

Contact Analysis using MSC Nastran and Patran

NAS133 Workshops

March 2014

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.

Copyright © 2014 MSC.Software Corporation. All Rights Reserved. This notice shall be marked on any reproduction of this documentation, in whole or in part. Any reproduction or distribution of this document, in whole or in part, without the prior written consent of MSC.Software Corporation is prohibited.

The MSC.Software corporate logo, Adams, Dytran, Easy5, Fatigue, Laminate Modeler, Marc, Mentat, MD Nastran, Patran, MSC, MSC Nastran, Mvision, Patran, SimDesigner, SimEnterprise, SimManager, SimXpert and Sofy are trademarks or registered trademarks of the MSC.Software Corporation in the United States and/or other countries. NASTRAN is a registered trademark of NASA. All other trademarks belong to their respective owners.

CONTENTS

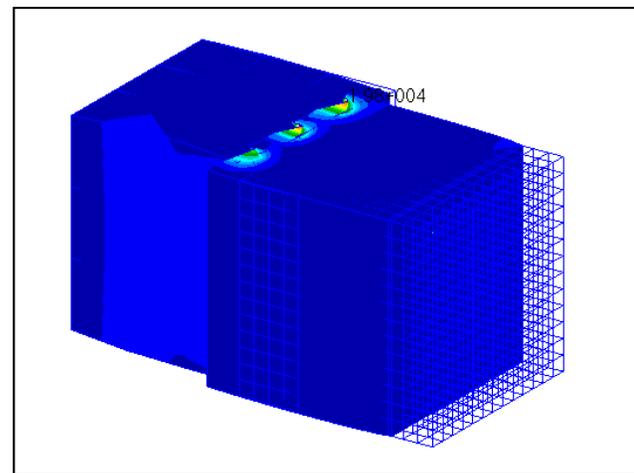
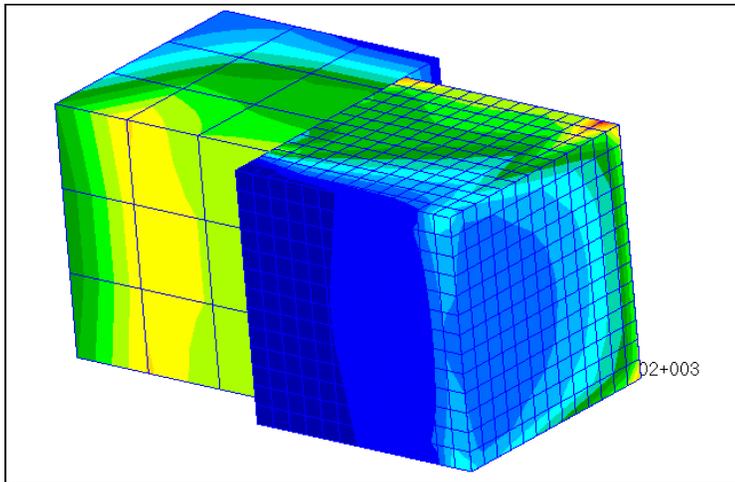
Workshop

- 1 Solid-to-Solid Contact
- 1a Deformable to Rigid Contact
- 2 Solid-to-Shell Contact
- 3 Shell Edge-to-Edge Glued Contact
- 4 Natural Frequency Analysis with Glued Contact
- 5 Shell Face to Face Contact
- 6 Curved Surface Contact
- 7 Interference Fit
- 8 Bolted Joint Analysis



WORKSHOP 1

SOLID-TO-SOLID CONTACT



- **Workshop Objectives**

- To perform a start to finish solid to solid contact analysis with MSC Nastran
- To gain familiarity with the Patran contact analysis GUI
- To investigate how to conduct a contact analysis with two bodies that have different mesh densities

- **Software Version**

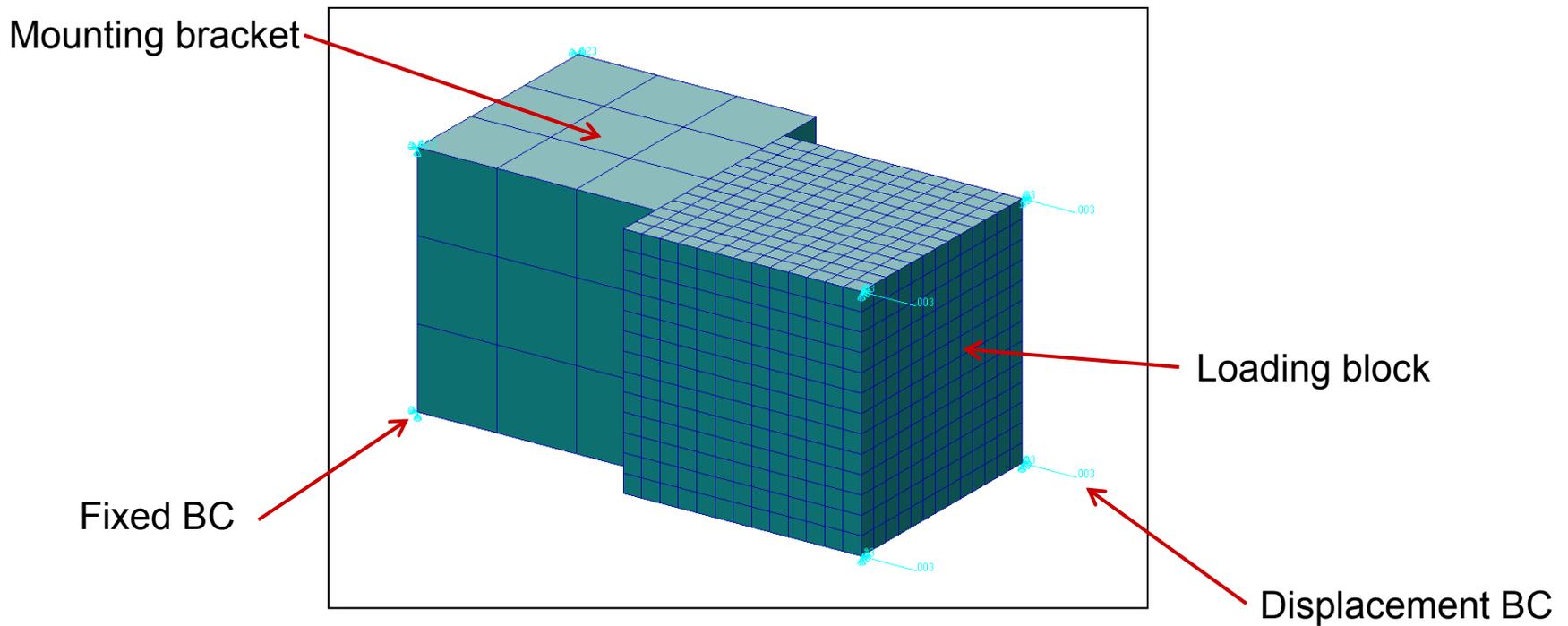
- MSC Nastran 2013.1
- Patran 2013

- **Required Files**

- **search_order_start.ses**

- **Problem Statement**

- This problem will include a mounting bracket that has a coarse mesh and a loading block that is more finely meshed. The mounting bracket has a fixed boundary condition of 0 on its left face. The loading block has a force on its right face pushing it a total displacement of .003 inches into the mounting bracket.



• Suggested Steps

1. Create a new database called **Contact_Search_order.db**
2. Build the starting model by running the session file **search_order_start.ses**.
3. Review the load and constraints on the model.
4. Define two deformable contact bodies using the Contact Bodies/Pairs Tool based on connectivity
5. Set up and Run the Analysis.
 - Set Solution Type to SOL101
 - Request MASTER/DBALL for Results Output Format.
 - Request contact results output
 - Run the analysis
6. Review the MSC Nastran input file.
7. Attach the Master result file.
8. Plot deformation and von Mises stress results
9. Create a New Body Pair between the two Deformable Bodies and select automatic contact detection
10. Rerun the Analysis File

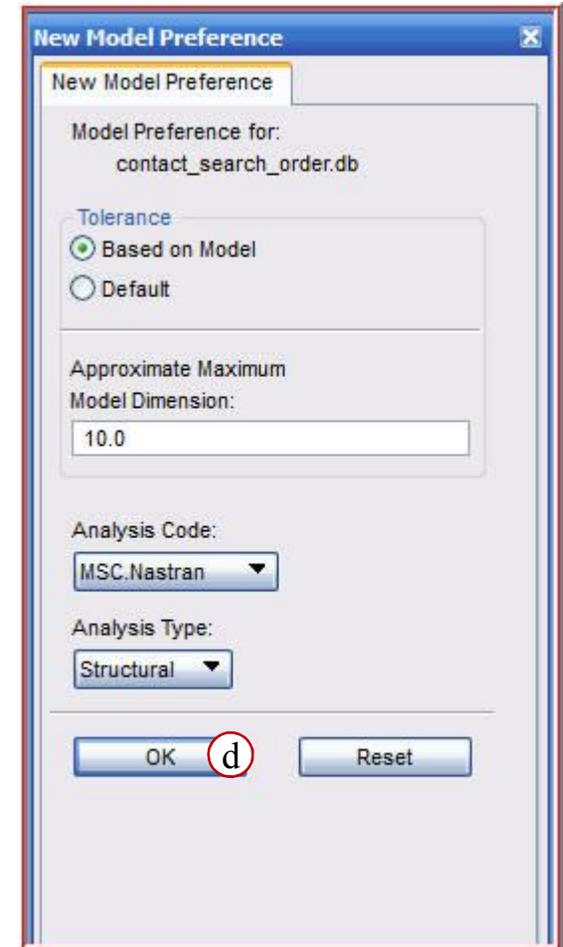
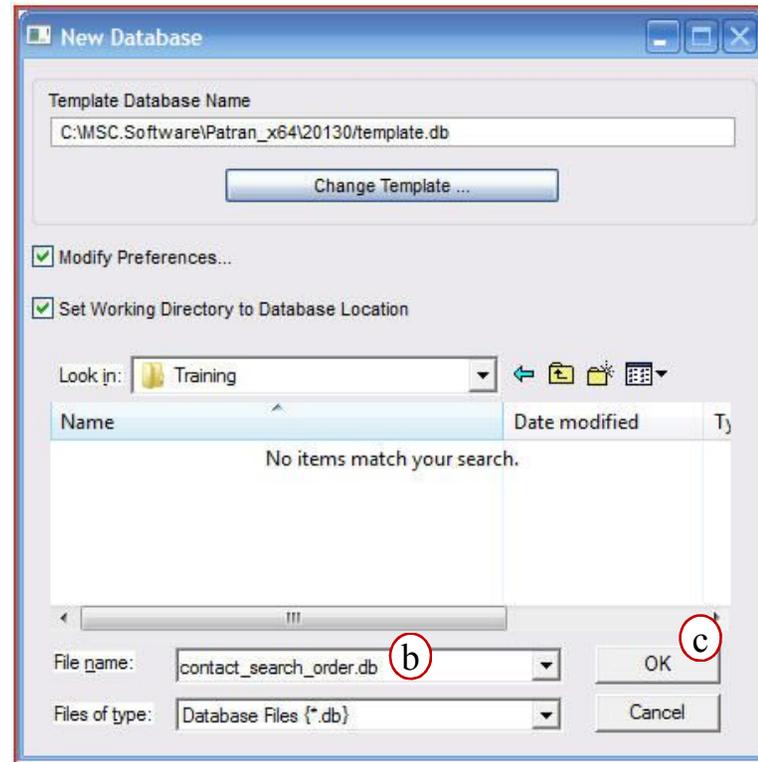
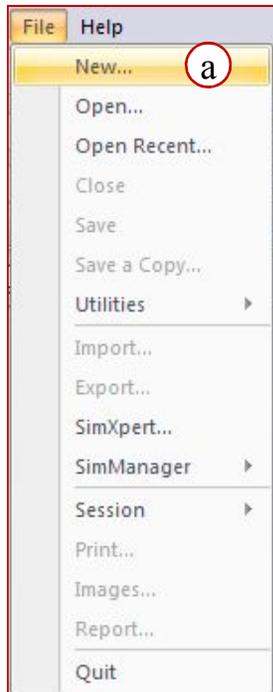
- **Suggested Steps (Cont.)**

11. Attach the New Masters Result File
12. Plot the new deformation and von Mises stress Results
13. Make a Marker Plot of Contact Status.
14. Plot Normal Contact Force as a Vector Plot on both bodies.
15. Display a Fringe Plot of Contact Status on both bodies.
16. Display a Fringe Plot of Normal Contact Stress on both bodies.

Step 1. Create New Database

Create a new database:

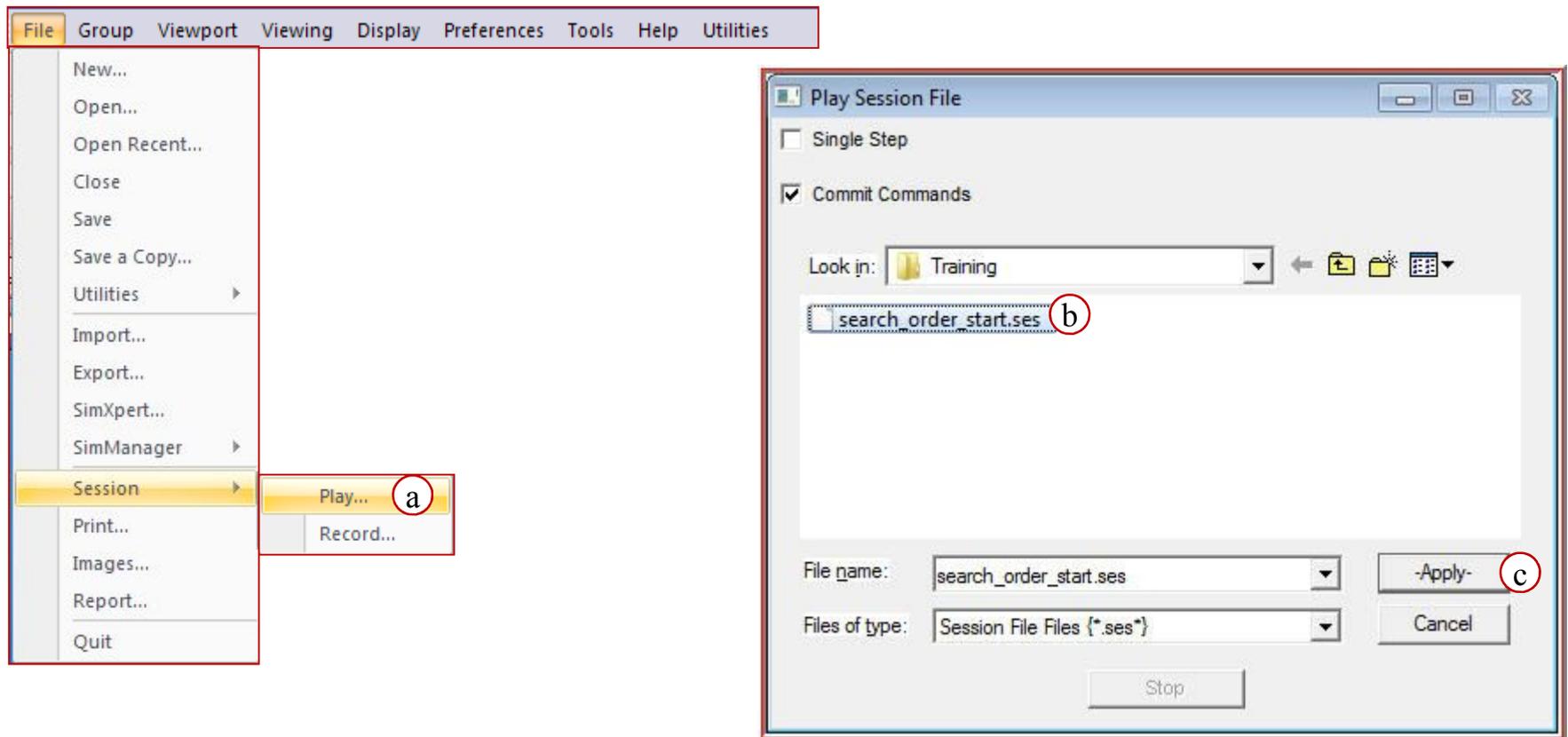
- a. Pull down **File > New**.
- b. Enter **contact_search_order** as the file name.
- c. Click **OK**.
- d. For *New Model Preference*, accept all defaults by clicking **OK**.



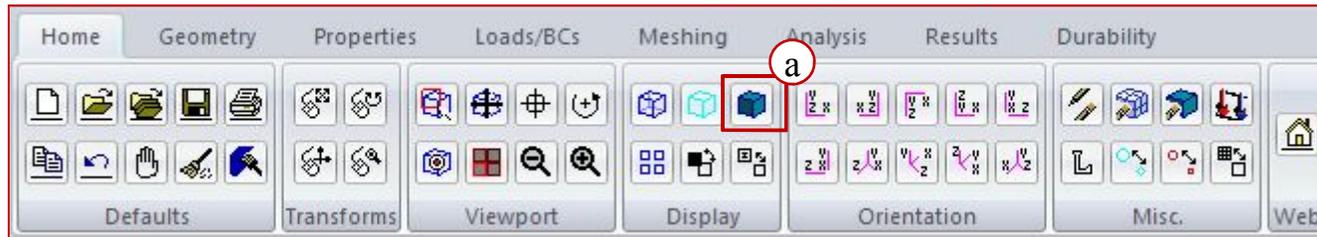
Step 2. Run the Session File

Build a starting model by running a session file:

- a. Pull down **File > Session > Play...**
- b. Select **search_order_start.ses**
- c. Click **Apply** to play the session file.

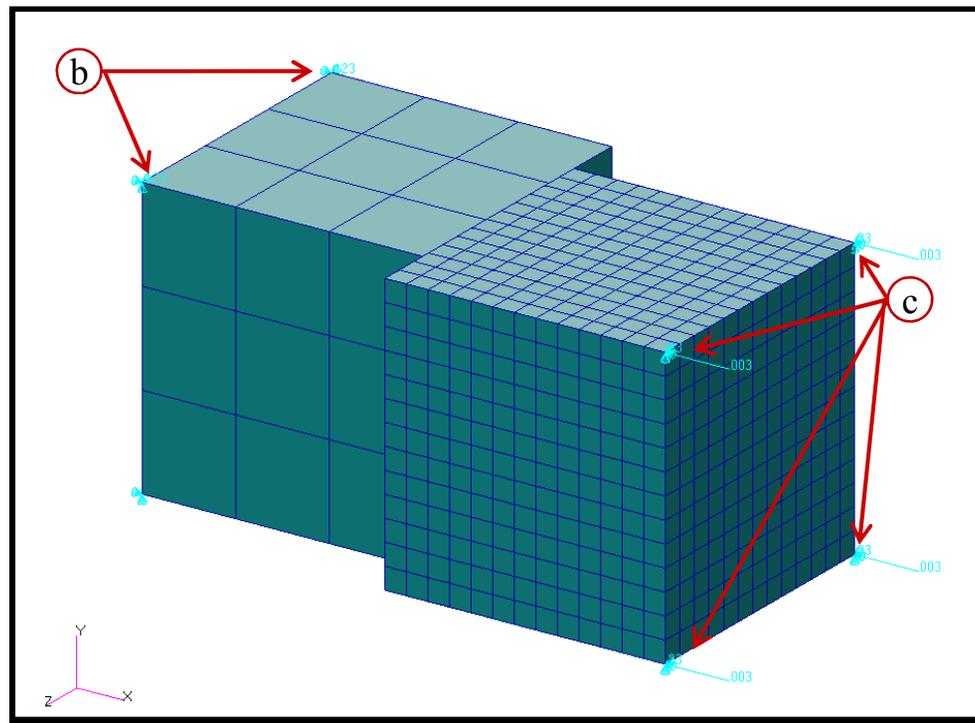


Step 3. Review the Model



Review the model:

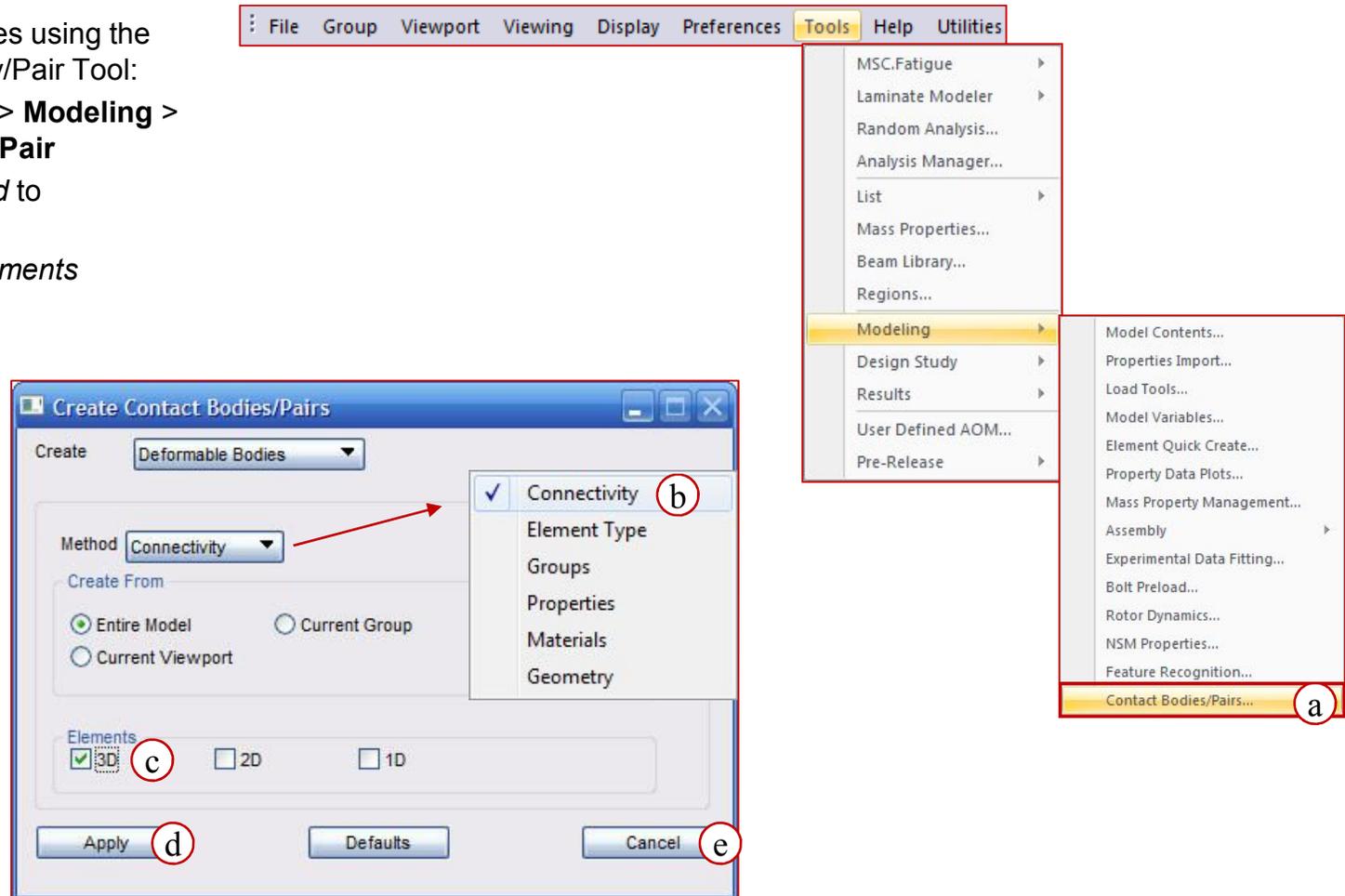
- a. Click **Smooth Shaded** in the *Display* group.
- b. Review boundary conditions: The left solid block is fixed at one face.
- c. Review loading: An enforced displacement of 0.003 is applied to the right block to push it into the left block.



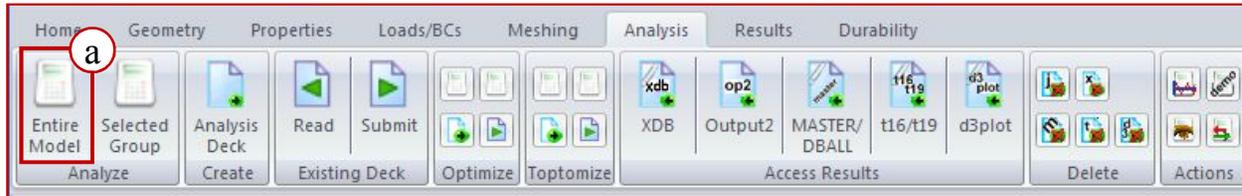
Step 4. Define Two Deformable Contact Bodies

Define Contact Bodies using the Create Contact Body/Pair Tool:

- Pull down **Tools > Modeling > Contact Bodies/Pair**
- Pull down *Method* to **Connectivity**
- Check **3D** for *Elements*
- Click **Apply**
- Click **Cancel**



Step 5. Set Up and Run the Analysis

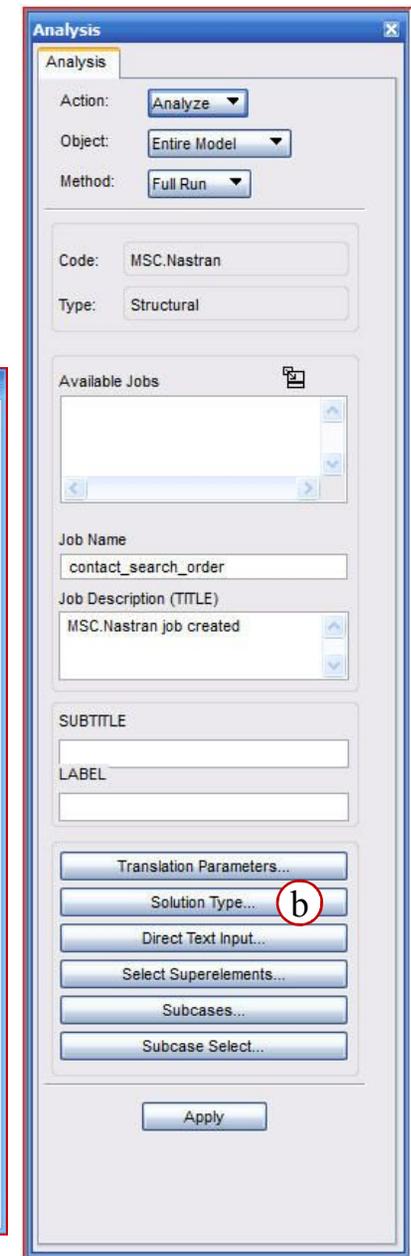
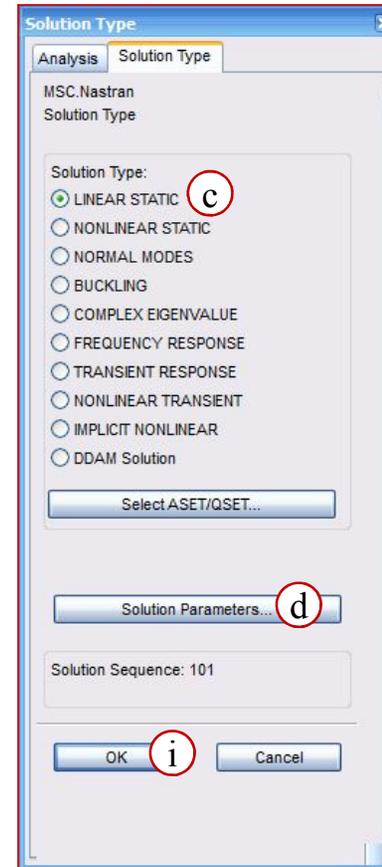
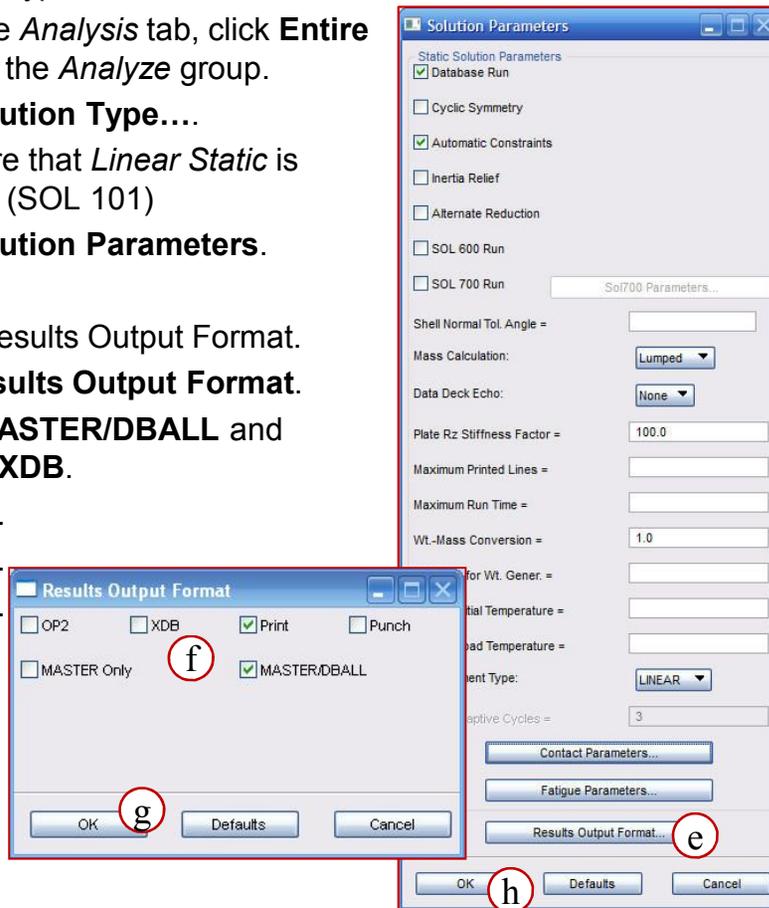


Set Solution Type to SOL101:

- Under the *Analysis* tab, click **Entire Model** in the *Analyze* group.
- Click **Solution Type....**
- Make sure that *Linear Static* is selected. (SOL 101)
- Click **Solution Parameters**.

Select the Results Output Format.

- Click **Results Output Format**.
- Check **MASTER/DBALL** and uncheck **XDB**.
- Click **OK**.
- Click **OK**.
- Click **OK**.



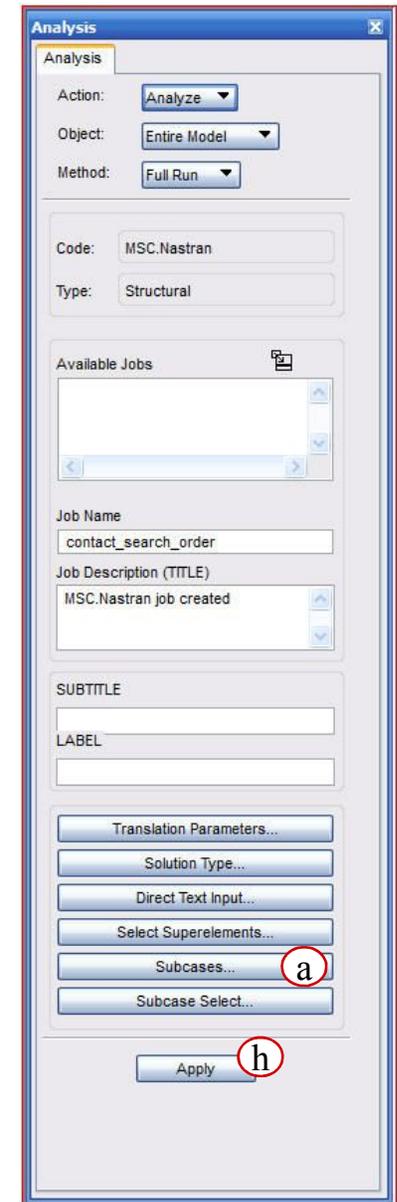
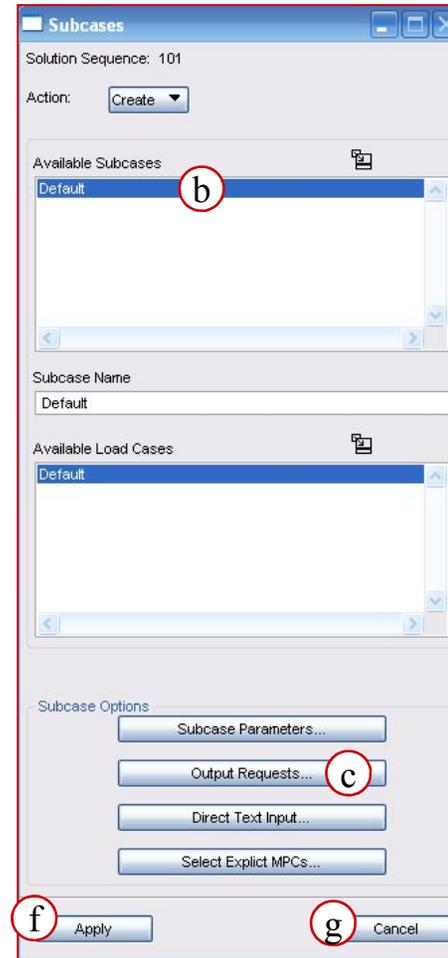
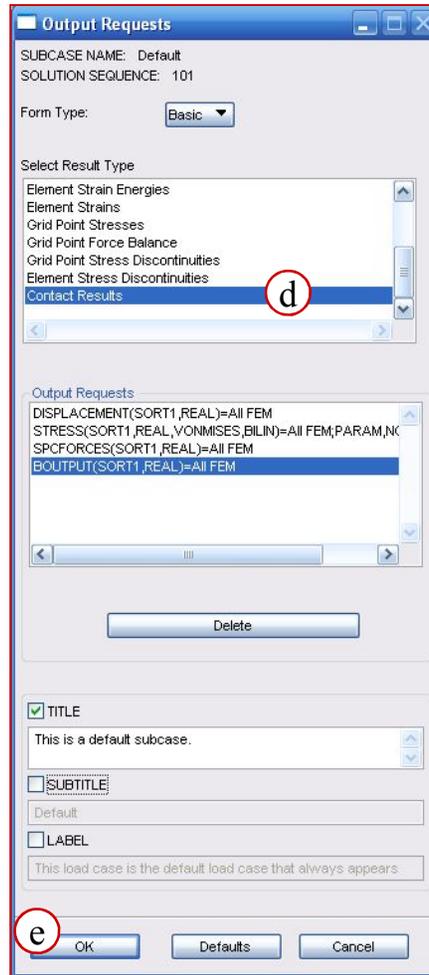
Step 5. Set Up and Run the Analysis (Cont.)

