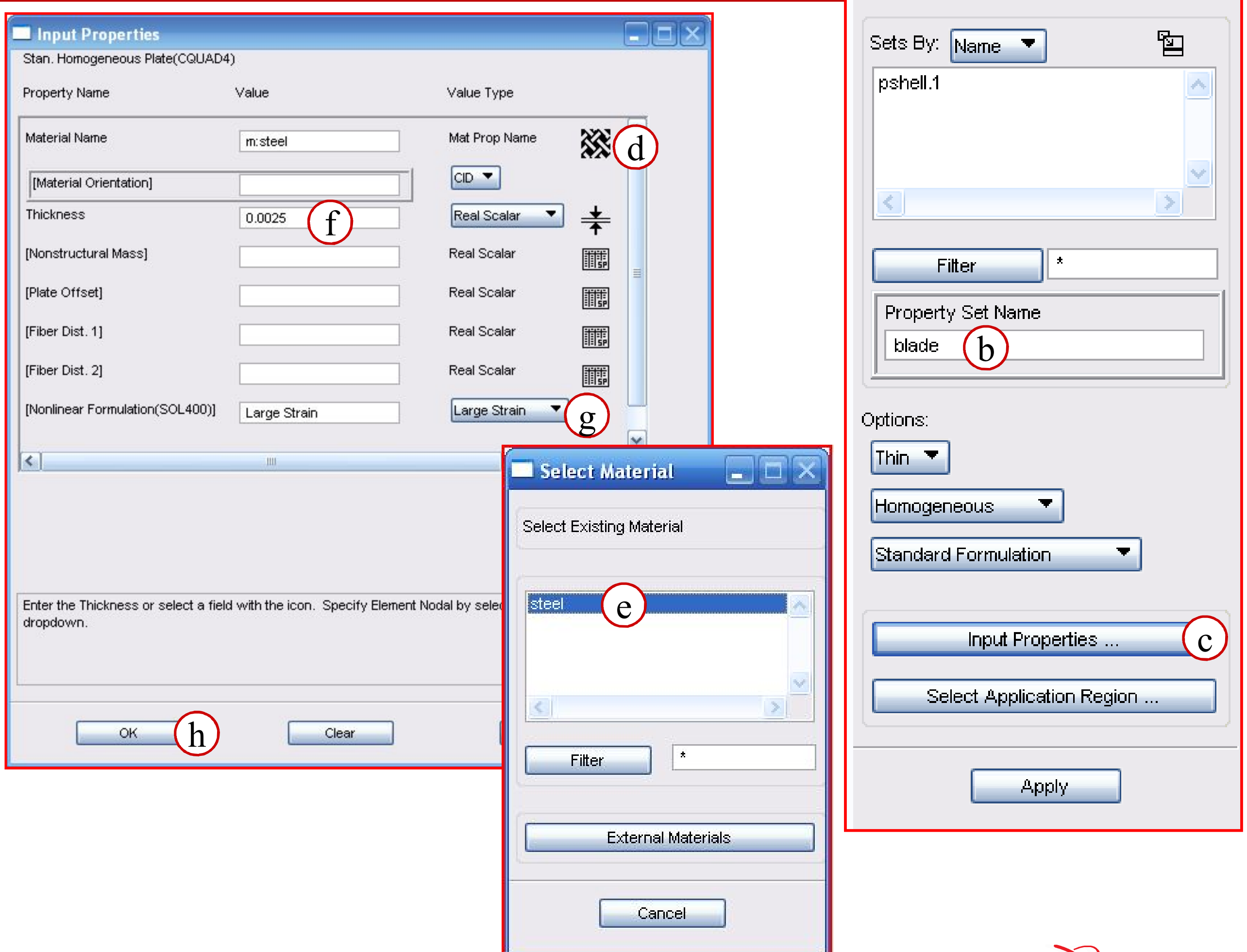


Step 8. Create the Element Properties



Create a nonlinear shell element property with steel as the material and thickness of **0.0025**.

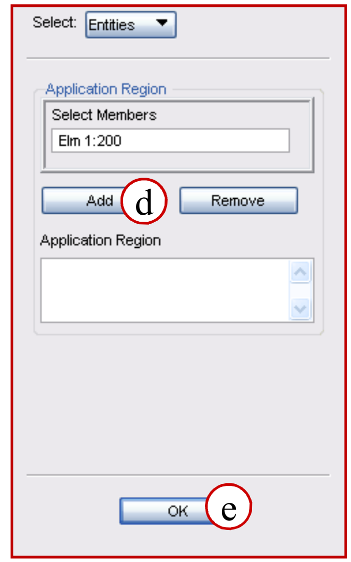
a. Click **Shell** in the *2D Properties* group.
b. Enter ***blade*** for the *Property Set Name.*
c. Click **Input Properties**.
d. Click on the **Mat Prop Name** icon.
e. Select **steel**.
f. Enter **0.0025** for *Thickness*.
g. Pull down **Large Strain** for *Nonlinear Formulation (SOL400).*
h. Click **OK**.

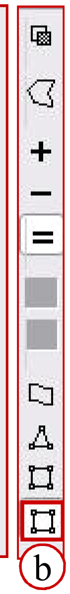


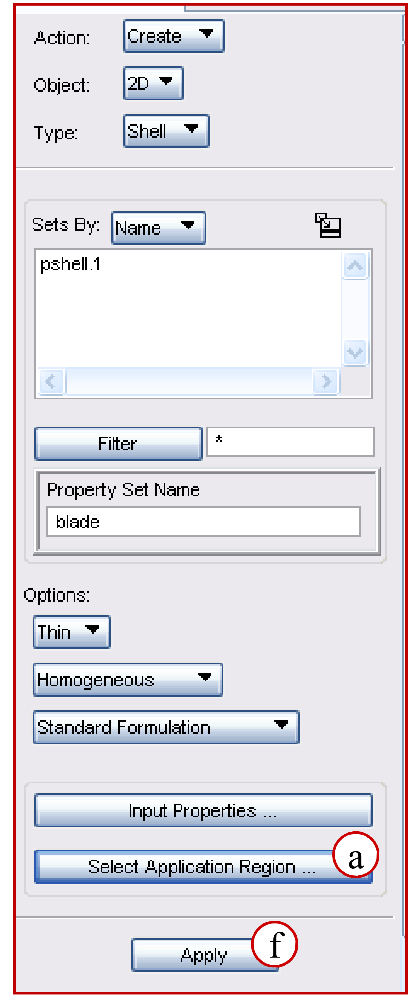
MSC Software

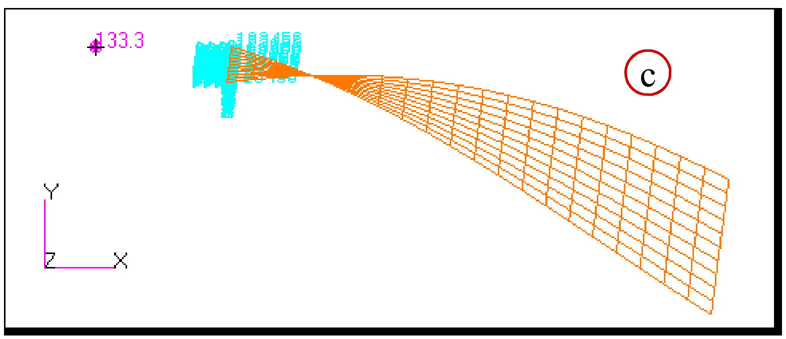
Step 8. Create the Element Properties (Cont.)

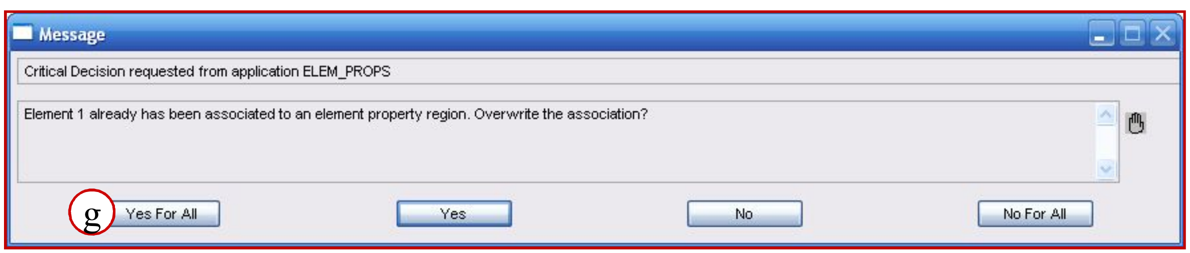
a. Click **Select Application Region**.
b. Click **Shell element** on the *Picking Filters* toolbar.
c. Select all the elements.
d. Click **Add**
e. Click **OK**.
f. Click **Apply**.
g. Click on **YES for ALL** on the pop-up Message form.



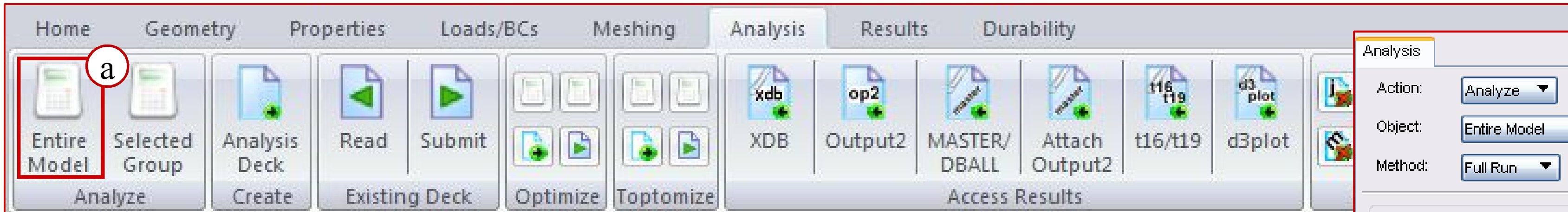








Step 9. Set up the Analysis



Create and run an implicit nonlinear analysis:

a. Under the *Analysis* tab click on **Entire Model** in the *Analyze* group.

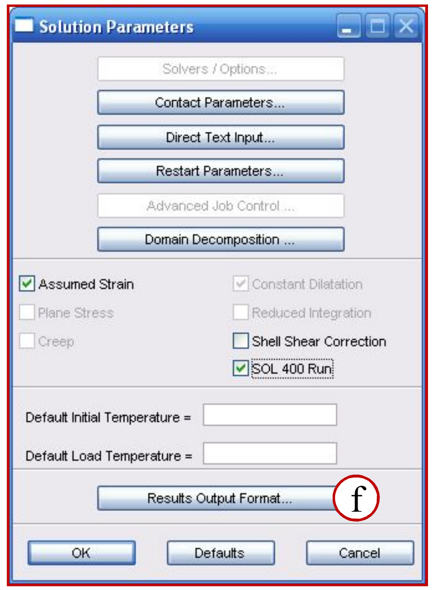
b. Enter **blade** for *Job Name*.

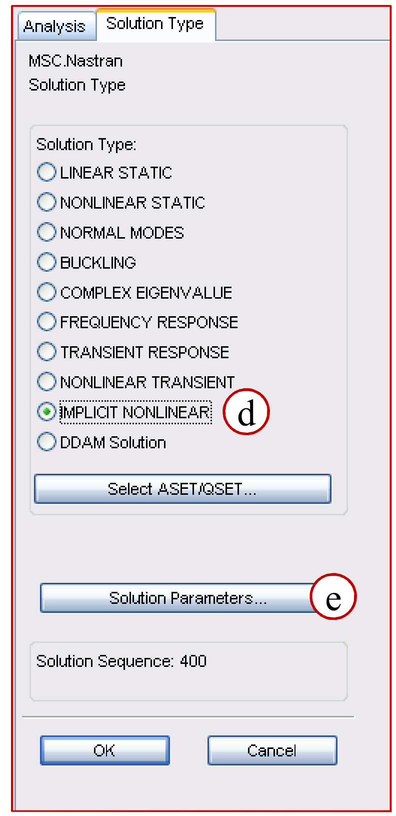
c. Click on **Solution Type**.

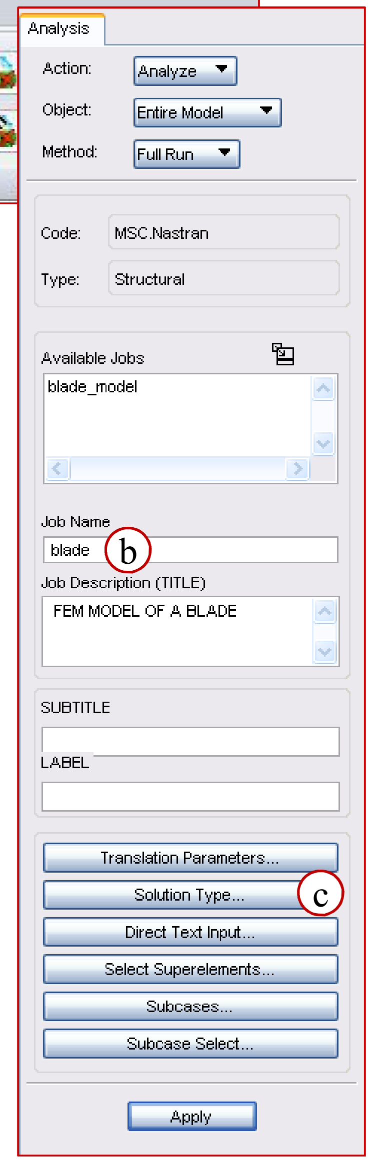
d. Select **IMPLICIT NONLINEAR**.

e. Click **Solution Parameters**.

f. Click **Results Output Format**.



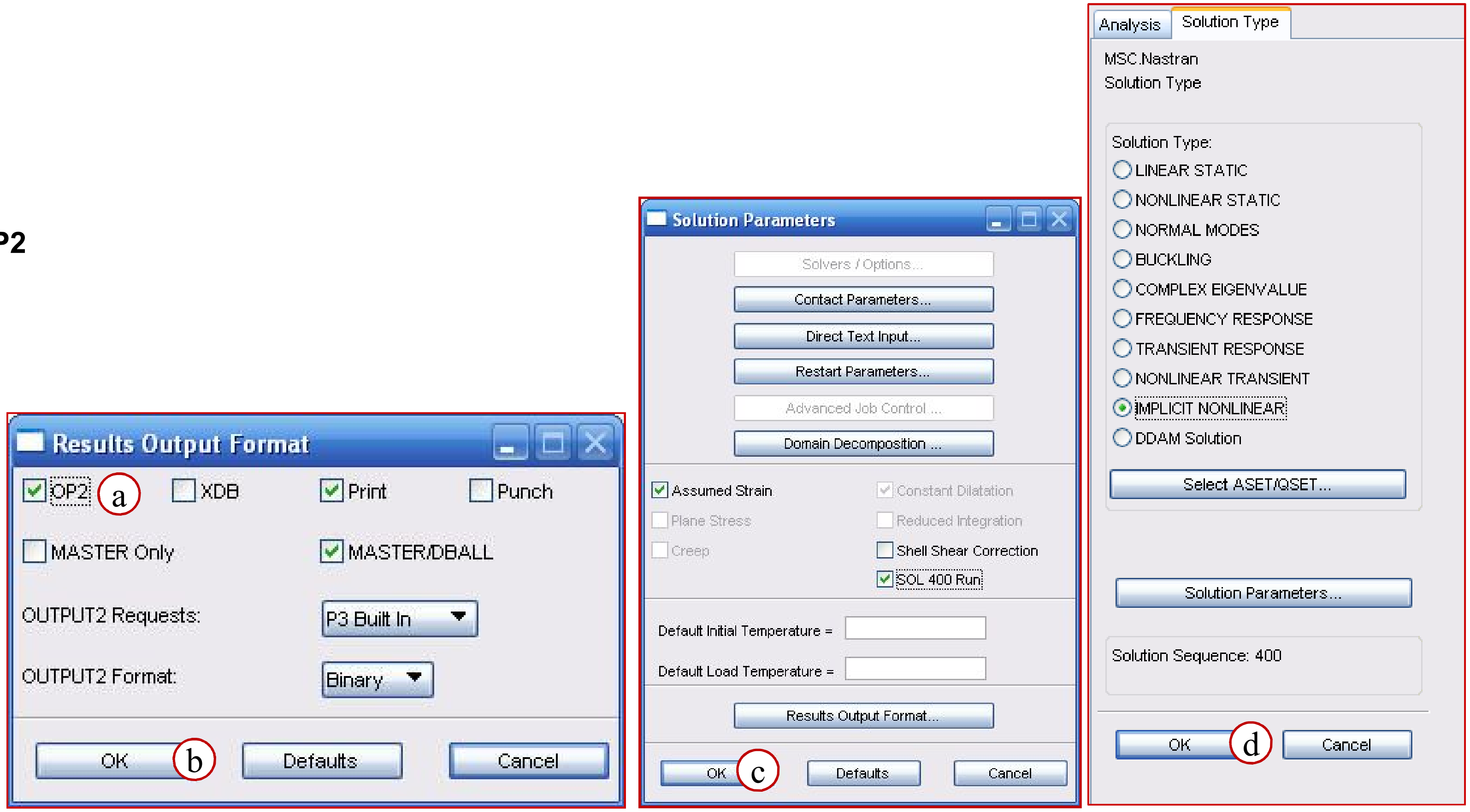




MSC Software

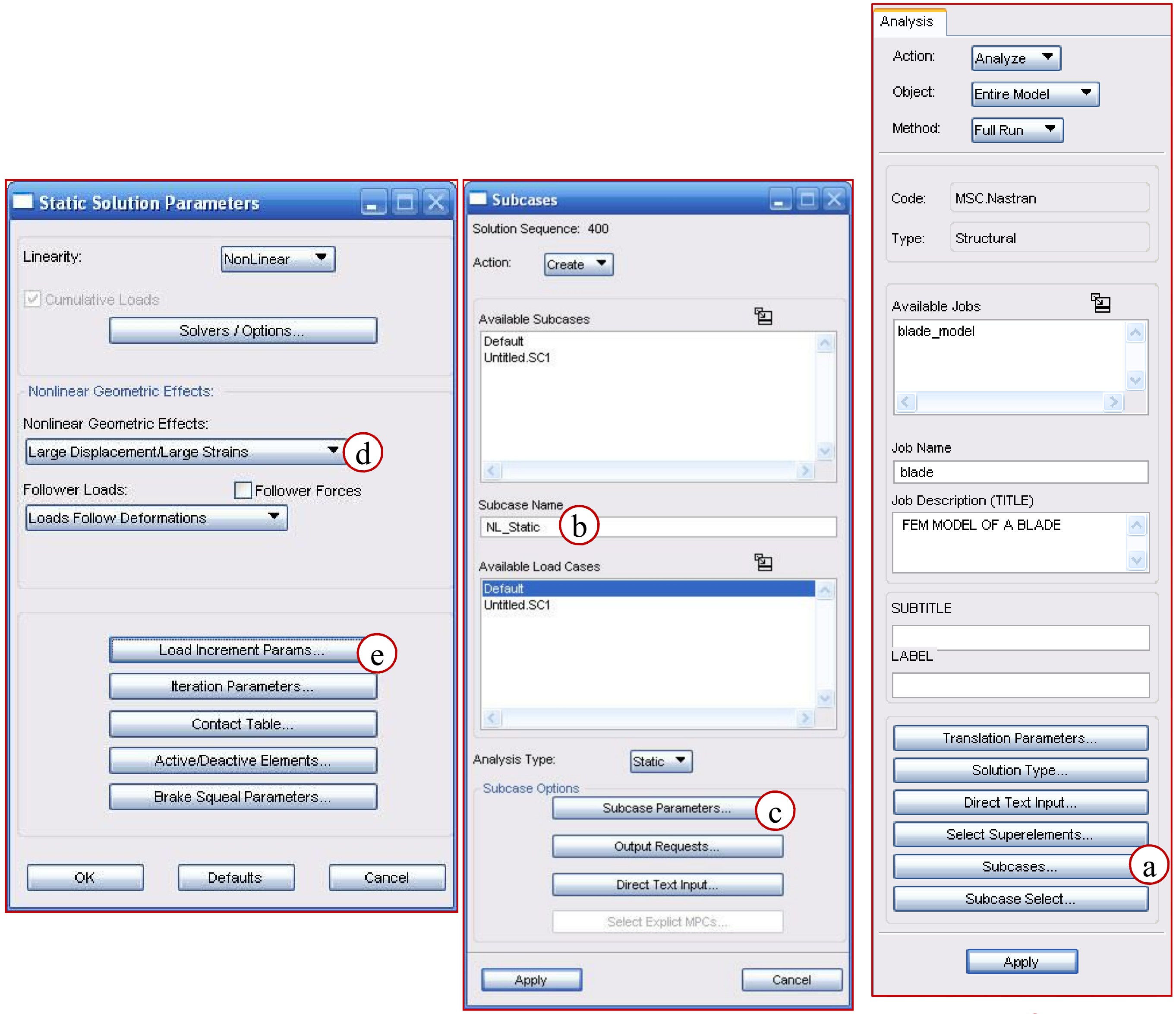
Step 9. Set up the Analysis (Cont.)

a. Select **OP2**
b. Click **OK**
c. Click **OK**
d. Click **OK**



Step 9. Set up the Analysis (Cont.)