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:

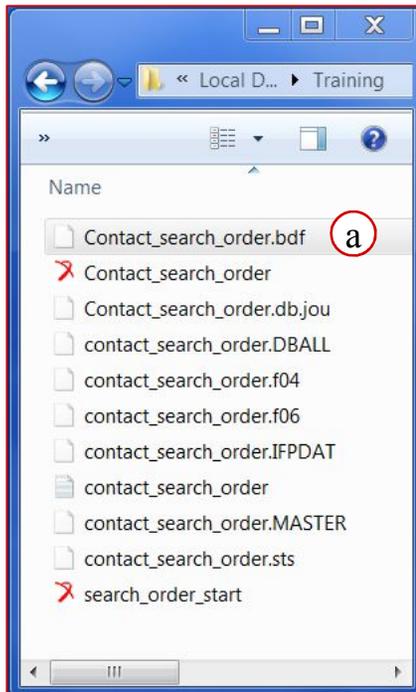
- h. Click **Apply**.



Step 6. Review MSC Nastran Input File

Review the MSC Nastran input file:

- a. In the training directory locate and open the file **contact_search_order.bdf** with any text editor.
- b. Note the Case Control entry **BCONTACT = ALLBODY** indicates that all contact bodies can potentially contact each other.



```
$ NASTRAN input file created by the Patran 2013 64-Bit input
file
$ translator on
$ Direct Text Input for Nastran System Cell Section
NASTRAN SYSTEM(316)=19
$ Replace the 19 in the above line with 7
$ to get a restart DBALL instead of what is currently written
$ Direct Text Input for File Management Section
$ Direct Text Input for Executive Control
$ Linear Static Analysis, Database
SOL 101
CEND
$ Direct Text Input for Global Case Control Data
TITLE = MSC.Nastran job created on
ECHO = NONE
SUBCASE 1
  SUBTITLE=Default
  NLPARM = 1
  BCONTACT = ALLBODY
  SPC = 2
  LOAD = 1
  DISPLACEMENT (SORT1,REAL)=ALL
  SPCFORCES (SORT1,REAL)=ALL
  STRESS (SORT1,REAL,VONMISES,BILIN)=ALL
  BOUTPUT (SORT1,REAL)=ALL
$ Direct Text Input for this Subcase
```

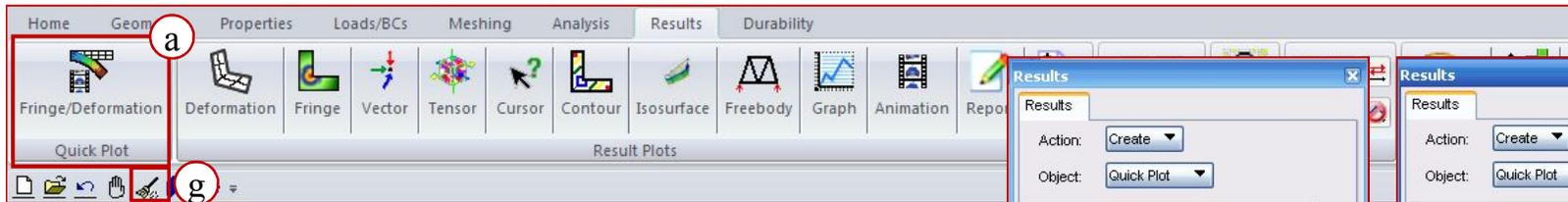
Step 7. Attach Master Result File

The image shows the MSC Nastran software interface. The top ribbon has tabs for Home, Geometry, Properties, Loads/BCs, Meshing, Analysis, Results, and Durability. The 'Analysis' tab is active, and the 'Access Results' group is highlighted with a red box. A red circle labeled 'a' is around the 'MASTER/DBALL' icon. To the right, the 'Analysis' dialog box is open. It has a 'Results' tab selected. The 'Action' dropdown is set to 'Access Results', 'Object' to 'Attach MASTER', and 'Method' to 'Result Entries'. The 'Code' is 'MSC.Nastran' and 'Type' is 'Structural'. Under 'Available Jobs', 'contact_search_order' is selected with a red circle labeled 'b'. The 'Job Name' field contains 'contact_search_order' and the 'Job Description (TITLE)' field contains 'MSC.Nastran job created'. At the bottom of the dialog, the 'Apply' button is circled in red with a red circle labeled 'c'.

Attach the Master result file:

- Under the *Analysis* tab, click **Master/DBALL** in the *Access Results* group.
- Select **contact_search_order** under *Available Jobs*.
- Click **Apply**.

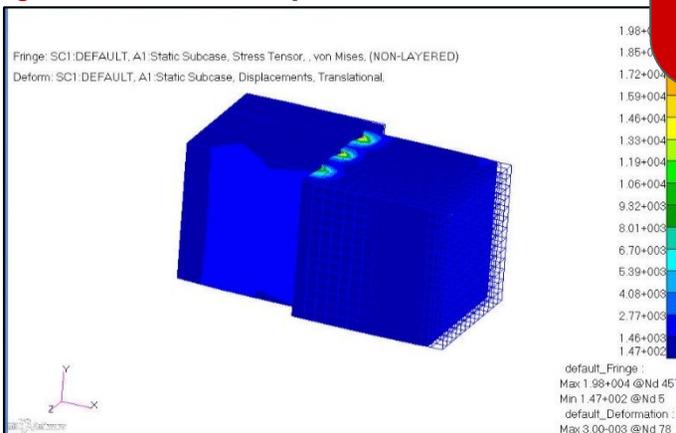
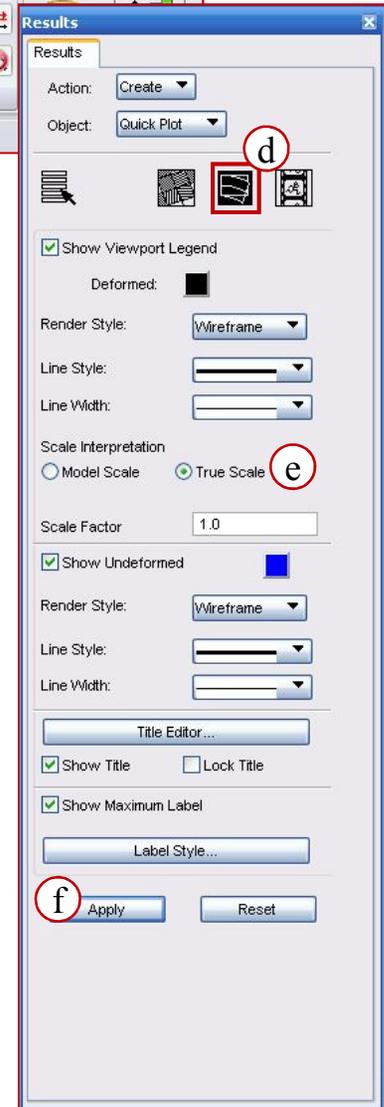
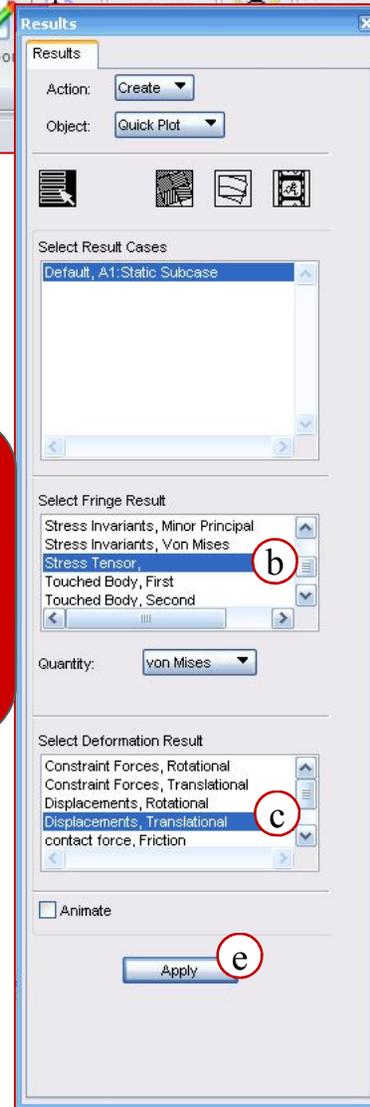
Step 8. Plot Deformation and Stress Results



Plot deformation and von Mises stress:

- a. Under the *Results* tab click **Fringe/Deformation** in the *Quick Plot* group.
- b. Select **Stress Tensor** for *Fringe Result*.
- c. Select **Displacements, Translational** for *Deformation Result*.
- d. Click the **Deform Attributes** icon
- e. Select **True Scale**.
- f. Click **Apply**.
- g. Click **Reset Graphics**

The master and slave have been incorrectly selected, causing this result. We will now switch the bodies selected as master and slave.

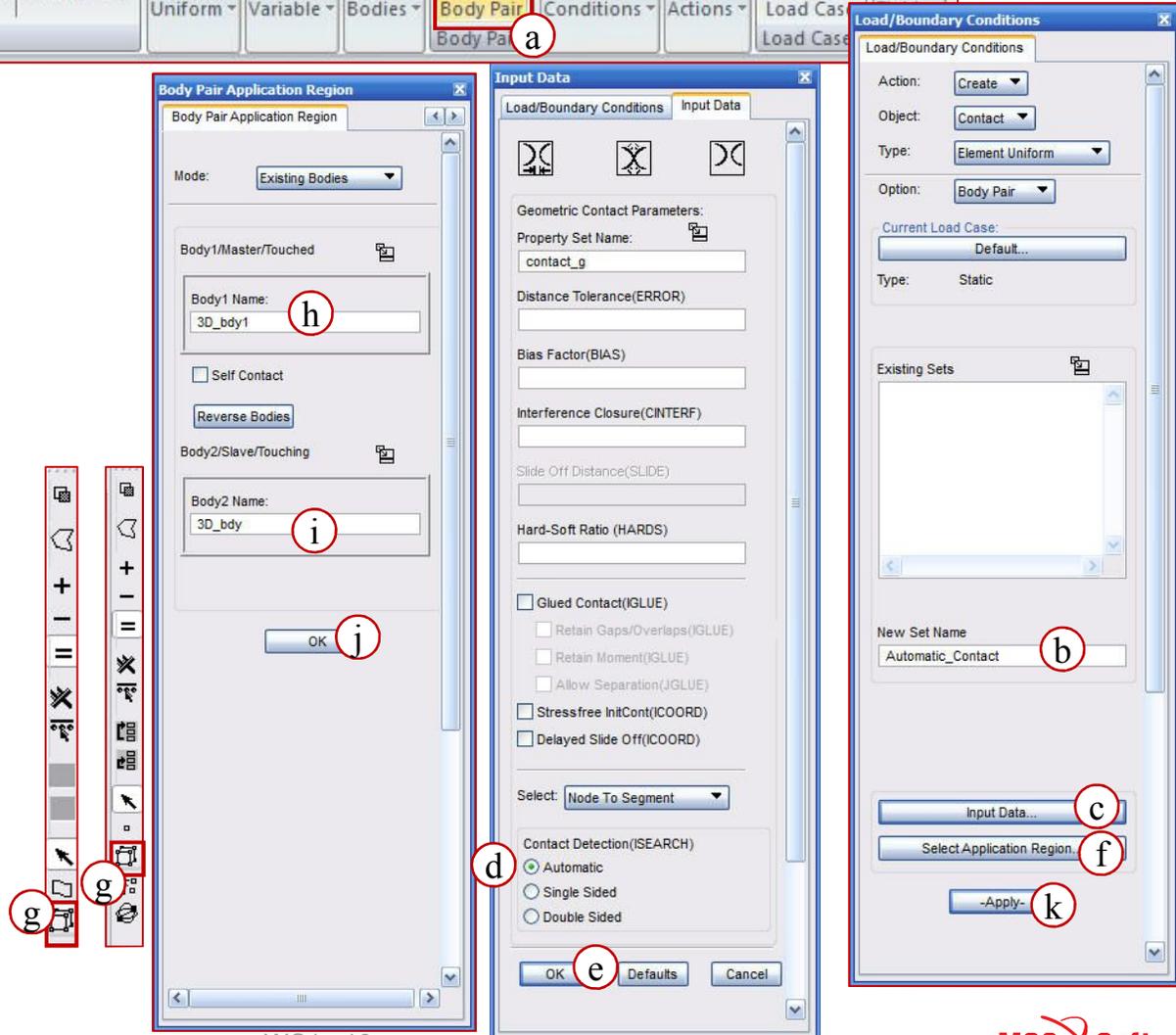


Step 9. Create Body Pair

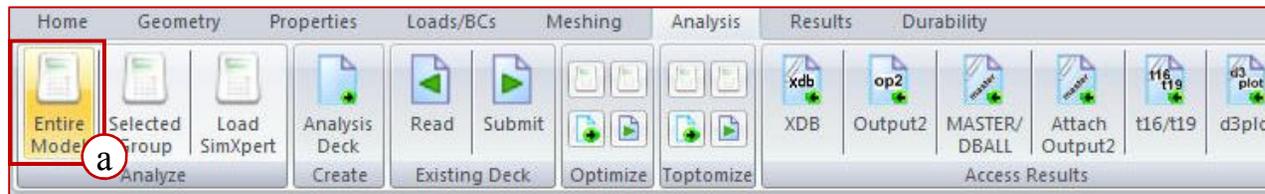


Create a new Body Pair and select automatic Contact Detection:

- a. Under the *Load/BCs* tab click **Create Body Pair** in the *Body Pairs* group.
- b. Enter **Automatic_Contact** for *New Set Name*.
- c. Click **Input Data...**
- d. Select **Automatic** for *Contact Detection*
- e. Click **OK**
- f. Click **Select Application Region**
- g. Click **FEM** and then **Elements** on the *Picking Filters* toolbar
- h. Click in the *Body1 Name* box and select any element in **3D_bdy1** (the more finely meshed body)
- i. Repeat steps g and h with *Body2 Name* and **3D_bdy** (the more coarsely meshed body)
- j. Click **OK**
- k. Click **Apply**

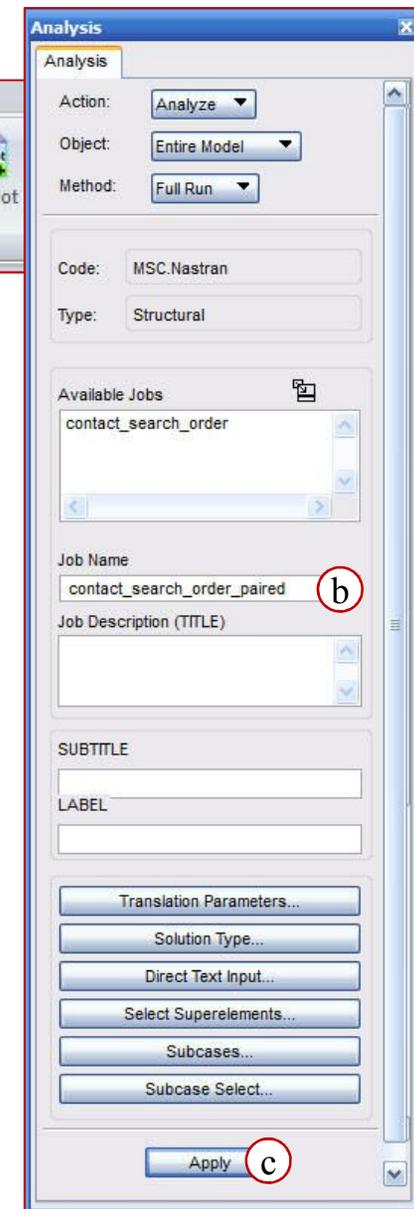


Step 10. Rerun the Analysis



Rerun the Analysis

- Under the *Analysis* tab click **Entire Model** in the *Analyze* group.
- Enter **contact_search_order_paired** for *Job Name*
- Click **Apply**.



Step 11. Attach New Master Result File

Attach the Master result file:

- Under the *Analysis* tab, click **MASTER/DBALL Attachment** in the *Delete* Group.
- Select the previous attachment under *Existing Files*
- Click **Apply**.
- Click **MASTER/DBALL** in the *Access Results* Group.
- Click **contact_search_order_paired** under *Available Jobs*
- Click **Apply**.

Step 12. Plot New Deformation and Stress Results

The screenshot shows the MSC Software interface with the Results tab selected. The 'Quick Plot' group contains several icons, with 'Fringe/Deformation' circled in red and labeled 'a'. Two 'Results' dialog boxes are open. The first dialog box shows 'SC0: A1:Static Subcase' and 'SC1:DEFAULT, A1:Static Subcase' selected in the 'Select Result Cases' list, with 'Stress Tensor' selected in the 'Select Fringe Result' list, and 'Displacements, Translational' selected in the 'Select Deformation Result' list. The second dialog box shows 'True Scale' selected under 'Scale Interpretation' and 'Apply' circled in red and labeled 'g'.

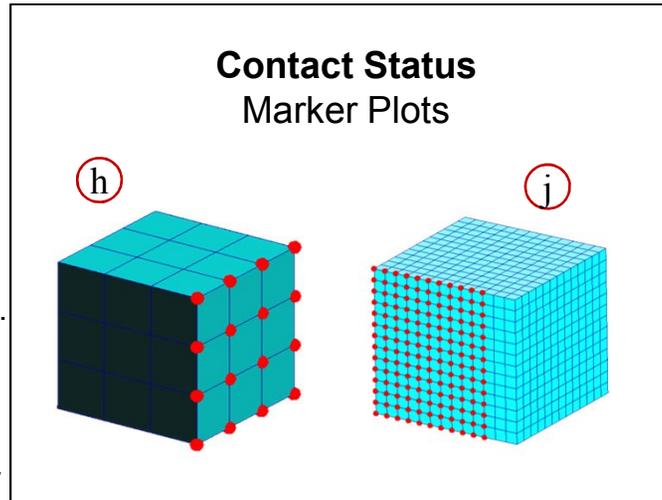
Plot deformation and von Mises stress:

- Under the *Results* tab click **Fringe/Deformation** in the *Quick Plot* group.
- Select **SC1:DEFAULT, A1:Static Subcase** for *Select Result Cases*.
- Select **Stress Tensor** for *Select Fringe Result*.
- Select **Displacements, Translational** for *Select Deformation Result*.
- Click the **Deform Attributes** icon
- Select **True Scale**.
- Click **Apply**.

Step 13. Make Marker Plot of Contact Status

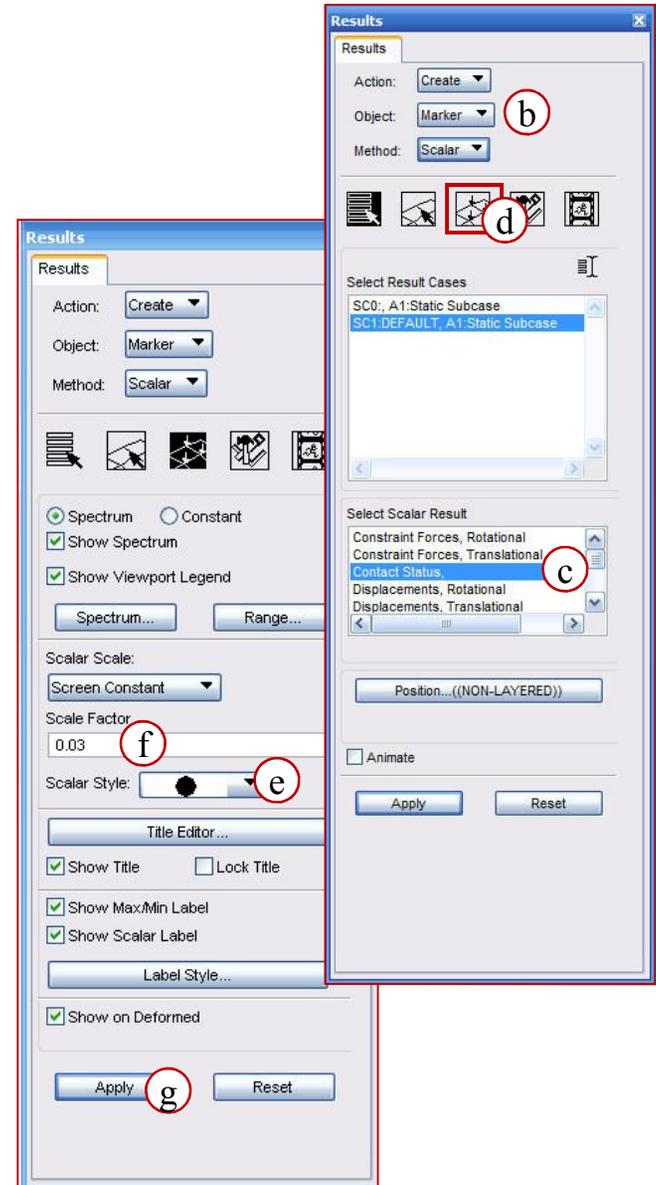
Create a Marker Plot of Contact Status on the mounting bracket.

- a. View just the mounting bracket (on the left) by pulling down **Group > Post** and posting the **mounting_bracket** group. Click **Apply**. (Not shown)
- b. Pull down **Create > Marker > Scalar**.
- c. Select **Contact Status** for the *Scalar Result*.
- d. Click the **Display Attributes** icon.
- e. Pull down *Scalar Style*, pick the **Solid Sphere**.
- f. Set *Scale Factor* to .03
- g. Click **Apply**.
- h. The solid sphere markers appear on the mounting bracket as shown.



Create a marker plot of Contact Status on the loading block:

- i. View just the loading block (on the right) by pulling down **Group > Post** and posting the **loading_block** group. Click **Apply**. (Not shown)
- j. When the block is displayed it should have the solid sphere markers as shown above.
- k. When finished observing the markers, click **Reset Graphics** to remove them.



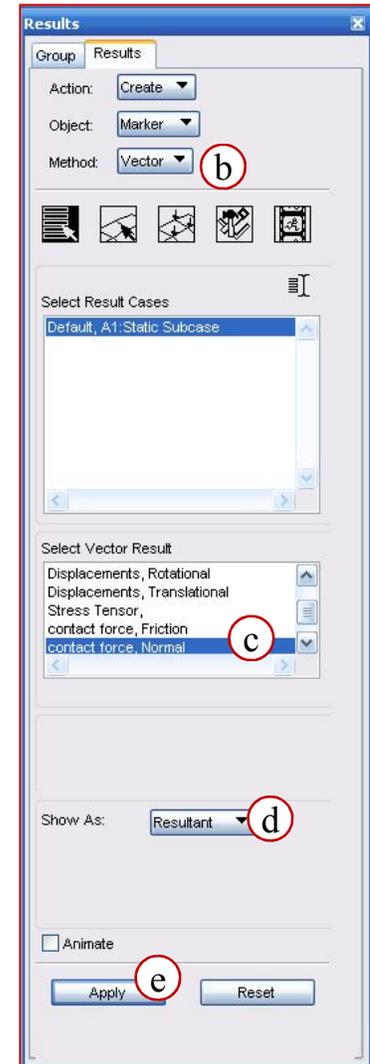
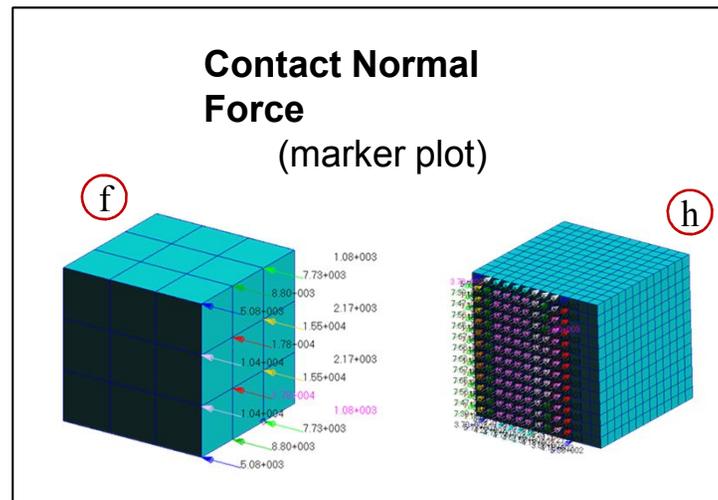
Step 14. Plot Normal Contact Force

Plot Normal Contact Force Vector on the mounting bracket:

- a. View just the mounting bracket by pulling down **Group** > **Post** and posting the group **mounting_bracket**. Click **Apply**. (not shown)
- b. Pull down **Vector** for *Method*
- c. For the *Vector Result* select **Contact Force, Normal**.
- d. Pull down **Resultant** for *Show As*.
- e. Click **Apply**.
- f. The Normal Force Vectors will appear on the mounting bracket as shown.

Plot Normal Contact Force Vector on the loading block:

- g. View just the loading block by pulling down **Group** > **Post** and posting the group **loading_block**. Click **Apply**. (not shown)
- h. The Normal Force Vectors are displayed on the loading block.
- i. When finished observing the vectors, click **Reset Graphics** to remove them.



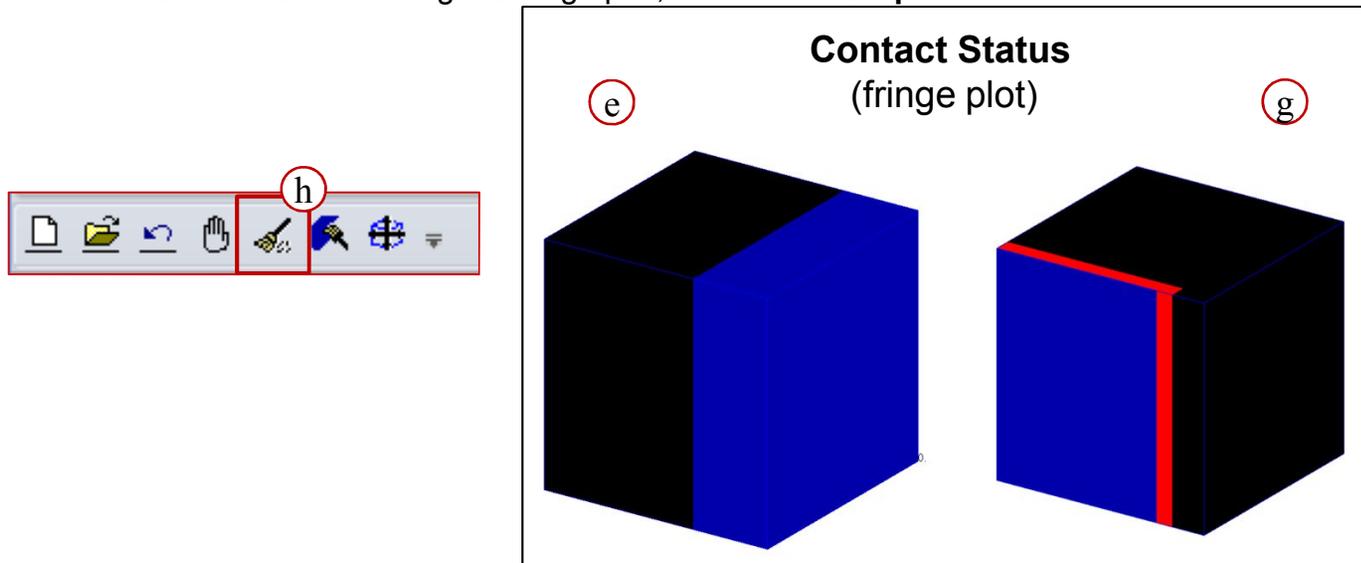
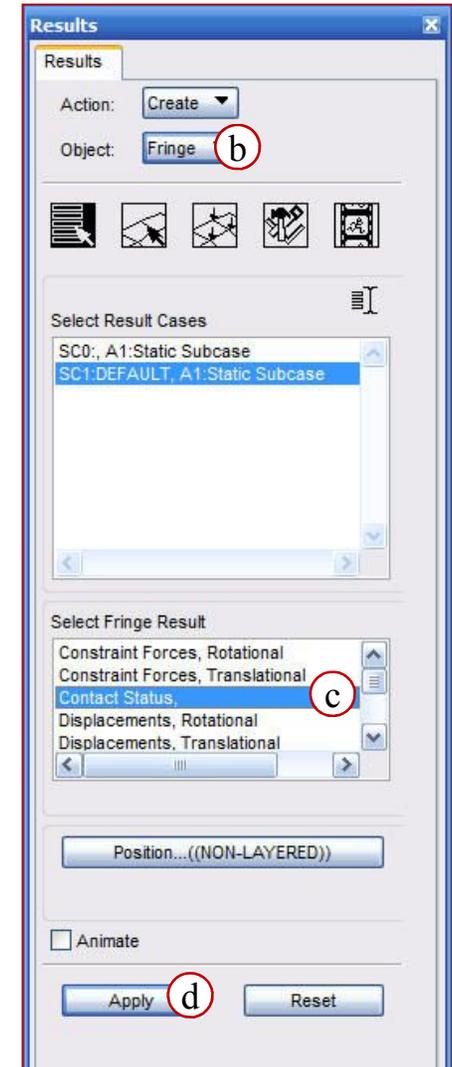
Step 15. Display a Fringe Plot of Contact Status

Display a fringe plot of Contact Status on the mounting bracket:

- View just the mounting bracket by pulling down **Group > Post** and posting the **Mounting_bracket** group. Click **Apply**. (Not shown)
- Pull down **Object** to **Fringe**.
- For the **Fringe Result** select **Contact Status**.
- Click **Apply**.
- The fringe plot will appear on the mounting bracket as shown.

Display a fringe plot of Contact Status on the loading block:

- View just the loading block by pulling down **Group > Post** and posting the **Loading_block** group. Click **Apply**. (Not shown)
- When the block appears it will be displayed with the fringe plot.
- When finished observing the fringe plot, click **Reset Graphics** to clear it.



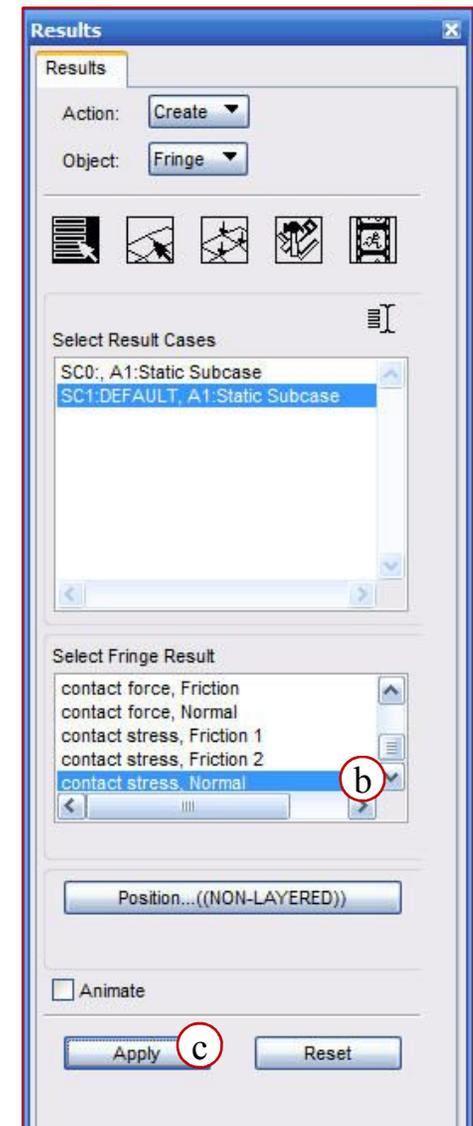
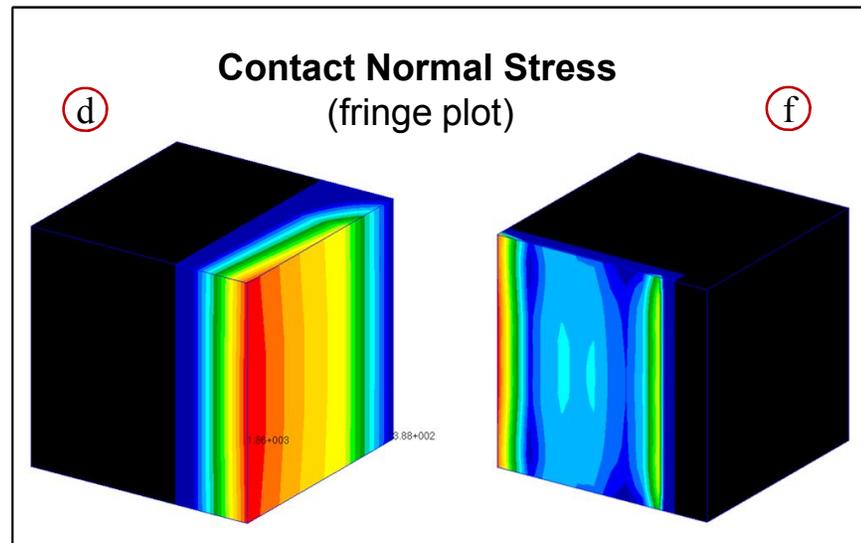
Step 16. Display a Fringe Plot of Normal Contact Stress

Display a fringe plot of Normal Contact Stress on the mounting bracket:

- View just the mounting bracket by pulling down **Group > Post** and posting the **mounting_bracket** group. Click **Apply**. (Not shown)
- For the *Fringe Result* select **Contact Stress, Normal**.
- Click **Apply**.
- The normal stress fringe plot will appear on the mounting bracket as shown.

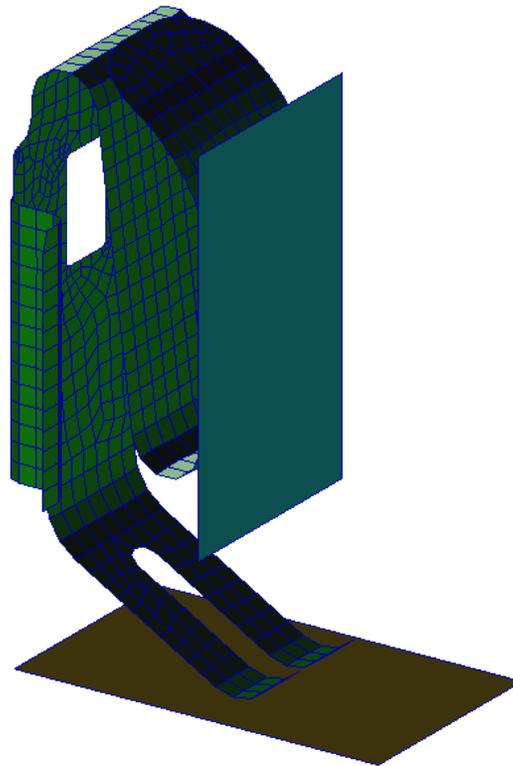
Display a fringe plot of Normal Contact Stress on the loading block:

- View just the loading block by pulling down **Group > Post** and posting the **loading_block** group. Click **Apply**. (Not shown)
- When the block appears it will be displayed with the fringe plot.



WORKSHOP 1A

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**

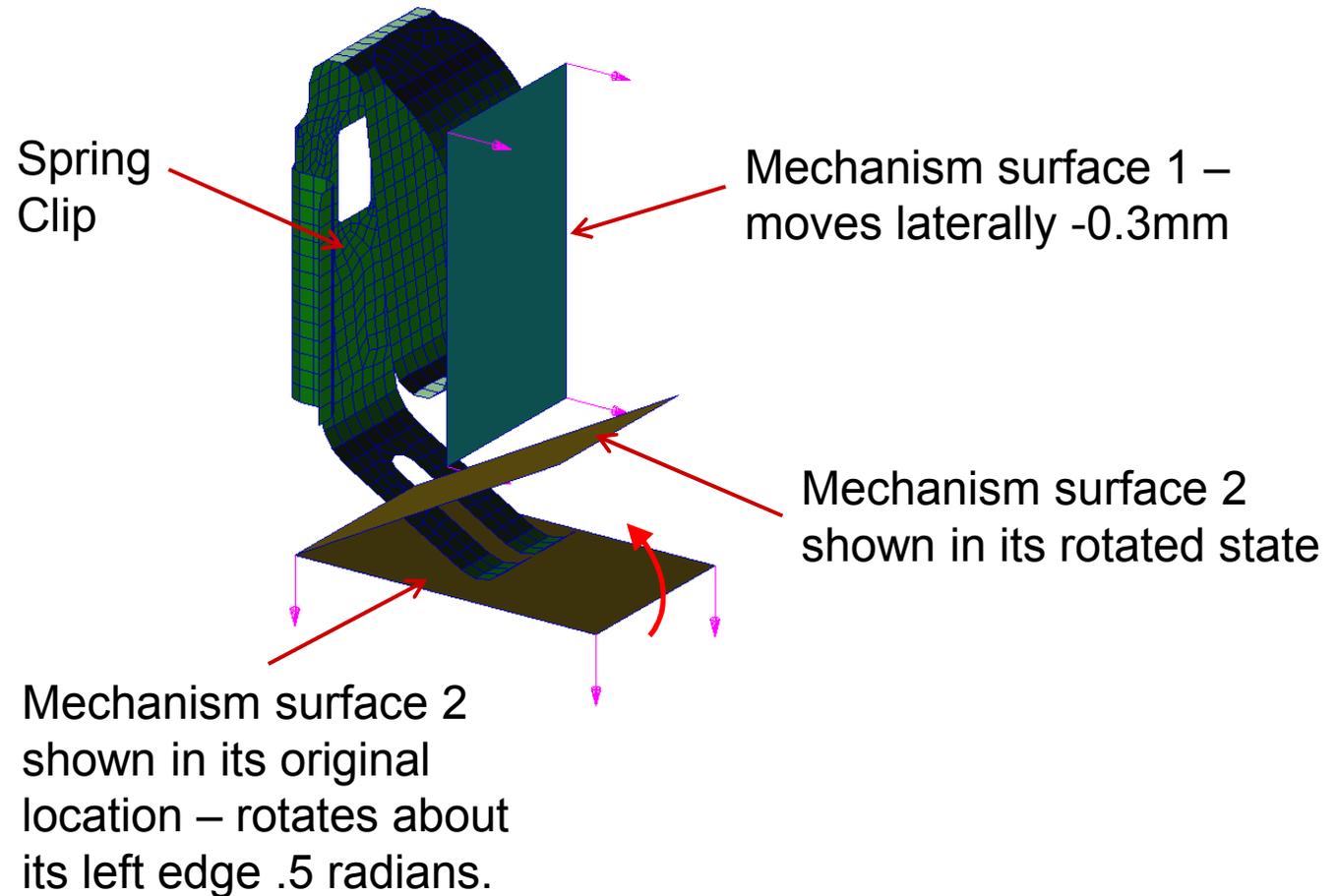
- MSC Nastran 2013.1
- Patran 2013

- **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



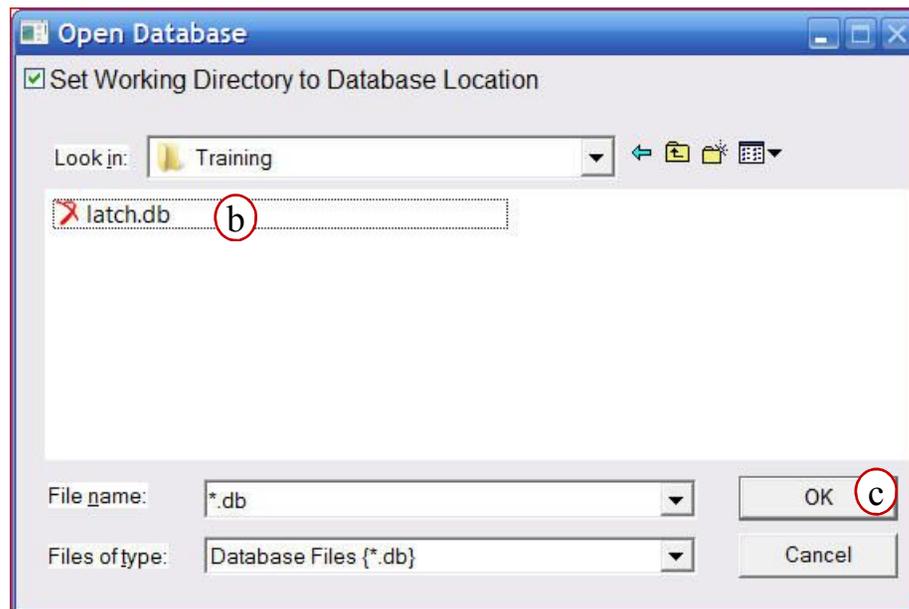
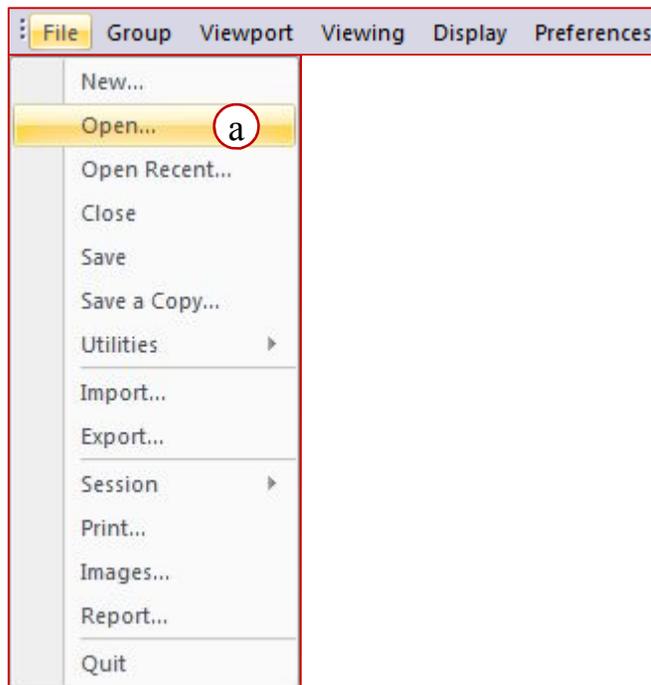
• Suggested Steps

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 MASTER/DBALL 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 MASTER 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

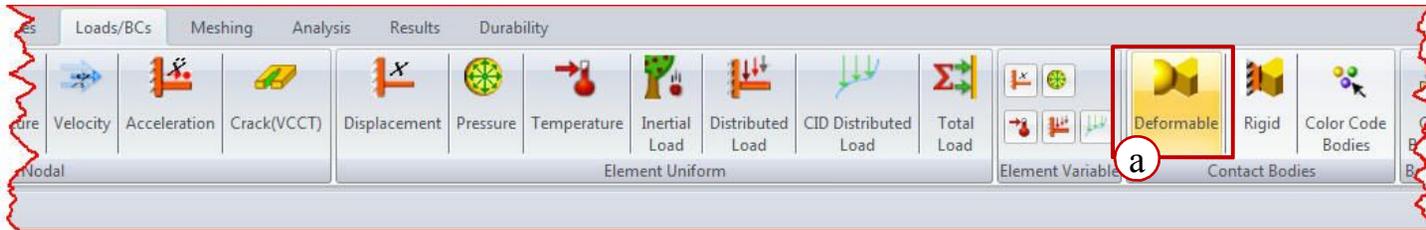
Step 1. Open the Existing Database

Open the database **latch.db**:

- a. Pull down **File > Open**.
- b. Select **latch.db** as the database file.
- c. Click **OK**.

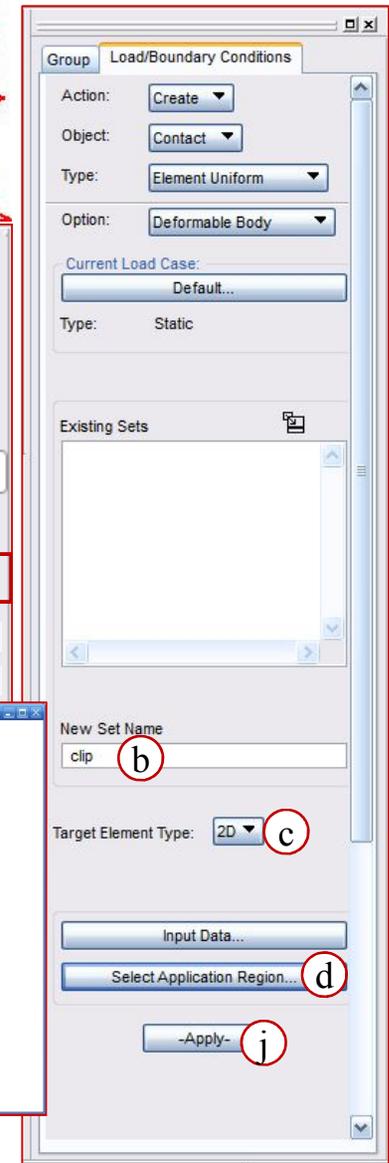
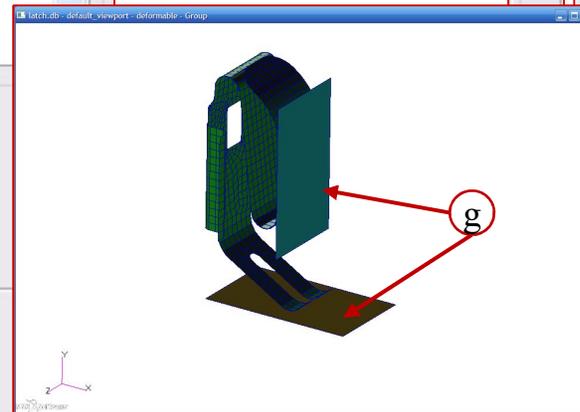
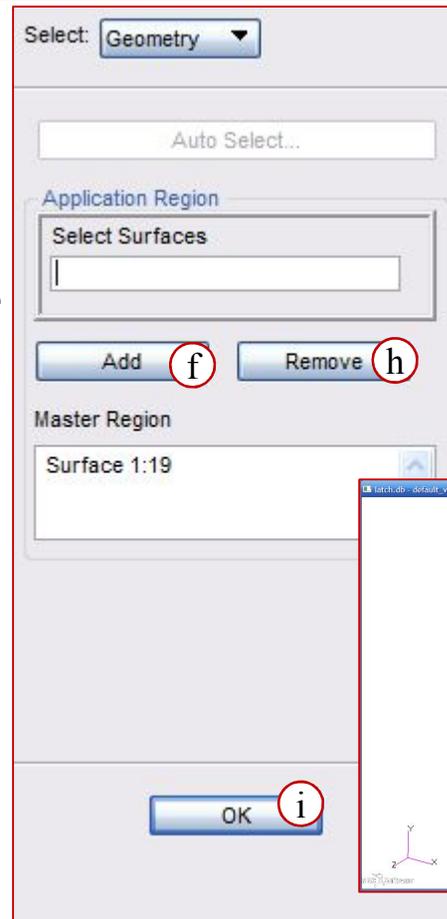


Step 2. Create Deformable Body

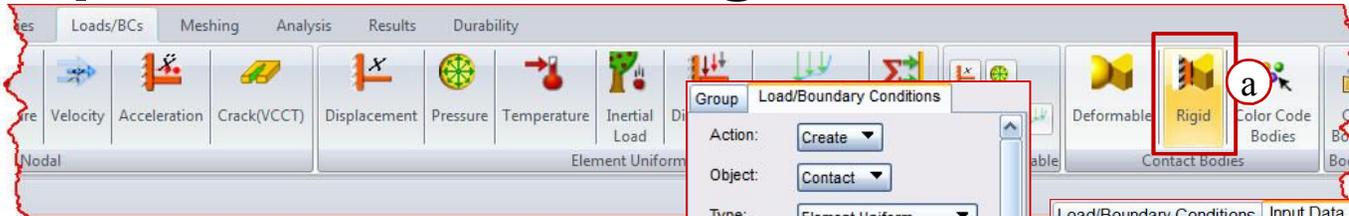


Create a deformable contact body for the spring clip:

- 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**.



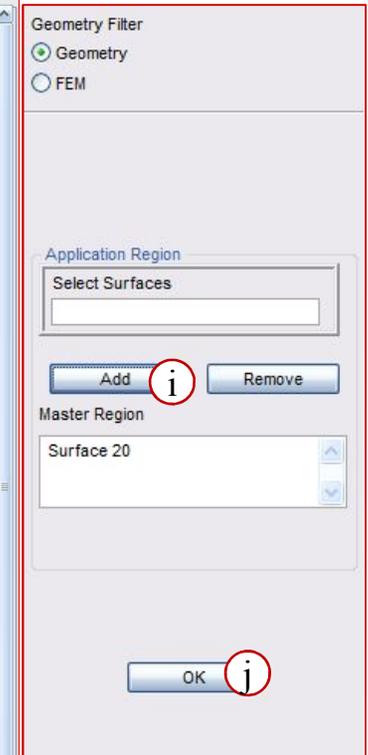
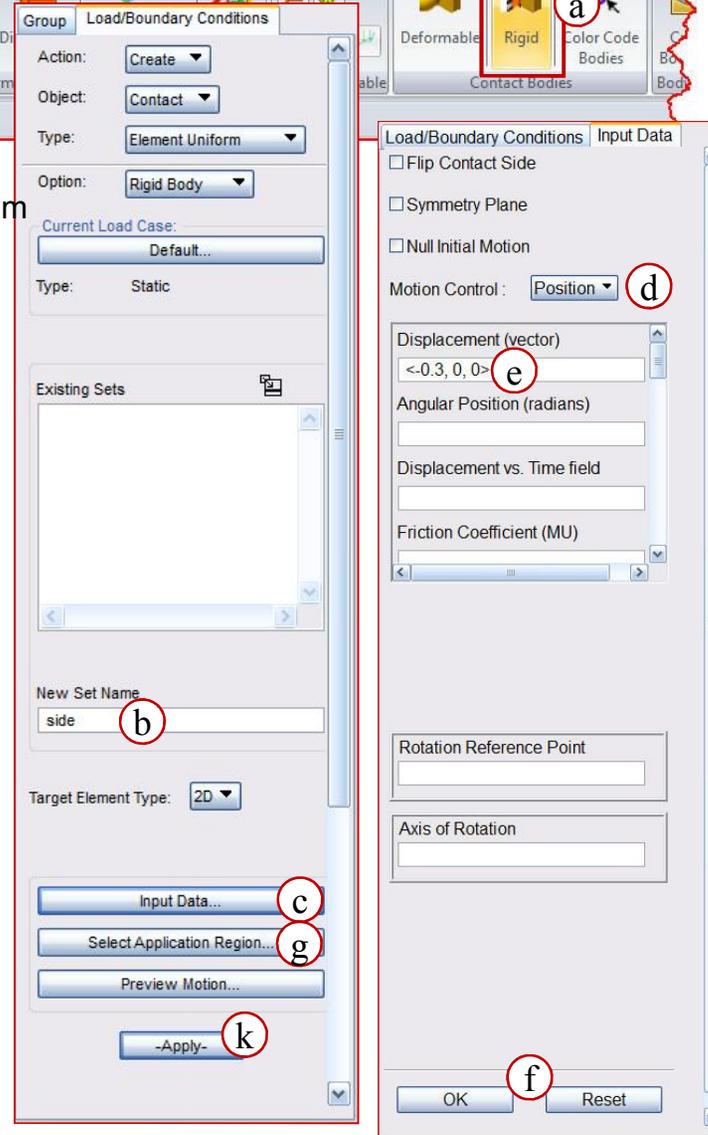
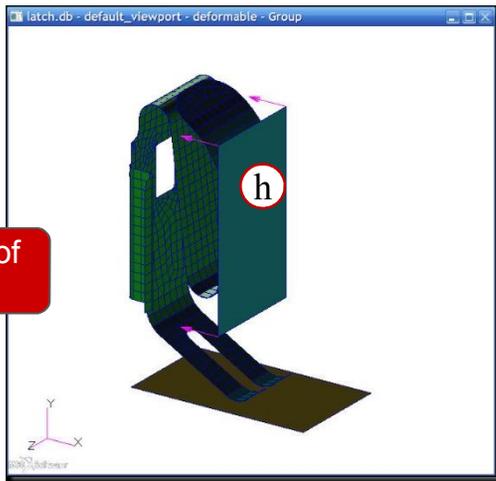
Step 3. Create Two Rigid Contact Bodies



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 **Position**
- e. Enter **<-0.3, 0, 0>** for *Displacement(vector)*.
- 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

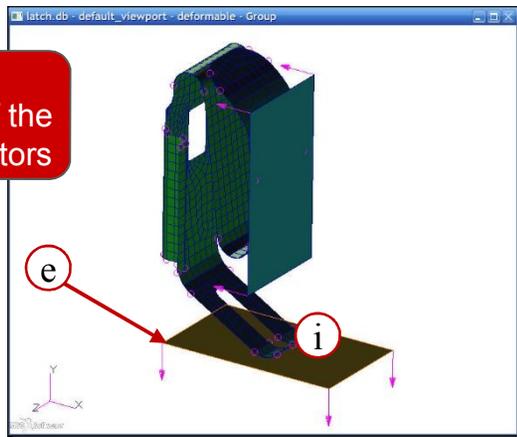


Step 3. Create Two Rigid Contact Bodies (Cont.)

Create the rigid contact bodies (cont.):

- Enter **bottom** for *New Set Name*.
- Click **Input Data**.
- Enter $\langle 0, 0, 0 \rangle$ for *Displacement(vector)*.
- Enter **0.5** for *Angular Velocity*.
- Select **Node 9999** on the corner of the bottom surface for the *Rotation Reference Point*.
- Enter $\langle 0 \ 0 \ 1 \rangle$ for the *Axis of Rotation*.
- Click **OK**.
- Click **Select Application Region**.
- Select the bottom surface.
- Click **Add**.
- Click **OK**.
- Click **Apply**.

Note the direction of the normal vectors



The screenshot shows the MSC Software interface for creating a rigid contact body. The 'Load/Boundary Conditions' panel is active, showing the 'Create' action, 'Contact' object, 'Element Uniform' type, and 'Rigid Body' option. The 'Input Data' panel is also visible, showing the 'Displacement (vector)' set to $\langle 0, 0, 0 \rangle$, 'Angular Position (radians)' set to 0.5, and 'Motion Control' set to 'Position'. The 'Rotation Reference Point' is set to 'Node 9999' and the 'Axis of Rotation' is set to $\langle 0 \ 0 \ 1 \rangle$. The 'Application Region' panel shows 'Surface 21' selected. Callouts a through k point to various fields and buttons in the interface.

Step 4. Create Contact Pairs

Create contact pairs between the clip and rigid bodies:

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

Step 4. Create Contact Pairs (Cont.)

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

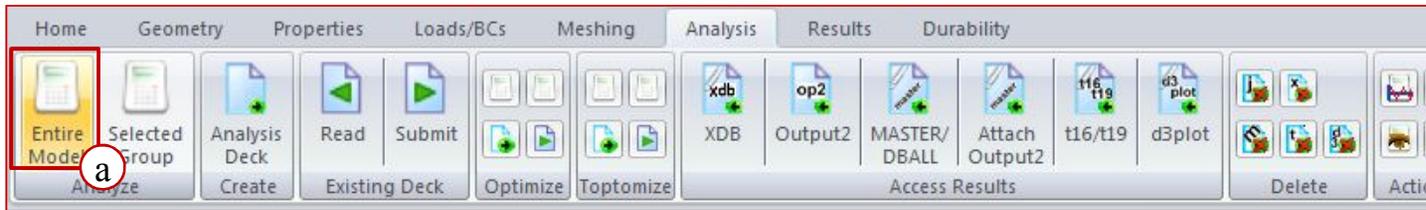
- Enter **bottom_contact** for *New Set Name*.
- Click **Select Application Region**.
- Select the **bottom** contact body for *Body1/Master/Touched*.
- Select the **clip** contact body for *Body2/Slave/Touching*.
- Click **OK**.
- Click **Apply**.

Step 5. Apply Loads and Boundary Conditions

Fix the spring clip around the rectangular hole:

- Click **Displacement Constraint** in the *Nodal* group.
- Enter **hold** for *New Set Name*.
- Click **Input Data**.
- Enter **<0, 0, 0>** for *Translations*.
- Enter **<0, 0, 0>** for *Rotations*.
- Click **OK**.
- Click **Select Application Region**.
- Click **Curve or Edge** on the *Picking Filters Toolbar*.
- Select the surface edges that bound the hole.
- Click **Add**.
- Click **OK**.
- Click **Apply**.

Step 6. Set Up and Run the Analysis

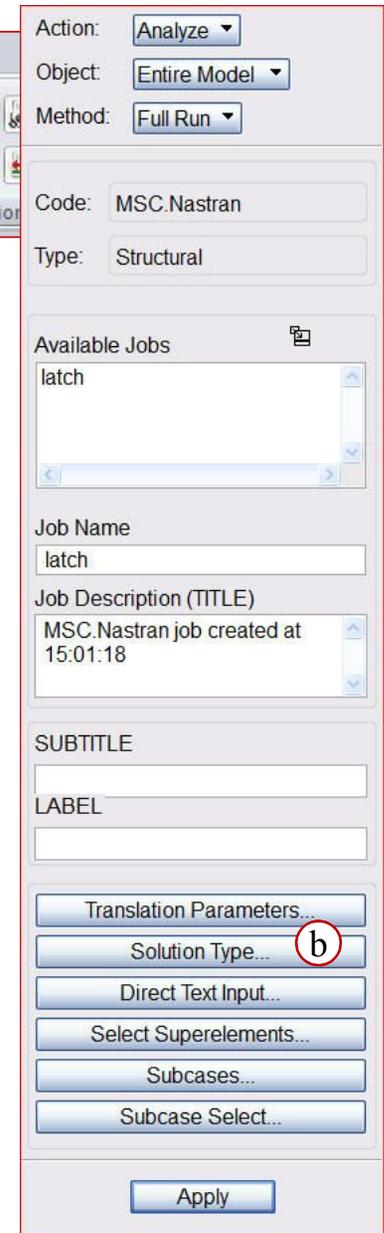
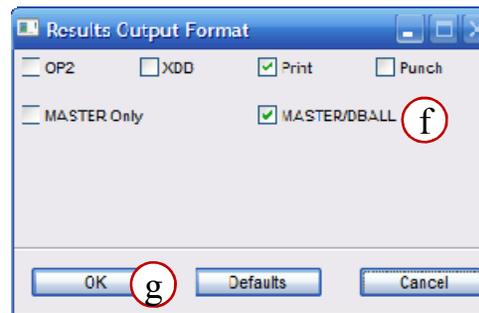
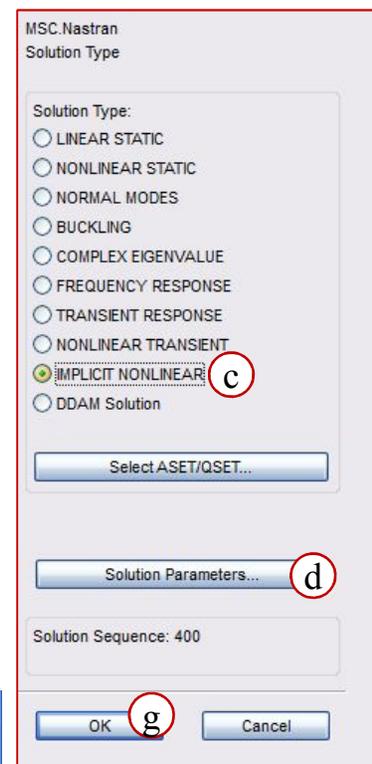
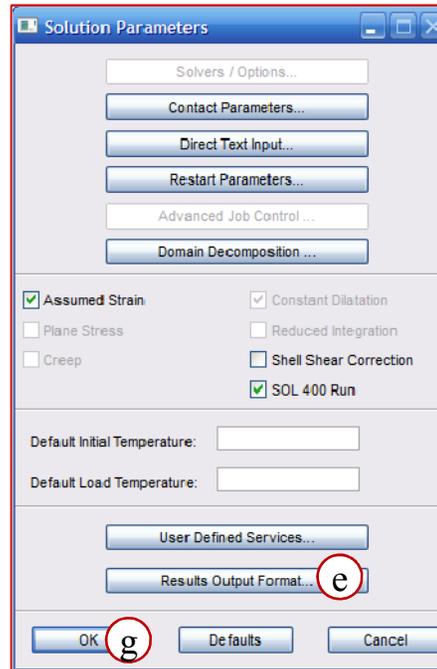


Set Solution Type to SOL400:

- 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. Ensure **MASTER/DBALL** is checked.
- g. Close all sub-menus by clicking **OK**.



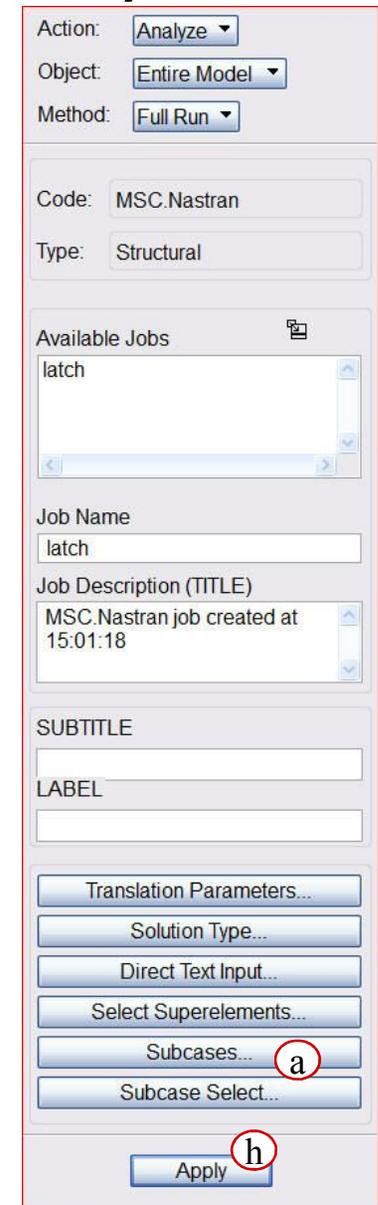
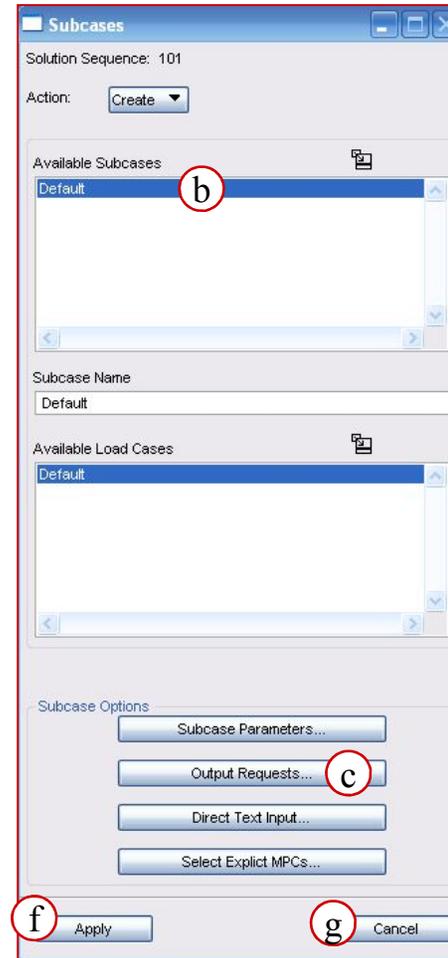
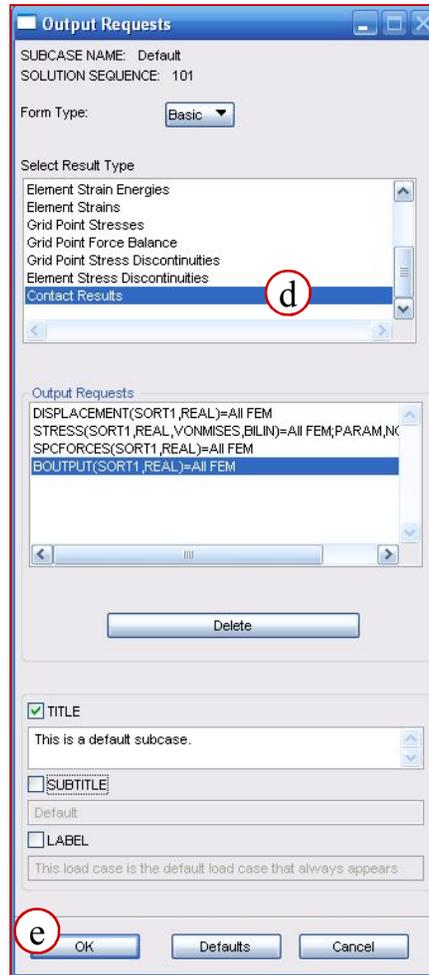
Step 6. Set Up and Run the Analysis (Cont.)