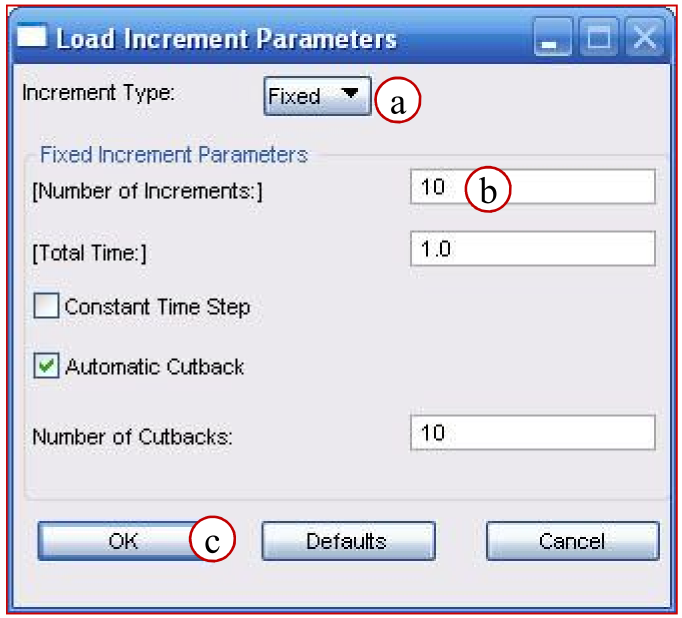
a. Click on **Subcases**.
b. Enter **NL_Static** for *Subcase Name*.
c. Click on **Subcase Parameters**.
d. Pull down **Large Displacement/ Large Strains** for *Nonlinear Geometric Effects*.
e. Click on **Load Increment Parameters**.

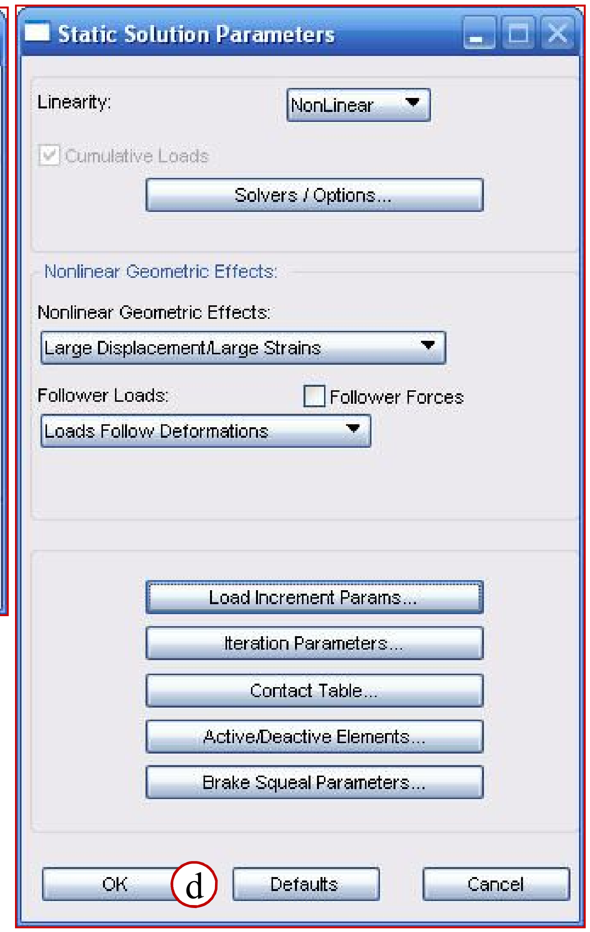


MSC Software

Step 9. Set up the Analysis (Cont.)

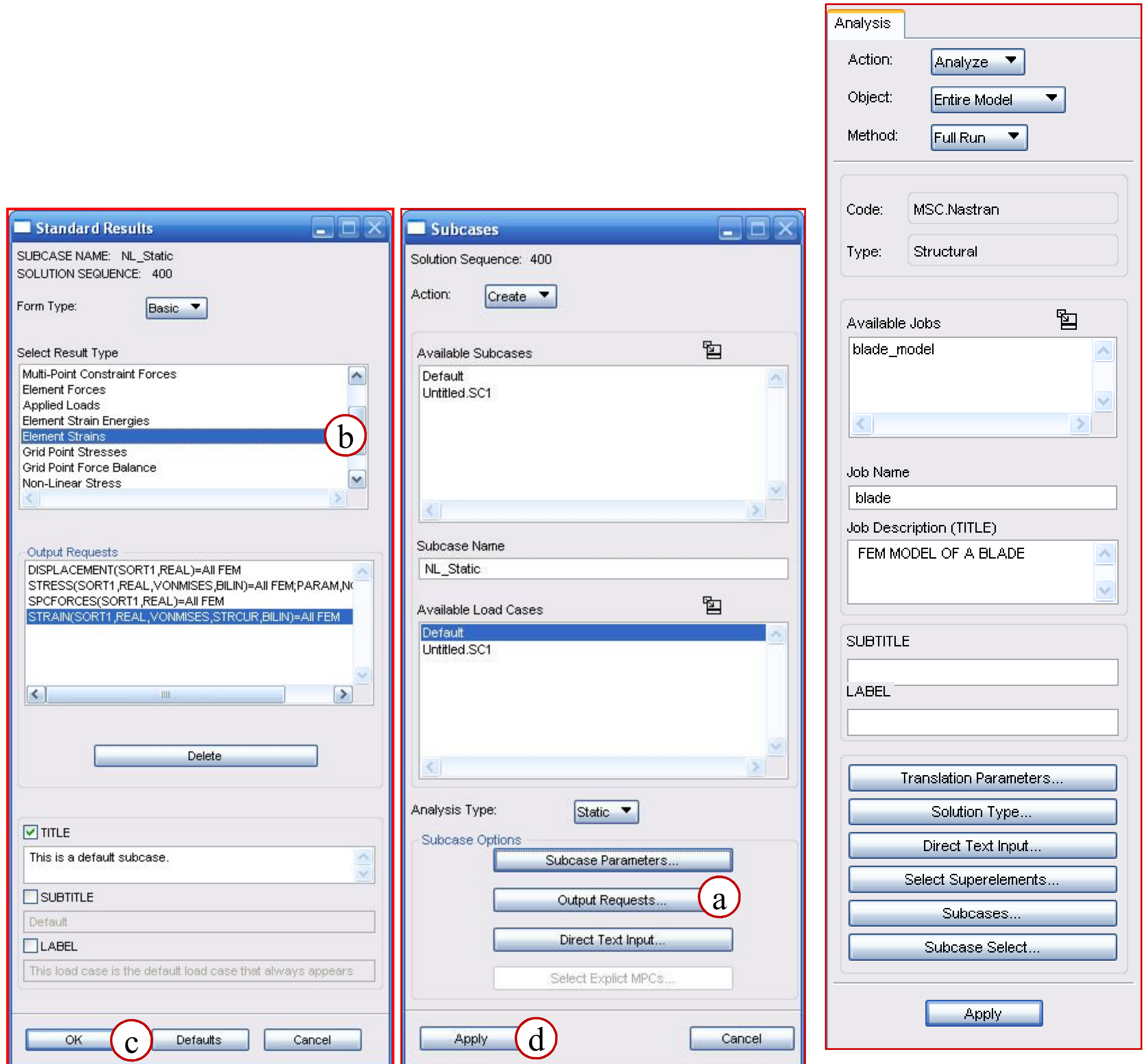
a. Pull down **Fixed** for *Increment Type*
b. Enter ***10*** for *Number of Increments*
c. Click **OK**
d. Click **OK**





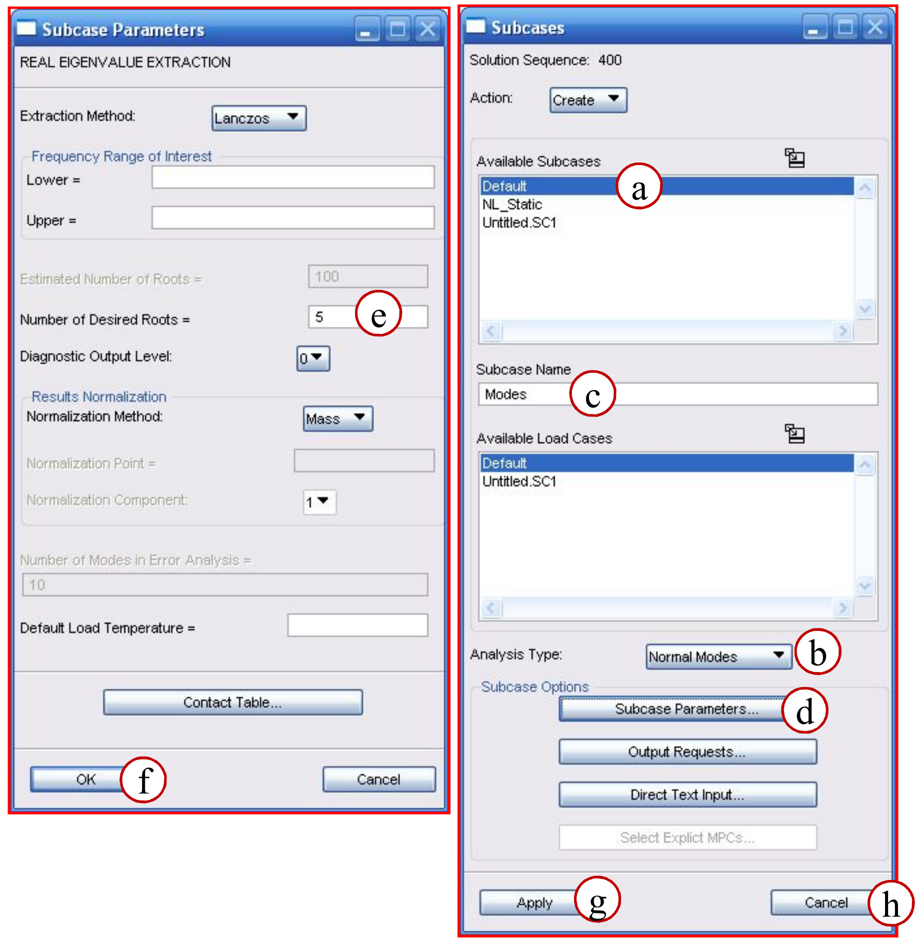
Step 9. Set up the Analysis (Cont.)

a. Click on **Output Requests**.
b. Select **Element Strains**.
c. Click **OK**.
d. Click **Apply**.



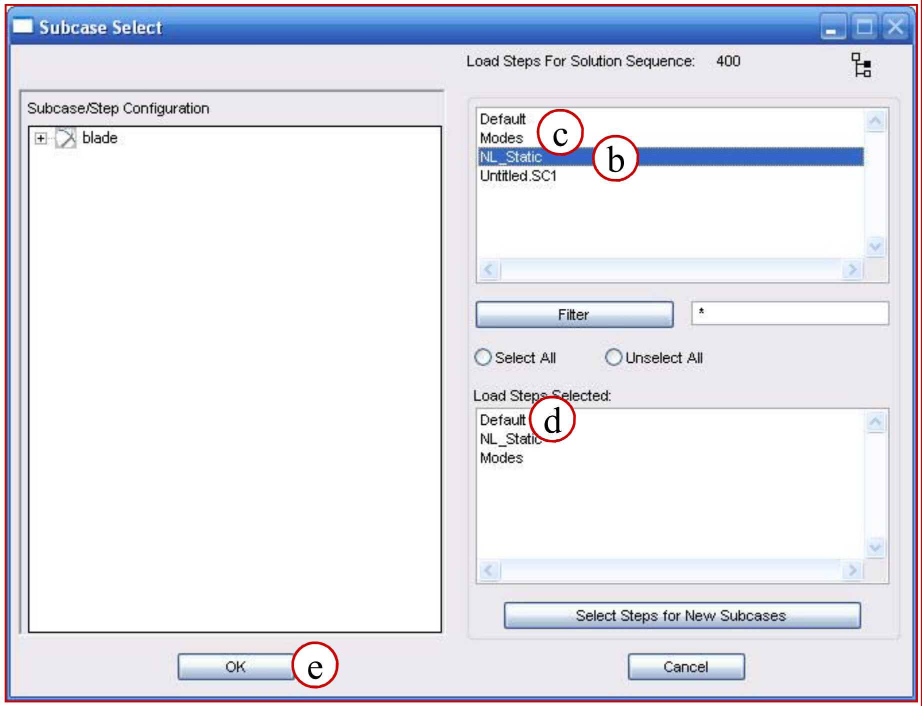
Step 9. Set up the Analysis (Cont.)

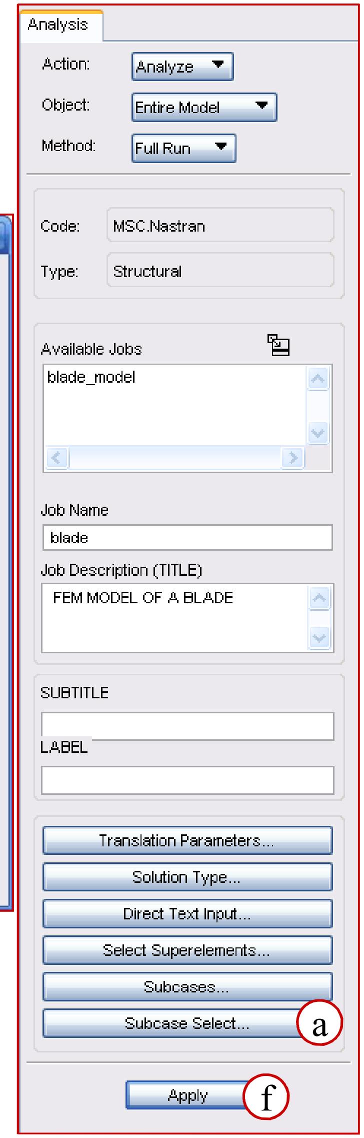
a. Click on **Default** under *Available Subcases.*
b. Select **Normal Modes** as *Analysis Type*.
c. Enter ***Modes*** for *Subcase Name.*
d. Click on **Subcase Parameters**.
e. Enter ***5*** for *Number of Desired Roots.*
f. Click **OK**.
g. Click **Apply**.
h. Click **Cancel**.



Step 9. Set up the Analysis (Cont.)

a. Click on **Subcase Select**.
b. Select **NL_Static**.
c. Select **Modes**.
d. Unselect **Default**.
e. Click **OK**.
f. Click **Apply**.

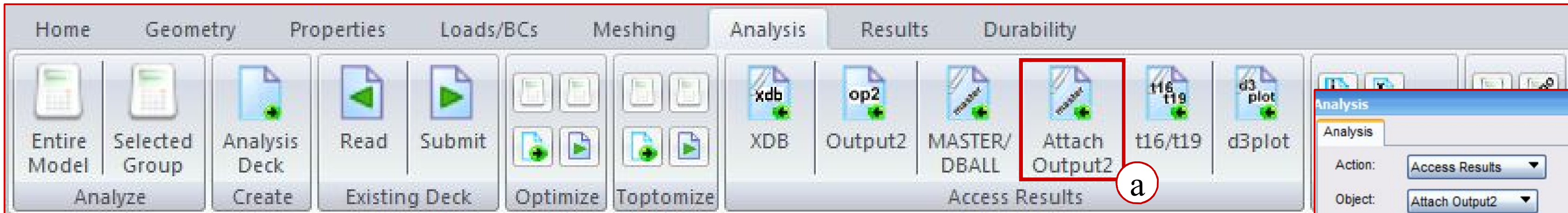




Be sure to select NL_Static before Modes (Analysis Chaining: nonlinear static analysis followed by normal modes analysis).

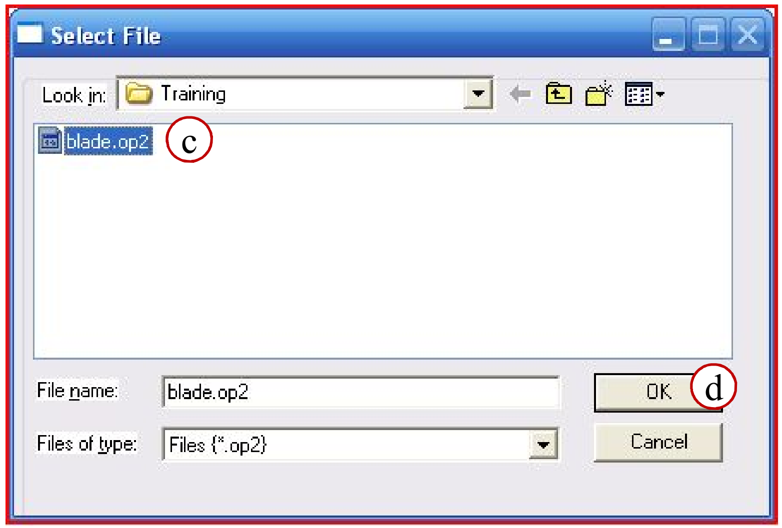
MSC Software

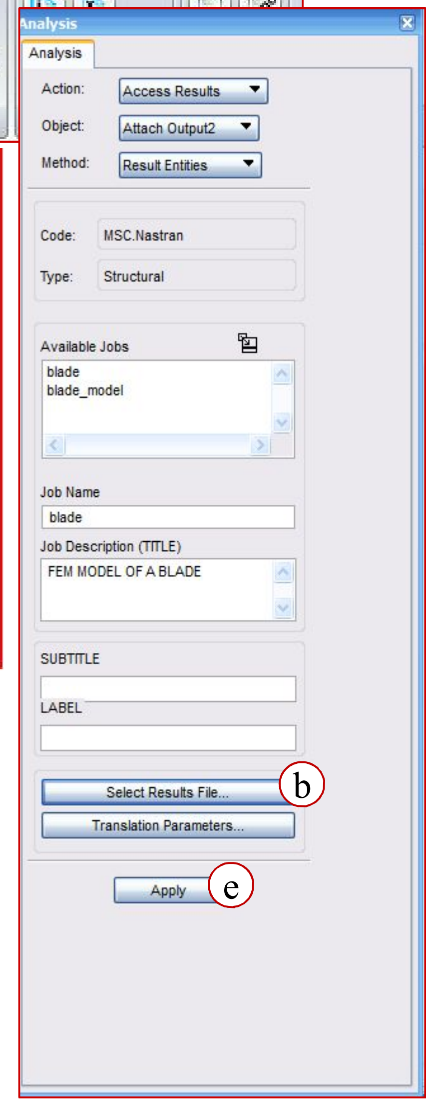
Step 10. Attach the Results



Once the analysis is complete, attach the results to the database.

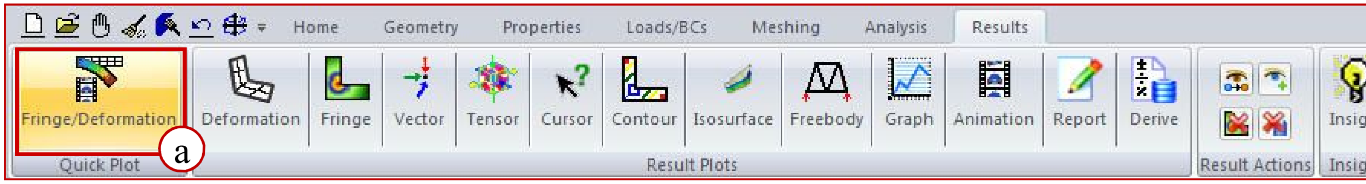
a. Under the *Analysis* tab click **Attach Output2** in the *Access Results* group.
b. Click **Select Results File**.
c. Select **blade.op2**.
d. Click **OK**.
e. Click **Apply**.





MSC Software

Step 11. Plot the Stress Results



Post-process the von Mises stresses.

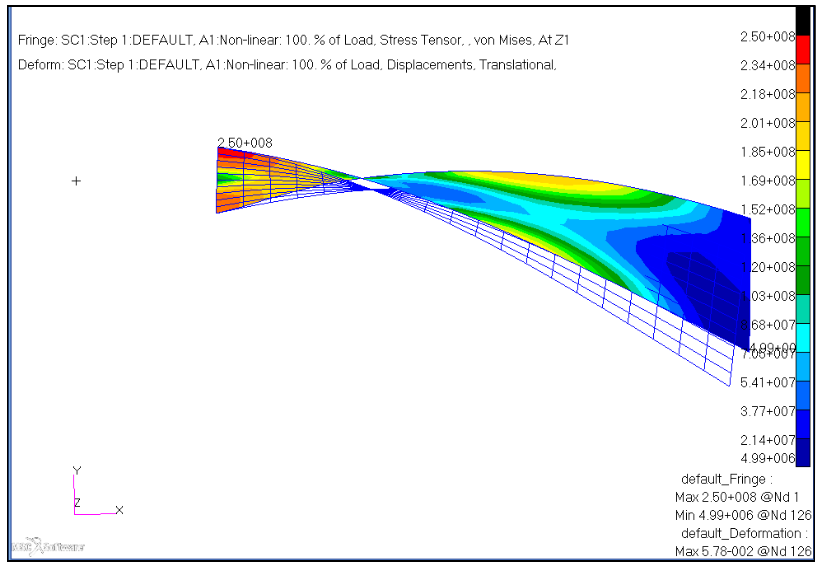
a. Under the *Results* tab click on **Fringe/Deformation** in the *Quick Plot* group.