Request contact results output:

- Click **Subcases...**
- Click on **Default** under *Available Subcases*.
- Click **Output Requests**.
- Select **Contact Results**.
- Click **OK**.
- Click **Apply**.
- Click **Cancel**.

To run the analysis:

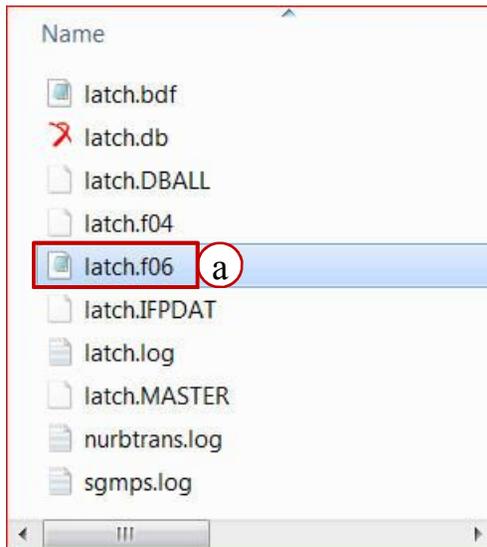
- Click **Apply**.



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



```
*** USER WARNING MESSAGE 8166 (MCN1AB2D)
      MASTER BODY IS SET TO RIGID AND IT CANNOT BE FLIPPED WHEN ISEARCH IS SET TO 0 OR
2 IN BCBODY1 ID=3.
*** USER WARNING MESSAGE 8166 (MCN1AB2D)
      MASTER BODY IS SET TO RIGID AND IT CANNOT BE FLIPPED WHEN ISEARCH IS SET TO 0 OR
2 IN BCBODY1 ID=2.

      now moving rigid surfaces into initial contact with deformable bodies
```

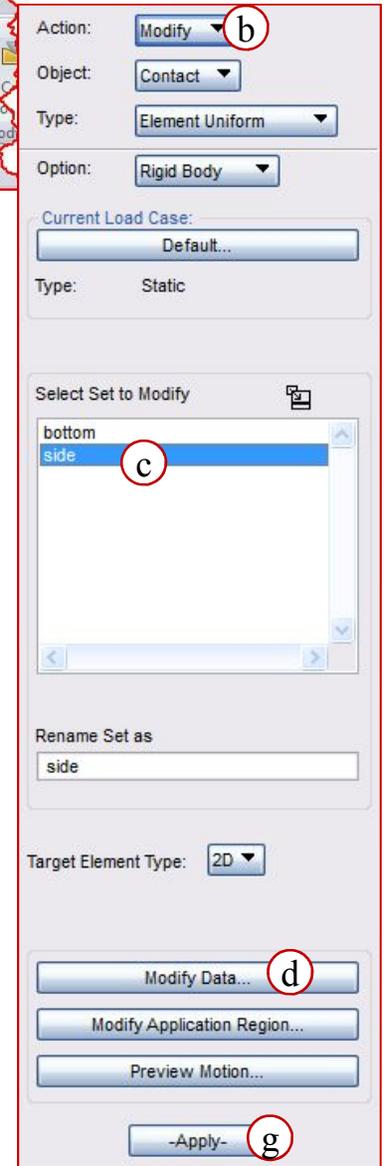
b error: check the position and velocity of contact body ID= 3

```
[User Information]:
During the approach stage of a CONTACT analysis, a rigid
body did not contact a deformable body within 1000 trials.
The time step per trial is chosen in such a way that the
displacement of the rigid body does not exceed 100 times
the error tolerance calculated by the Contact.
The initial distance between the rigid and deformable
contact bodies is probably too large or the rigid body
velocity has an incorrect direction.
=====
```

```
*** SYSTEM FATAL MESSAGE 8109 (MCN1INID)
      Error encountered while sending DISTRT data to the 3D Contact Component
0FATAL ERROR
```

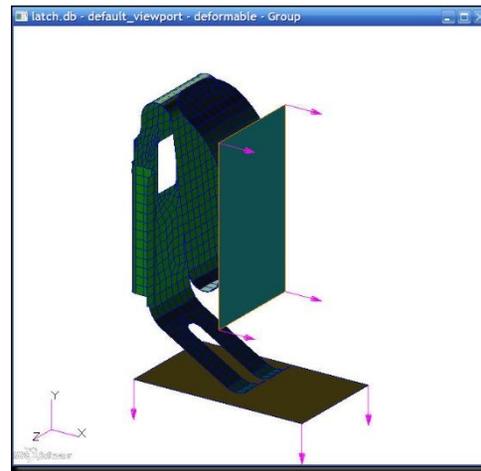
An error is encountered due to the direction of contact for the 'side' contact body.

Step 8. Correct the Direction of Contact

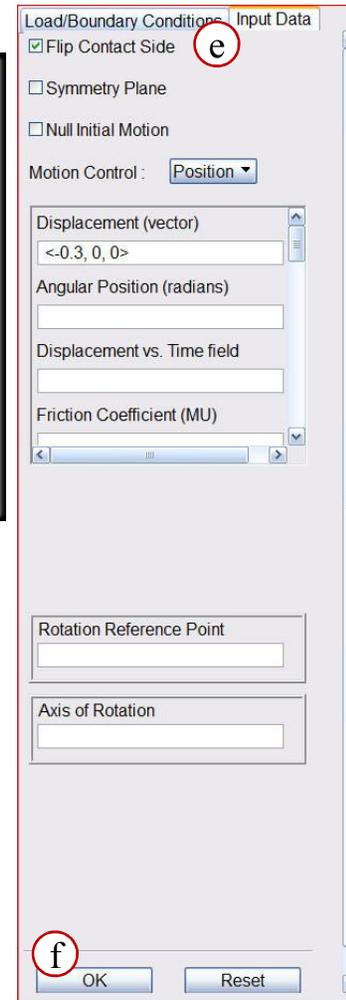


Reverse the contact side for the 'side' rigid body:

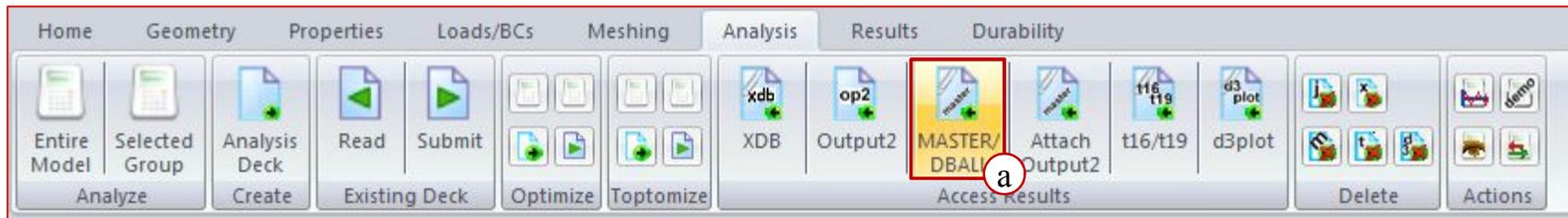
- Under the *Loads/BCs* tab, click **Rigid** in the *Contact Bodies* group.
- Pull down *Action* to **Modify**.
- Select the **side** contact body.
- Click **Modify Data**.
- Check **Flip Contact Side**.
- Click **OK**.
- Click **Apply**.



Note the normal vectors now point away from the clip

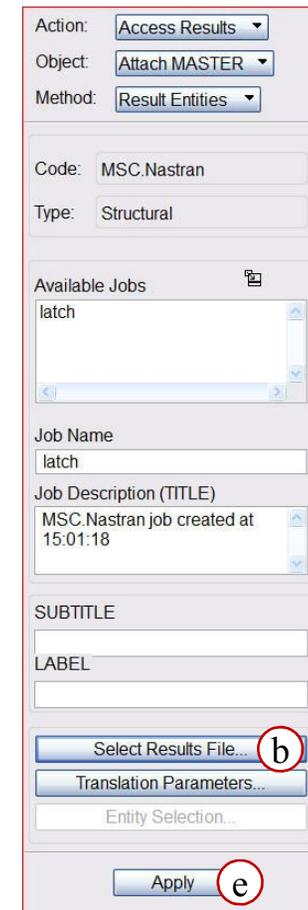
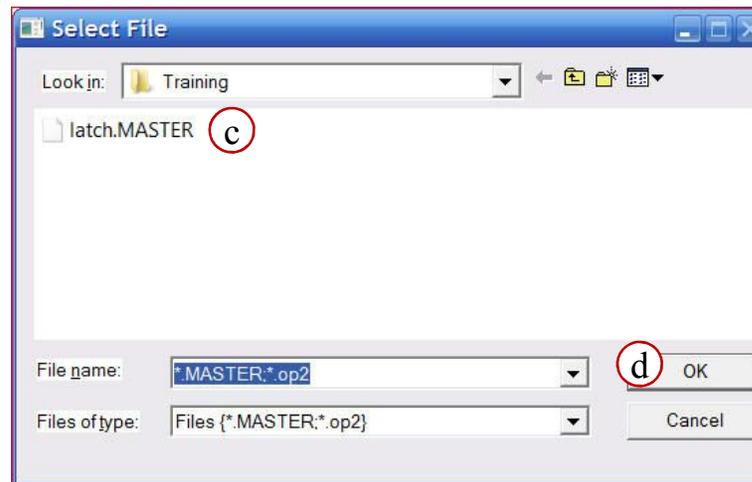


Step 9. Attach Master Result File

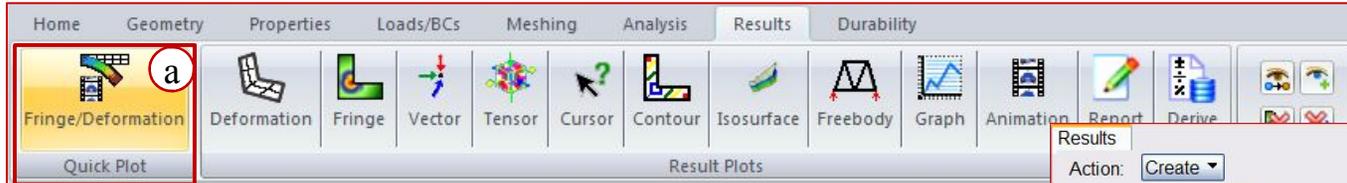


Attach the Master result file:

- Under the *Analysis* tab, click **Master/DBALL** in the *Access Results* group.
- Click **Select Results File...**
- Select the **latch** Master result file.
- Click **OK**.
- Click **Apply**.

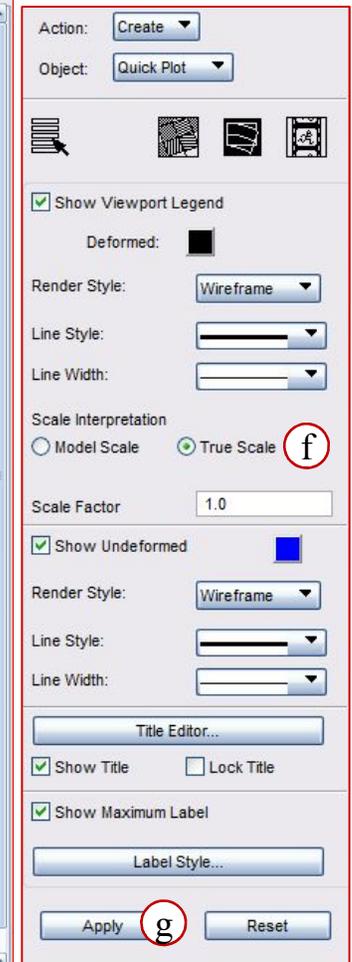
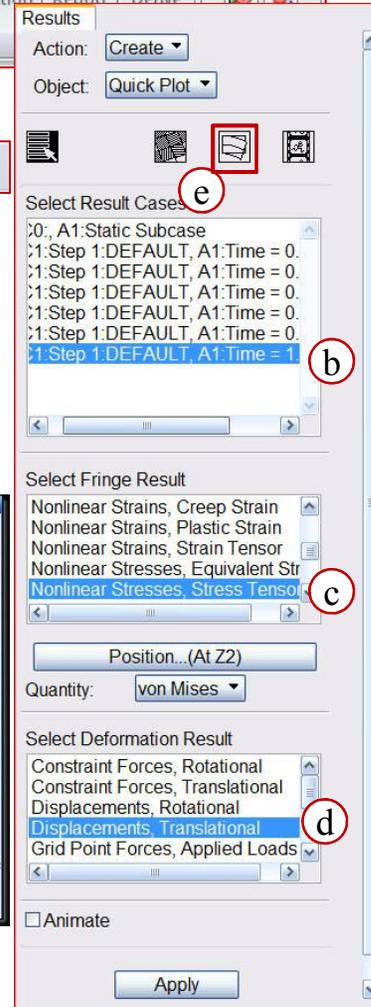
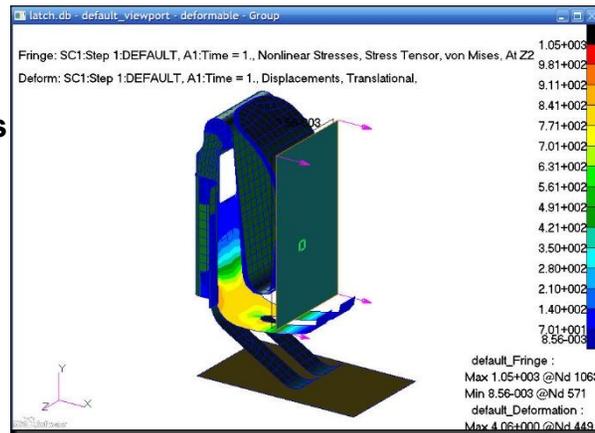


Step 10. Plot the Deformation and Stress Results



Plot deformation and von Mises stress:

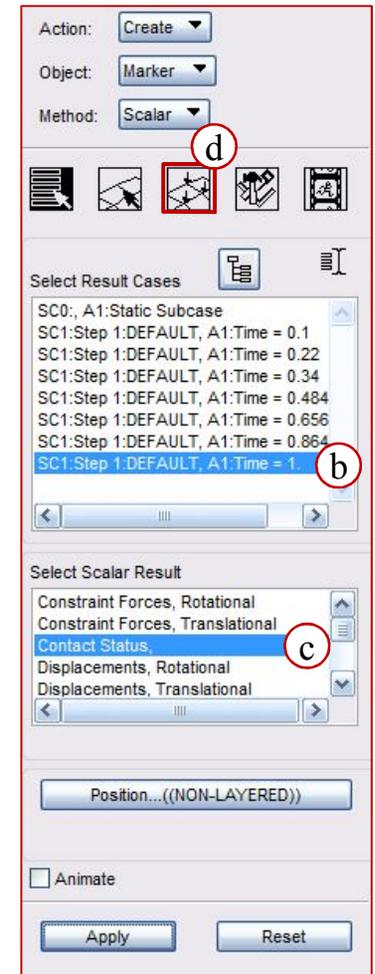
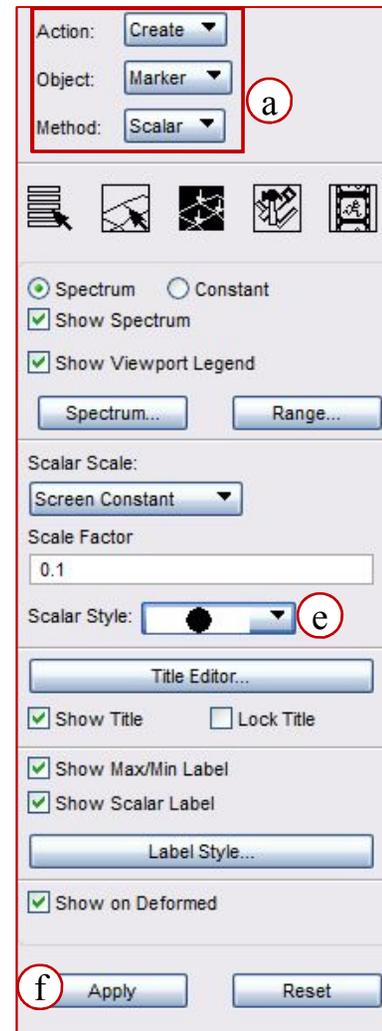
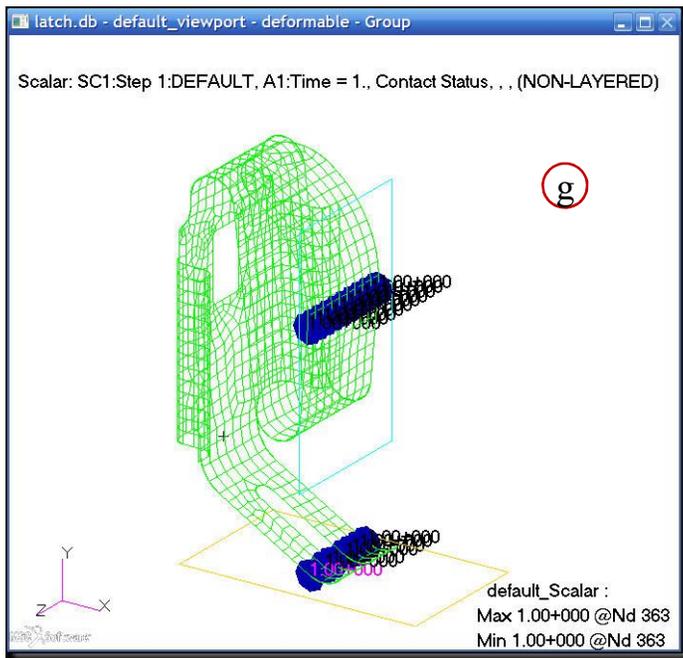
- Under the *Results* tab click **Fringe/Deformation** in the *Quick Plot* group.
- Select **SC1:...A1:Time = 1.0** for *Select Result Cases*.
- Select **Nonlinear Stresses, Stress Tensor** for *Select Fringe Result*.
- Select **Displacements, Translational** for *Select Deformation Result*.
- Click the **Deform Attributes** icon
- Select **True Scale**.
- Click **Apply**.
- When finished, click **Reset Graphics**.



Step 11. Creating Marker Plot of Contact Status

Create a Marker Plot of the Contact status:

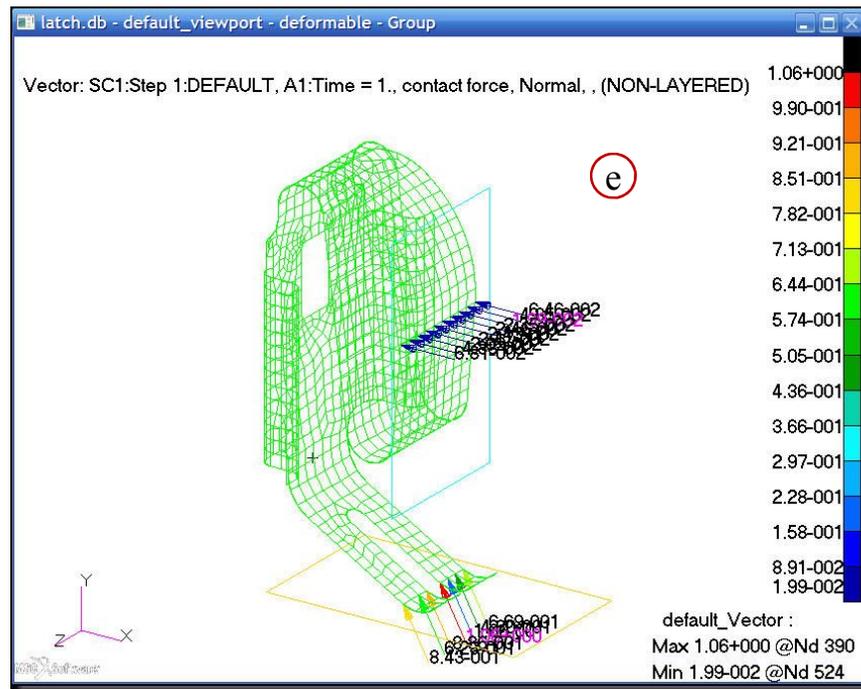
- Pull down **Create > Marker > Scalar**.
- Select **SC1:...A1:Time = 1.0** for *Select Result Cases*.
- Select **Contact Status** for the *Scalar Result*.
- Click the **Display Attributes** icon.
- Pull down *Scalar Style* and pick the **Solid Sphere**.
- Click **Apply**.
- The solid sphere markers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



Step 12. Plot Normal Contact Force

Plot the Normal Contact Force Vector for the clip:

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



Action: Create

Object: Marker

Method: Vector **a**

Select Result Cases

SC0.: A1:Static Subcase
 SC1:Step 1:DEFAULT, A1:Time = 0.1
 SC1:Step 1:DEFAULT, A1:Time = 0.22
 SC1:Step 1:DEFAULT, A1:Time = 0.34
 SC1:Step 1:DEFAULT, A1:Time = 0.484
 SC1:Step 1:DEFAULT, A1:Time = 0.656
 SC1:Step 1:DEFAULT, A1:Time = 0.864
 SC1:Step 1:DEFAULT, A1:Time = 1. **b**

Select Vector Result

Nonlinear Strains, Strain Tensor
 Nonlinear Stresses, Stress Tensor
 contact force, Friction
 contact force, Normal **b**

Show As: Resultant **c**

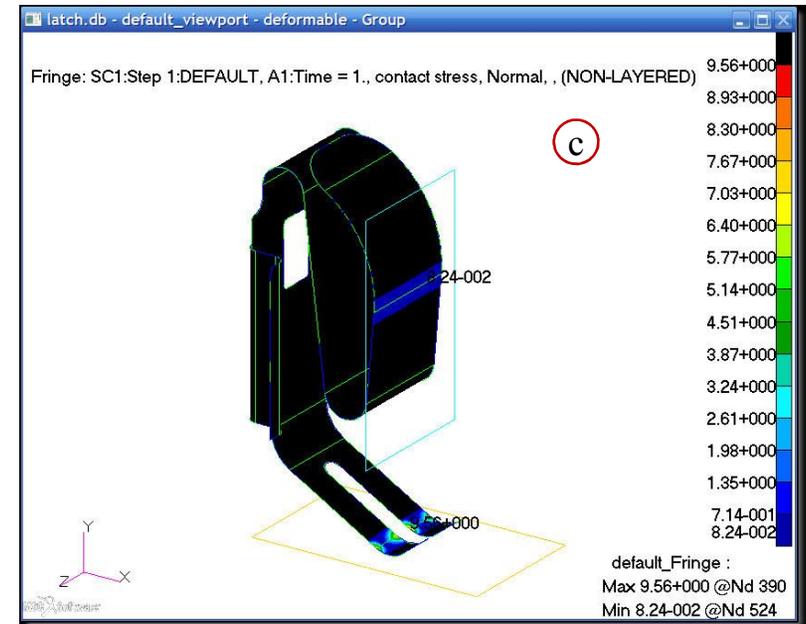
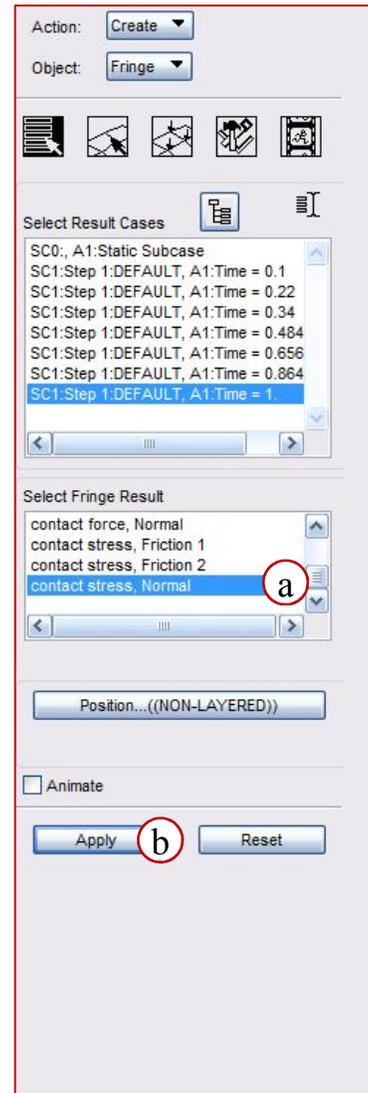
Animate **e**

Apply **d** Reset

Step 14. Display a Fringe Plot of Normal Contact Stress

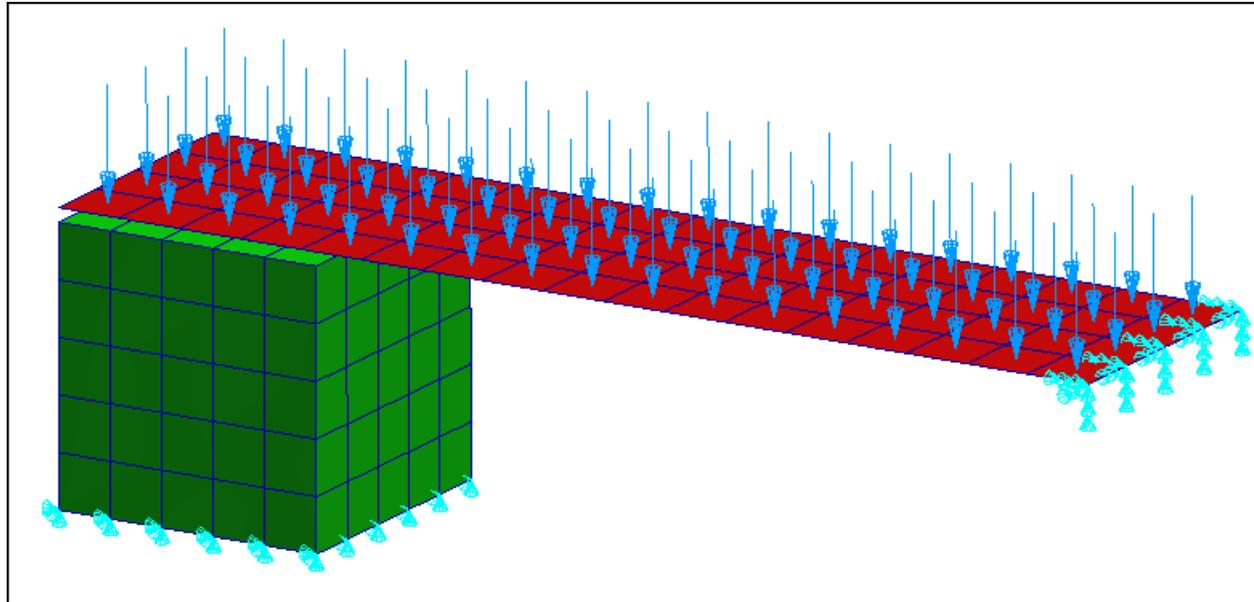
Display a fringe plot of Normal Contact Stress on the clip:

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



WORKSHOP 2

SOLID-TO-SHELL CONTACT



- **Workshop Objectives**

- Perform a start to finish solid to shell contact analysis with MSC Nastran
- Gain familiarity with the Patran contact analysis GUI

- **Software Version**

- MSC Nastran 2013.1
- Patran 2013

- **Files Required**

- **solid_shell_contact_start.ses**

• Suggested Steps

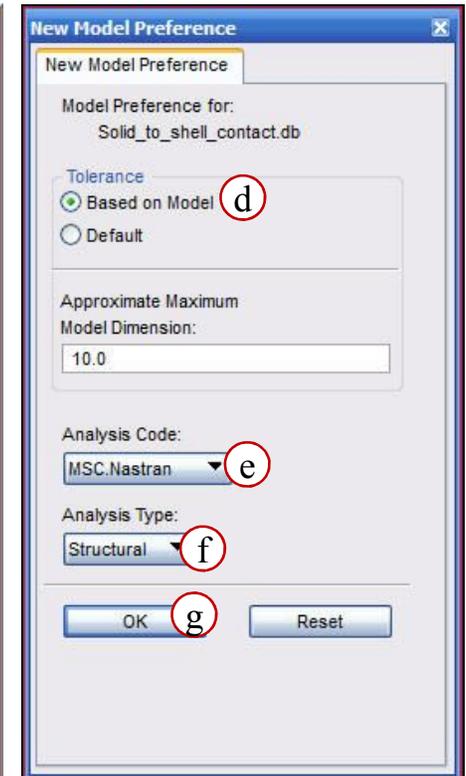
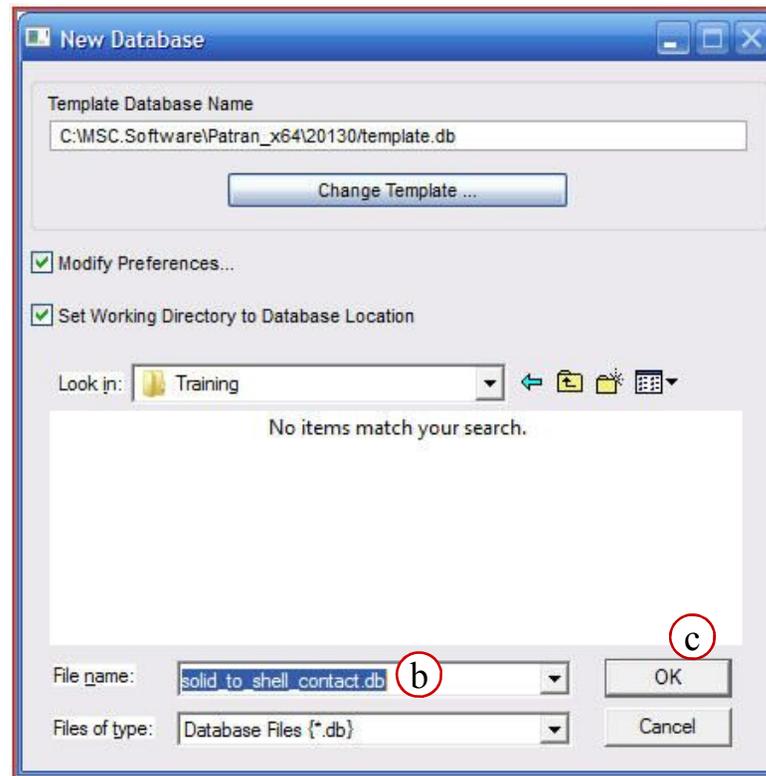
1. Create a new database called **solid_to_shell_contact.db**
2. Play the session file **solid_shell_contact_start.ses**. Review the model.
3. Define two deformable contact bodies using the Contact Bodies/Pairs Tool, one for the 3D solid and one for the 2D surface.
4. Create a body pair between the two deformable bodies and select automatic contact detection
5. Set up and run the analysis.
 - Set up a linear static analysis, SOL 101
 - Request MASTER output file format
 - Request contact results
6. View the MSC Nastran f06 result file to verify the analysis has run to completion.
7. Attach the MASTER result file in Patran.
8. Review the contact analysis results on the entire model and on the individual contact bodies.
 - Create fringe plots of von Mises and Normal Contact Stress
 - Create a vector plot of Normal Contact Force
 - Create a marker plot of Contact Status

Step 1. Create a New Database



Create a new database

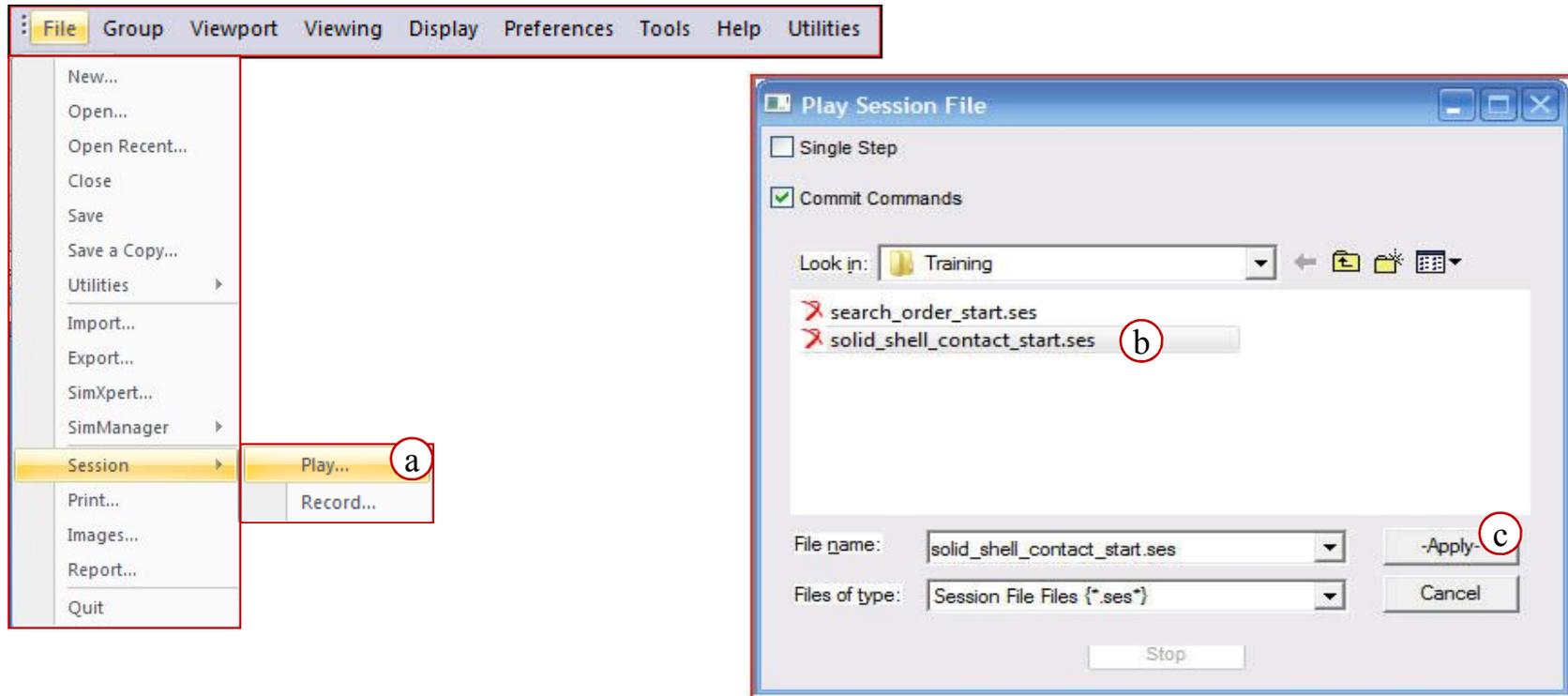
- a. Under the *Home* tab, click **New** in the *Defaults* group.
- b. Enter **solid_to_shell_contact** as the *File name*.
- c. Click **OK**.
- d. Select **Based on Model** for *Tolerance*.
- e. Pull down **MSC Nastran** as the *Analysis Code*.
- f. Pull down **Structural** as the *Analysis Type*.
- g. Click **OK**.



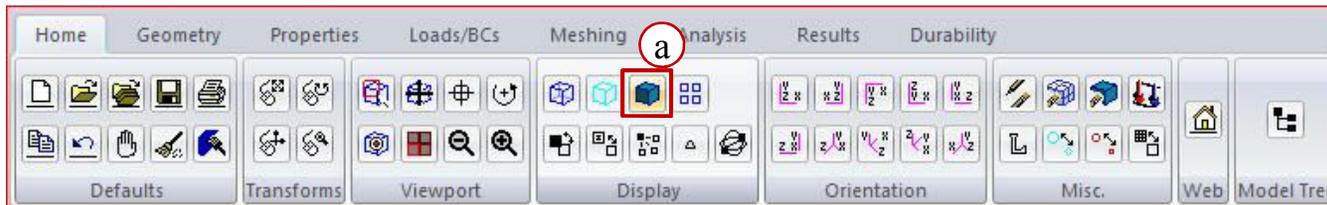
Step 2. Play the Session File

Play session file *solid_shell_contact_start.ses* to build the model.

- a. Pull down **File > Session > Play...**
- b. Select **solid_shell_contact_start.ses**
- c. Click **Apply** to play the session file.

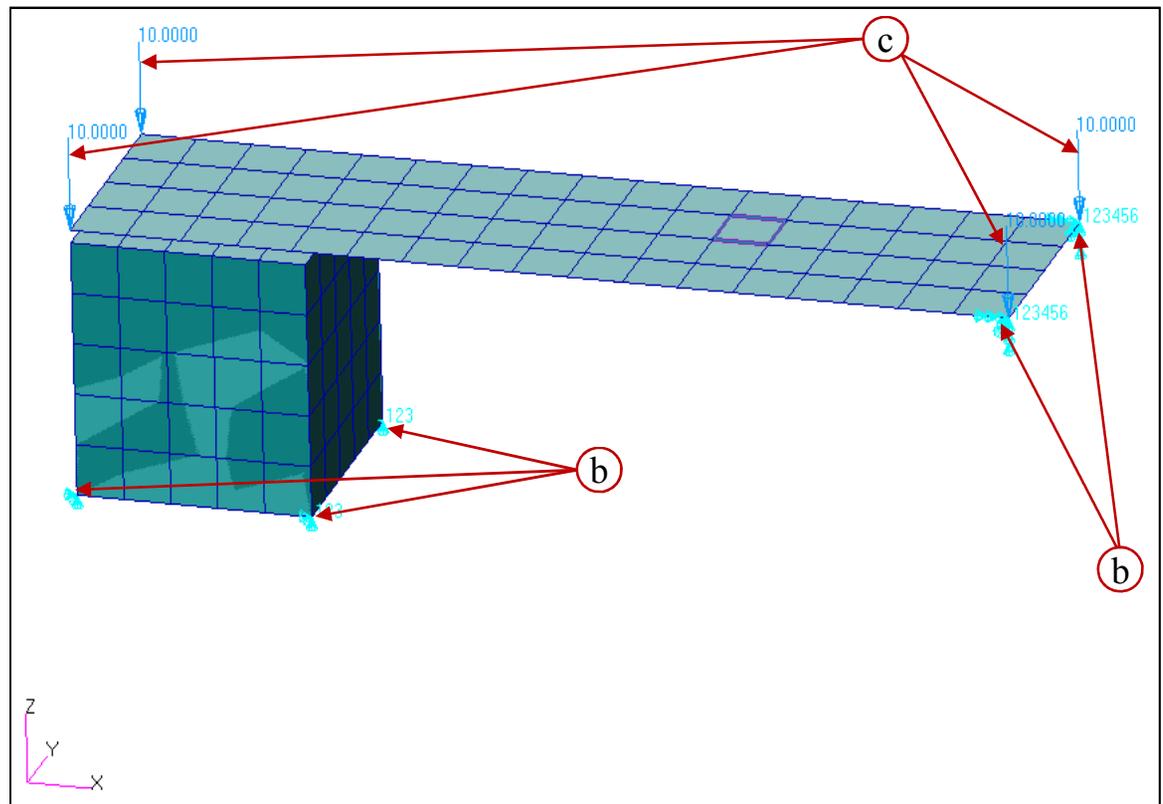


Step 2. Play the Session File (Cont.)



Review the model.

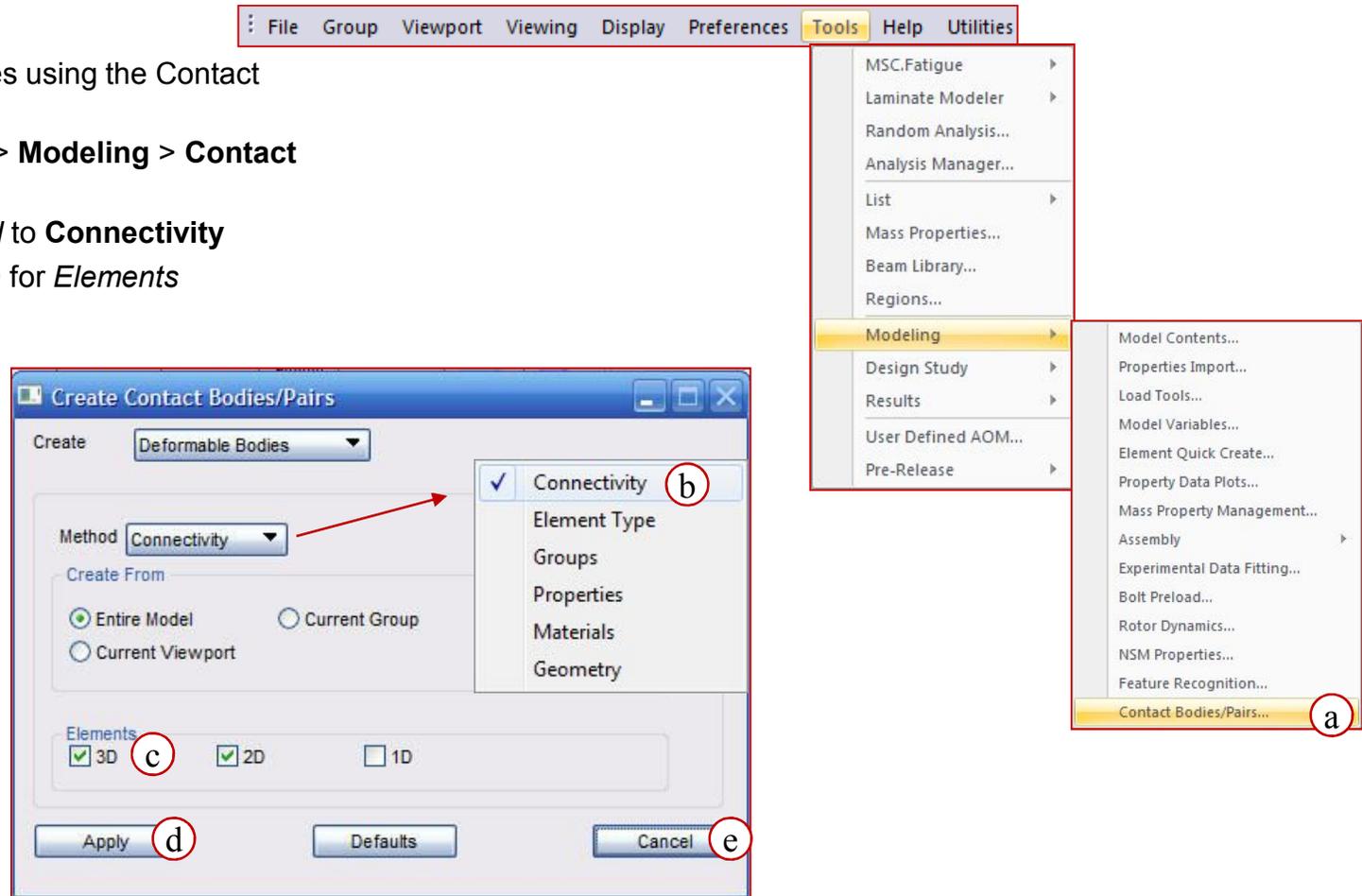
- a. Under the *Home* tab, click **Smooth Shaded** in the *Display* group.
- b. Review boundary conditions:
 - The solid block is fixed at the base.
 - The plate is fixed at the right edge.
- c. Review loading: A total load of 10 lbs is applied to the plate.



Step 3. Define Two Deformable Contact Bodies

Define Contact Bodies using the Contact Bodies/Pair Tool:

- Pull down **Tools > Modeling > Contact Bodies/Pair**
- Pull down *Method* to **Connectivity**
- Check **3D** and **2D** for *Elements*
- Click **Apply**
- Click **Cancel**

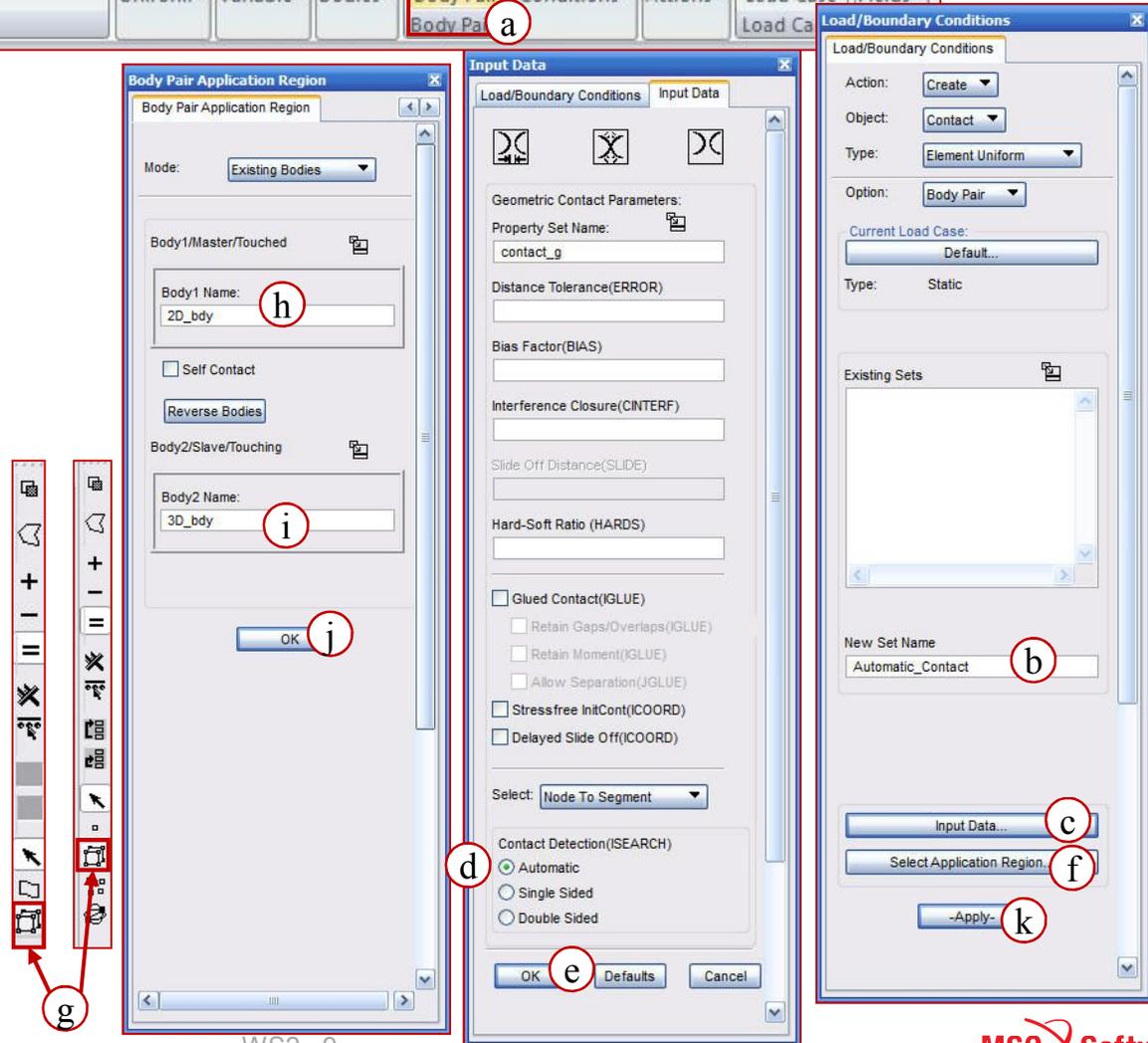


Step 4. Create Body Pair

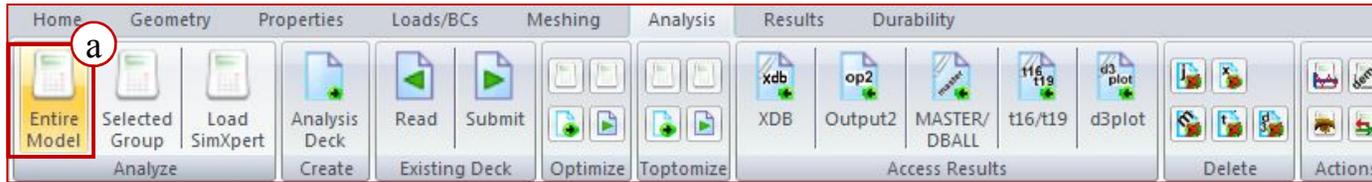


Create a body pair and select automatic contact detection:

- a. Under the *Load/BCs* tab click **Create Body Pair** in the *Body Pairs* group.
- b. Enter **Automatic_Contact** for *New Set Name*.
- c. Click **Input Data...**
- d. Select **Automatic** for *Contact Detection*
- e. Click **OK**
- f. Click **Select Application Region**
- g. Click **FEM entity** and then **Element** on the *Picking Filters* toolbar
- h. Click in the *Body1 Name* box and select any element in **2D_bdy** (the plate).
- i. Repeat steps g and h for *Body2 Name* this time selecting an element in **3D_bdy** (the block).
- j. Click **OK**
- k. Click **Apply**

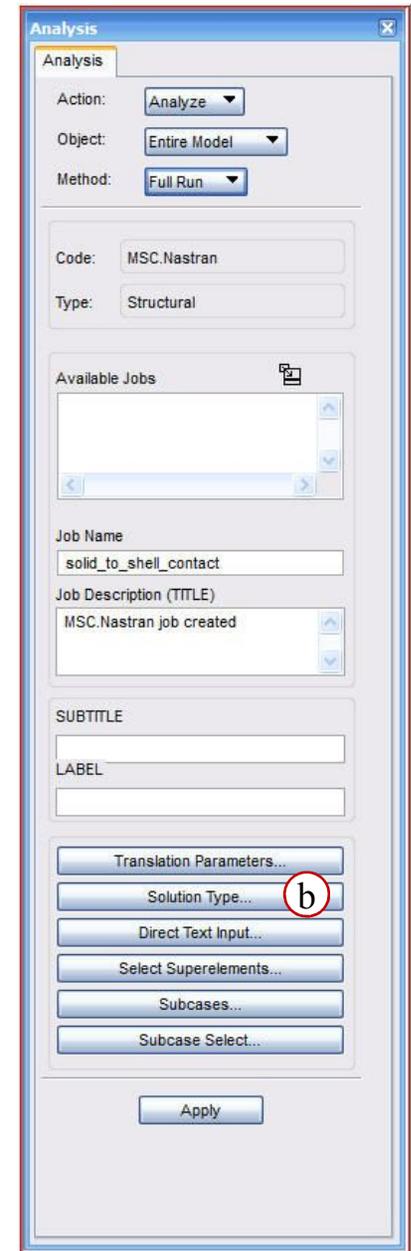
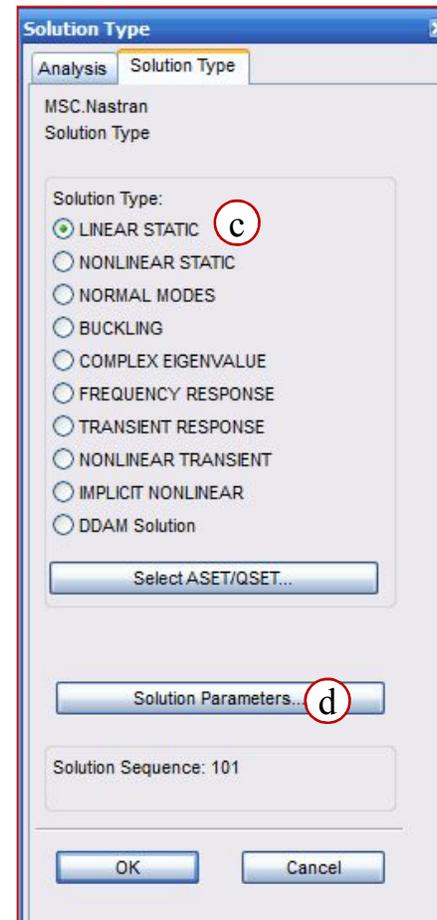


Step 5. Set Up and Run the Analysis



Set up a linear static analysis.

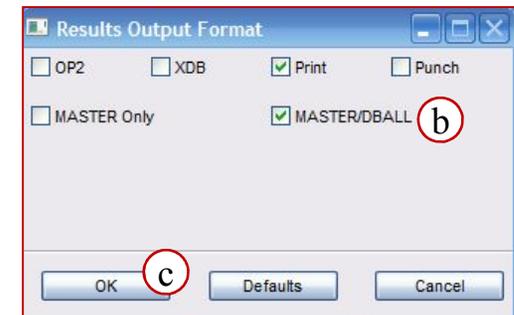
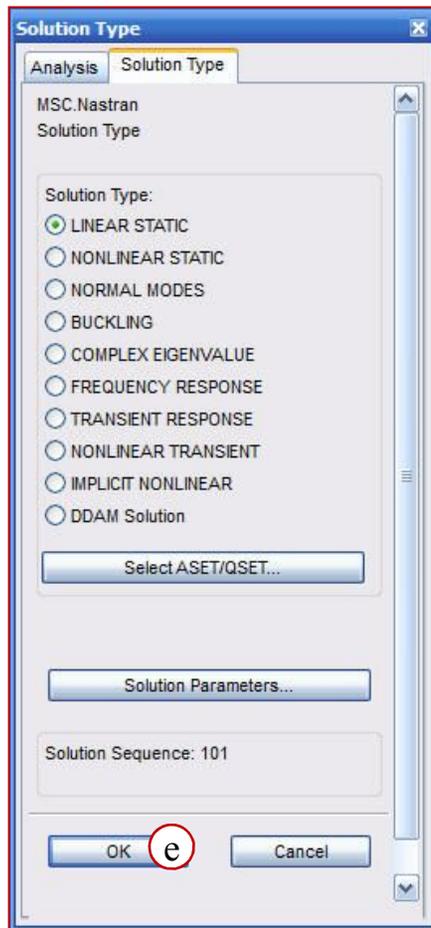
- a. Under the *Analysis* tab, click **Entire Model** in the *Analyze* group.
- b. Click **Solution Type....**
- c. Select **LINEAR STATIC**.
- d. Click **Solution Parameters.**



Step 5. Set Up and Run the Analysis (Cont.)

Request MASTER results output format to capture contact results.

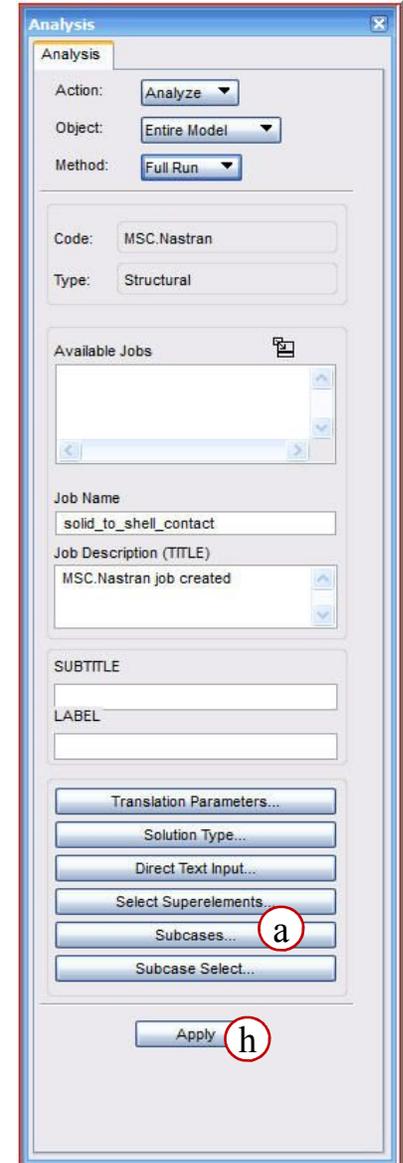
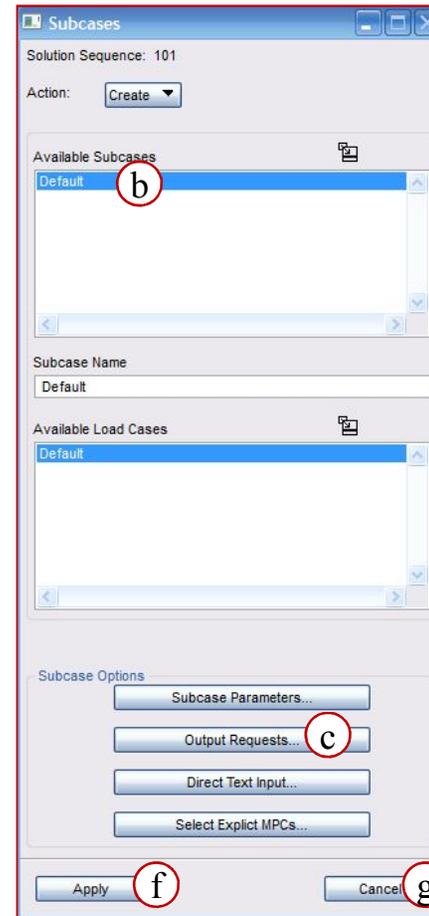
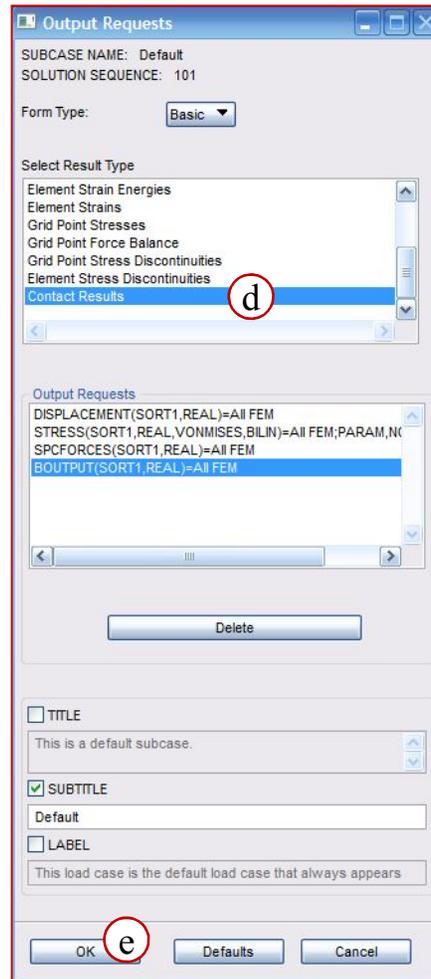
- a. Click **Results Output Format**.
- b. Check **MASTER/DBALL** and uncheck **XDB**.
- c. Click **OK**.
- d. Click **OK**.
- e. Click **OK**.



Step 5. Set Up and Run the Analysis (Cont.)

Request contact results.

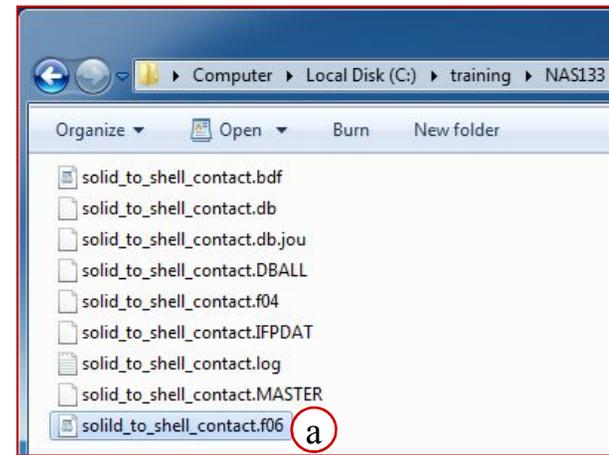
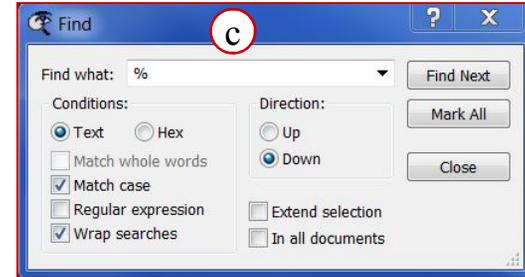
- a. Click **Subcases...**
- b. Highlight **Default** in *Available Subcases*.
- c. Click **Output Requests**.
- d. Scroll down and highlight **Contact Results**.
- e. Click **OK**.
- f. Click **Apply**.
- g. Click **Cancel**.
- h. Click **Apply** to submit the analysis.



Step 6. View the Result File

After the analysis runs to completion, review the .f06 results file

- a. Locate the file **solid_to_shell_contact.f06**
- b. Open **solid_to_shell_contact.f06** with any text editor
- c. Search for the string %.
- d. Scroll down the load steps, note that Load Step 1.0000 has been reached. Then close the .f06 file.



```

0      (b)  N O N - L I N E A R   I T E R A T I O N   M O D U L E   O U T P U T
      STIFFNESS UPDATE TIME-1892160000.00 SECONDS          SUBCASE      1          STEP      0
      ITERATION TIME          0.02 SECONDS

      LOAD NO.      - - ERROR FACTORS - -      CONV  ITR MAT NO.      AVG      TOTL      - - - - - DISP - - - - - LINE_S NO. TOT TOT
      STEP INC ITR  DISP      LOAD      WORK      RATE  DIV DIV BIS  R_FORCE  WORK      AVG      MAX      AT GRID C  FACT NO  QNV  KUD  ITR
%1.00000E-01  1  1  1.00E+00  1.24E-13  1.00E+00  1.000  0  1  0  1.77E-07  2.388E-05  6.61E-06  -1.047E-04      227 3  1.00 0  0  0  1
*** SYSTEM INFORMATION MESSAGE 8137 (NL3CON)
      SEPARATION condition has been detected. Additional iterations will be performed.
%1.00000E-01  1  2  5.97E-05  8.02E-14  4.66E-05  0.564  0  1  0  2.16E-09  2.388E-05  6.61E-06  -1.047E-04      227 3  1.00 0  0  0  2 (d)
*** SYSTEM INFORMATION MESSAGE 8137 (NL3CON)
      SEPARATION condition has been detected. Additional iterations will be performed.
%1.00000E-01  1  3  1.44E-01  1.22E-13  7.53E-03  0.599  0  1  0  1.76E-15  2.424E-05  6.90E-06  -1.070E-04      227 3  1.00 0  0  0  3
%1.00000E-01  1  4  8.72E-02  7.90E-14  3.38E-03  0.599  0  1  0  4.69E-12  2.440E-05  6.89E-06  -1.094E-04      227 3  1.00 0  0  0  4
    
```

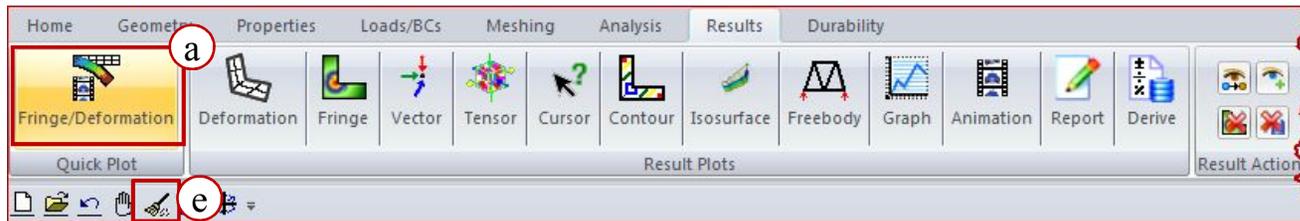
Step 7. Attach the Result File

The screenshot shows the Patran software interface. The 'Access Results' group in the 'Results' tab is highlighted, with the 'MASTER/DBALL' file icon circled in red and labeled 'a'. The 'Analysis' dialog box is open, showing the 'Available Jobs' list with 'solid_to_shell_contact' selected, circled in red and labeled 'b'. The 'Apply' button at the bottom of the dialog is circled in red and labeled 'c'.

Attach the MASTER results file in Patran.