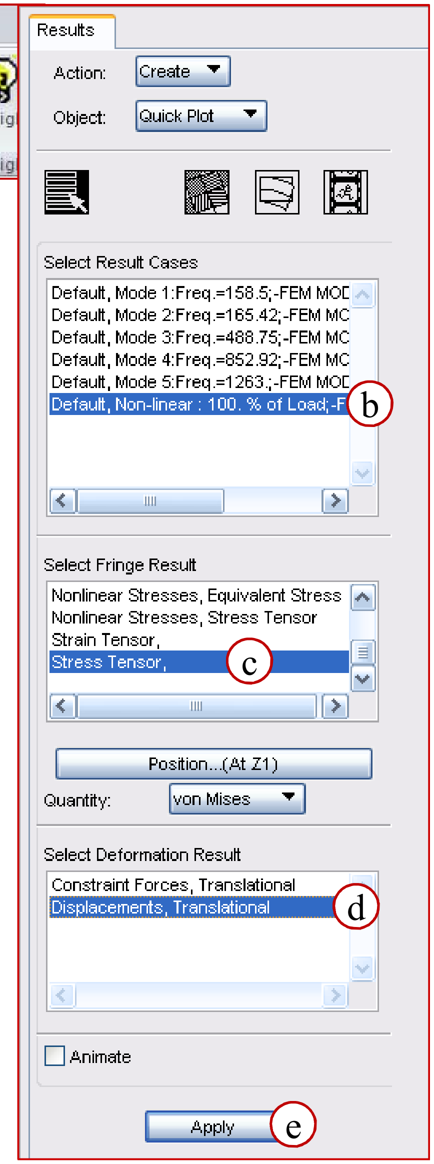
b. Select the **Non-linear** *Result Case.*

c. Select **Stress Tensor,** for the *Fringe Result*.

d. Select **Displacements, Translational** for the *Deformation Result.*

e. Click **Apply.**

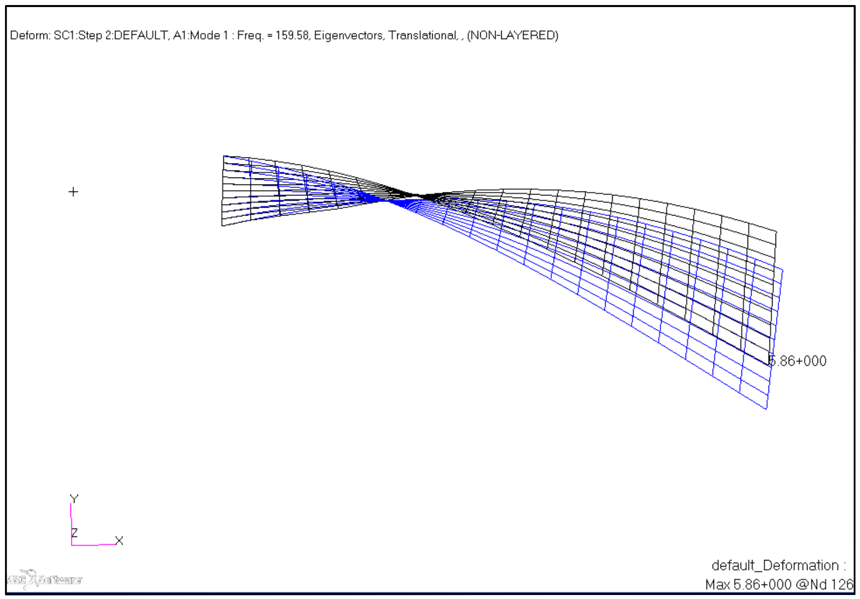




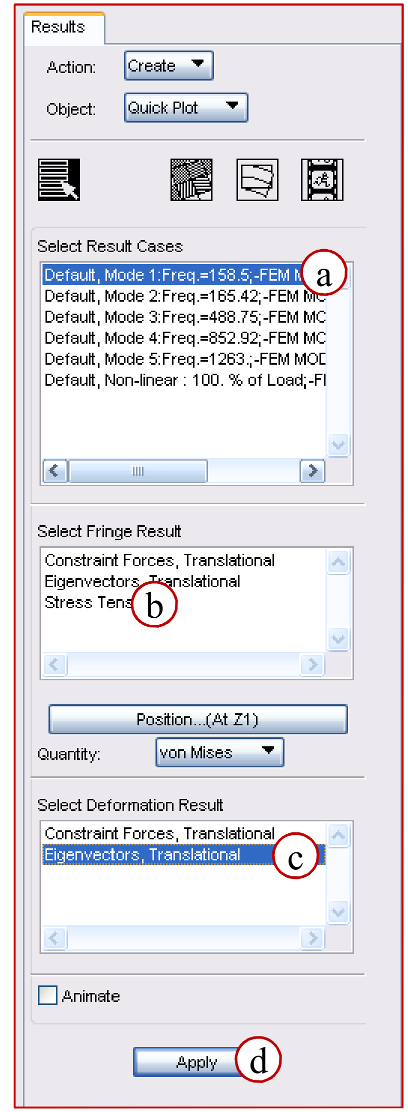
Step 12. Plot the Modes

Post process the normal modes.

a. Select the **Mode 1** *Result Case*.
b. Unselect **Stress Tensor,** for the *Fringe Result.*
c. Select **Eigenvectors, Translational** for the *Deformation Result.*
d. Click **Apply**.



Hold Ctrl to Unselect an Entry

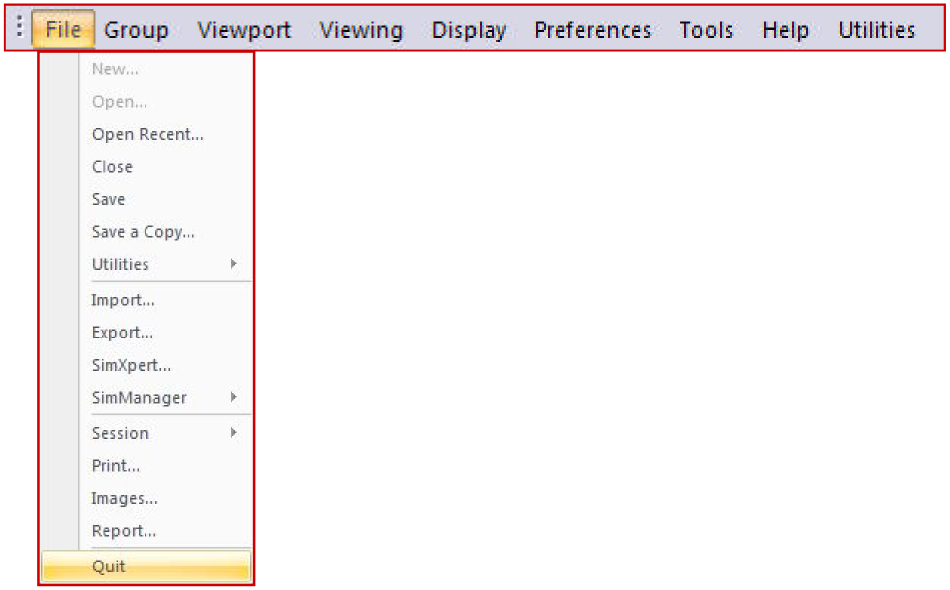


Normal Modes Results Summary

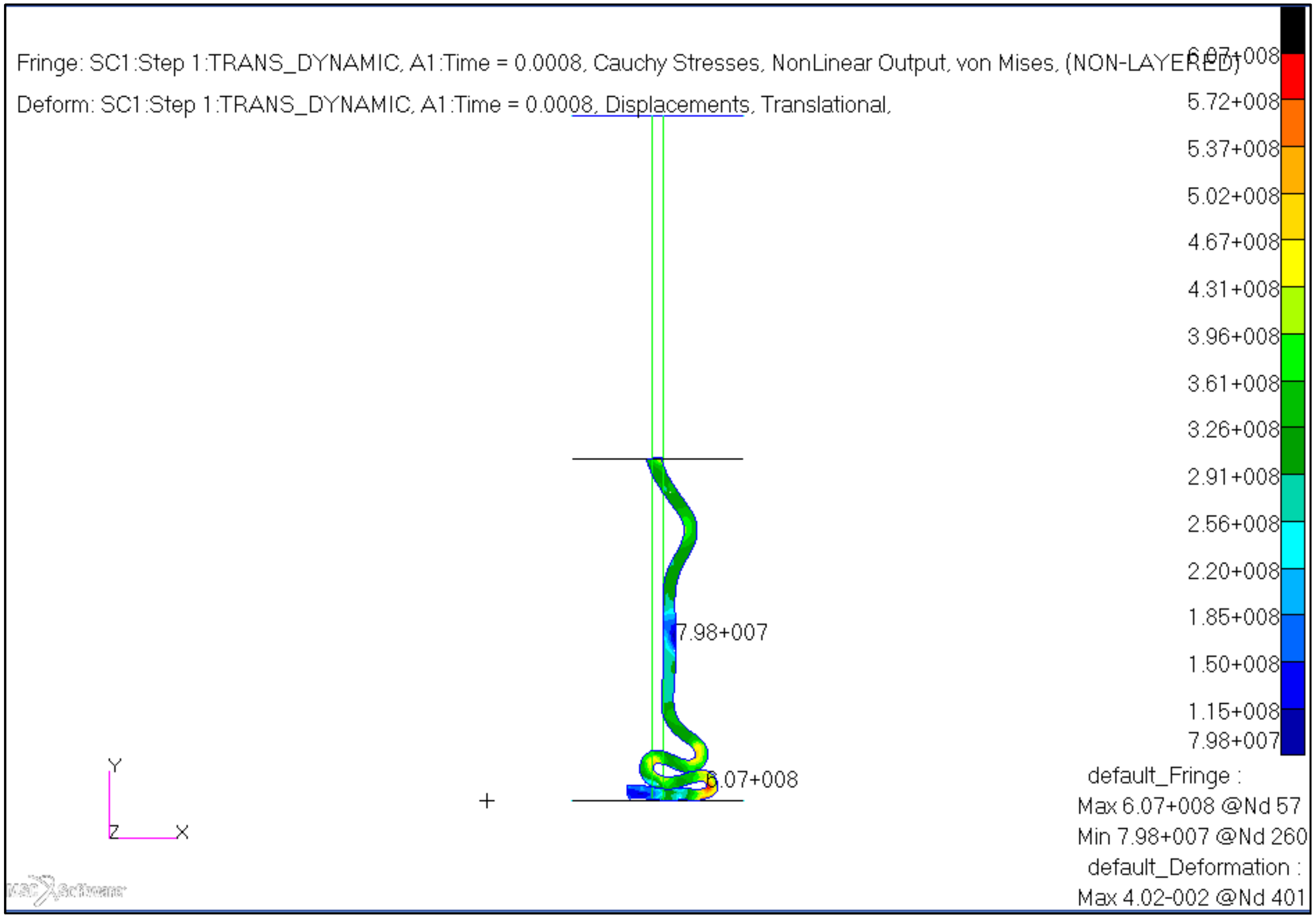
Mode	Frequency (Pre-Stiffened)	Frequency (un-stiffened*)
1	1.59E2	8.83E1
2	1.65E2	3.00E2
3	4.89E2	8.44E2
4	8.53E2	1.15E3
5	1.26E3	1.98E3

* *Obtained by changing the rotational speed to 1.E-6*

Step 13. Quit Patran



WORKSHOP B2
DYNAMIC COLLAPSE OF A CYLINDER



- **Workshop Objectives**
  - Perform Nonlinear Transient Dynamic Analysis on a cylinder.
  - Analyze the dynamic collapse of the cylinder. Examine the resultant stresses and deformations
- **Software Version**
  - Patran 2013
  - MSC Nastran 2013.1
- **Files Required**
  - cylinder_model.ses

- **Problem Description**
  - In this exercise, we analyze the dynamic collapse of a cylinder. The 0.08 meter long cylinder (radius = 0.02 m ; thickness = 0.0131 m) is compressed axially by two rigid bodies, one of which is fixed and the other moving with a velocity of 50 m/s. The purpose of the analysis is to examine the stresses and deformations. The cylinder is made of an elasto-plastic material with linear work hardening. We model the cylinder here with a 100x3 mesh of 4-noded, 2D solid axisymmetric (CQUADX) elements.
  - The non-linear material properties for the elasto-plastic material are as shown:
    - Elastic Modulus = **2.05E11**. Poisson Ratio = **0.3**. Density = **7800**.

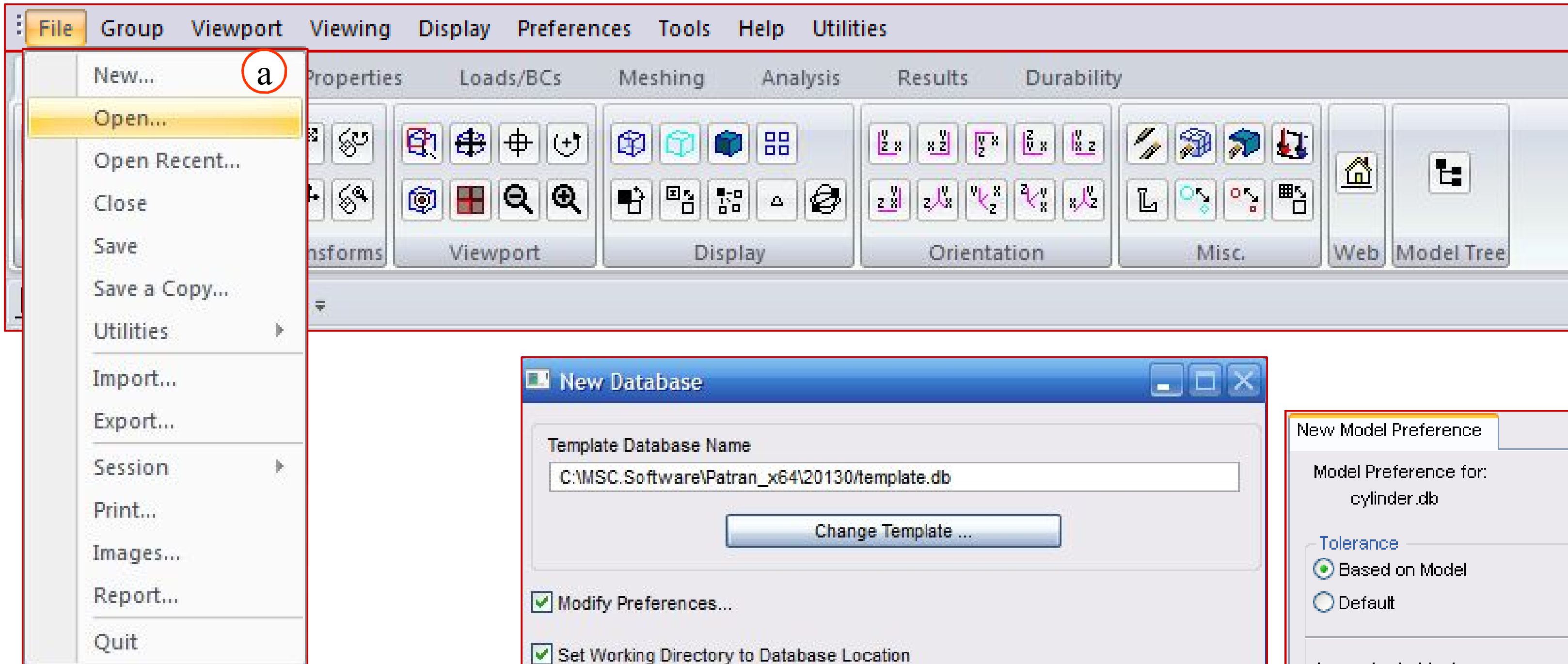
Strain(e)	Stress
0.0	2.0E8
1.0	6.0E8
2.0	6.1E8

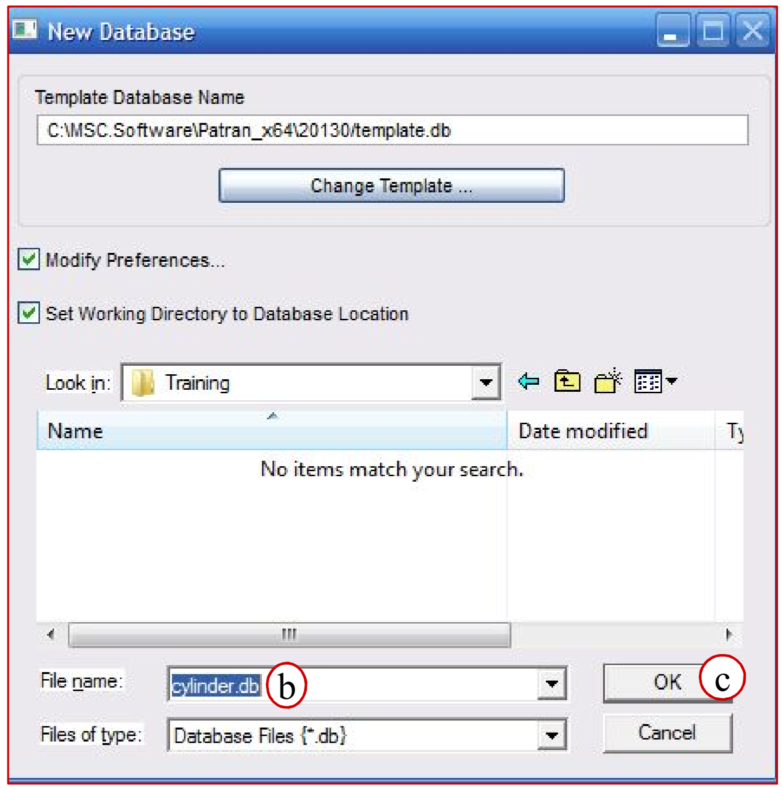
- **Suggested Exercise Steps**
  1. Open a New Database.
  2. Run the cylinder_model.ses file.
  3. Create a time dependent load case.
  4. Define the cylinder as a Deformable Contact Body.
  5. Define the two walls as Rigid Contact Bodies.
     - Velocity of -**50** for **upper_wall**.
  6. Preview the rigid body motion.
  7. Create material property with tabular input for plastic as shown in the problem description.
  8. Create a material with elastic and plastic properties.
     - Elastic Modulus = **2.05E11**. Poisson Ratio = **0.3**. Density = **7800**.
  9. Apply 2D Solid properties to the cylinder.

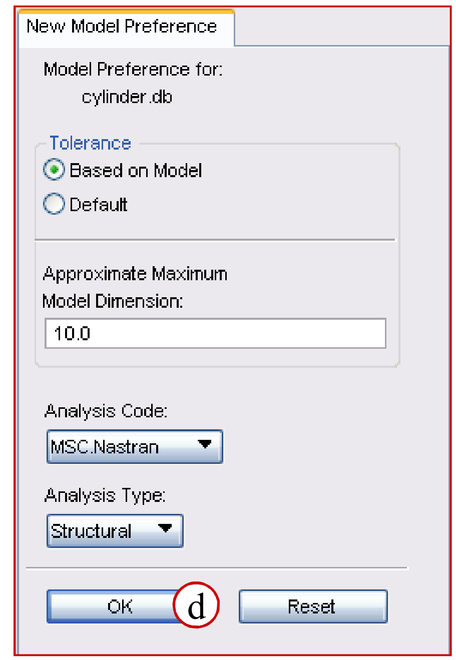
- **Suggested Exercise Steps (Cont.)**

10. Set up and run the implicit nonlinear analysis.
    - Set friction type to **Bilinear Coloumb**.
    - Select the .op2 results output format.
    - Create a Trans_Dynamic subcase and apply these properties: Time Step Size = **2E-6**. Total Time = **8E-4**. Matrix Update Method = **Pure Full Newton**. Number of Iterations per Update = **-1**. Turn on Displacement Error and Vector Component Method in load increment parameters.
    - Select Element Strains, Non-Linear Stress and Contact Results output request.
    - Unselect the Default subcase and select the Trans_Dynamic subcase.
    - Edit the exported input file.
11. Access the nonlinear results.
12. View Results.
    - Create a **logarithmic strains** fringe result and **displacements, translational** deformation result quick plot for the final result case.
    - Create a **Cauchy stresses**, **NonLinear Output** fringe result and displacements, translational deformation result quick plot for the final result case.
13. Quit Patran.

Step 1. Open a New Database







Open a new database named **cylinder.db**:

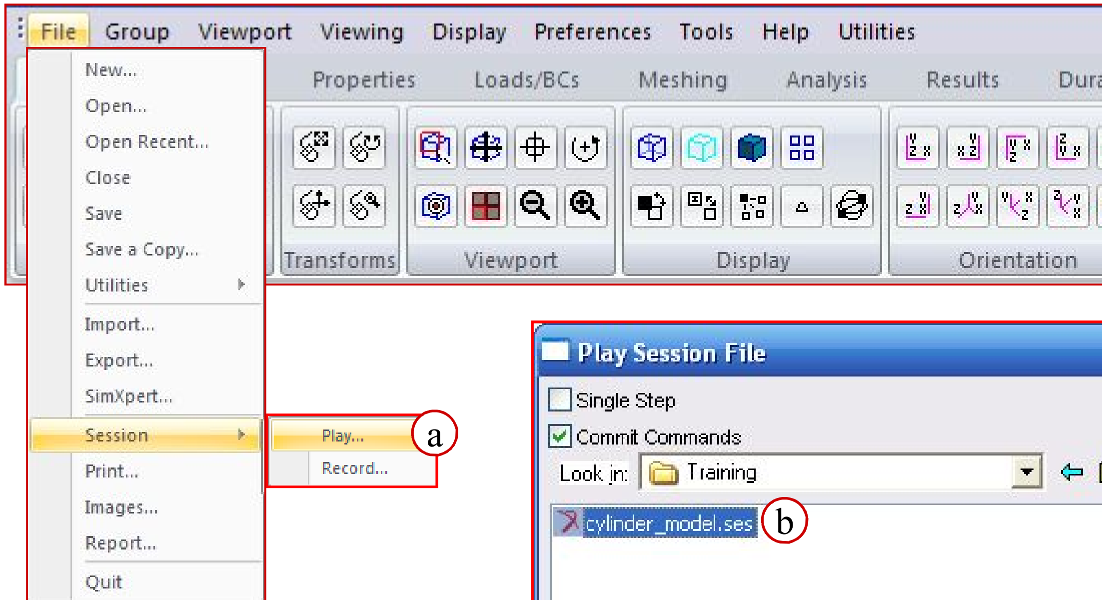
a. Pull down *File* > **New.**

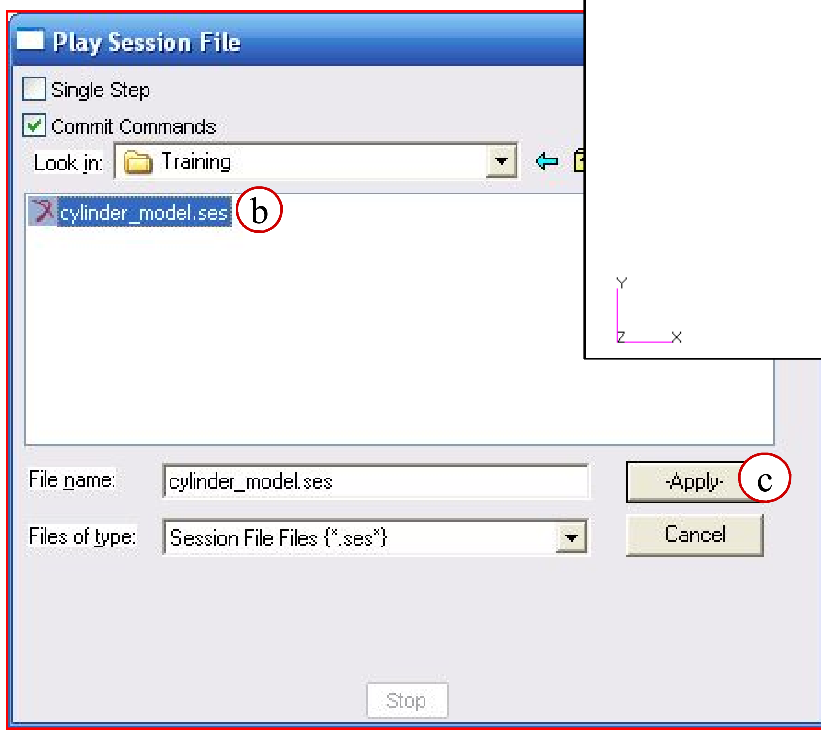
b. Type **cylinder** as *File name.*

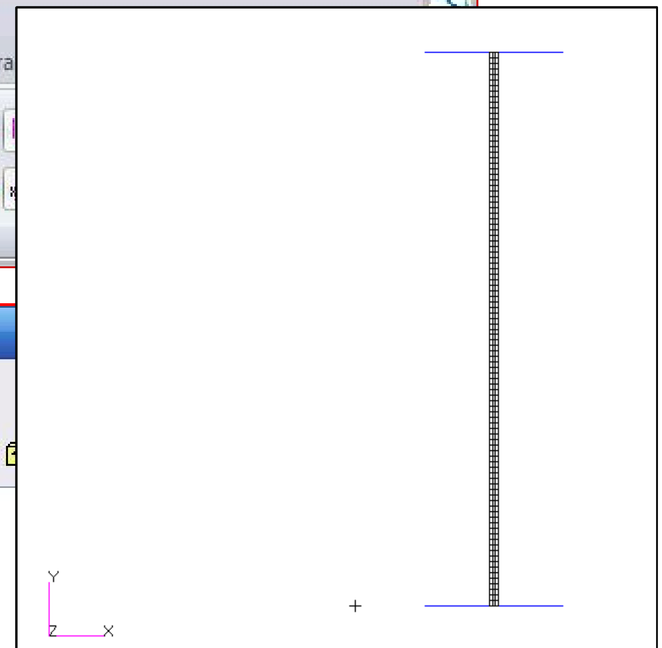
c. Click **OK.**

d. Click **OK** to select **MSC.Nastran** as the *Analysis Code*.

Step 2. Run the Provided Session File







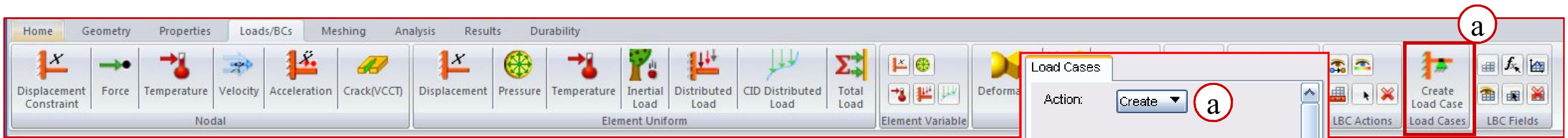
Run the provided session file.

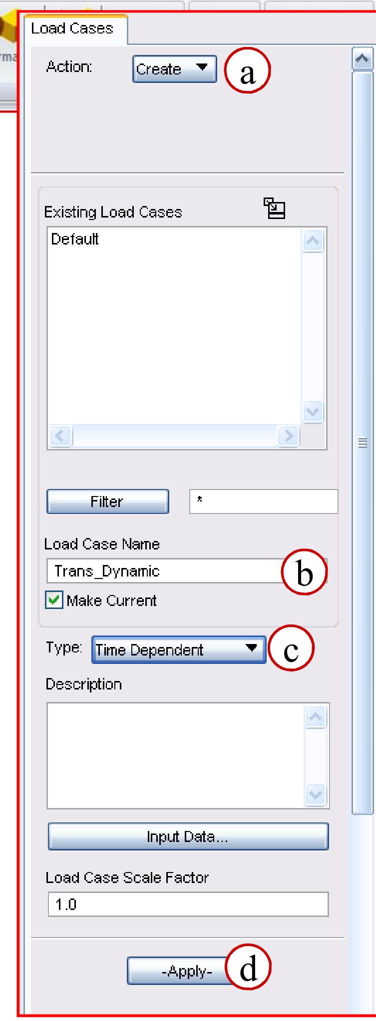
a. Pull down *File* > *Session* > **Play**.

b. Select **cylinder_model.ses** as the *File name*.

c. Click **Apply**.
(This action will run the session file. Please do not interrupt it.)

Step 3. Create a Time Dependent Load Case





Create a Time Dependent Load Case.

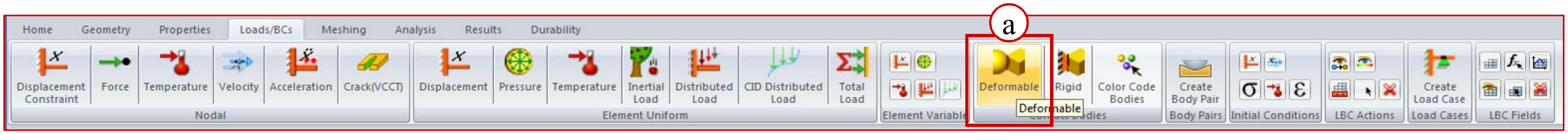
a. Under the *Loads/BCs* tab, click on **Create Load Case** in the *Load Cases* group.

b. Enter **Trans_Dynamic** for *the Load Case Name*.

c. Pull down **Time Dependent** for the load case *Type*.

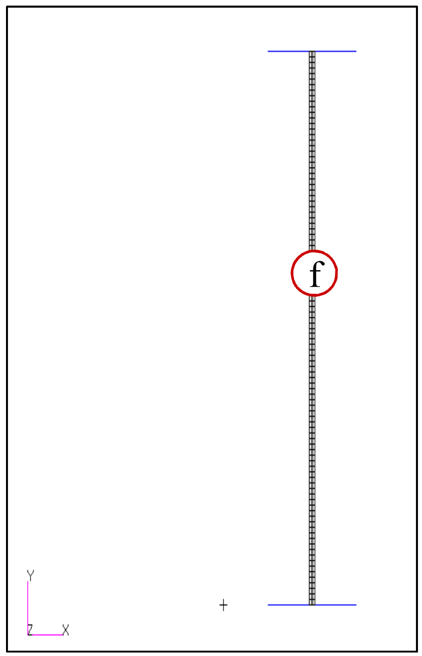
d. Click **Apply**.

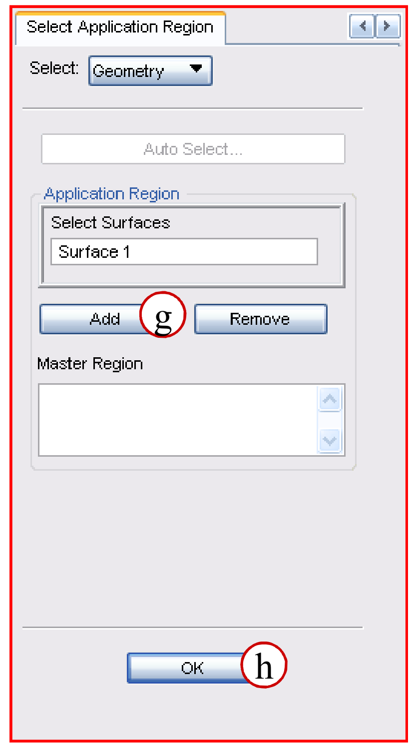
Step 4. Define the Deformable Contact Body

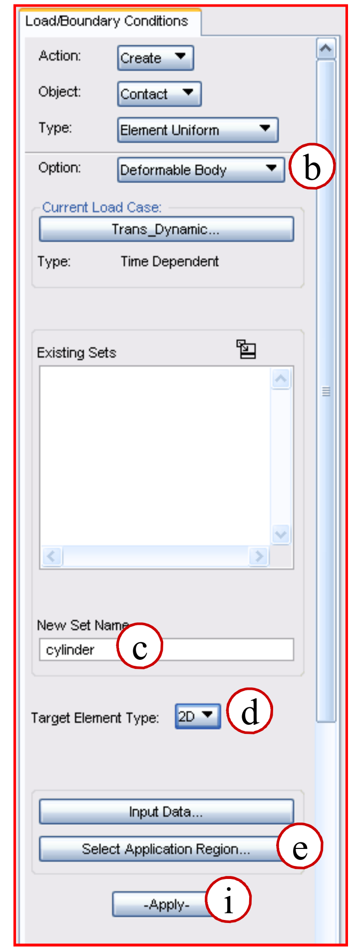


Create a deformable contact body for the cylinder.

a. Click **Deformable** in the *Contact Bodies* group.
b. Verify **Deformable Body** as the *Option*.
c. Enter **cylinder** as the *New Set Name*.
d. Select **2D** as the *Target Element Type*.
e. Click **Select Application Region**.
f. Select the cylinder surface.
g. Click **Add**.
h. Click **OK**.
i. Click **Apply**.



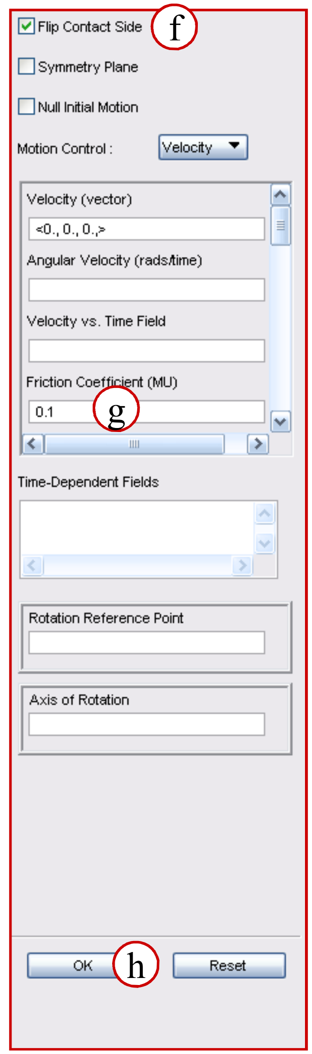


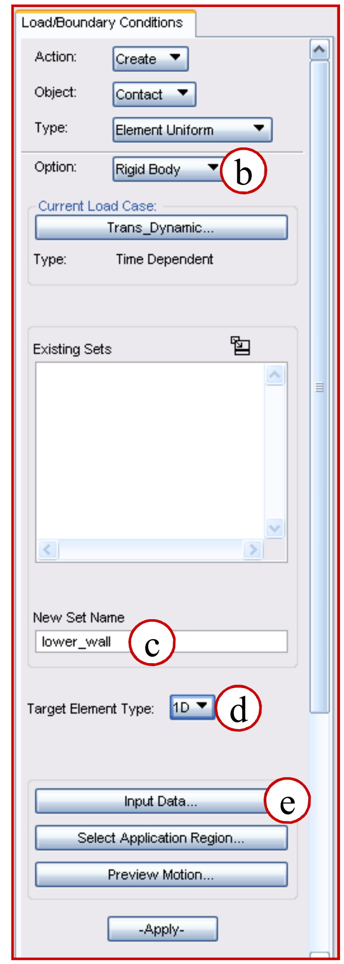


Step 5. Define the Rigid Contact Bodies

Create rigid contact bodies.

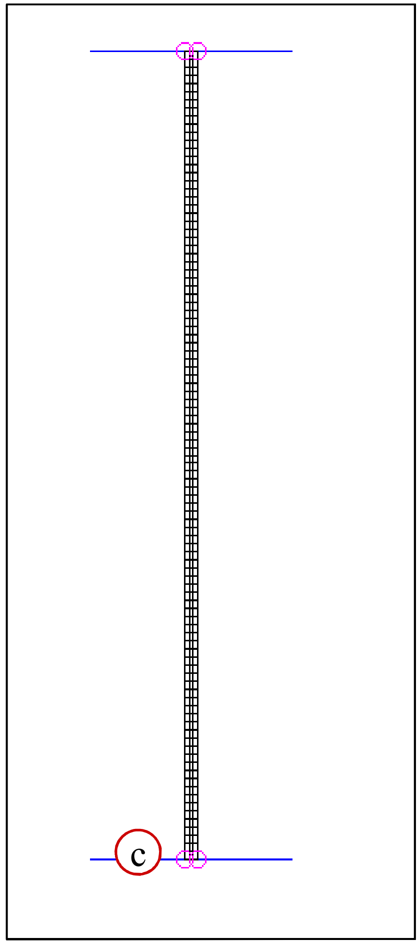
a. Click **Rigid** in the *Contact Bodies* group.
b. Verify **Rigid Body** as the Option.
c. Enter **lower_wall** as the *New Set Name*.
d. Select **1D** as the *Target Element Type*.
e. Click **Input Data**.
f. Check *Flip Contact Side* (Flipping the Contact Side is needed, because of the orientation of the curve representing this wall).
g. Enter **0.1** for the *Friction Coefficient (MU)*.
h. Click **OK**.

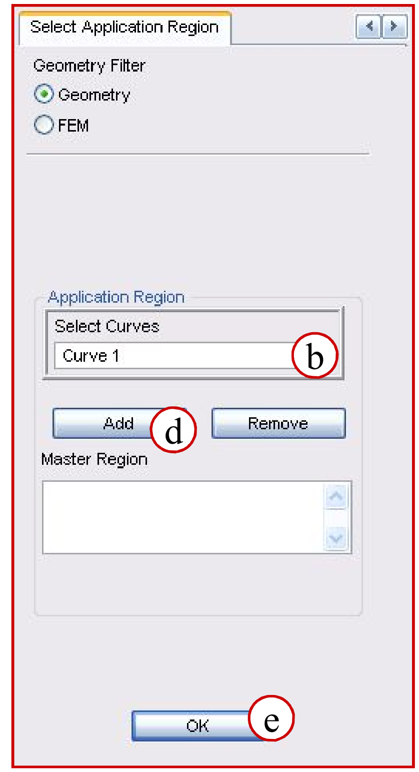


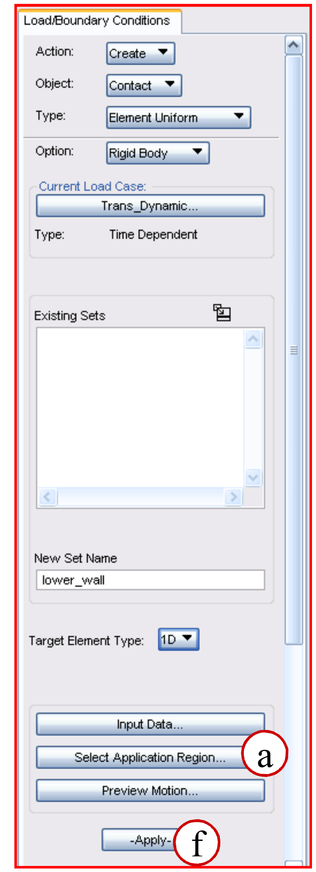


Step 5. Define the Rigid Contact Bodies (Cont.)

a. Click **Select Application Region**.
b. Click in the *Select Curves* box
c. Select the bottom curve defining the rigid surface (Curve 1).
d. Click **Add**.
e. Click **OK**.
f. Click **Apply**.

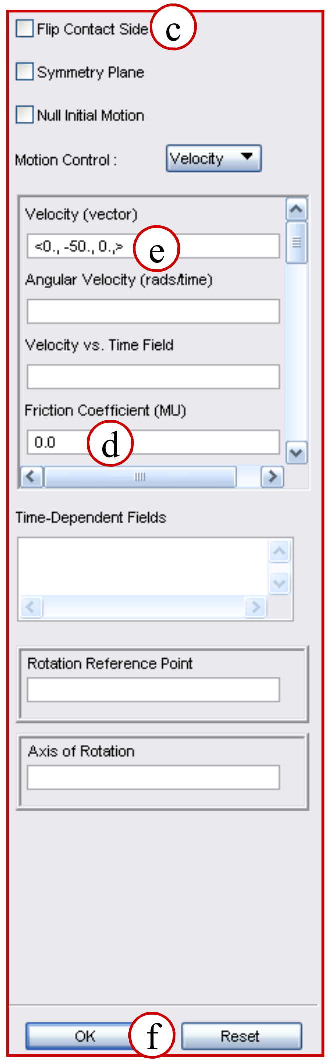


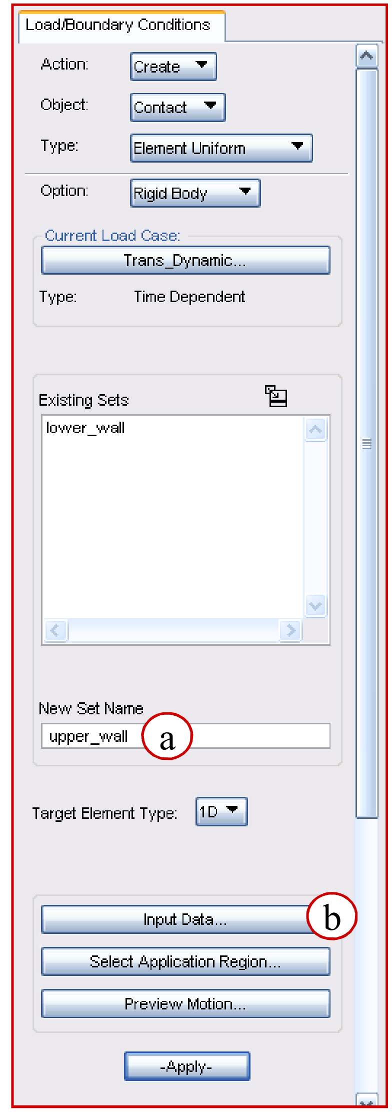




Step 5. Define the Rigid Contact Bodies (Cont.)

a. Enter **upper_wall** as the *New Set Name*.
b. Click **Input Data**.
c. Uncheck *Flip Contact Side*.
d. Enter **0.0** for the *Friction Coefficient (MU)*.
e. Enter **<0, -50, 0>** for the *Velocity (vector)*.
f. Click **OK**.