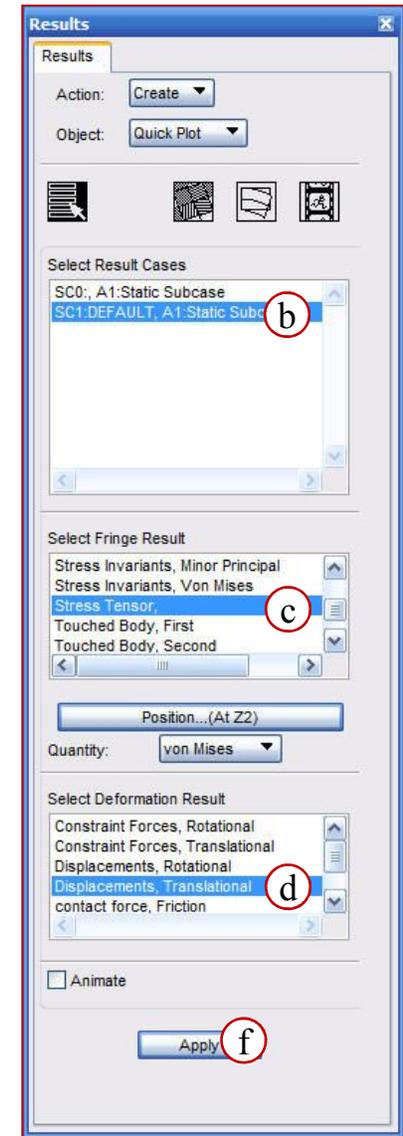
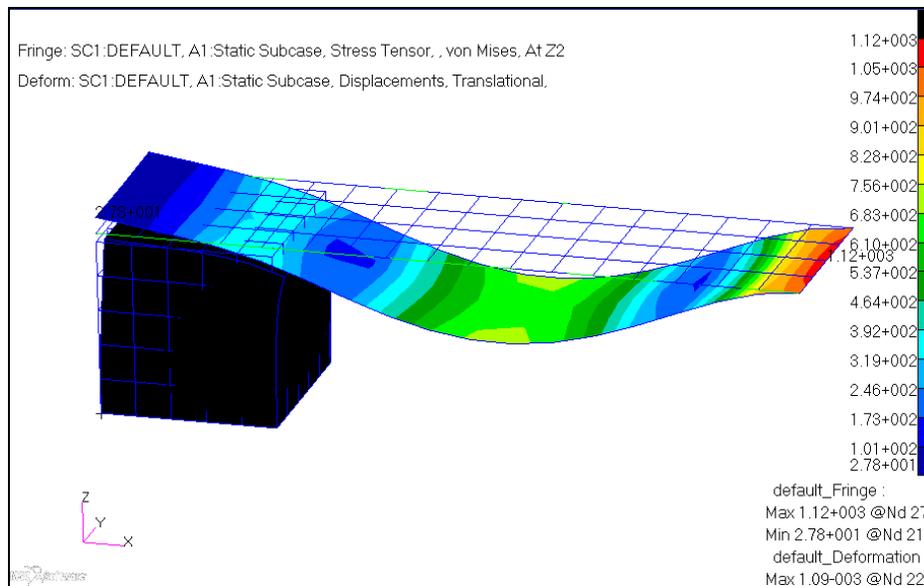
- Click **MASTER / DBALL** in the *Access Results* group.
- Select **solid_to_shell_contact** from *Available Jobs*.
- Click **Apply**

Step 8. Review Analysis Results



Plot deformation and shell von Mises stress

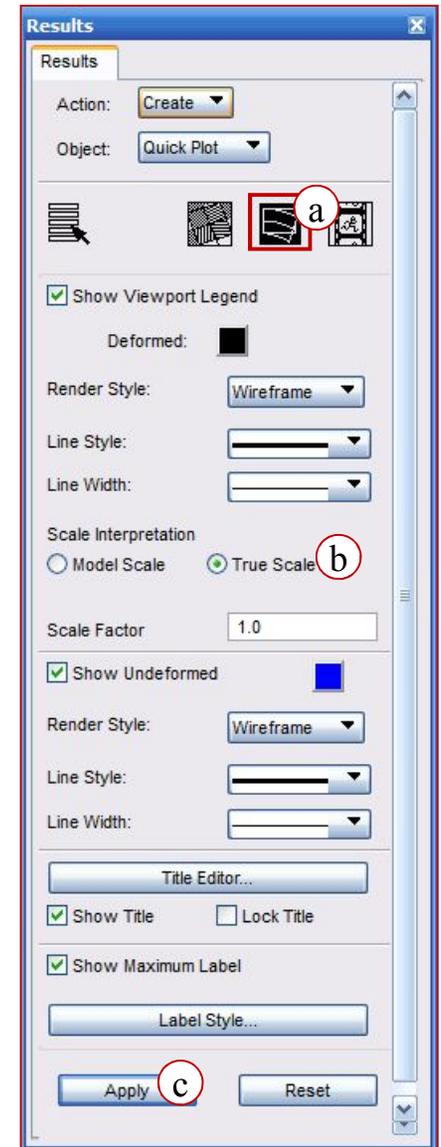
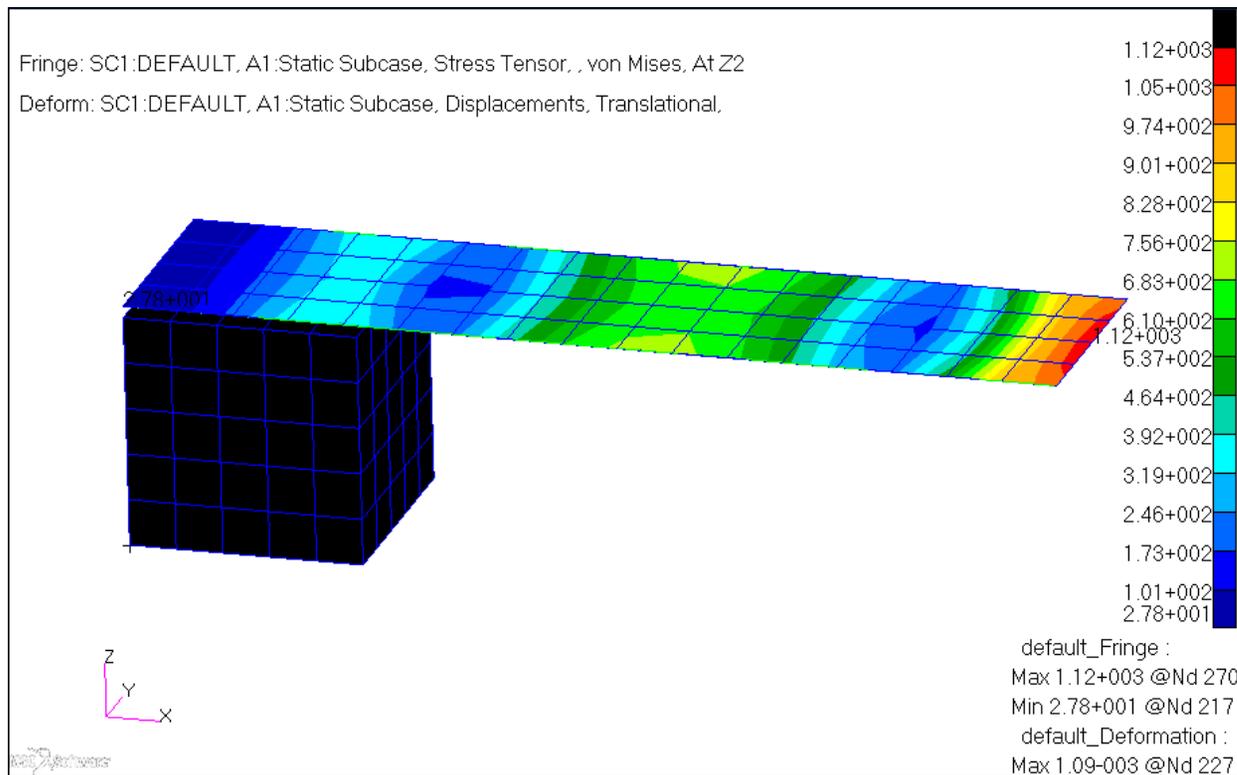
- Under the *Results* tab, click **Fringe/Deformation** in the *Quick Plot* group.
- Select **SC1:DEFAULT, A1:Static Subcase** for *Select Result Cases*.
- Highlight **Stress Tensor** for *Fringe Result*.
- Highlight **Displacements, Translational** for *Deformation Result*.
- Click **Reset Graphics** to clear the shading.
- Click **Apply**.



Step 8. Review Analysis Results (Cont.)

Set deformation scale to true scale.

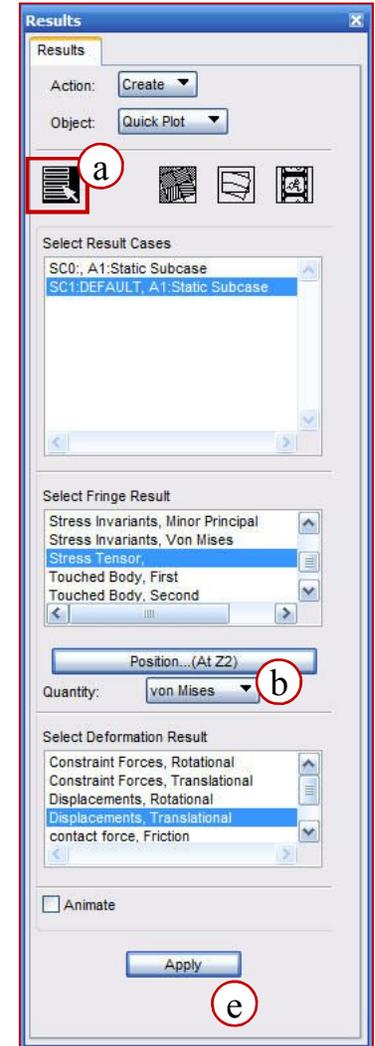
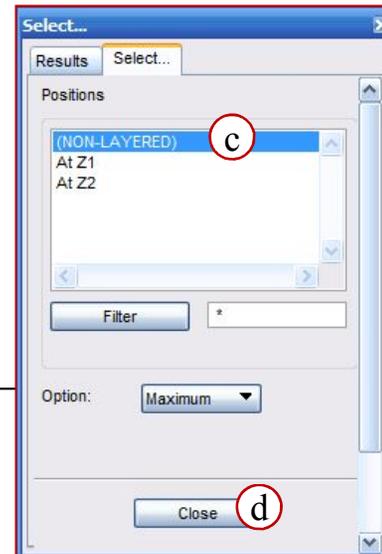
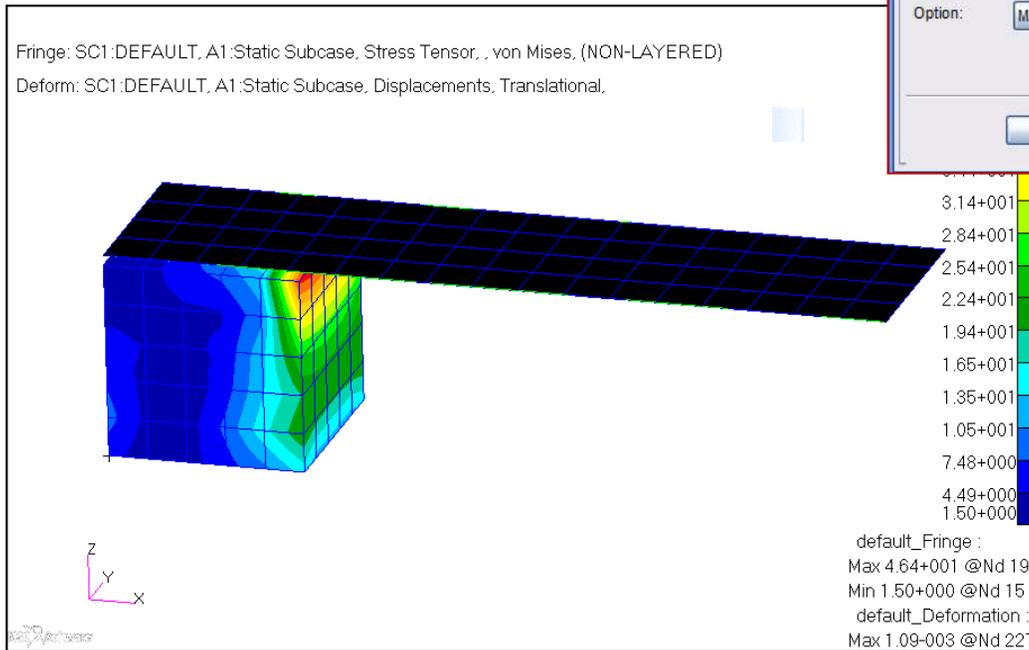
- Click **Deform Attributes**.
- Select **True Scale**.
- Click **Apply**.



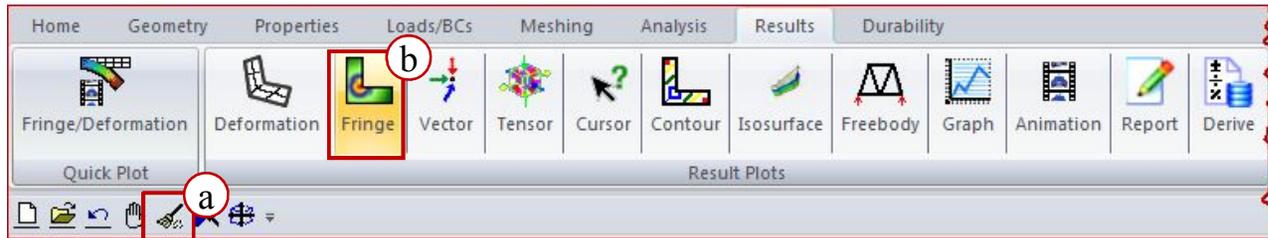
Step 8. Review Analysis Results (Cont.)

Plot solid von Mises stresses

- a. Click **Select Results**.
- b. Click **Position...(At Z2)**.
- c. Highlight **NON-LAYERED**.
- d. Click **Close**.
- e. Click **Apply**.

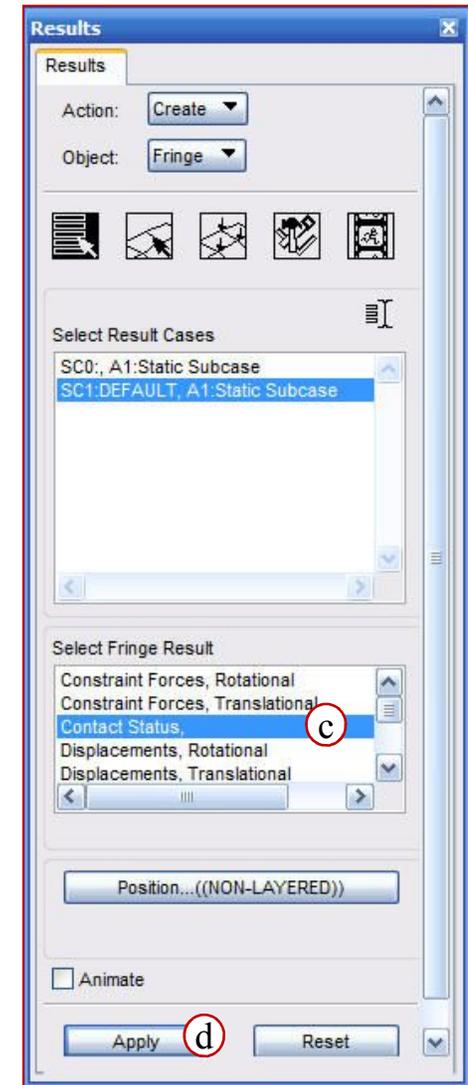
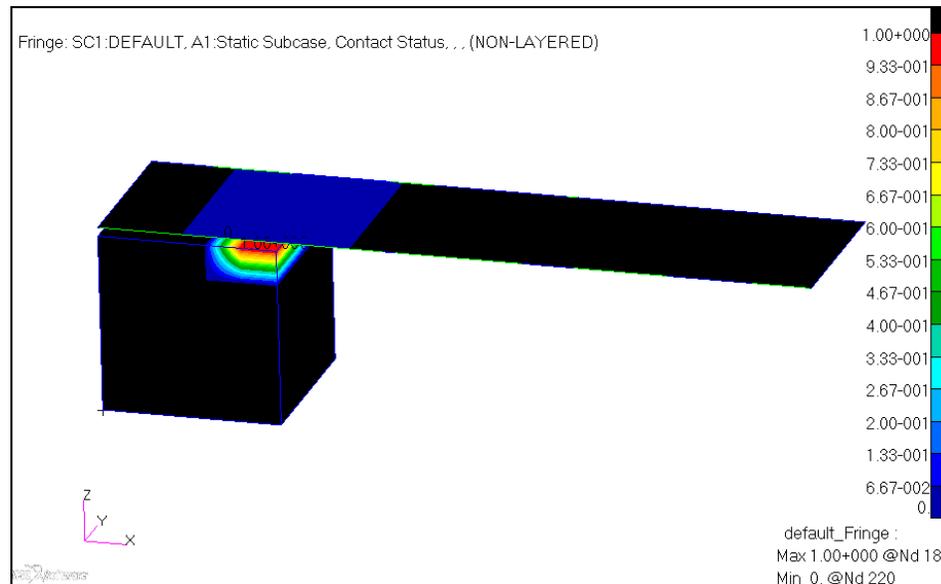


Step 8. Review Analysis Results (Cont.)



Plot contact status of both contact bodies

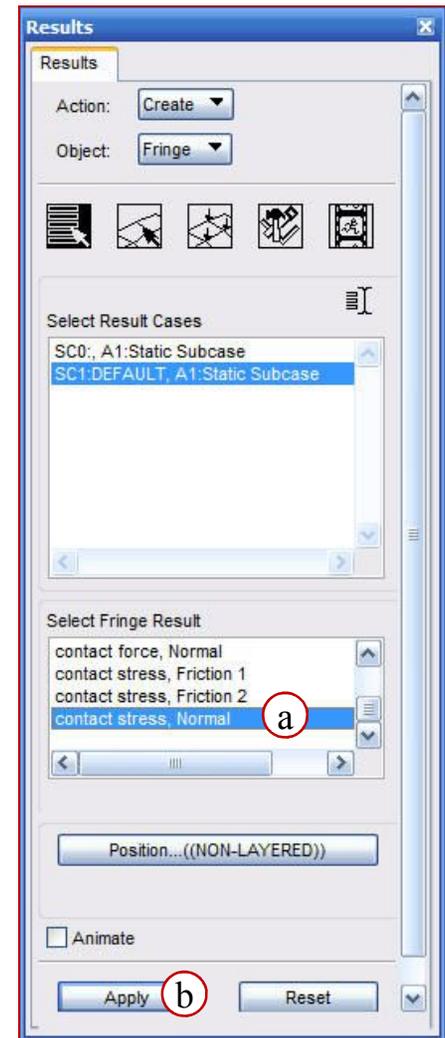
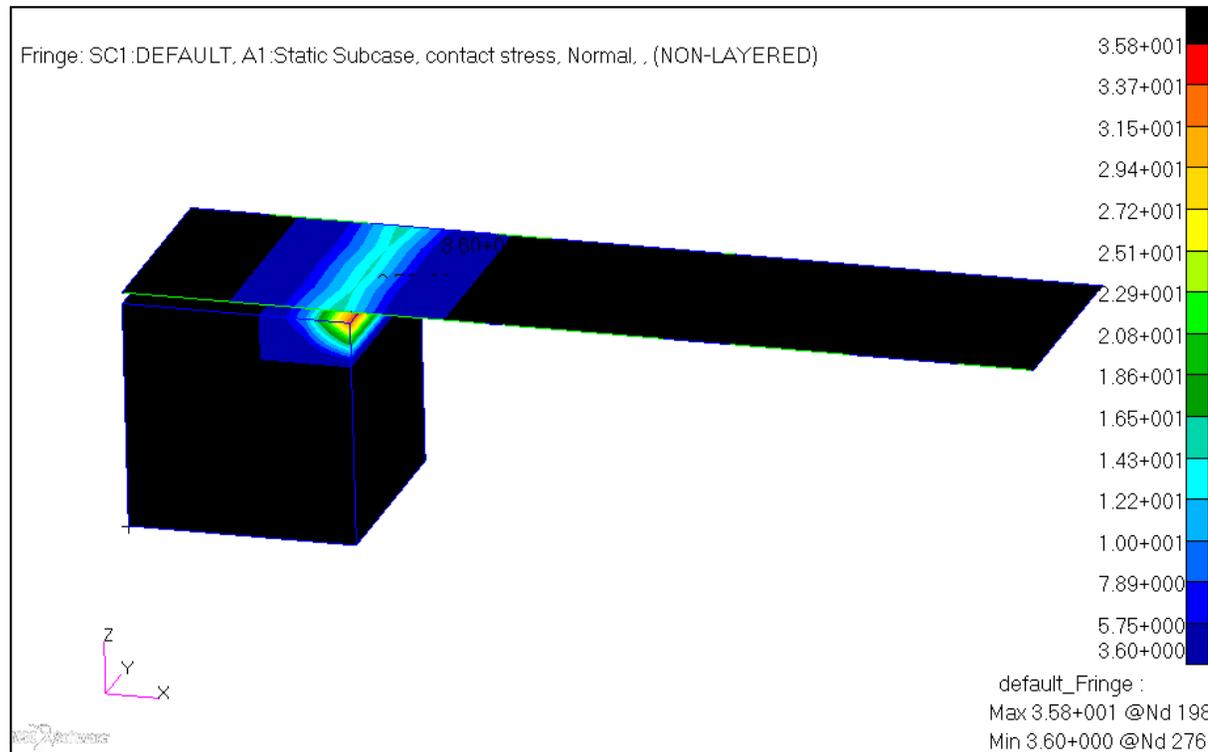
- a. Click **Reset Graphics**.
- b. Click **Fringe** in the *Results Plots* group.
- c. Highlight **Contact Status** under *Select Fringe Result*.
- d. Click **Apply**.



Step 8. Review Analysis Results (Cont.)

Plot contact normal stress on both contact bodies

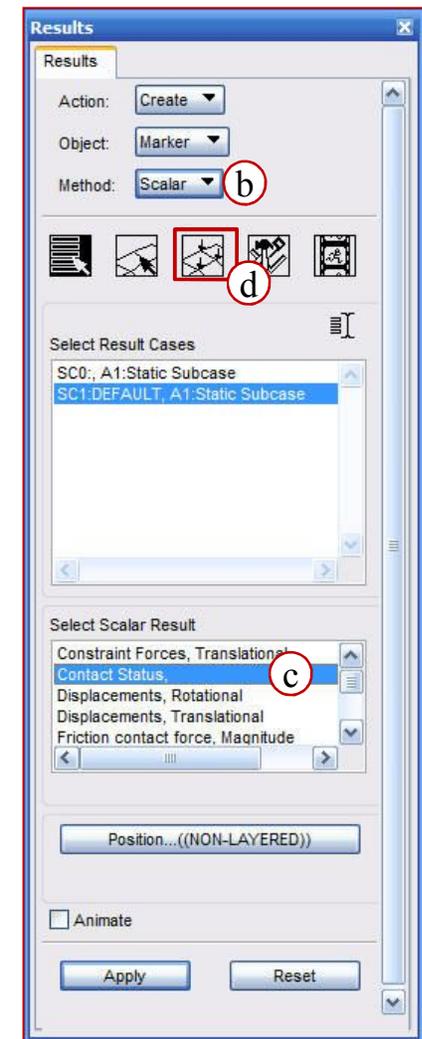
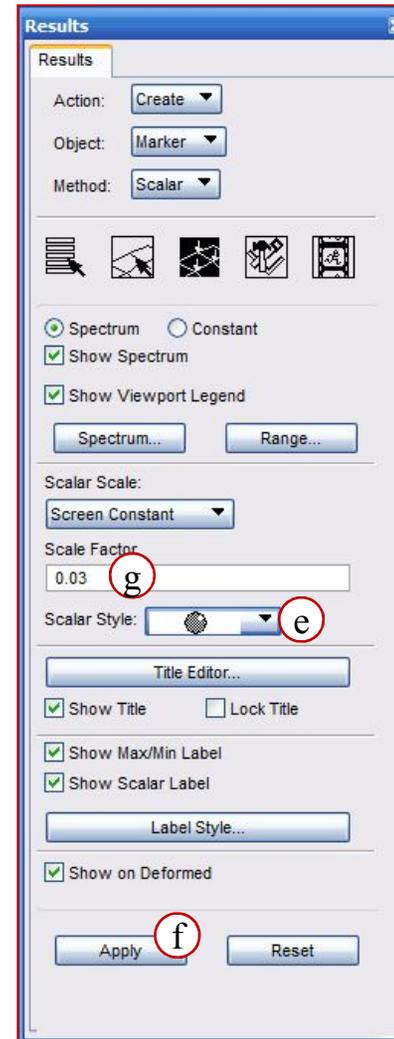
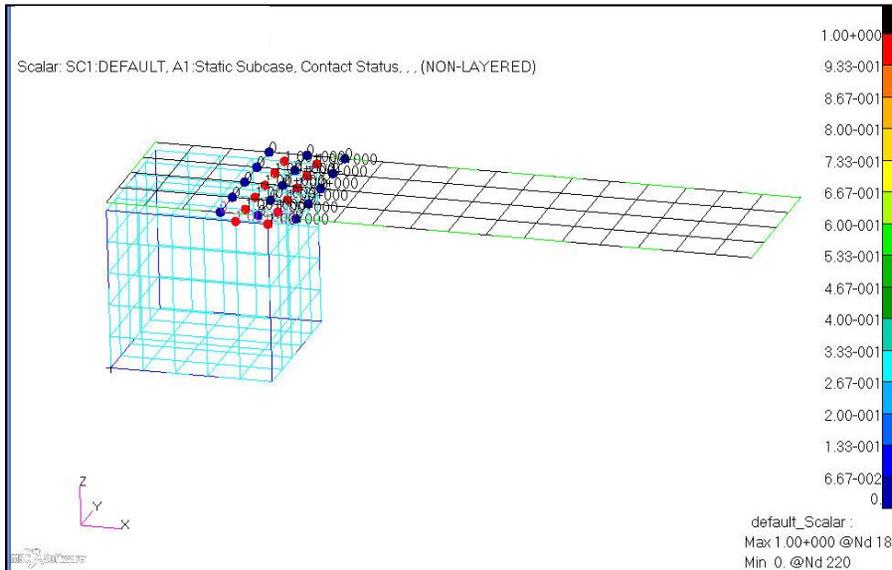
- Highlight **contact stress, Normal** under *Select Fringe Result*.
- Click **Apply**.



Step 8. Review Analysis Results (Cont.)

Plot contact status on both contact bodies

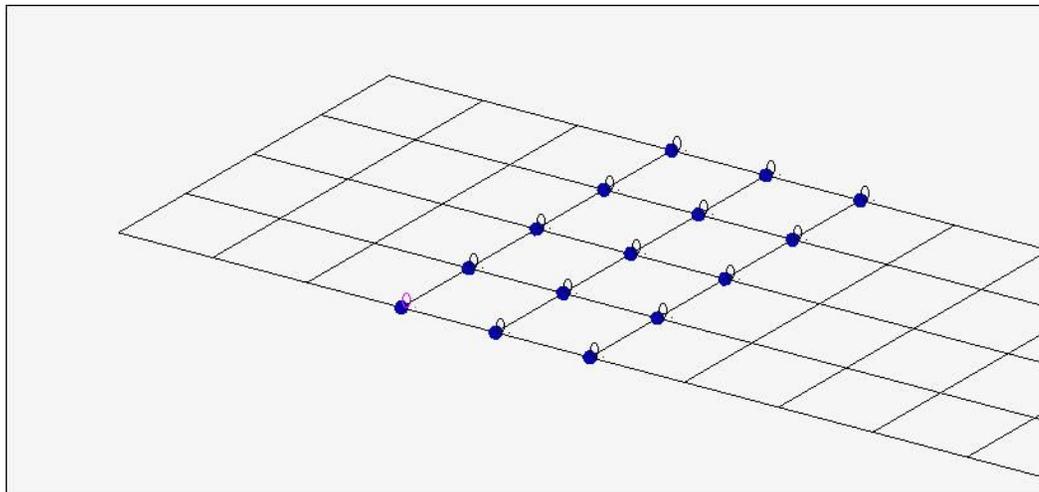
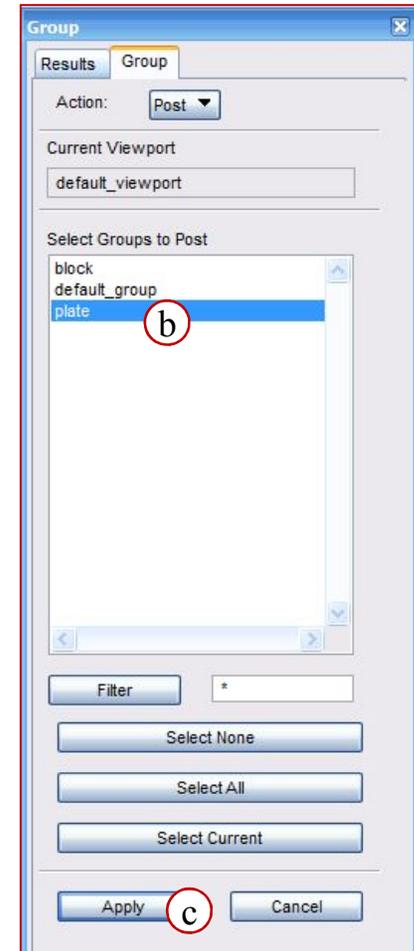
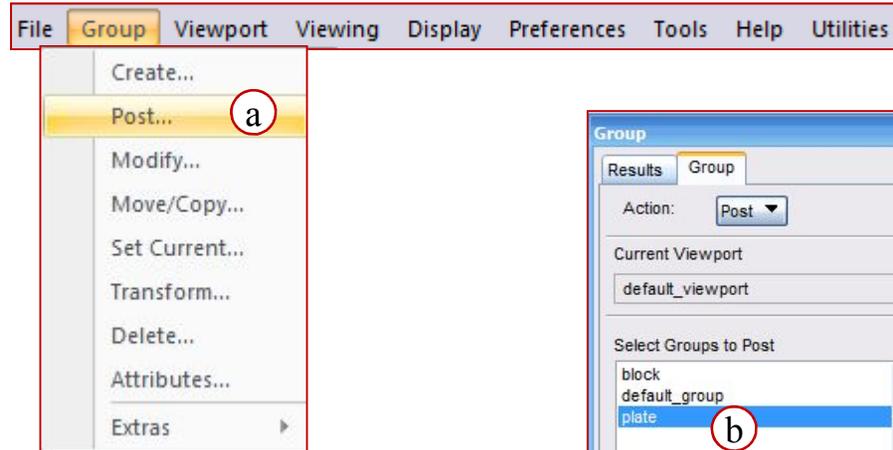
- Click **Reset Graphics**.
- Pull down **Scalar** for *Method*.
- Highlight **Contact Status**.
- Click **Display Attributes**.
- Pull down *Scalar Style* to the solid sphere (or choose your own).
- Click **Apply**.
- Adjust the *Scale Factor* as desired and click **Apply** again.



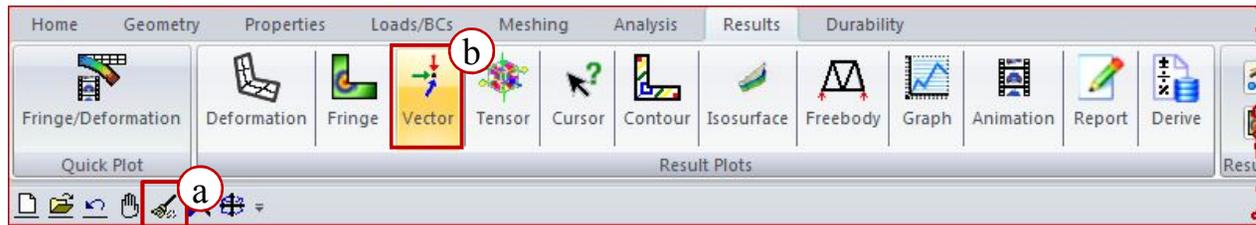
Step 8. Review Analysis Results (Cont.)

Examine the contact bodies individually.
First post the plate.

- a. Pull down **Group** > **Post**.
- b. Highlight **plate**
- c. Click **Apply**.

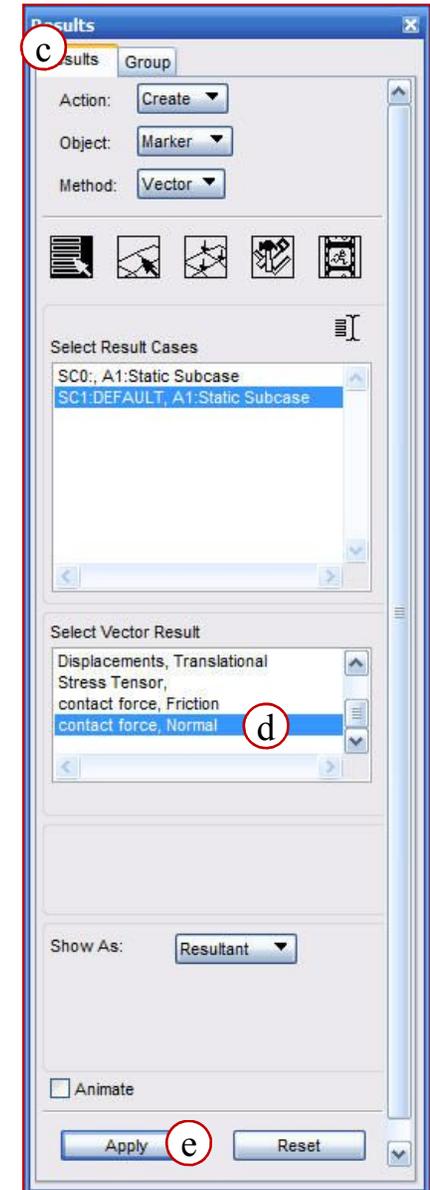
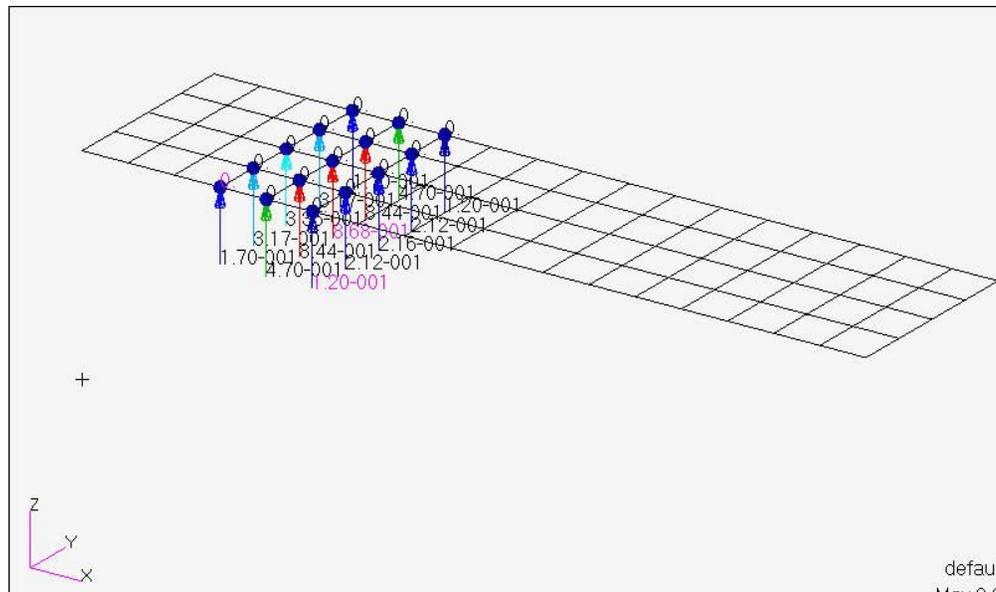


Step 8. Review Analysis Results (Cont.)