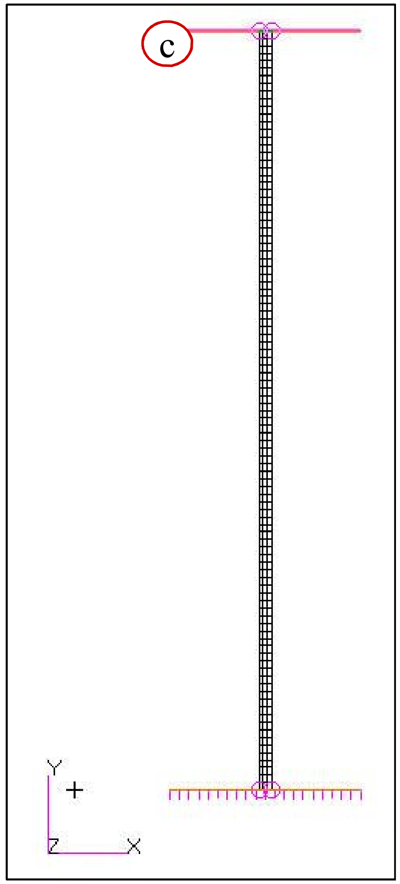


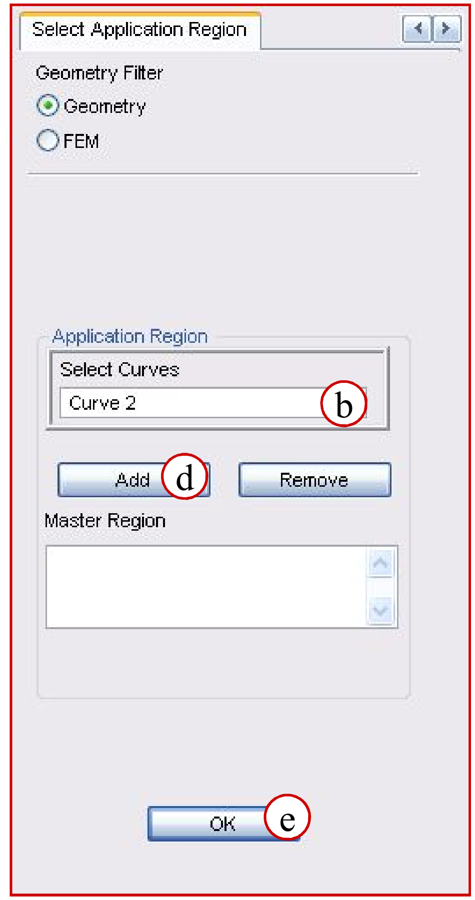


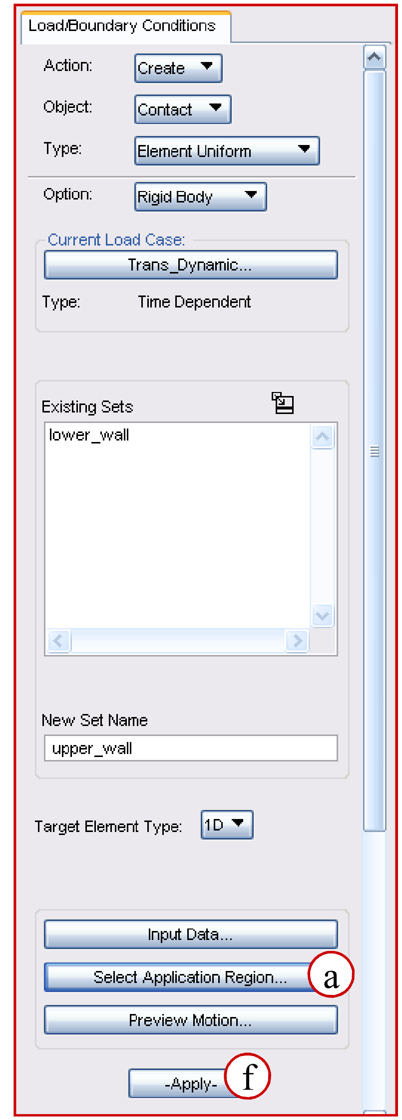
MSC Software

Step 5. Define the Rigid Contact Bodies (Cont.)

a. Click **Select Application Region**
b. Click in the *Select Curves* box.
c. Select the top curve defining the rigid surface (Curve 2).
d. Click **Add**.
e. Click **OK**.
f. Click **Apply**.





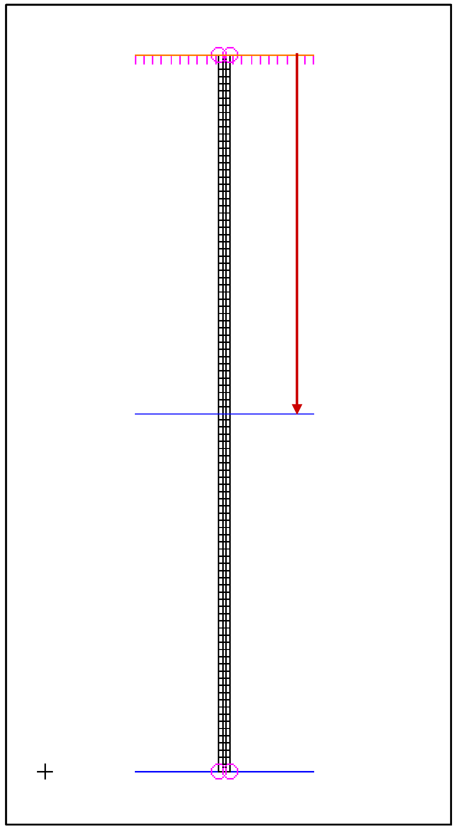


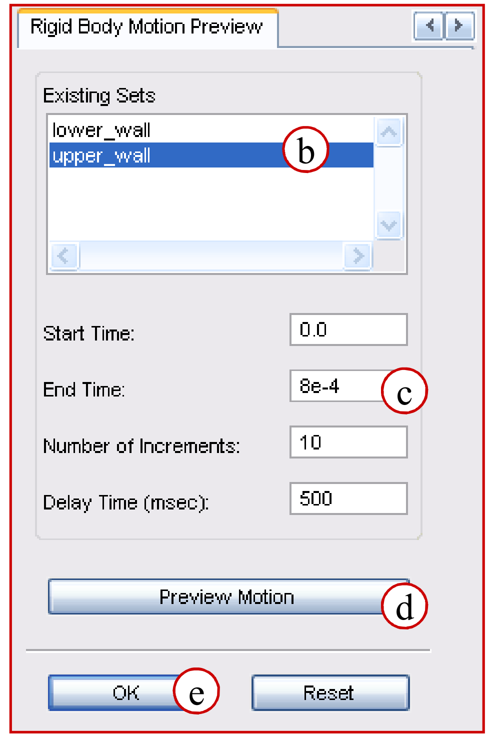
MSC Software

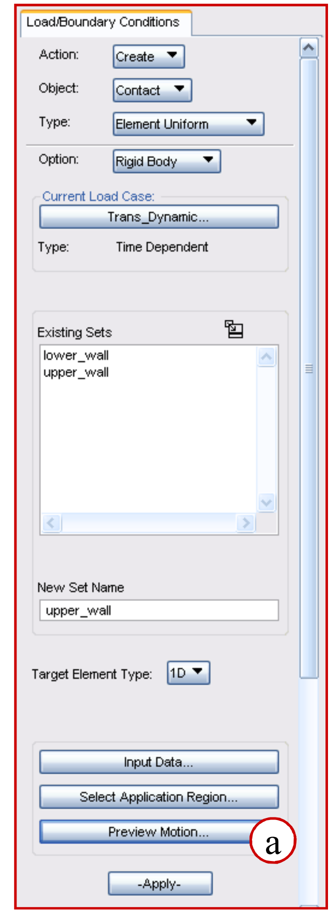
Step 6. Preview the Rigid Body Motion

Preview the Rigid Body Motion.

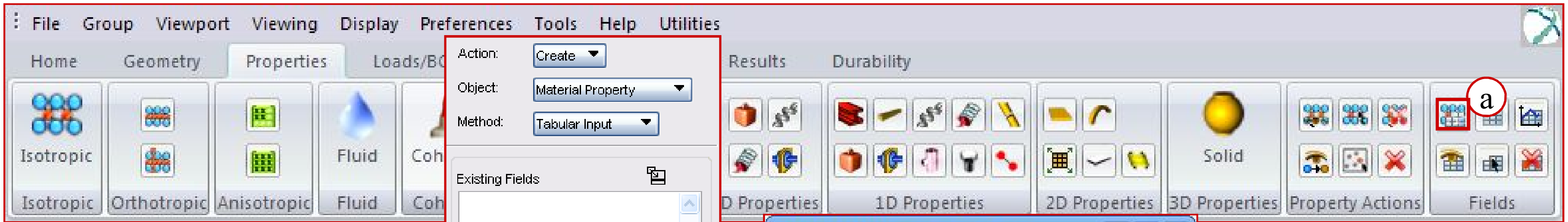
a. Click **Preview Motion**.
b. Select **upper_wall**.
c. Enter **8e-4** for *End Time*.
d. Click **Preview Motion**.
e. Click **OK** after reviewing the motion.

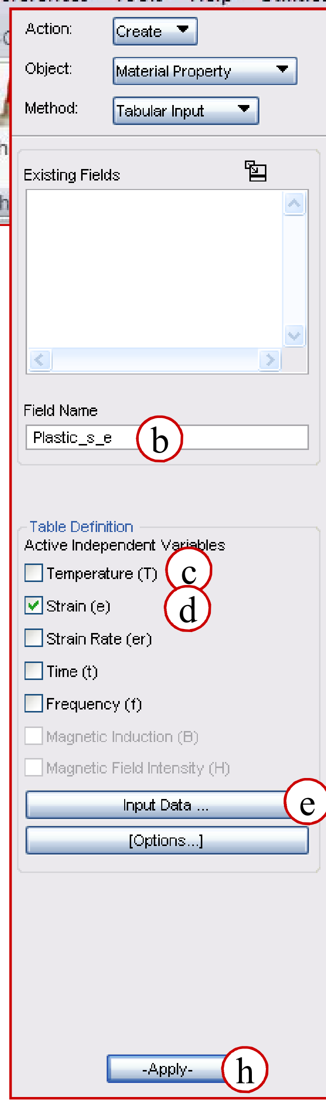


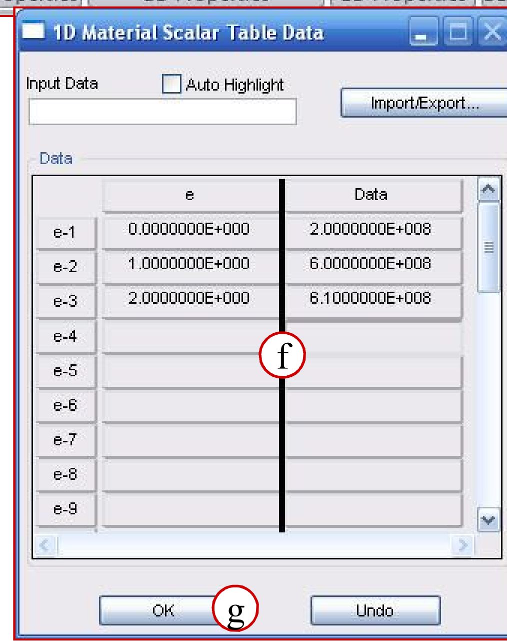




Step 7. Create Material Property with Tabular Input







Create a field for the elastic-plastic material data.

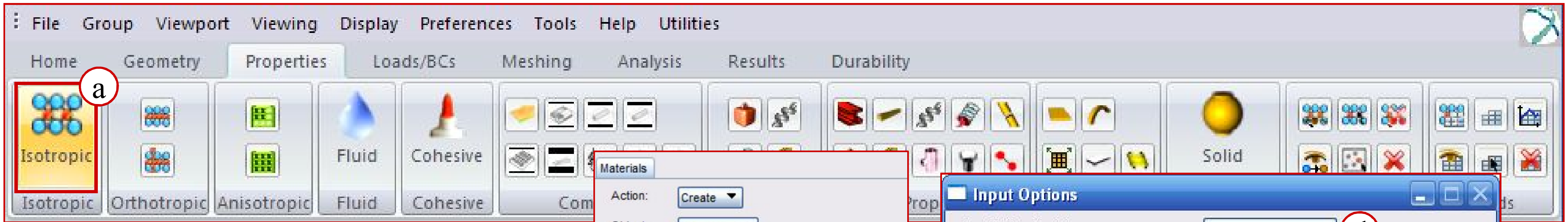
a. Under the *Properties* tab, click on **Material** in the *Fields* group.
b. Enter **Plastic_s_e** as the *Field Name*.
c. Uncheck *Temperature (T)* under *Active Independent Variables*.
d. Check *Strain (e)*.
e. Click **Input Data**.
f. Enter the data as shown.
g. Click **OK**.
h. Click **Apply**.

Strain(e)	Stress
0.0	2.0E8
1.0	6.0E8
2.0	6.1E8

The *1D Material Scalar Table Data* form needs to be filled out as shown in the table. To fill in the table, click on the cell you wish to edit, enter the value in the *Input Data* databox and press <Enter>. The table will automatically tab down.

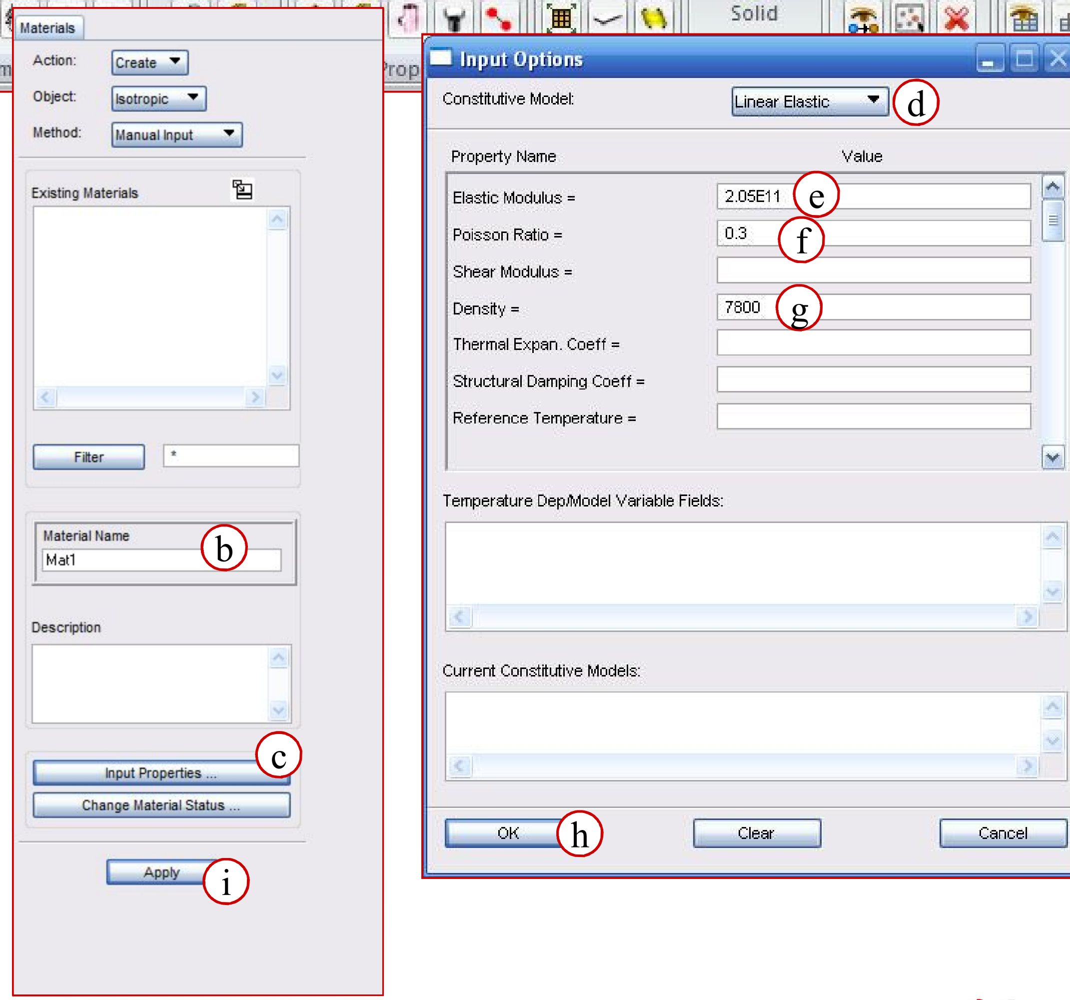
MSC Software

Step 8. Create the Material with Elastic and Plastic Properties



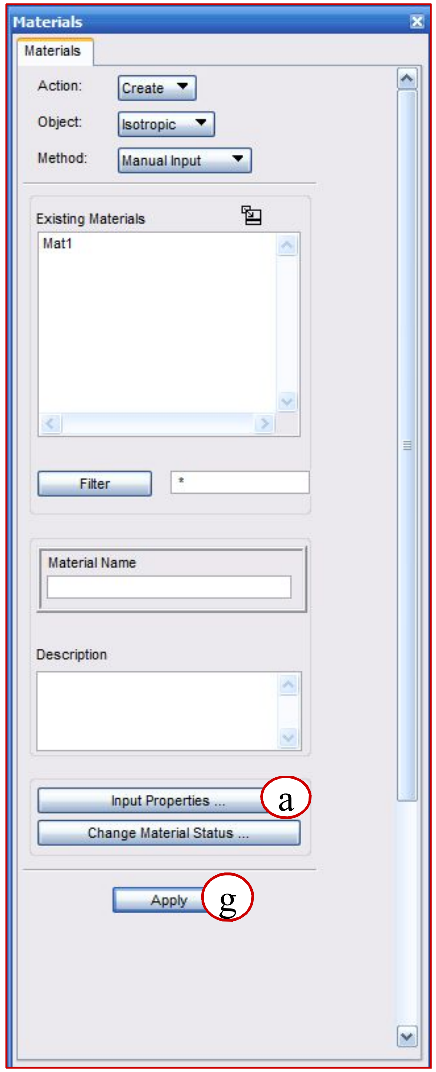
Create a material, with elastic and plastic properties.

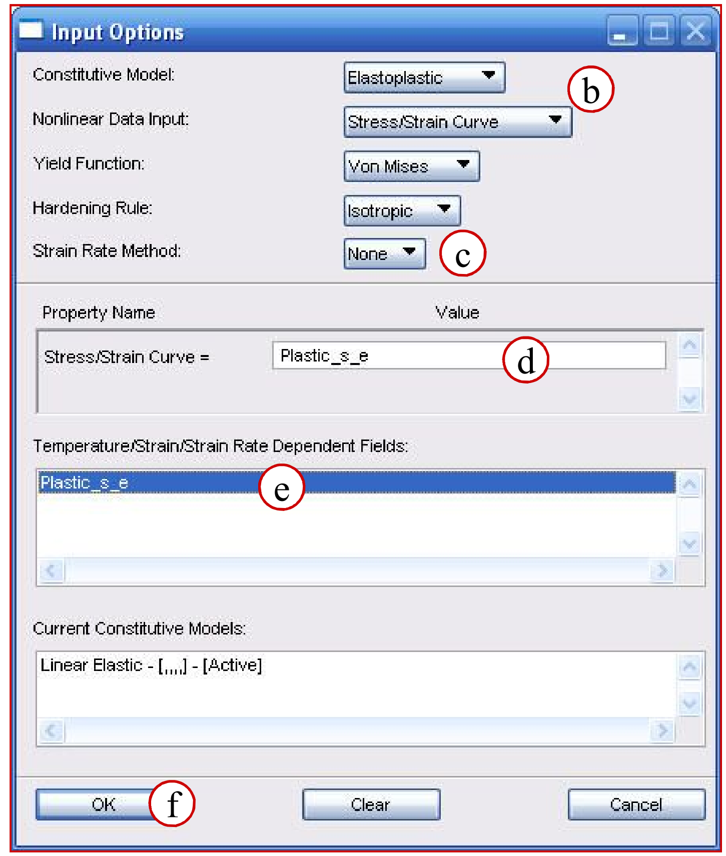
a. Click on **Isotropic** in the *Isotropic* group.
b. Enter **Mat1** as the *Material Name*.
c. Click **Input Properties**.
d. Select **Linear Elastic** as the *Constitutive Model*.
e. Enter **2.05E11** as the *Elastic Modulus*.
f. Enter **0.3** as the *Poisson Ratio*.
g. Enter **7800** as the *Density*.
h. Click **OK**.
i. Click **Apply**.



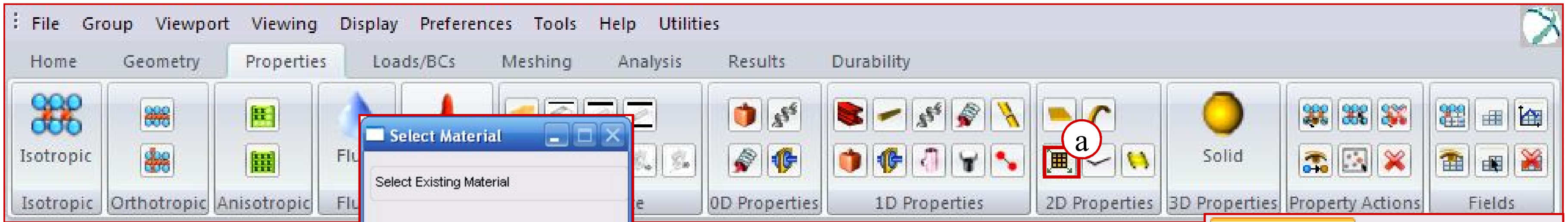
Step 8. Create the Material with Elastic and Plastic Properties (Cont.)

a. Click **Input Properities.**
b. Pull down **Elastoplastic** for *Constitutive Model* and **Stress/Strain Curve** for *Nonlinear Data Input.*
c. Verify that *Strain Rate Method* is set to **None**.
d. Click in the **Stress / Strain Curve** panel.
e. Select **Plastic_s_e**.
f. Click **OK**.
g. Click **Apply**.



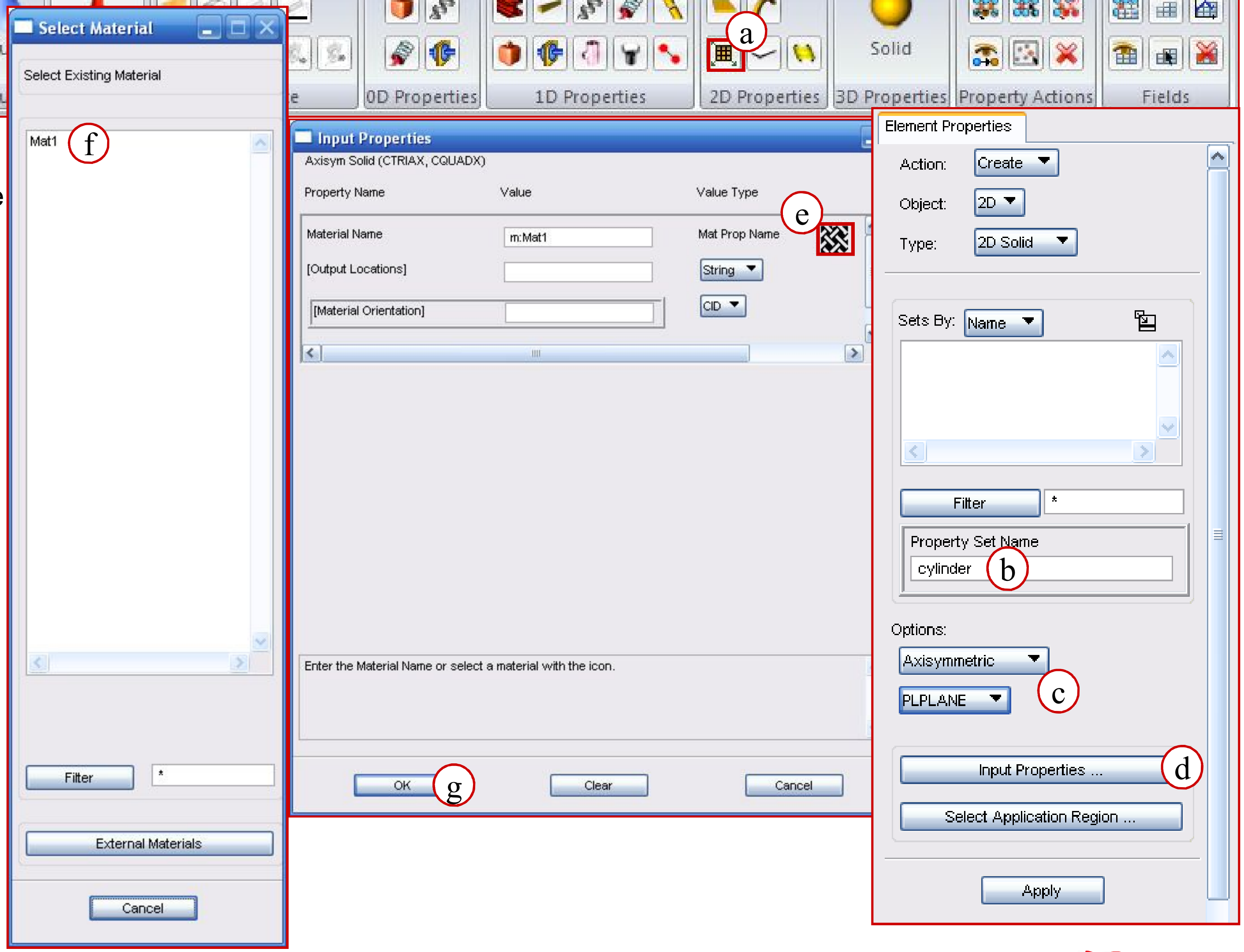


Step 9. Apply Properties



Apply properties:

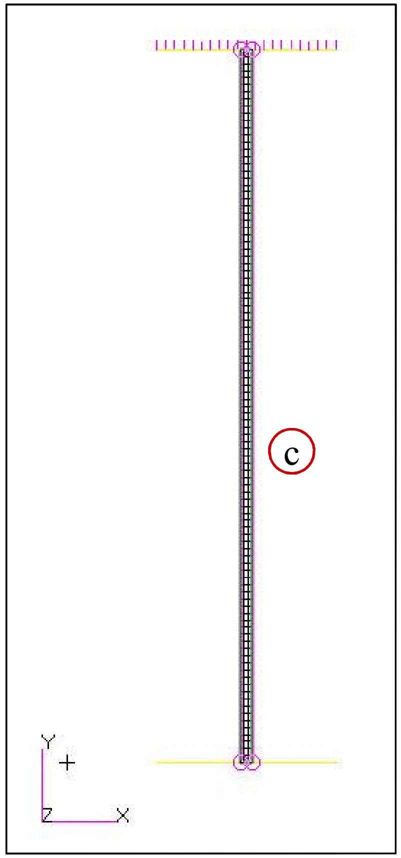
a. Click on **2D Solid** in the *2D Properties* group.
b. Enter **cylinder** for *Property Set Name*.
c. Modify *Options* to **Axisymmetric** and **PLPLANE**.
d. Click **Input Properties.**
e. Click the **Mat Prop Name** Icon.
f. Select **Mat1**.
g. Click **OK**.

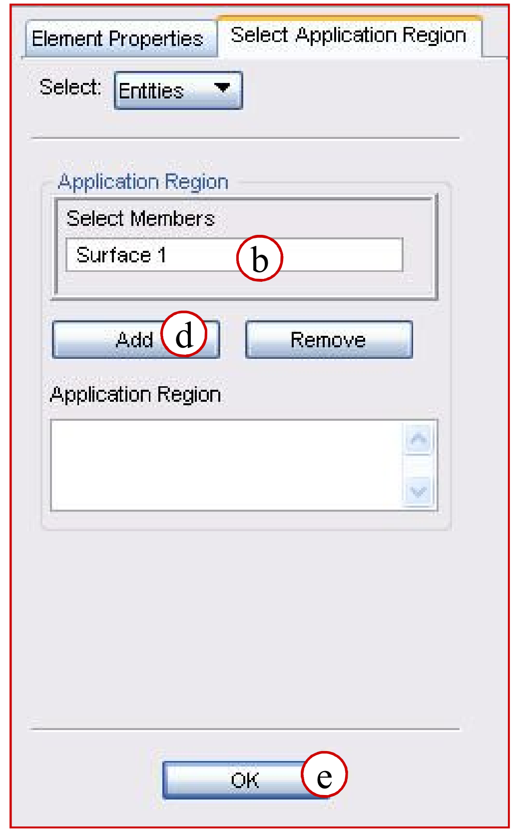


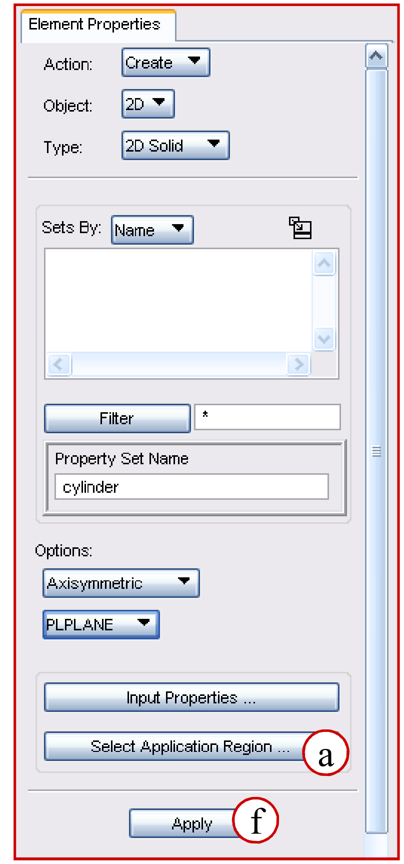
MSC Software

Step 9. Apply Properties (Cont.)

a. Click **Select Application Region**.
b. Click in the **Select Members** box.
c. Select the cylinder (Surface 1)
d. Click **Add**.
e. Click **OK**.
f. Click **Apply**.

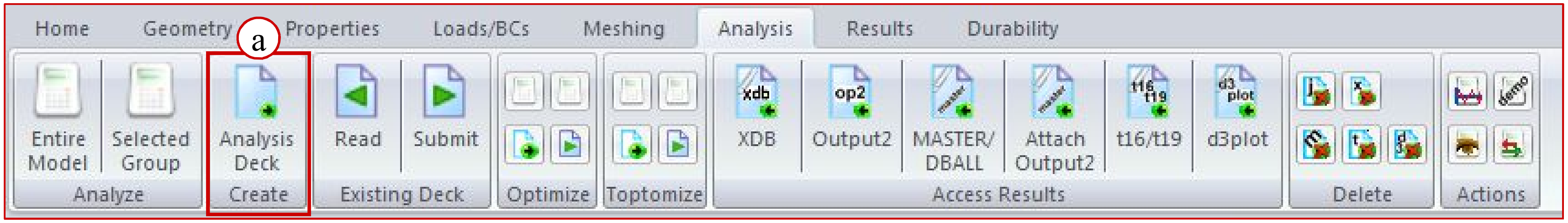






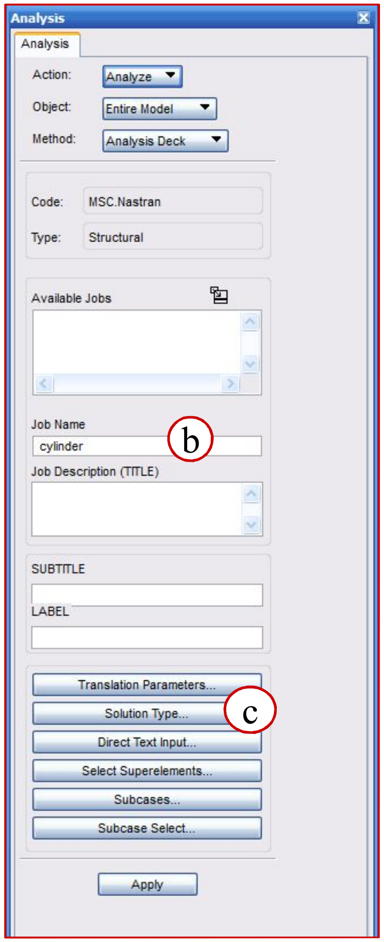
MSC Software

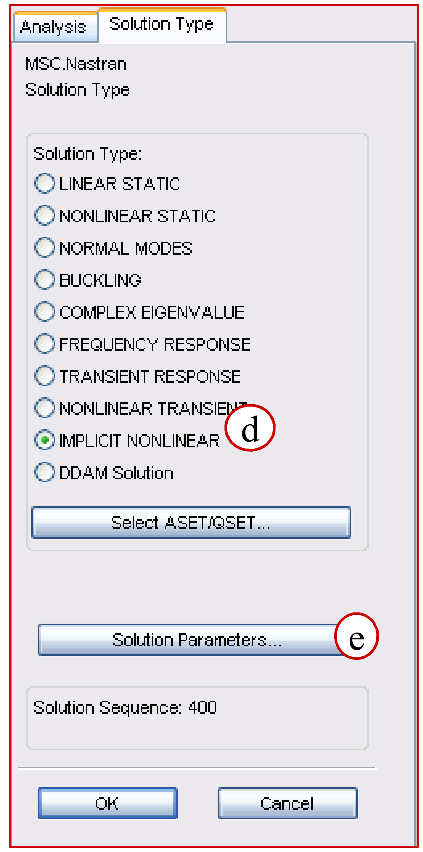
Step 10. Set Up the Analysis

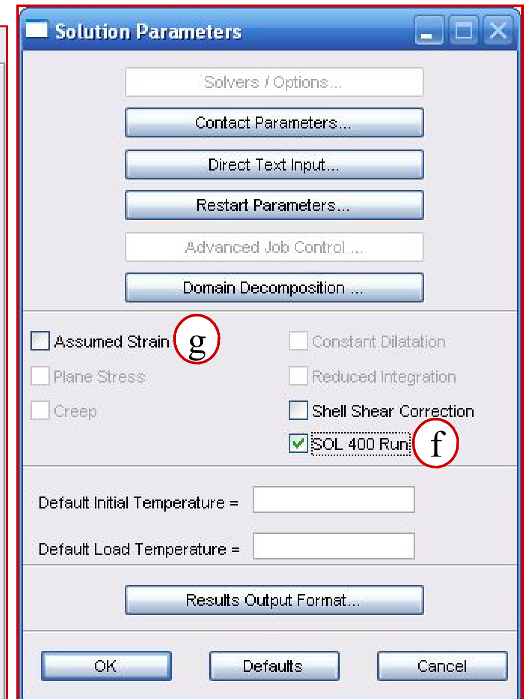


Set up the nonlinear Analysis job.

a. Under the *Analysis* tab, click on **Analysis Deck** in the *Create* Group.
b. Enter **cylinder** as the *Job Name.*
c. Click **Solution Type**.
d. Select **Implicit Nonlinear** as *Solution Type.*
e. Click **Solution Parameters.**
f. Verify that *SOL400 Run* is checked.
g. Uncheck *Assumed Strain.*

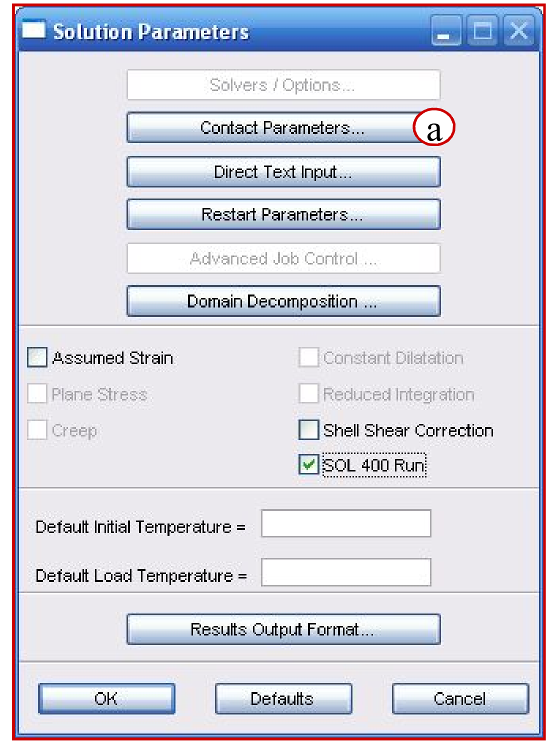


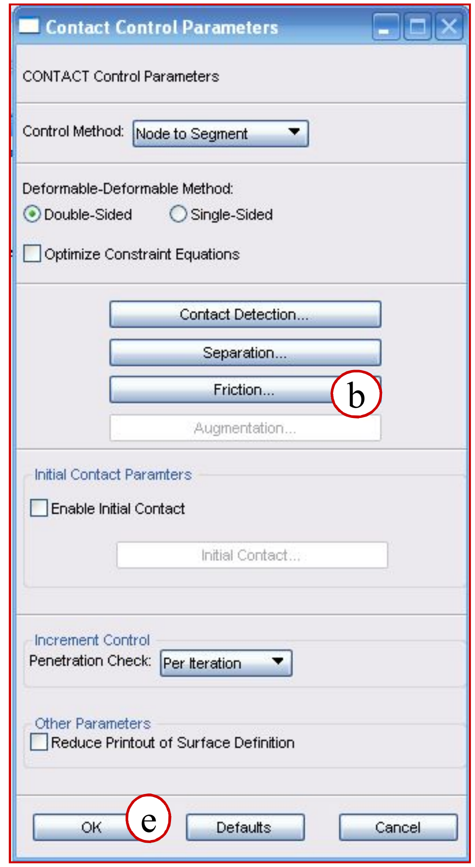


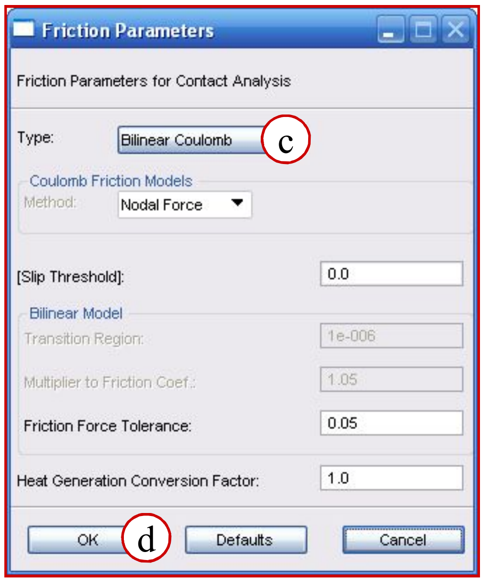


Step 10. Set Up the Analysis (Cont.)

a. Click **Contact Parameters**.
b. Click **Friction…**
c. Pull down **Bilinear Coulomb** for *Type*.
d. Click **OK**.
e. Click **OK**.

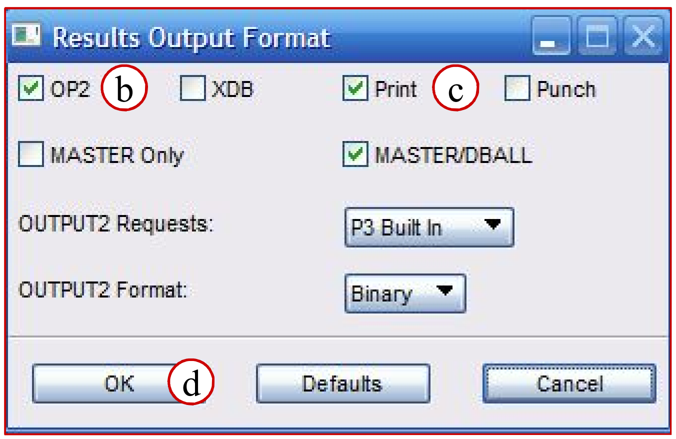


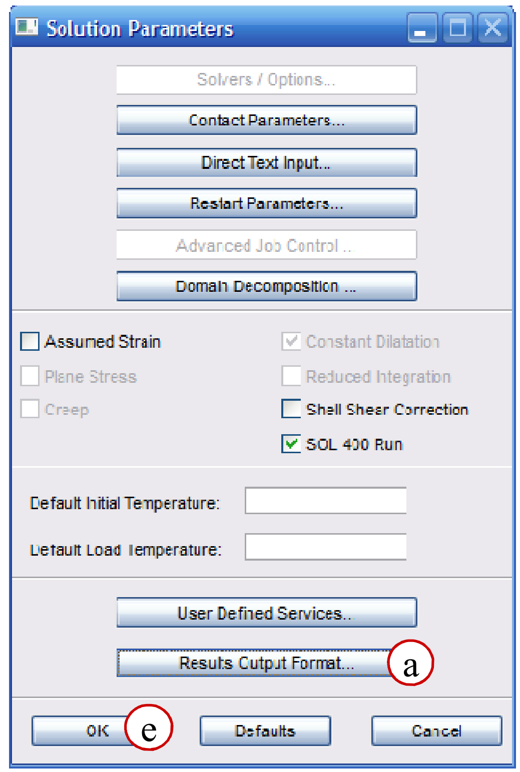


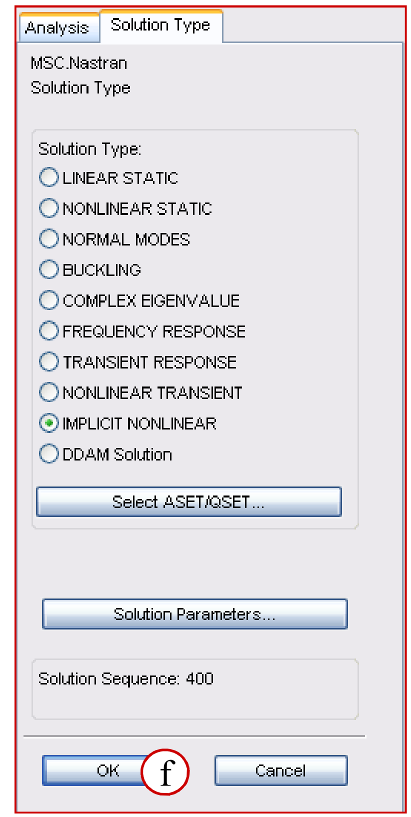


Step 10. Set Up the Analysis (Cont.)

a. Click **Results Output Format**.
b. Check *OP2.*
c. Check *Print.*
d. Click **OK**.
e. Click **OK**.
f. Click **OK**.

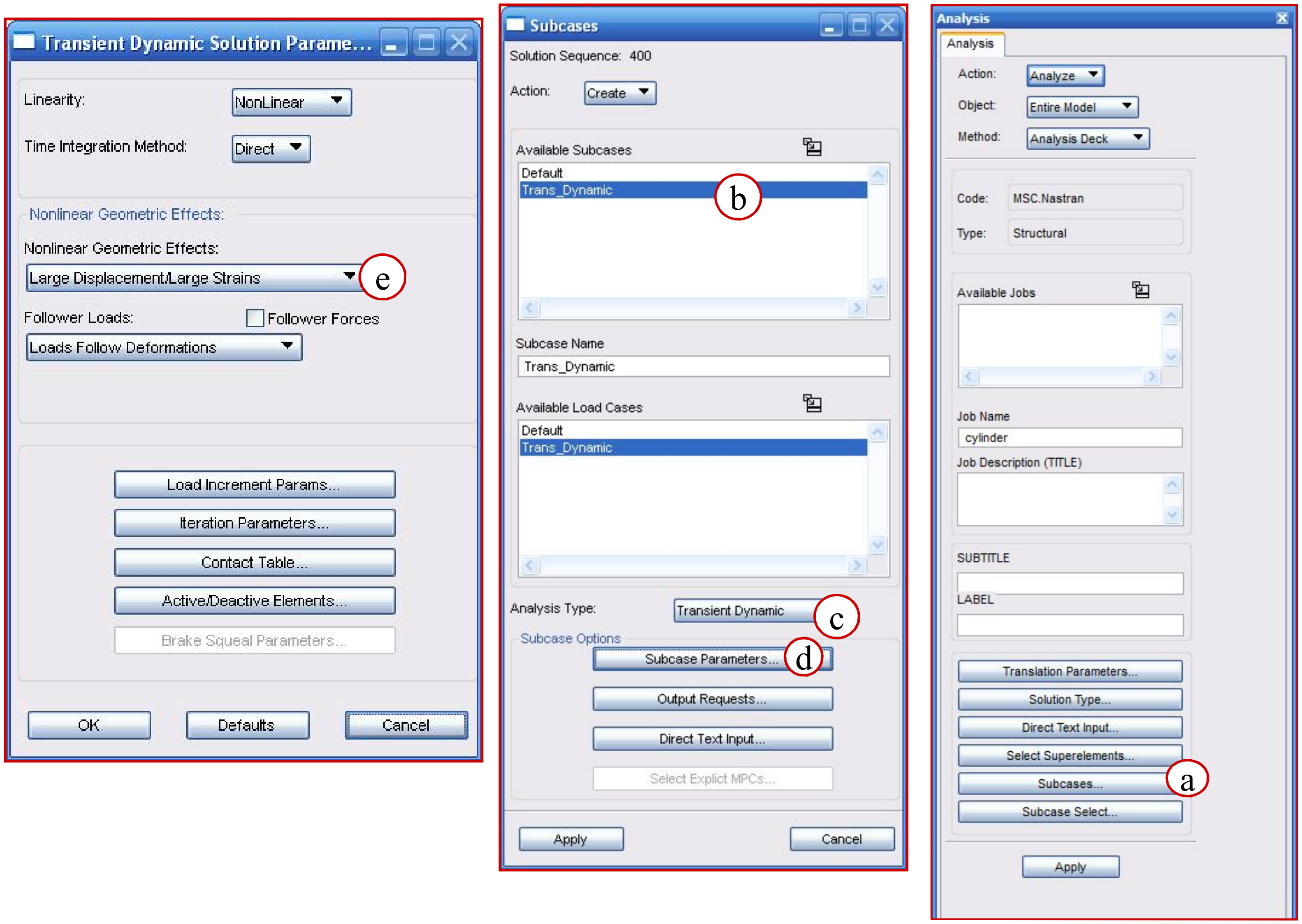






Step 10. Set Up the Analysis (Cont.)