Examine the plate contact body

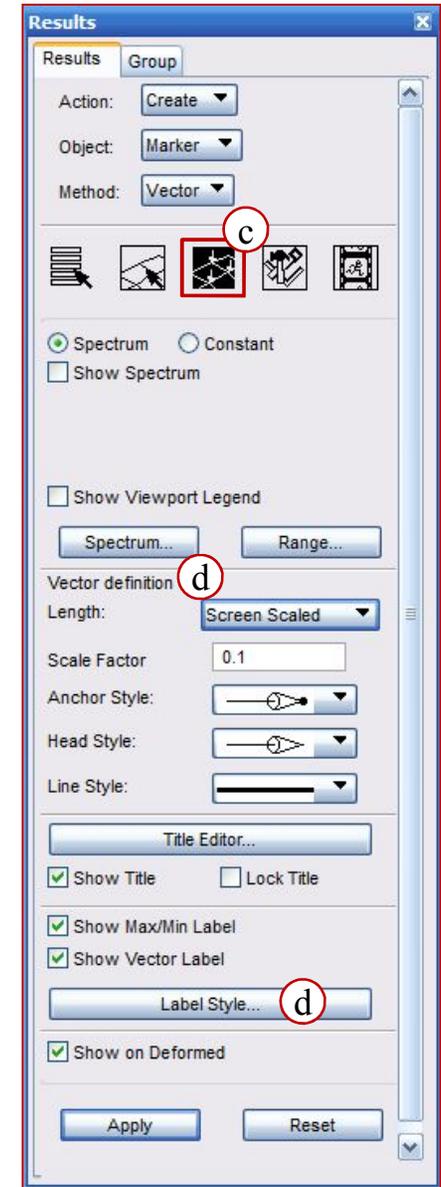
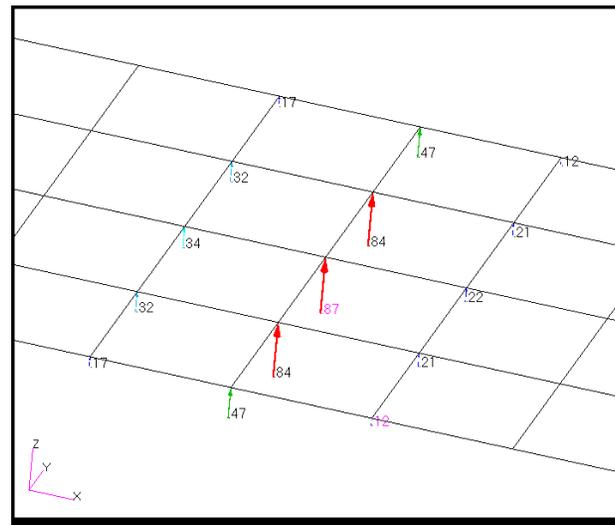
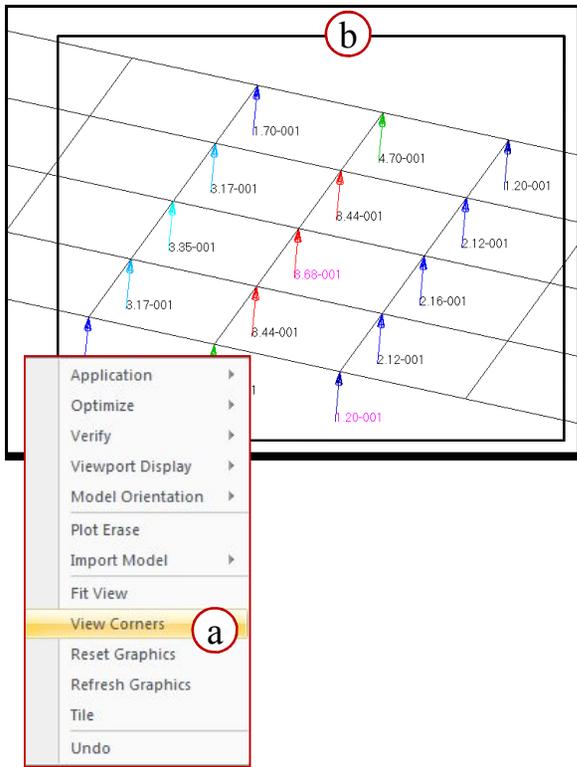
- Click **Reset Graphics**.
- Click **Vector** in the *Result Plots* group.
- Click the **Results** tab.
- Highlight **contact force, Normal**
- Click **Apply**



Step 8. Review Analysis Results (Cont.)

Examine the plate contact body

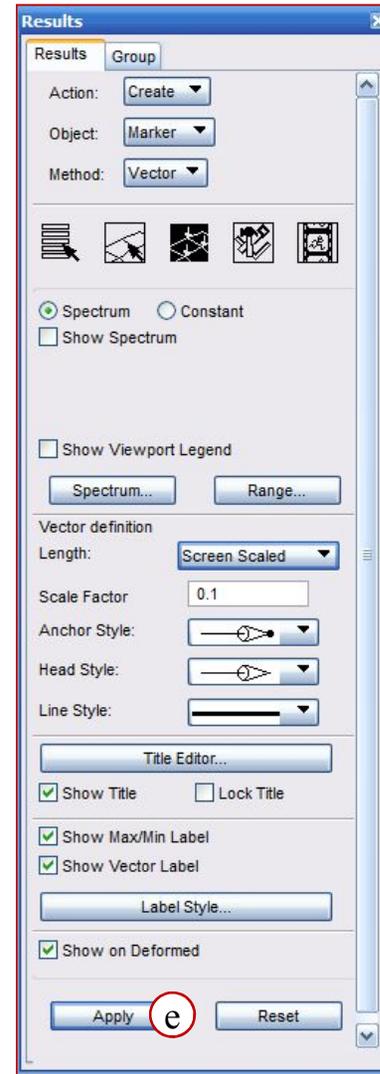
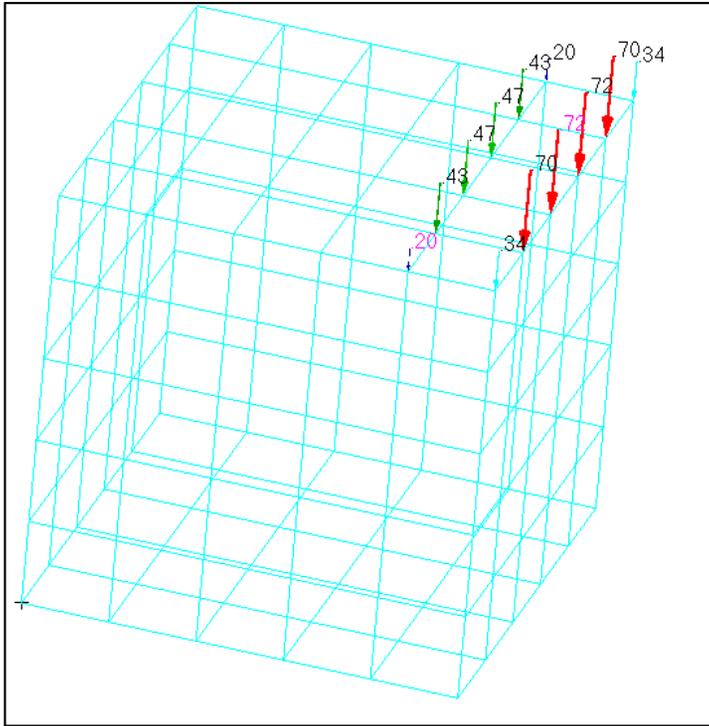
- Right-click in a blank part of the viewport and select **View Corners** from the context menu.
- Drag a rectangle to zoom in on the contact normal forces acting on the plate
- On the *Results* form, click **Display Attributes**
- Experiment with *Vector Definition* and *Label Style* to help visualize the contact force results



Step 8. Review Analysis Results (Cont.)

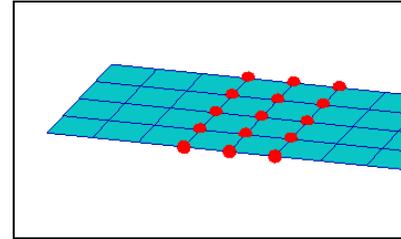
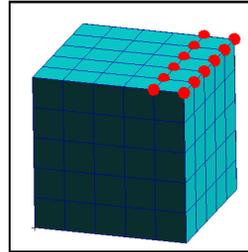
Now examine the block

- a. Click the **Group** tab.
- b. Highlight **block**.
- c. Click **Apply**.
- d. Click the **Results** tab.
- e. Click **Apply**.

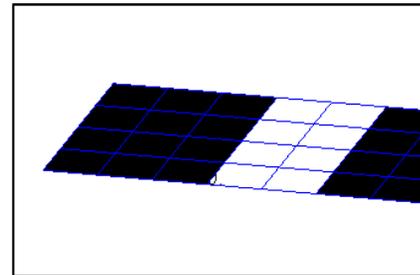
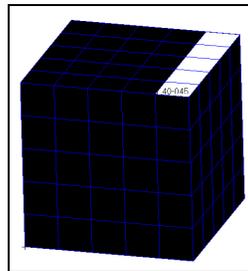


Step 8. Review Analysis Results (Cont.)

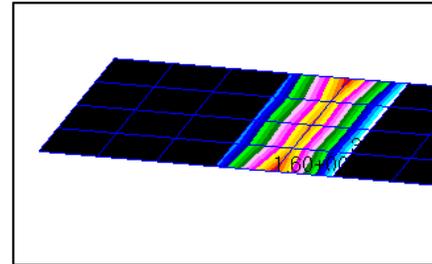
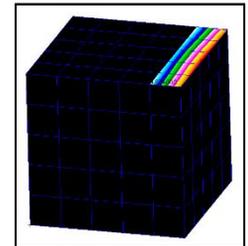
Continue to plot contact analysis results on individual contact bodies on your own.



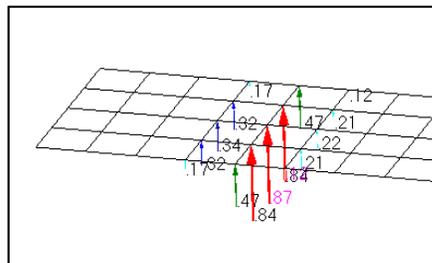
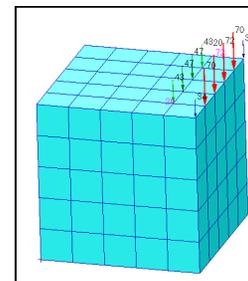
Contact Status
(marker plot)



Contact Status
(fringe plot)



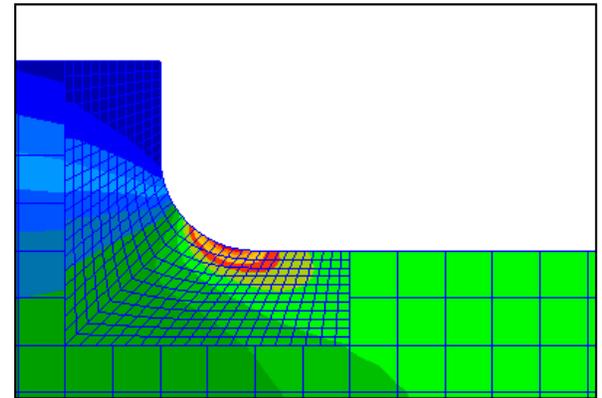
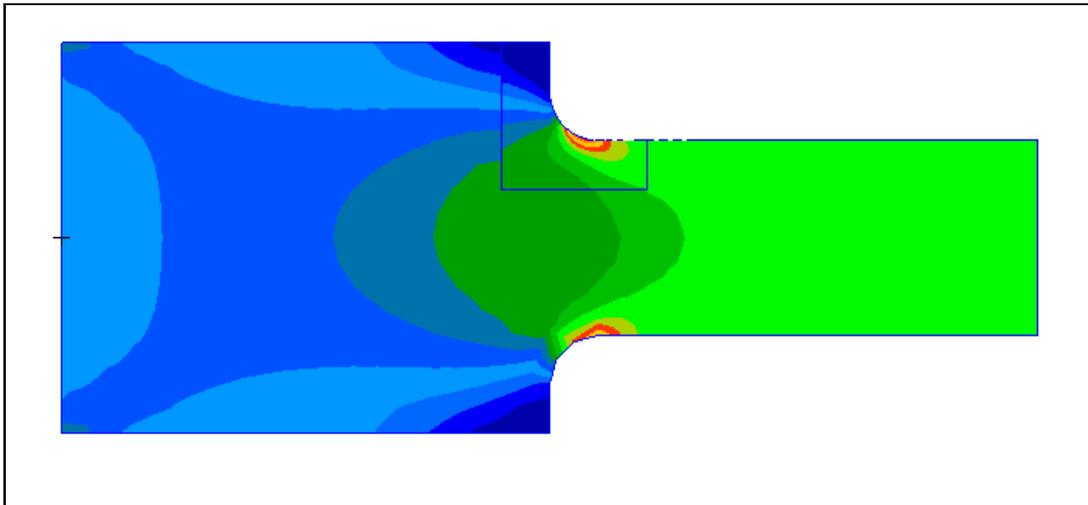
Contact
Normal Stress
(fringe plot)



Contact
Normal Force
(marker plot)

WORKSHOP 3

SHELL EDGE-TO-EDGE GLUED CONTACT



- **Workshop Objectives**

- Learn to create a shell edge-to-edge glued contact with moment carrying capability.

- **Software Version**

- MSC Nastran 2013.1
- Patran 2013

- **Required Files**

- **shell_edge_glue_start.bdf**

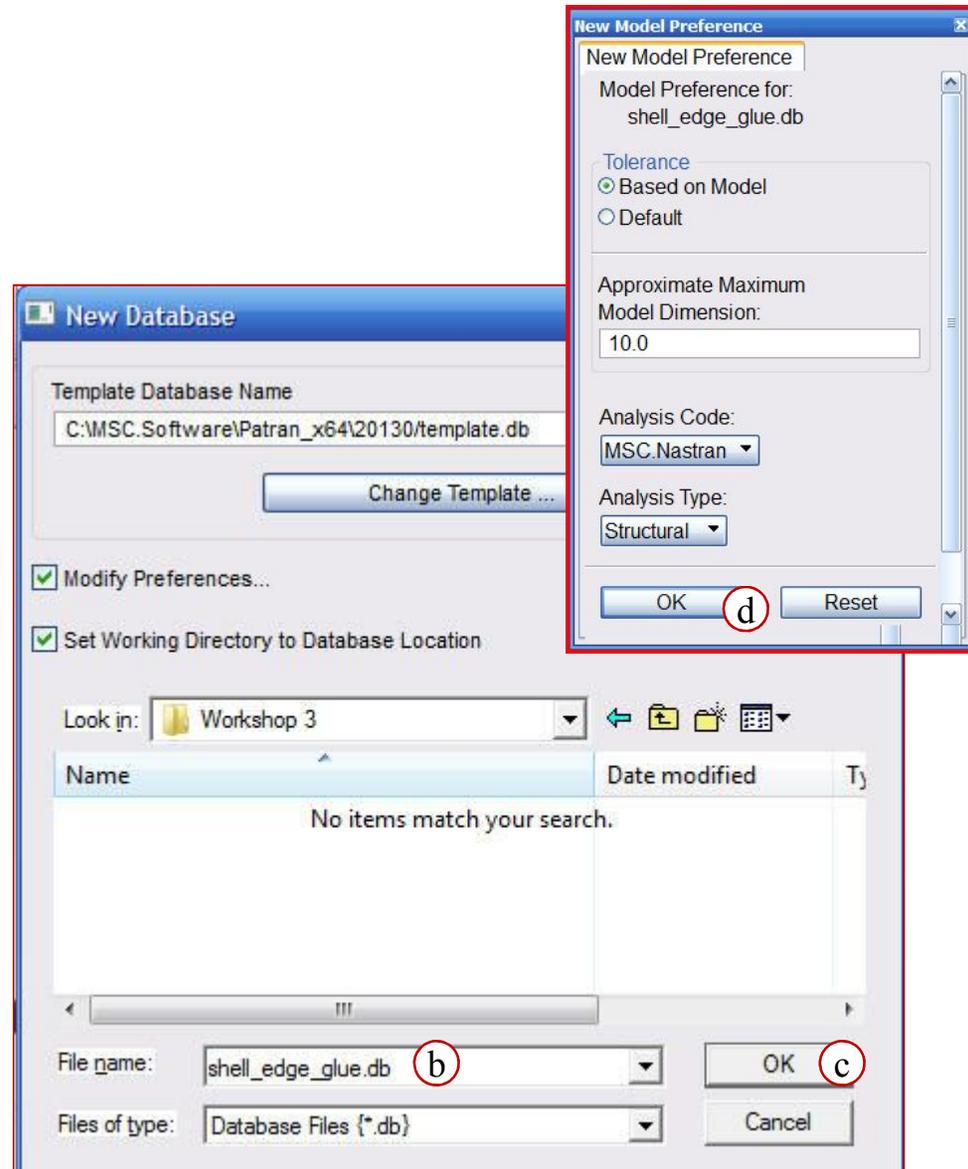
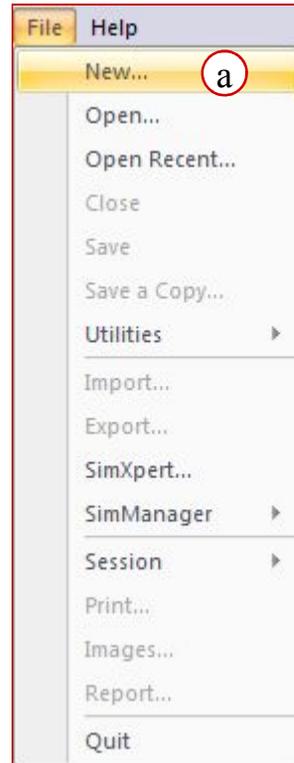
• Suggested Steps

1. Create a new database.
2. Import the MSC Nastran input file.
3. Review the model.
4. Define two deformable contact bodies using the Contact Bodies/Pairs Tool, one for the coarse mesh global model and one for the fine mesh local model.
5. Create a New Body Pair between the two Deformable Bodies
 - Check Glued Contact, Retain Moment, Stress free InitCont
 - Select automatic contact detection
 - Ignore thickness for both master & slave bodies
6. Set up a linear static analysis and run the analysis
7. Review the entries BCONPRG and BCONECT in the MSC Nastran input file.
8. Review MSC Nastran f06 result file.
9. Attach the MASTER result file.
10. Review analysis results. Plot deformation and von Mises stress results.

Step 1. Create a New Database

Create a new database:

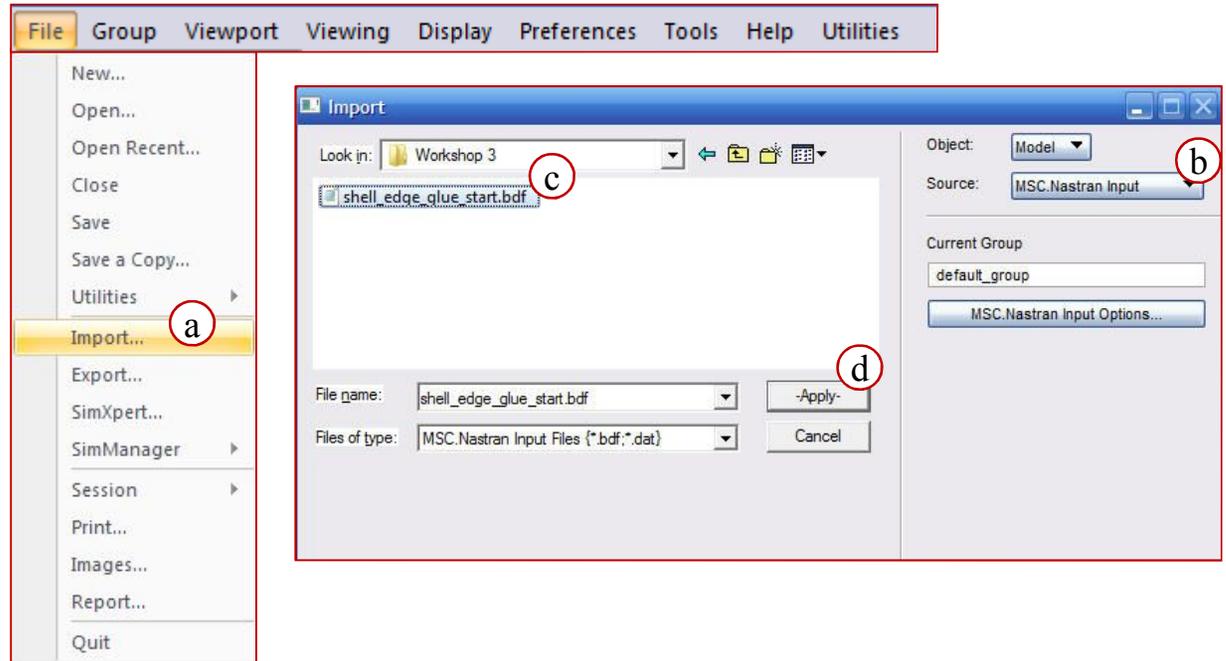
- a. Pull down **File > New**.
- b. Enter **shell_edge_glue**. for the *File name*.
- c. Click **OK**.
- d. For *New Model Preference*, accept all defaults by clicking **OK**.



Step 2. Import a MSC Nastran Input File

Import the starting finite element model:

- a. Pull down **File > Import...**
- b. Pull down *Source* to **MSC. Nastran Input**.
- c. Select **shell_edge_glue_start.bdf**.
- d. Click **Apply** to import the model.
- e. Click **OK** to close the import summary form.

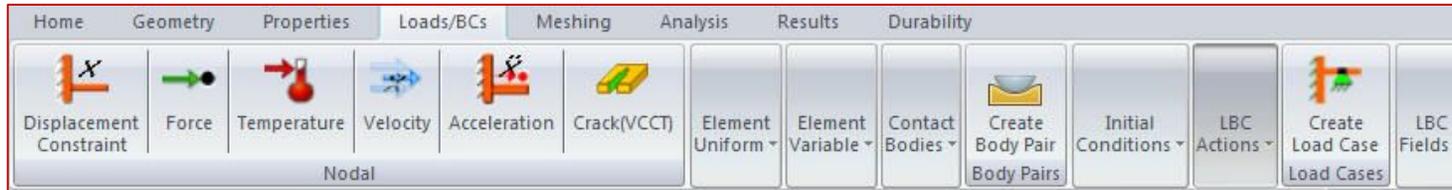


The screenshot shows the 'Nastran Input File Import Summary' dialog box. It contains a table with the following data:

	Imported	Imported with Warning	Not Imported
Nodes	993	0	0
Elements	884	0	0
Coordinate Frames	0	0	0
Material Propertie	1	0	0
Element Properties	2	0	0
Load Sets	2	0	0
Subcases	0	0	0
MPC Data	0	0	0
Comment Lines	12	0	0

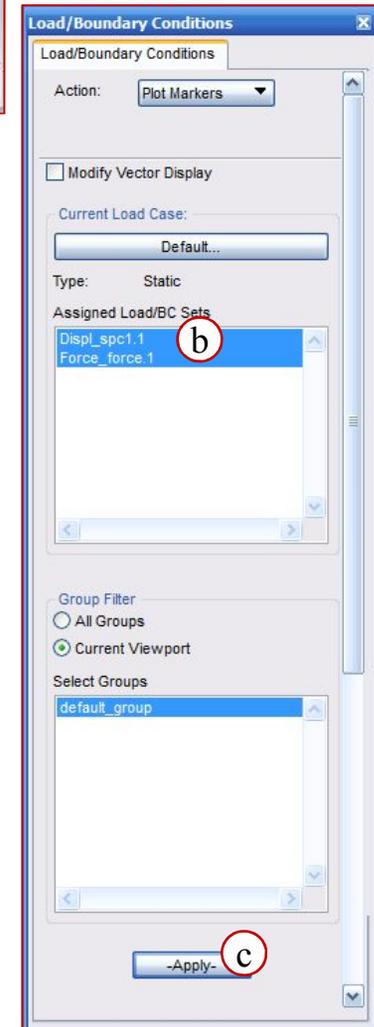
At the bottom of the dialog, there is a 'Reject Cards...' button and an 'OK' button circled in red with 'e'.

Step 3. Review the Model



Review the model:

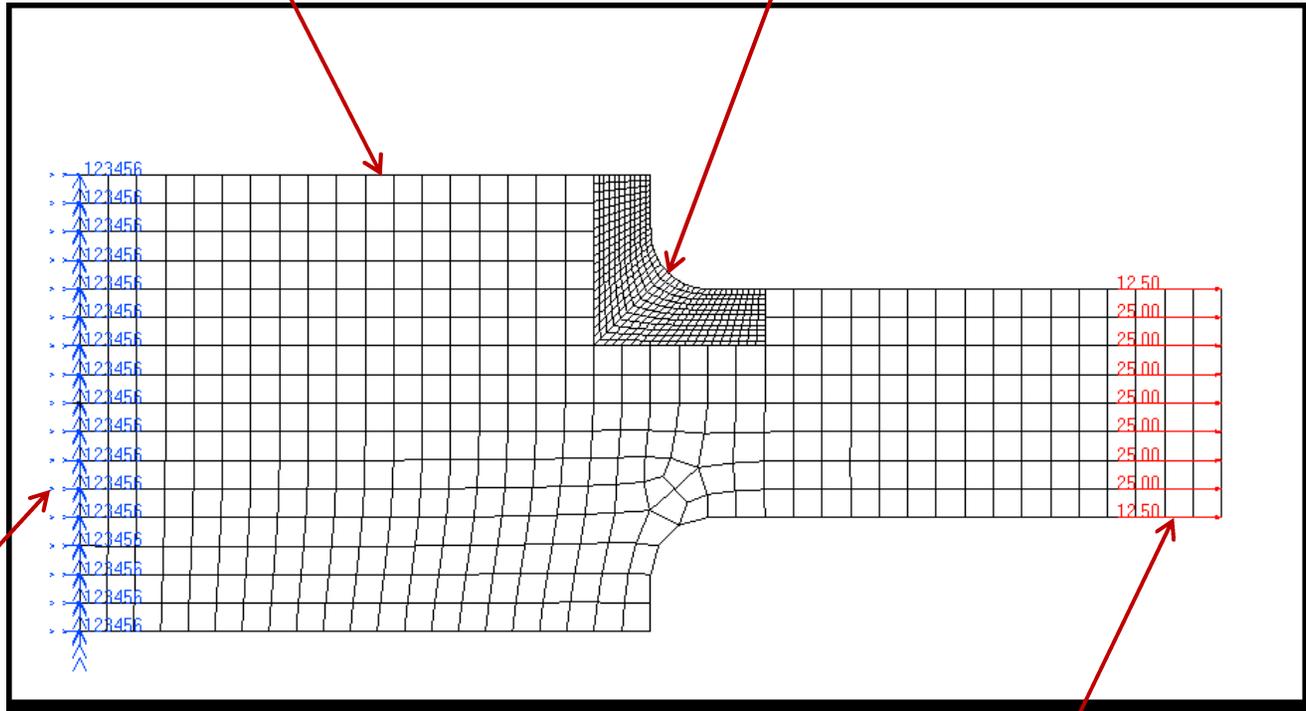
- Under the *Load/BCs* tab, click **Plot Markers** in the *LBC Actions* group.
- Shift-select both the **Displ_spc1.1** and **Force_force.1** Load/BC sets
- Click **Apply** to plot the loads and boundary conditions.