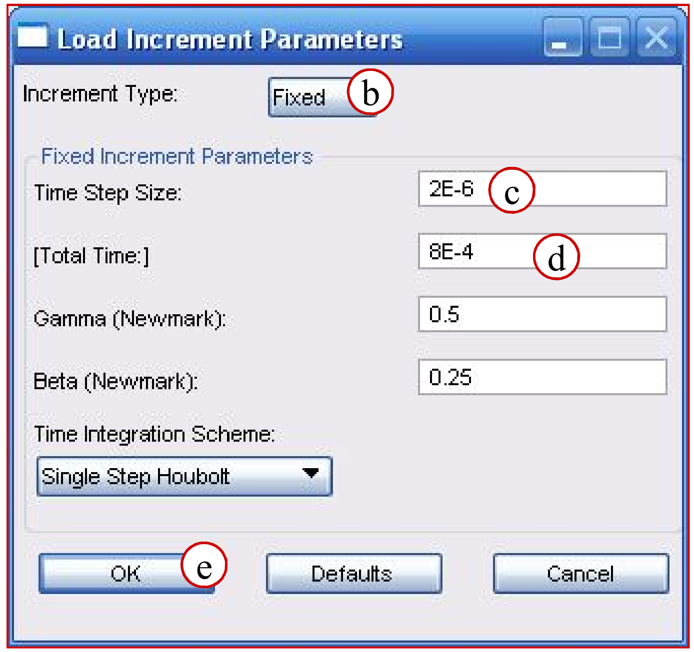
a. Click **Subcases**.
b. Click on **Trans_Dynamic** under *Available Subcases*.
c. Pull down **Transient Dynamic** for *Analysis Type*.
d. Click on **Subcase Parameters**.
e. Verify **Large Displacement/Large Strains** is selected for *Nonlinear Geometric Effects*.

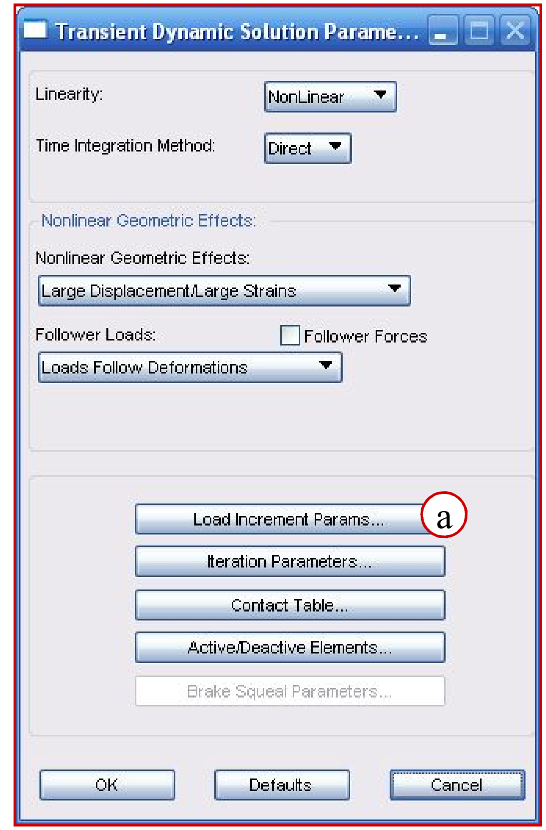


MSC Software

Step 10. Set Up the Analysis (Cont.)

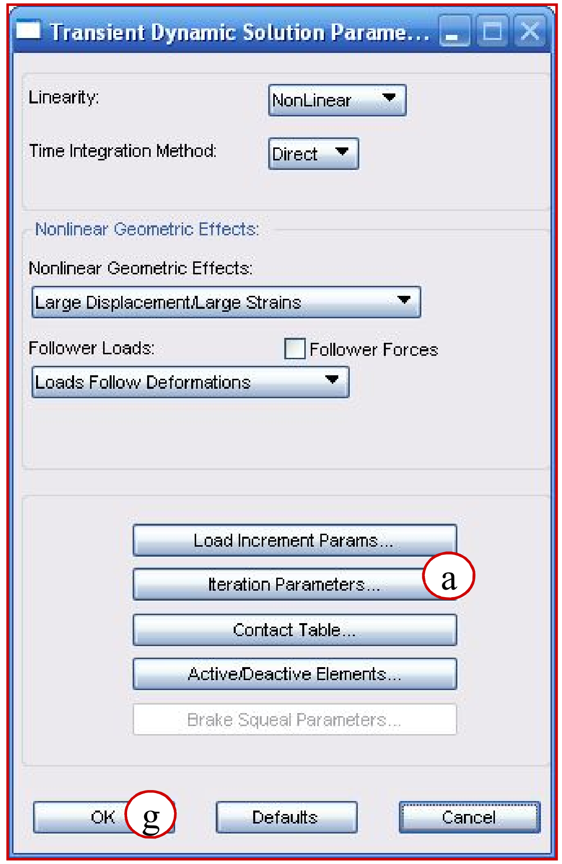
a. Click **Load Increment Parameters**.
b. Verify that *Increment Type* is set to **Fixed**.
c. Enter **2E-6** for *Time Step Size*.
d. Enter **8E-4** for *Total Time*.
e. Click **OK**.

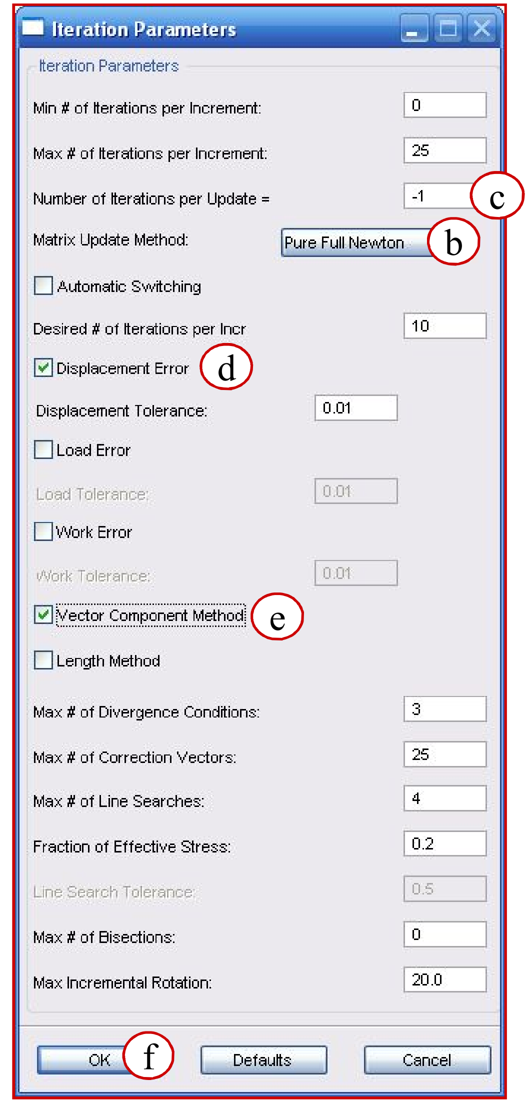




Step 10. Set Up the Analysis (Cont.)

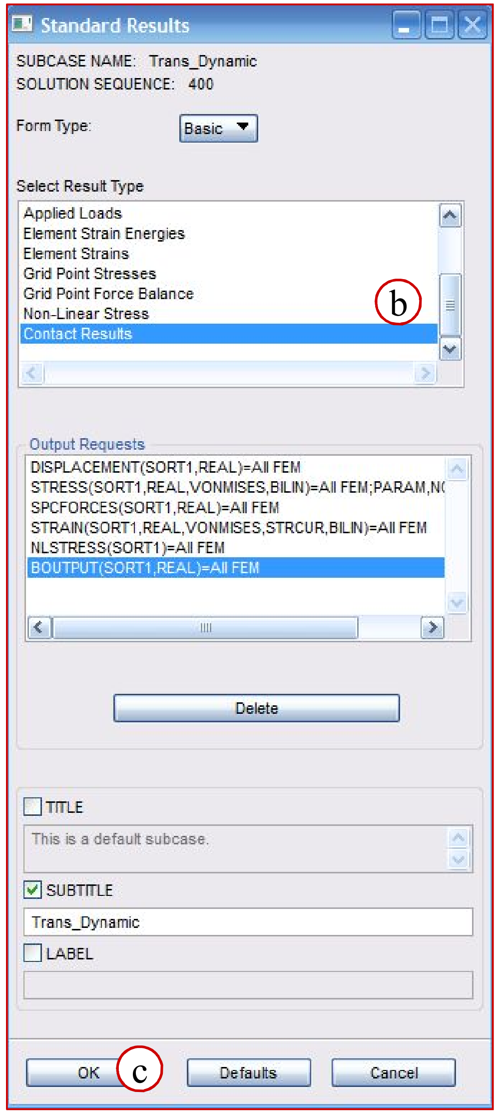
a. Click **Iteration Parameters**
b. Pull down **Pure Full Newton** for *Matrix Update Method*.
c. Enter **-1** for *Number of Iterations per Update*.
d. Check *Displacement Error*.
e. Check *Vector Component Method.*
f. Click **OK**.
g. Click **OK**.

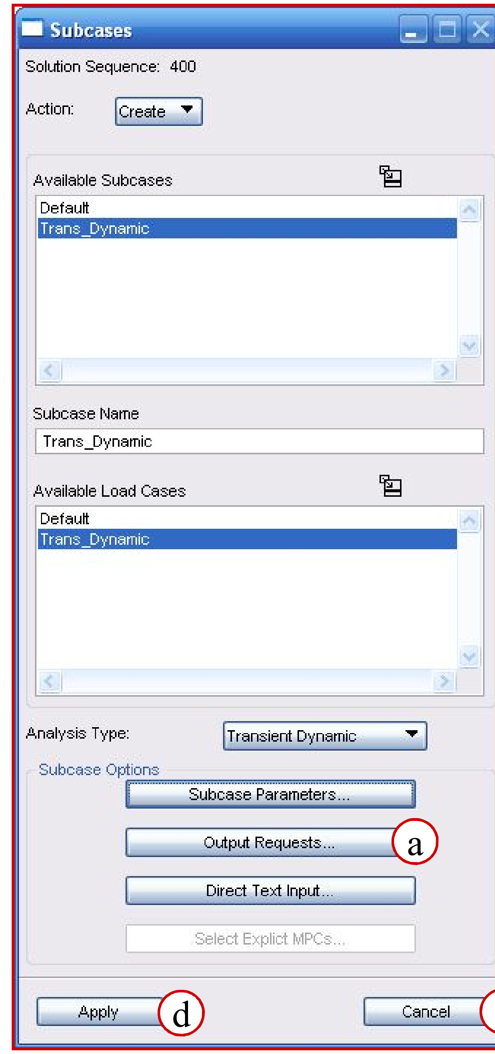




Step 10. Set Up the Analysis (Cont.)

a. Click **Output Requests**.
b. Select **Element Strains, Non-Linear Stress** and **Contact Results** under *Select Result Type.*
c. Click **OK**.
d. Click **Apply**.
e. Click **Cancel**.





Step 10. Set Up the Analysis (Cont.)

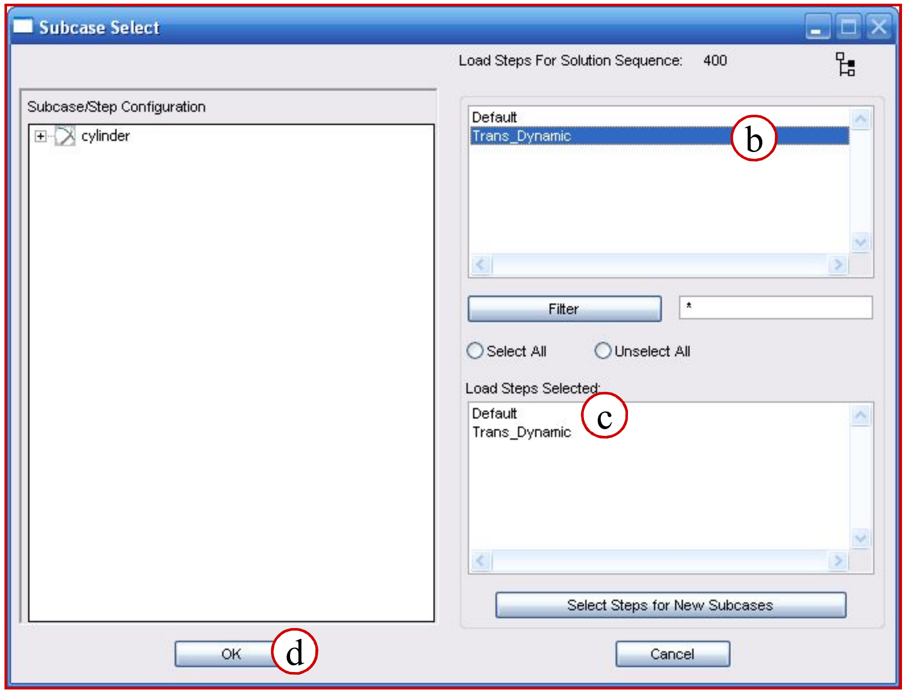
a. Click **Subcase Select**.

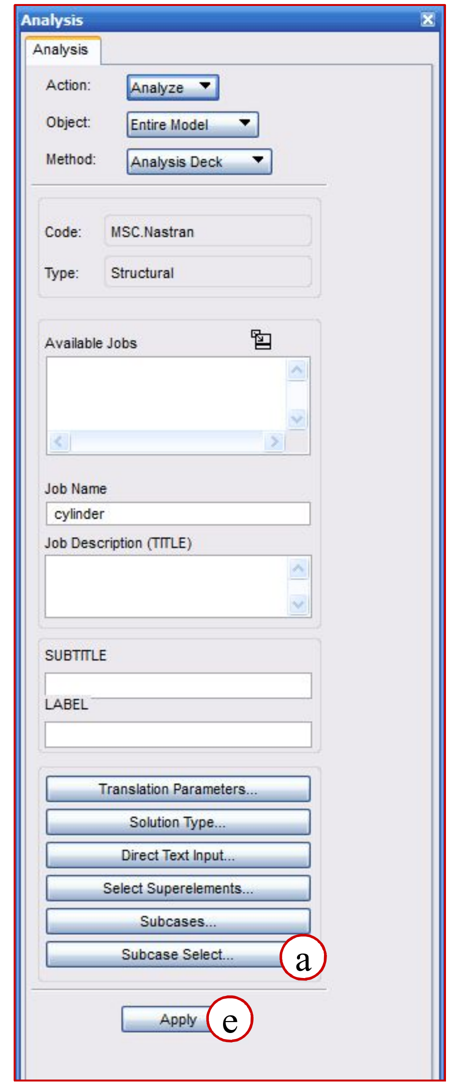
b. Select **Trans_Dynamic** from the Available Subcases.

c. Unselect **Default** by clicking on it under *Load Steps Selected*.

d. Click **OK**.

e. Click **Apply**.





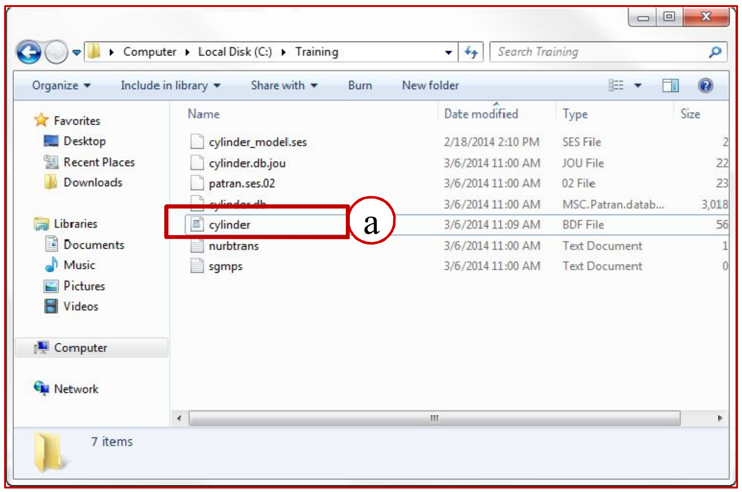
Step 10. Set Up the Analysis (Cont.)

Modify the **cylinder.bdf** file.

a. Open the **cylinder.bdf** file in a text editor. It will be located in the same folder you saved the Patran database file.

b. Locate the **“BCPARA 0 NLGLUE 0 FTYPE 6”** line.

c. Change the line to **“BCPARA, 0, NLGLUE, 1, NBODIES, 3, FTYPE, 6”**. Save the file as **cylinder.bdf**. This will set the friction type to **Bilinear Coulomb**.



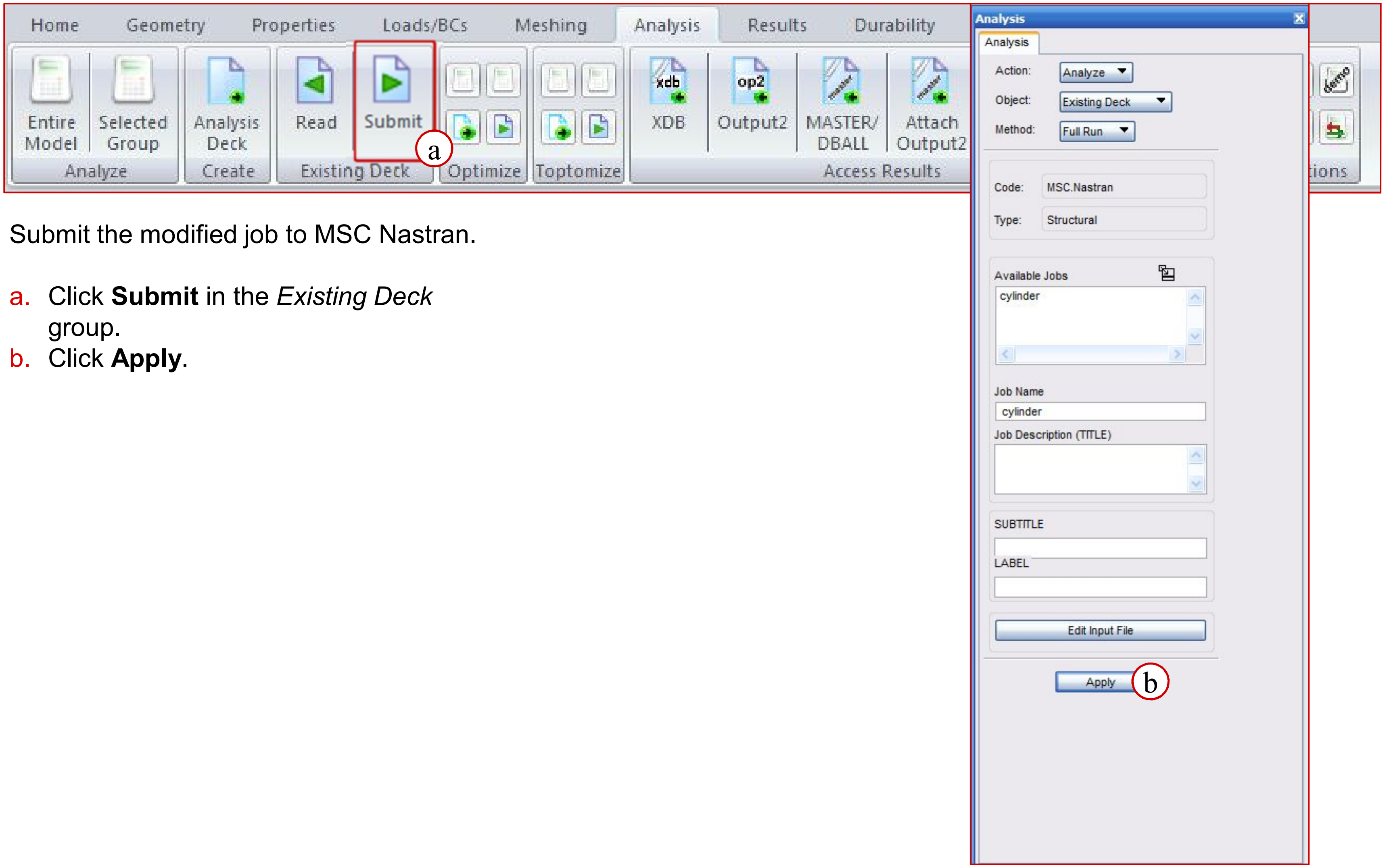

```
BEGIN BULK
$ Direct Text Input for Bulk Data
PARAM    POST     1
PARAM   PRTMAXIM YES
BCPARA    0       NLGLUE   0        FTYPE    6
PARAM    LGDISP   1
TSTEPNL  1        400     2.-6             PFNT    -1              UV
        .01
```

b

```
BEGIN BULK
$ Direct Text Input for Bulk Data
PARAM    POST     1
PARAM   PRTMAXIM YES
BCPARA,0,NLGLUE,1,NBODIES,3,FTYPE,6
PARAM    LGDISP   1
TSTEPNL  1        400     2.-6             PFNT    -1              UV
        .01
```

c

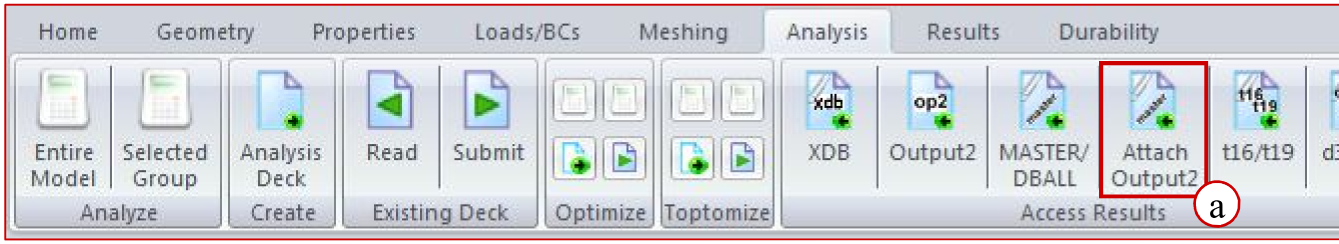
Step 10. Set Up the Analysis (Cont.)

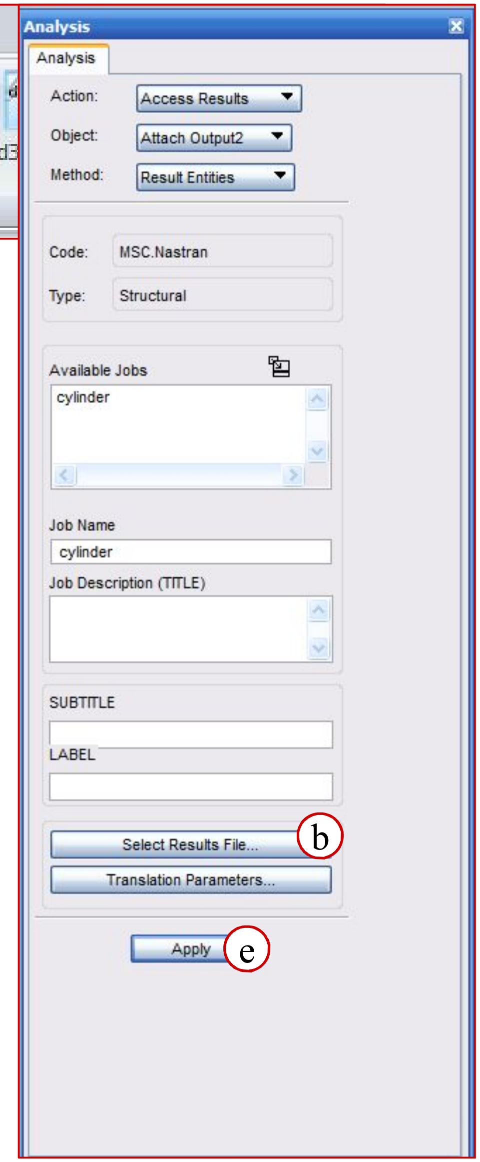


Submit the modified job to MSC Nastran.

a. Click **Submit** in the *Existing Deck* group.
b. Click **Apply**.

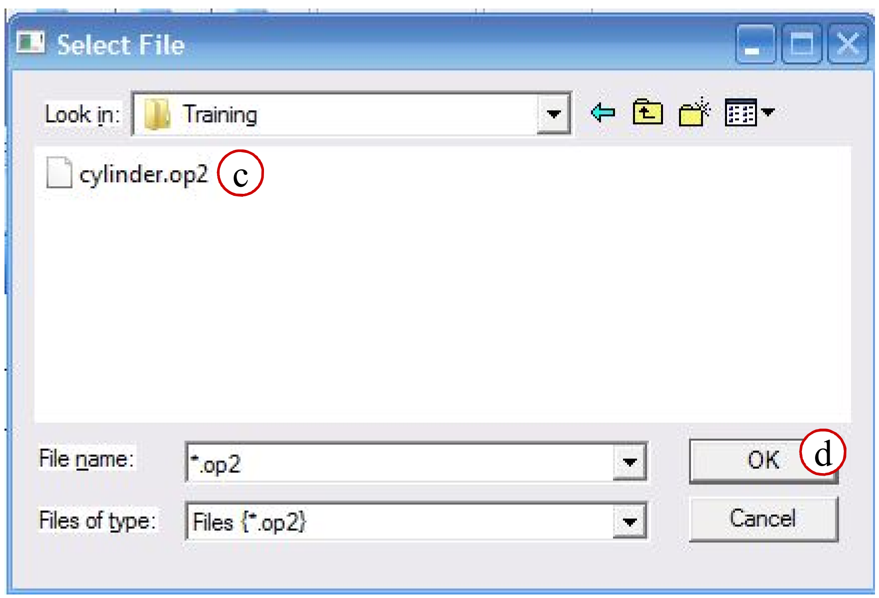
Step 11. Access the Nonlinear Results



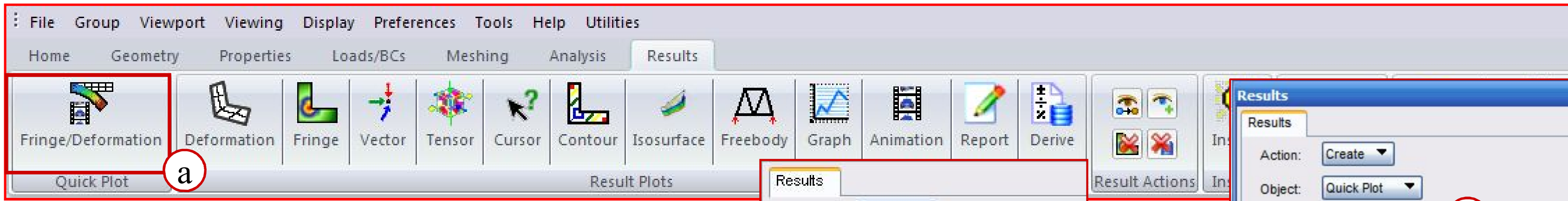


Read (Attach) results.

a. Click on **Attach Output2** in the *Access Results* group.
b. Click on **Select Results File**.
c. Select **cylinder.op2**.
d. Click **OK**.
e. Click **Apply**.

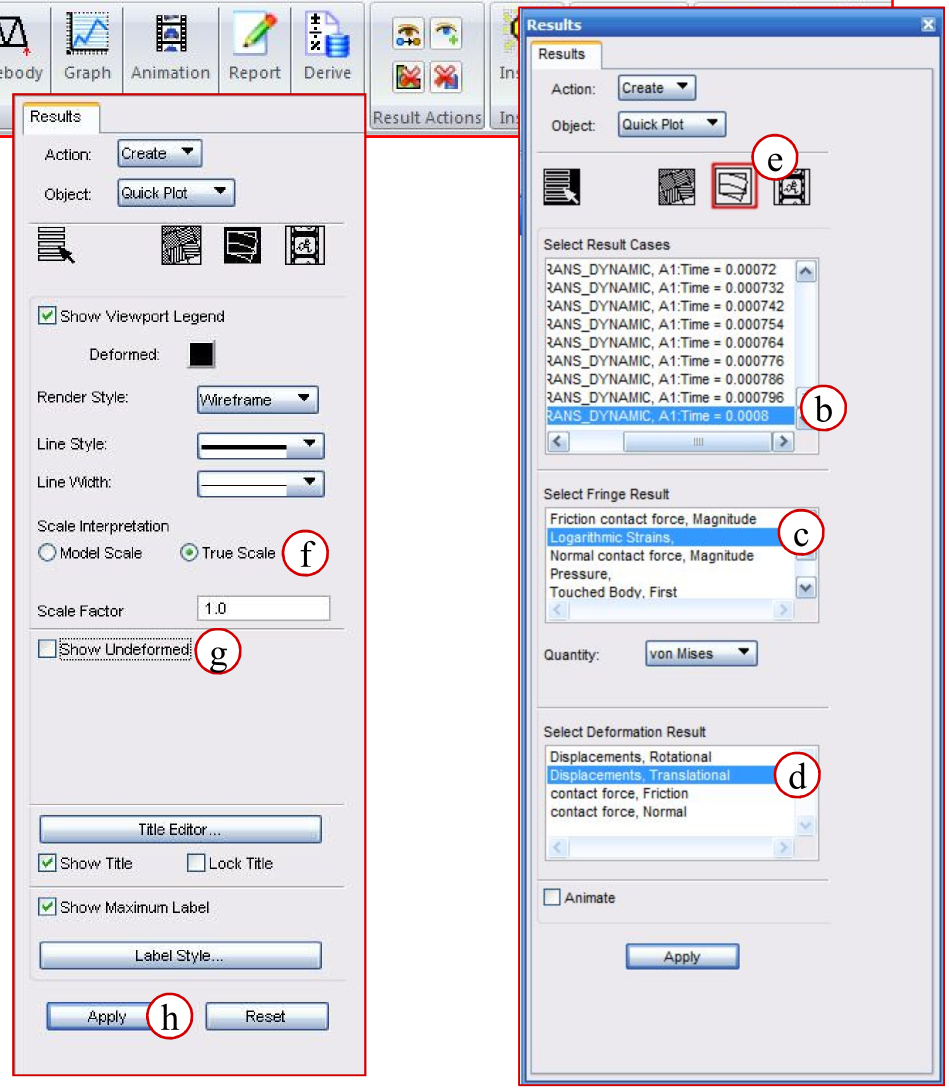


Step 12. View Results



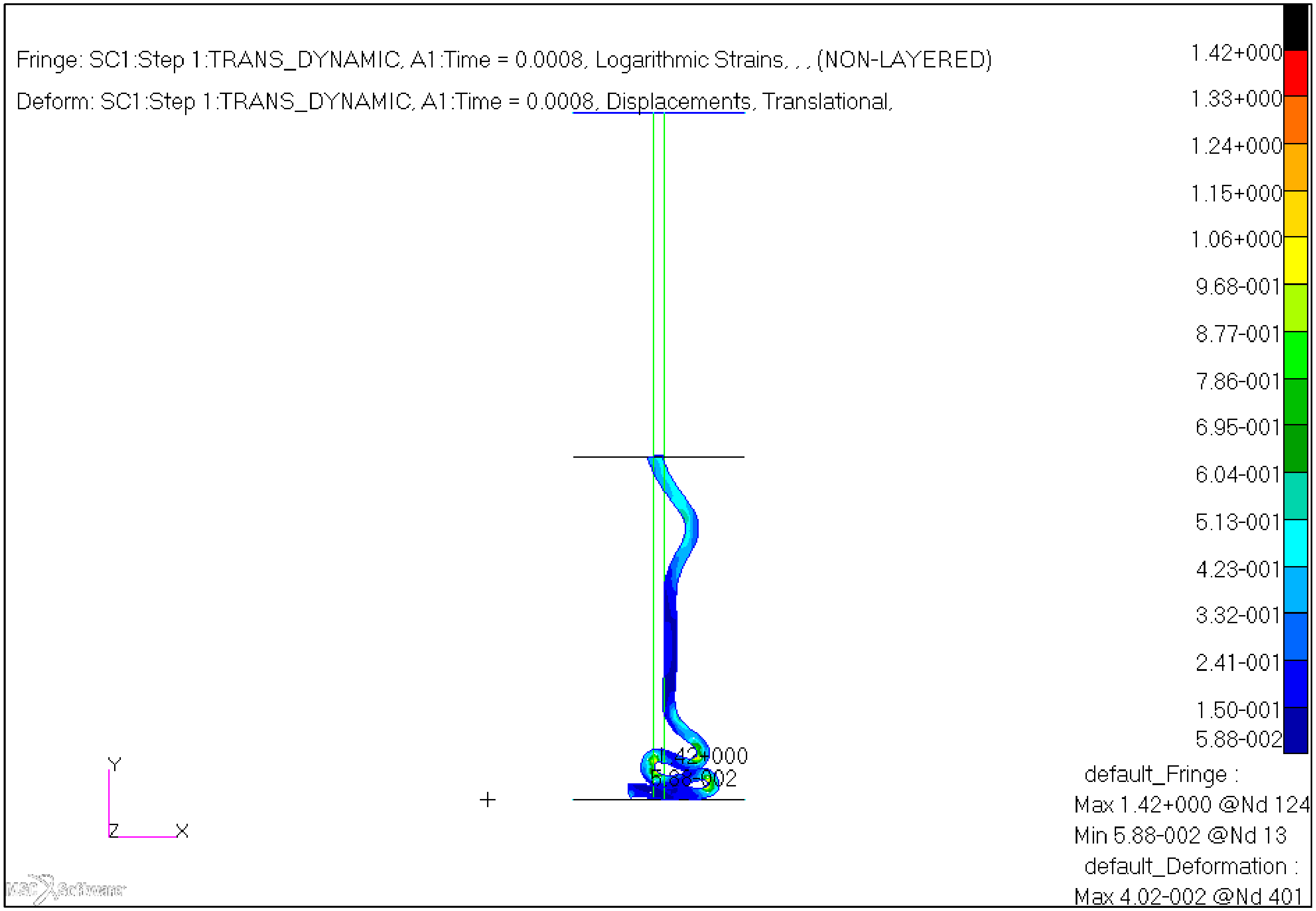
Create a logarithmic strains fringe plot and a displacements, translational deformation plot for the final result case:

a. Under the *Results* tab, click **Fringe/Deformation** in the *Quick Plot* group.
b. Under *Select Result Cases,* select the last result case, **Trans_Dynamic, A1:Time = 0.0008**.
c. Select **Logarithmic Strains,** as the *Fringe Result*.
d. Select **Displacements, Translational**.
e. Click the **Deform Attributes** icon.
f. Change the *Scale Interpretation* to **True Scale**.
g. Uncheck *Show Undeformed*
h. Click **Apply**.



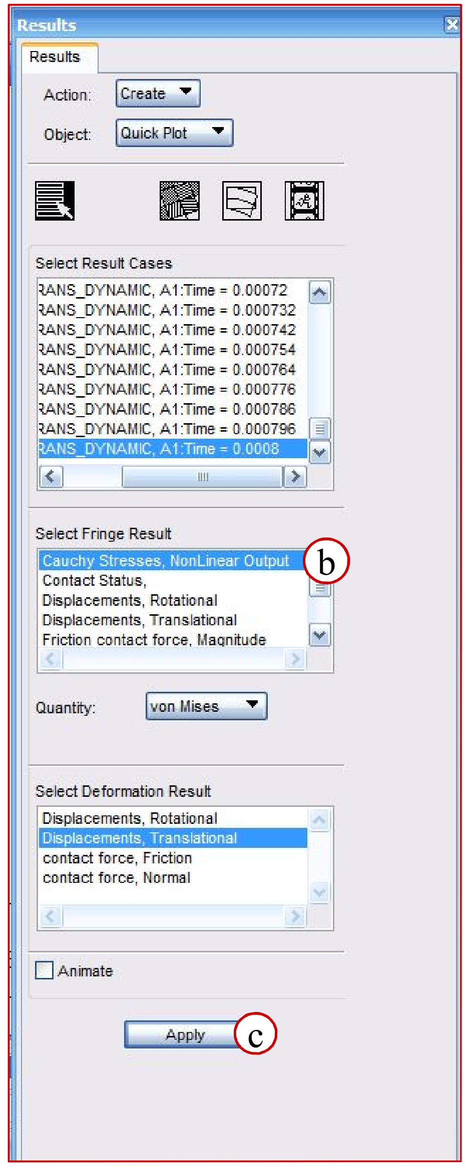
MSC Software

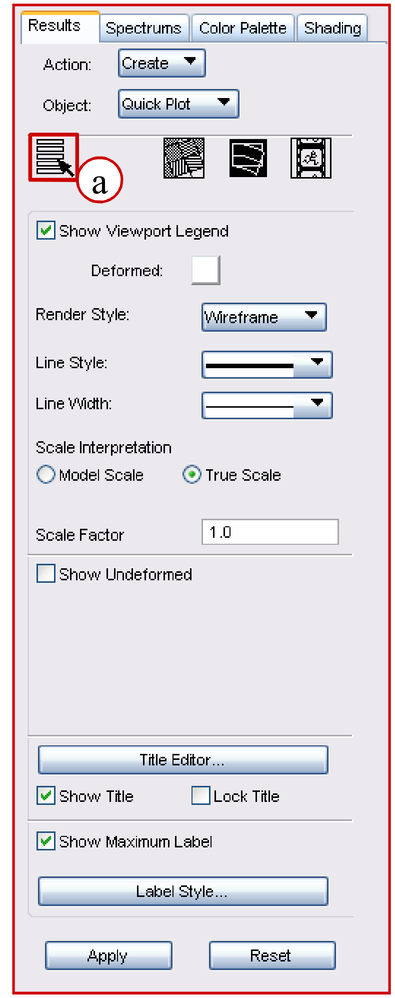
Step 12. View Results (Cont.)



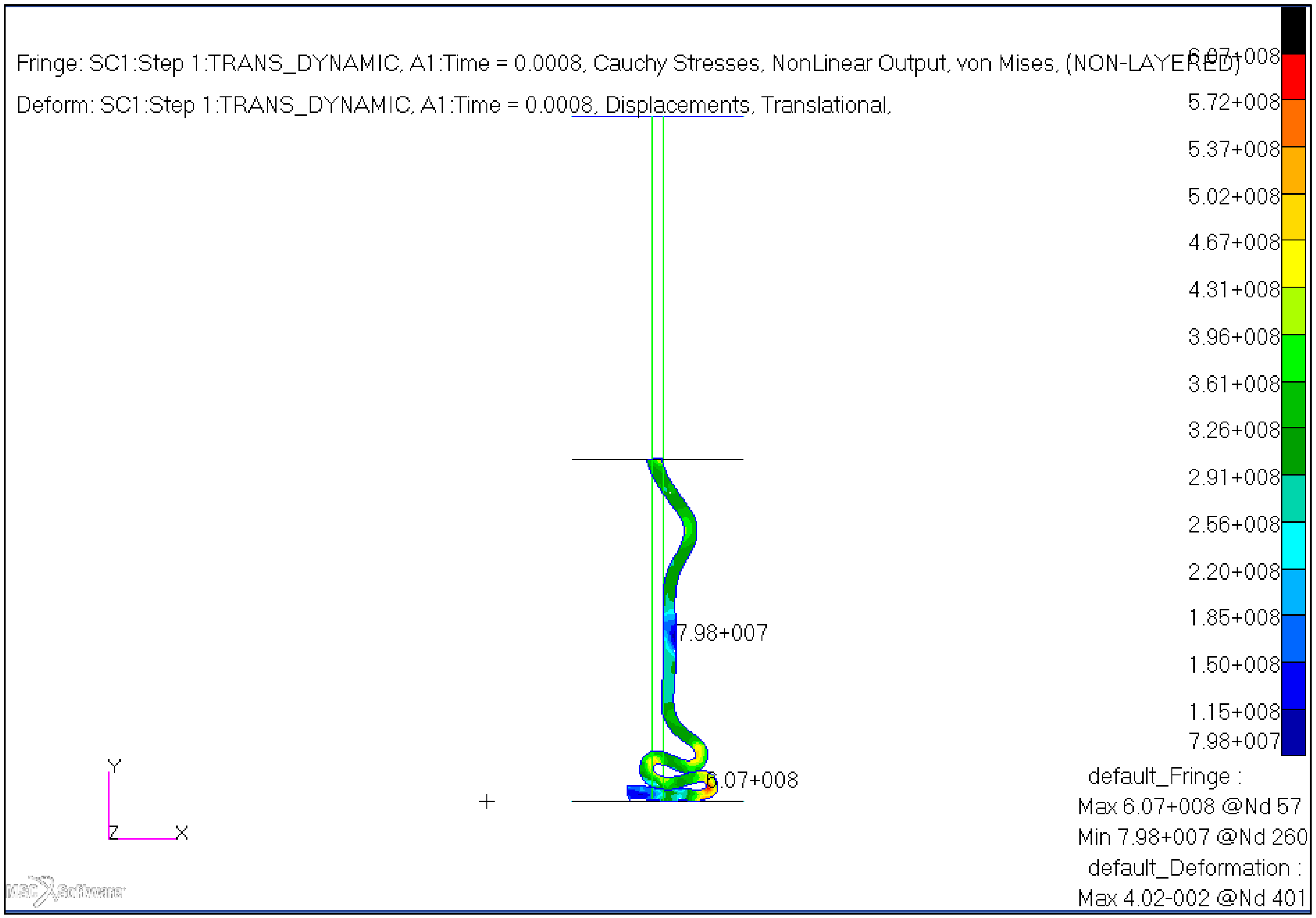
Step 12. View Results (Cont.)

a. Click the **Select Results** icon.
b. Select **Cauchy Stresses, NonLinear Output** as the *Fringe Result*.
c. Click **Apply**.





Step 12. View Results (Cont.)



Step 13. Quit Patran