Step 3. Review the Model

Coarse Mesh Global Model

Fine Mesh Local Model



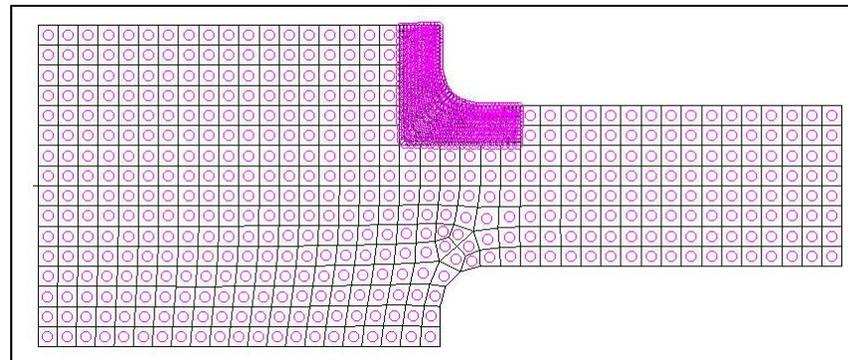
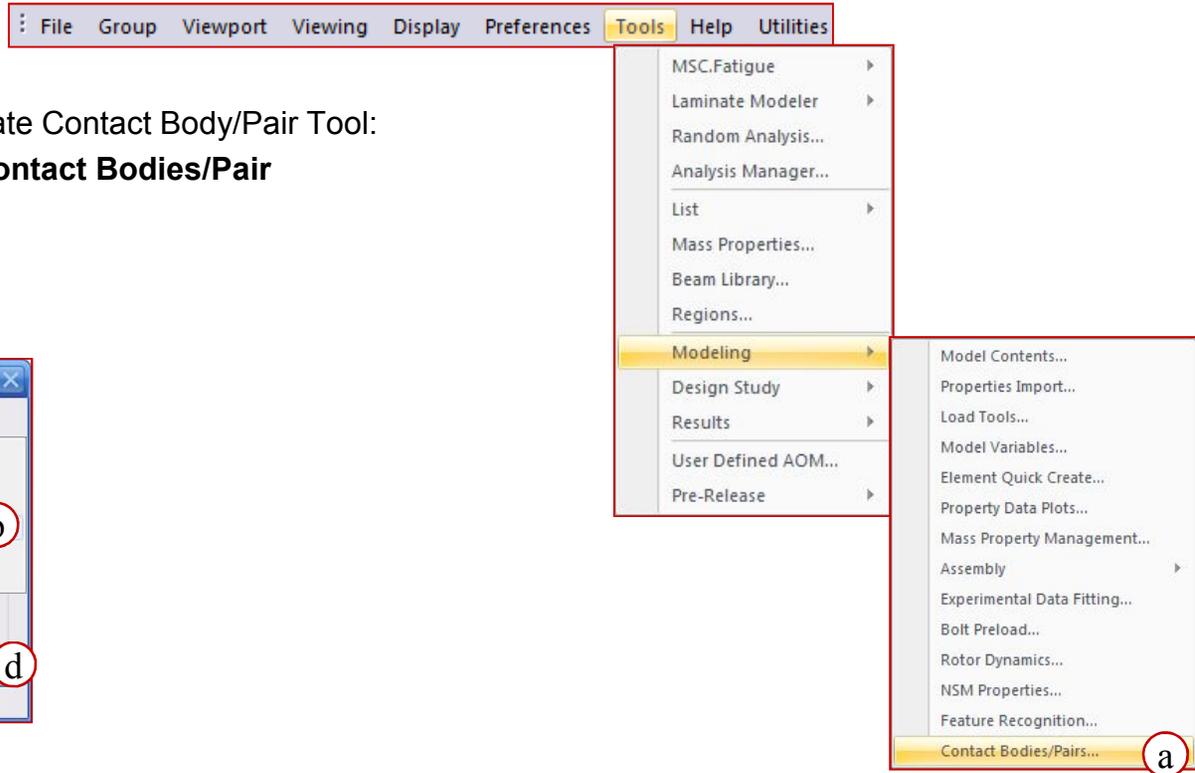
Fixed End

200 Lb Tensile Load

Step 4. Define Two Deformable Contact Bodies

Define Contact Bodies using the Create Contact Body/Pair Tool:

- Pull down **Tools > Modeling > Contact Bodies/Pair**
- Pull down *Method* to **Properties**
- Click **Apply**
- Click **Cancel**



Step 5. Create Body Pair

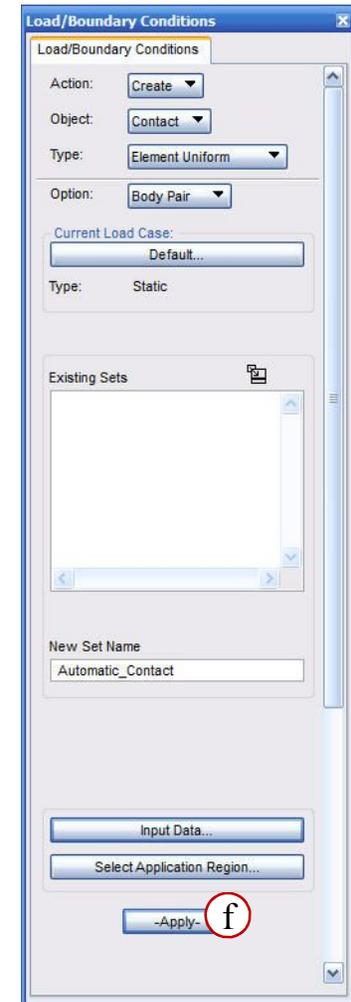
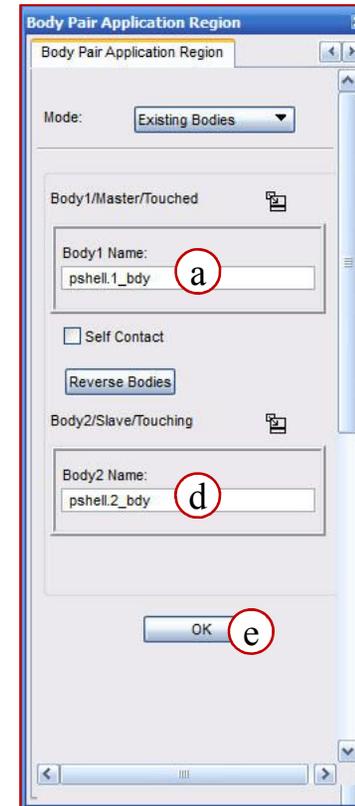
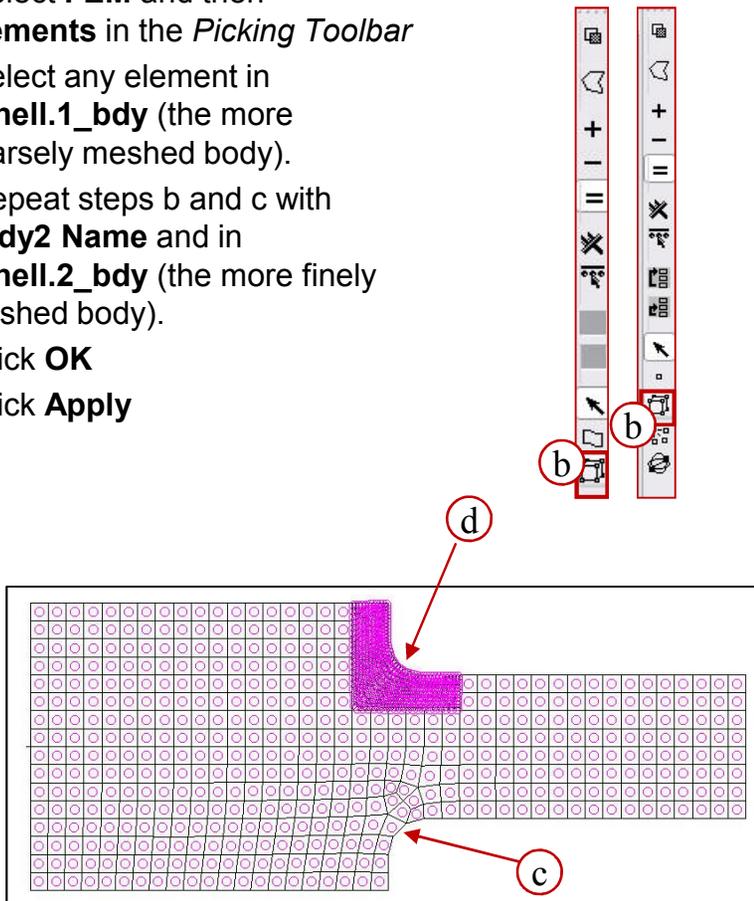
Create a New Body Pair:

- Under the *Load/BCs* tab click **Create Body Pair** in the *Body Pairs* group.
- Enter **Automatic_Contact** for *New Set Name*.
- Click **Input Data...**
- Check **Glued Contact(IGLUE)**, **Retain Moment(IGLUE)**, and **Stress free InitCont(ICOORD)**.
- Select **Automatic** for *Contact Detection (ISEARCH)*.
- Click **Contact Options**
- Check **Ignore Thickness** for both *Master* and *Slave* bodies
- Click **OK**
- Click **Select Application Region**

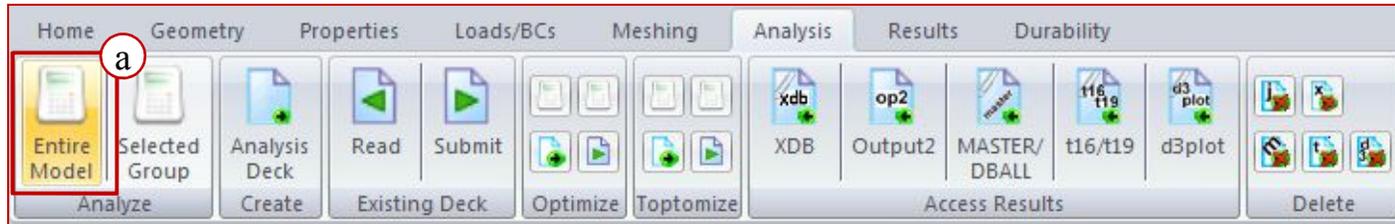
Step 5. Create Body Pair (Cont.)

Select Application Region:

- Click in the **Body1 Name** box
- Select **FEM** and then **Elements** in the *Picking Toolbar*
- Select any element in **pshell.1_bdy** (the more coarsely meshed body).
- Repeat steps b and c with **Body2 Name** and in **pshell.2_bdy** (the more finely meshed body).
- Click **OK**
- Click **Apply**

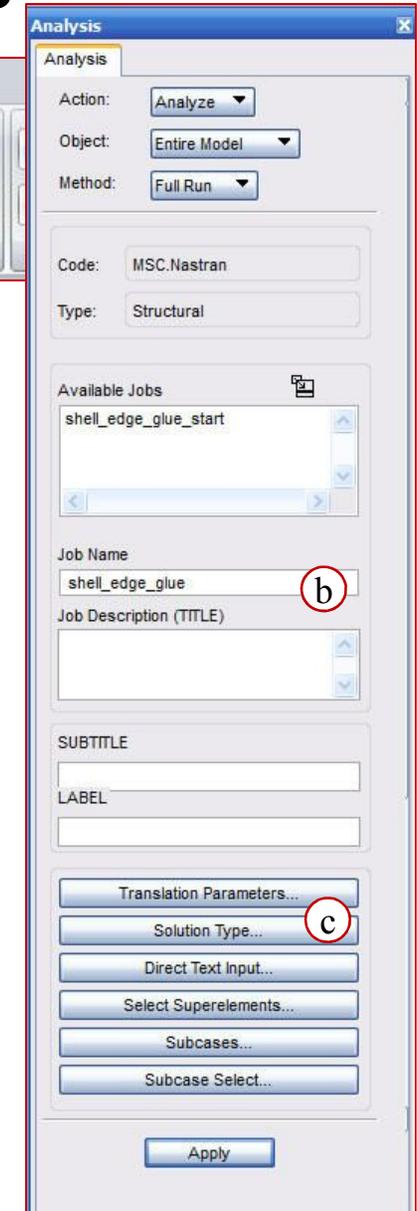
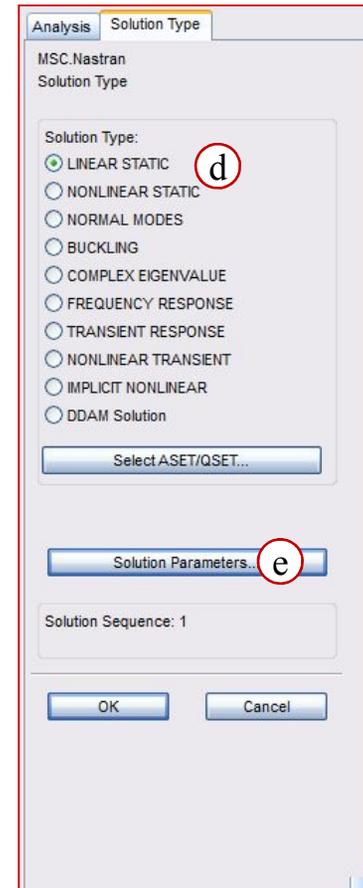


Step 6. Set up a Linear Static Analysis



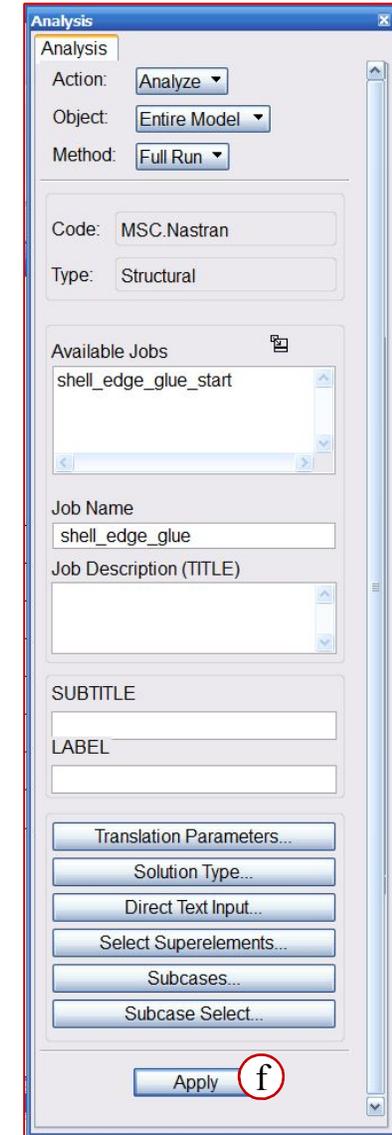
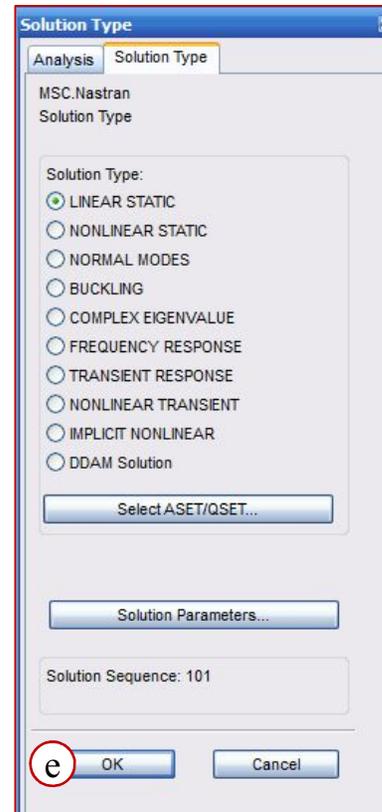
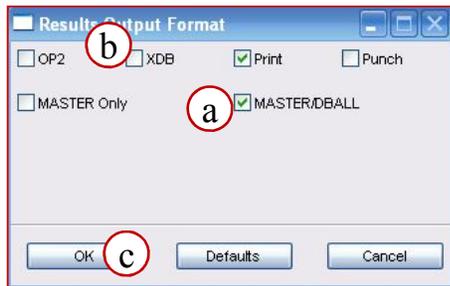
Set up Solution Type

- a. Under the *Analysis* tab, click **Entire Model** in the *Analyze* group.
- b. Enter `global_local_glue` for *Job Name*.
- c. Click **Solution Type....**
- d. Select **LINEAR STATIC**.
- e. Click **Solution Parameters**.
- f. Make sure that both **Database Run** and **Automatic Constraints** are checked under *Static Solution Parameters*.
- g. Click **Results Output Format**.



Step 6. Set up a Linear Static Analysis (Cont.)

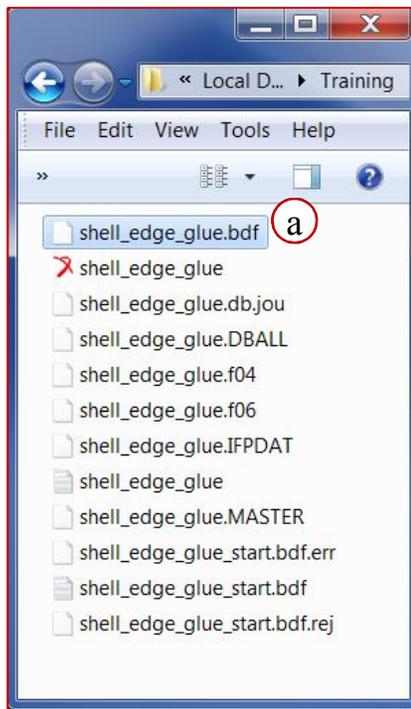
- a. Check **MASTER/DBALL**
- b. Uncheck **XDB**.
- c. Click **OK**.
- d. Click **OK** on the *Solution Parameters* form.
- e. Click **OK**.
- f. Click **Apply** to run the Analysis



Step 7. Review MSC Nastran Input File

While the job is running, review the MSC Nastran input file.

- a. Locate the file **shell_edge_glue.bdf**
- b. Open **shell_edge_glue.bdf** with any text editor.

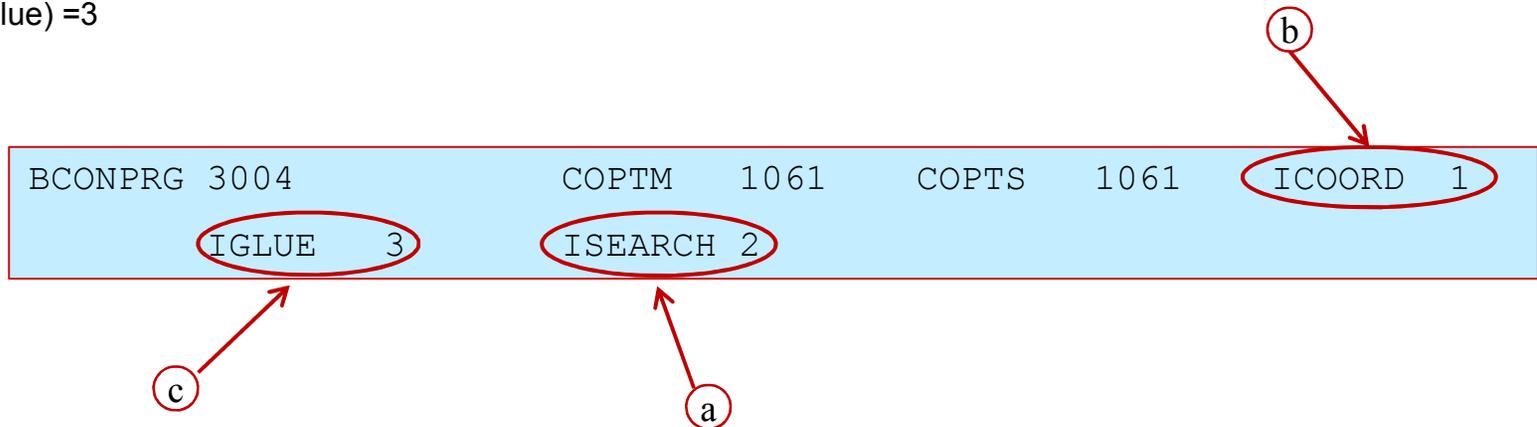


```
$ Direct Text Input for Nastran System Cell Section
NASTRAN SYSTEM(316)=19
$ Replace the 19 in the above line with 7
$ to get a restart DBALL instead of what is currently written
$ Linear Static Analysis, Database (b)
SOL 101
CEND
ECHO = NONE
    BCONTACT = 0
SUBCASE 1
$ Subcase name : Default
    SUBTITLE=Default
    NLPARM = 1
    BCONTACT = 1
    SPC = 2
    LOAD = 3
    DISPLACEMENT (SORT1, REAL)=ALL
    SPCFORCES (SORT1, REAL)=ALL
    STRESS (SORT1, REAL, VONMISES, BILIN)=ALL
BEGIN BULK
PARAM   PRTMAXIM YES
BCPARA  0      NLGLUE  0
NLPARM  1
BCTABL1 0      8005
BCONECT 8005  3004      2      1
BCONPRG 3004      COPTM  1061  COPTS  1061  ICOORD  1
          IGLUE  4
BCTABL1 1      8005
$ Elements and Element Properties for region : pshell.1
PSHELL  1      1      .1      1      1
$ Pset: "pshell.1" will be imported as: "pshell.1"
CQUAD4  1      1      1      2      23      22      0.      0.
```

Step 7. Review MSC Nastran Input File (Cont.)

Review the Input File

- a. On the BCONPRG entry check that ISEARCH is equal to 2
- b. Check that the ICOORD = 1
- c. Check that IGLUE (moment carrying glue) =3



Step 7. Review MSC Nastran Input File (Cont.)

The QRG entry for BCONECT and note concerning COPTS1 and COPTM1 have been copied below for convenience

- a. Notice that the BCBODY 2 is the SLAVE and BCBODY 1 is the MASTER.
- b. The note from the QRG concerning COPTS1 and COPTM1 is copied here for your convenience. Notice that in this case:
 - A = 1
 - B = 6
 - C=1
 - so COPTS1=CPTM1= **1061**.

BCONECT	ID	BCGPID	BCPPID	IDSLAVE	IDMASTER				
BCONECT 8005		3004		2	1				
BCONPRG 3004						COPTM 1061	COPTS 1061		ICOORD 1
		IGLUE 3				ISEARCH 2			

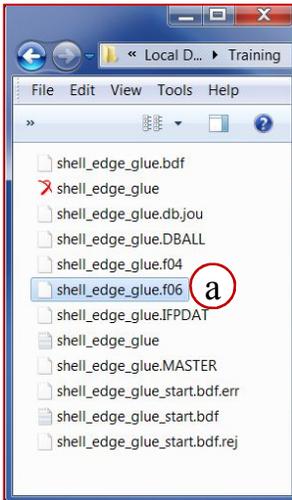
$$COPT = A + 10 \cdot B + 1000 \cdot C$$

- **A** – the outside of solid elements in the body
 - = 1: the outside will be in the contact description (default)
- **B** (flexible bodies) – the outside of shell elements in the body
 - = 1: top and bottom can contact, thickness included (default)
 - = 2: only bottom can contact, thickness included
 - = 3: only bottom can contact, thickness ignored
 - = 4: only top can contact, thickness included
 - = 5: only top can contact, thickness ignored
 - = 6: top and bottom can contact, thickness ignored
- **C** (flexible bodies) – the edges of the body
 - = 1: only the beam edges can contact (default)
 - = 10: only the free and hard shell edges can contact
 - = 11: beam edges and free and hard shell edges can contact

Step 8. Review MSC Nastran Result File

After completion of the run, review the .f06 results file.

- Locate the file **global_local_glue.f06**
- Open **global_local_glue.f06** with any text editor.
- Scroll down to the CASE CONTROL ECHO section. Notice no nonlinear increments or iterations were performed. A line solution was performed. Scan the rest of the .f06 file then close it.



```
0
0
COMMAND          CASE CONTROL ECHO
COUNT
1 ECHO = NONE
2 BCONTACT = 0
3 SUBCASE 1
4 SUBTITLE=DEFAULT
5 NLPARM = 1
6 BCONTACT = 1
7 SPC = 2
8 LOAD = 3
9 DISPLACEMENT(SORT1,REAL)=ALL
10 SPCFORCES(SORT1,REAL)=ALL
11 STRESS(SORT1,REAL,WONMISES,BILIN)=ALL
12 $ DIRECT TEXT INPUT FOR THIS SUBCASE
13 BEGIN BULK
INPUT BULK DATA ENTRY COUNT = 1927
TOTAL COUNT= 1913
^^^
^^^ >>> IFP OPERATIONS COMPLETE <<<
^^^
DEFAULT NONLINEAR TOLERANCES
NLPARM ID 1
NINC 10
EPSU 1.000000E-02 (not normally used)
EPSP 1.000000E-02
EPSW 1.000000E-02
(VALUE YOU ENTER OVERRIDE THE ABOVE DEFAULTS)

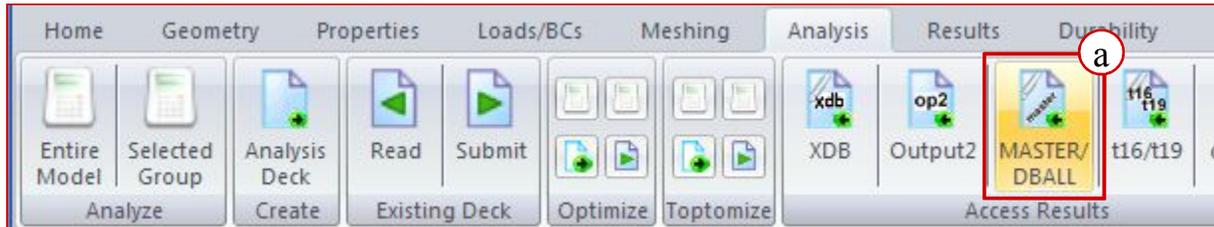
*** USER WARNING MESSAGE 4124 (IFS3P)
THE SPCADD OR MPCADD UNION CONSISTS OF A SINGLE SET.

MODEL SUMMARY
NUMBER OF GRID POINTS = 993
NUMBER OF CQUAD4 ELEMENTS = 884

flexible body # 2 (body_2 ) will detect flexible body # 1 (body_1 )
flexible body # 2 (body_2 ) will detect flexible body # 1 (body_1 )
SEPTEMBER 1, 2011 MD NASTRAN 7/15/10 PAGE 4

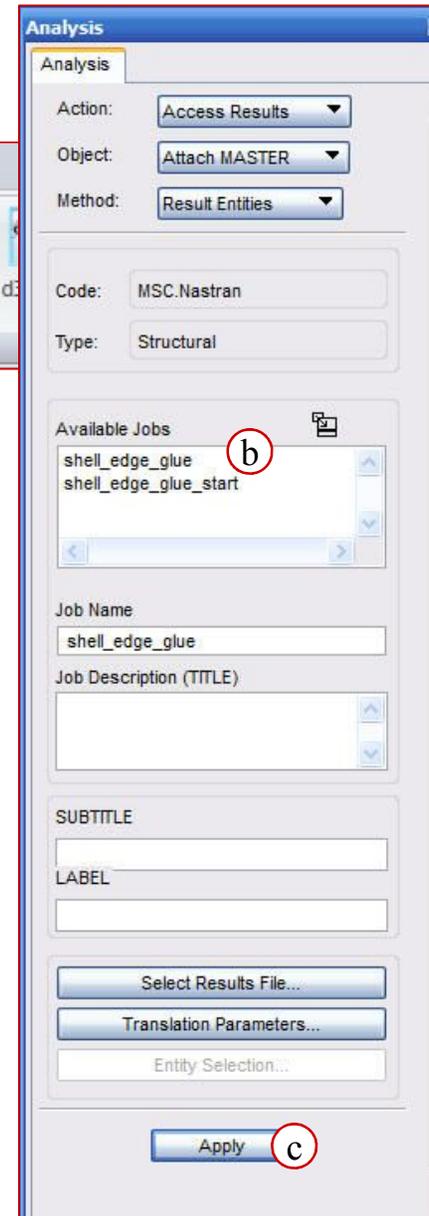
*** USER INFORMATION MESSAGE 7310 (VECPRN)
ORIGIN OF SUPERELEMENT BASIC COORDINATE SYSTEM WILL BE USED AS REFERENCE LOCATION.
RESULTANTS ABOUT ORIGIN OF SUPERELEMENT BASIC COORDINATE SYSTEM IN SUPERELEMENT BASIC SYSTEM COORDINATES.
OLOAD RESULTANT
SUBCASE/ DAREA ID LOAD TYPE T1 T2 T3 R1 R2 R3
0 1 FX 2.000000E+02 ---- ---- 0.000000E+00 0.000000E+00 0.000000E+00
FY ---- 0.000000E+00 ---- 0.000000E+00 0.000000E+00 0.000000E+00
FZ ---- ---- 0.000000E+00 0.000000E+00 0.000000E+00 ----
MX ---- ---- 0.000000E+00 ---- 0.000000E+00 ----
MY ---- ---- 0.000000E+00 ---- 0.000000E+00 ----
MZ ---- ---- 0.000000E+00 ---- 0.000000E+00
1 TOTALS 2.000000E+02 0.000000E+00 0.000000E+00 0.000000E+00 0.000000E+00 0.000000E+00
SEPTEMBER 1, 2011 MD NASTRAN 7/15/10 PAGE 5
0 SUBCASE 1
```

Step 9. Attach Results File



To attach the results file:

- a. Under the *Analysis* tab, click **MASTER/DBALL** in the *Access Results* group.
- b. Select **global_local_glue** from the *Available Jobs*
- c. Click **Apply**



Step 10. Review Analysis Results

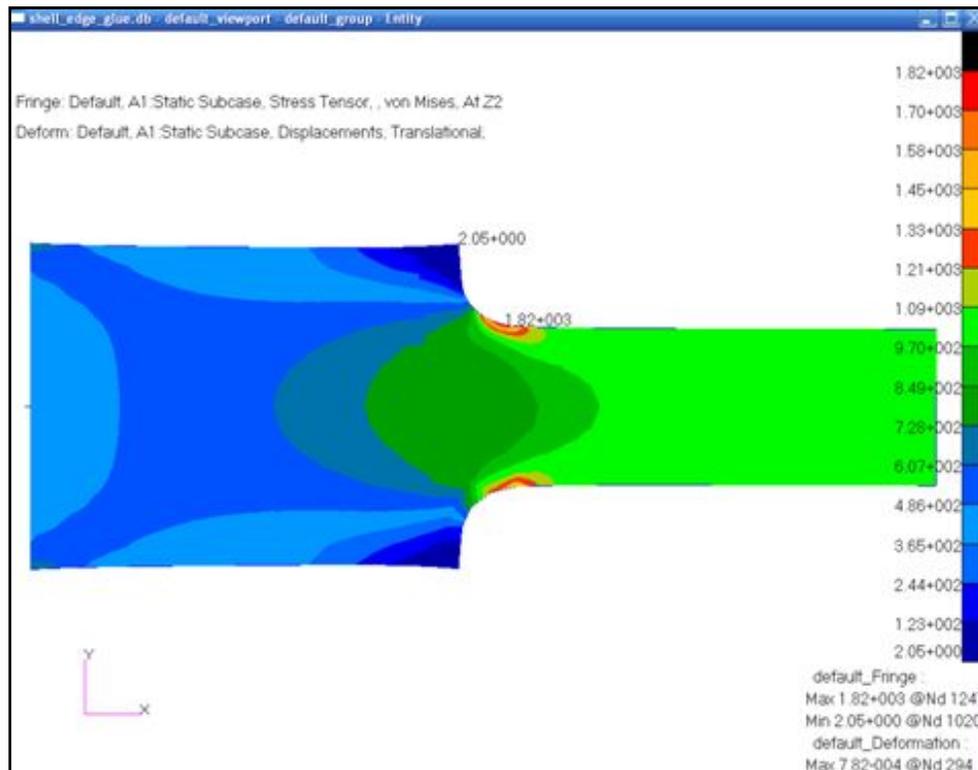
Plot deformation and von Mises stress

- Under the *Results* tab, click **Fringe/Deformation** in the *Quick Plot* group.
- Select **Stress Tensor** for *Fringe Result*.
- Select **Displacements, Translational** for *Deformation Result*.
- Click **Apply**.

Step 10. Review Analysis Results (Cont.)

Turn off the display of the undeformed shape.

- a. Click **Deform Attributes**
- b. Uncheck **Show Undeformed**.
- c. Click **Apply**.



Action: Create

Object: Quick Plot

a

Show Viewport Legend

Deformed:

Render Style: Wireframe

Line Style:

Line Width:

Scale Interpretation

Model Scale True Scale

Scale Factor: 0.1

Show Undeformed **b**

Title Editor...

Show Title Lock Title

Show Maximum Label

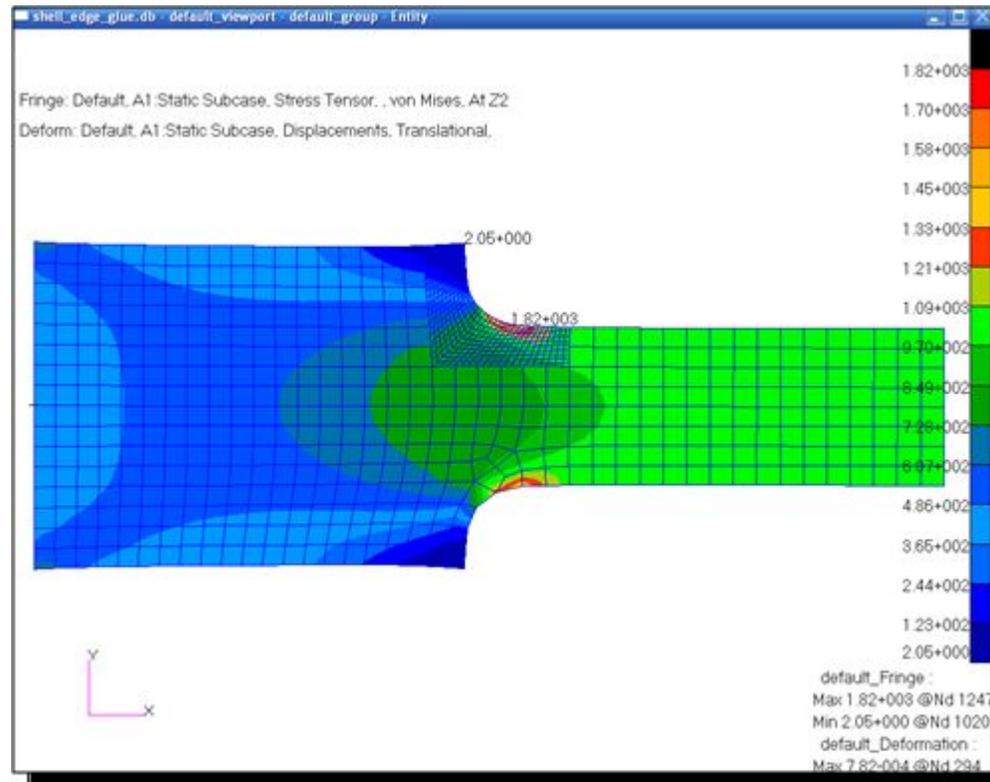
Label Style...

Apply **c** Reset

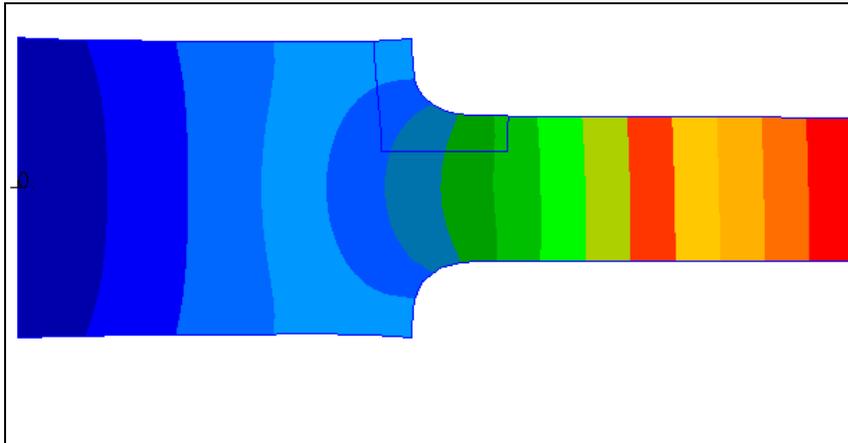
Step 10. Review Analysis Results (Cont.)

Display element edges.

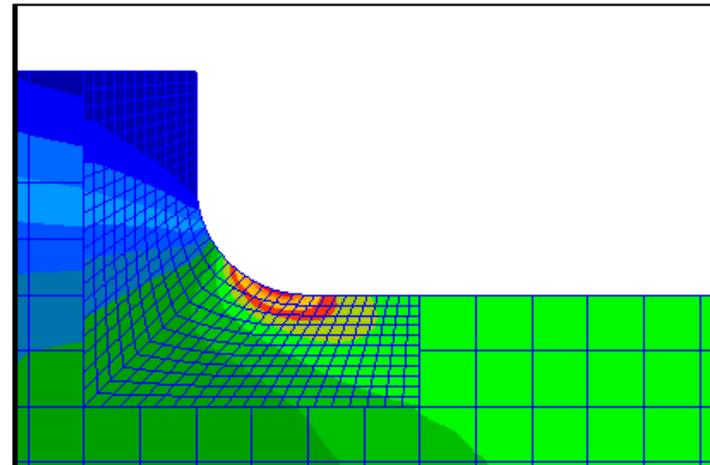
- a. Click **Fringe Attributes**
- b. Pull down *Display* to **Element Edges**.
- c. Click **Apply**.



Step 10. Review Analysis Results (Cont.)

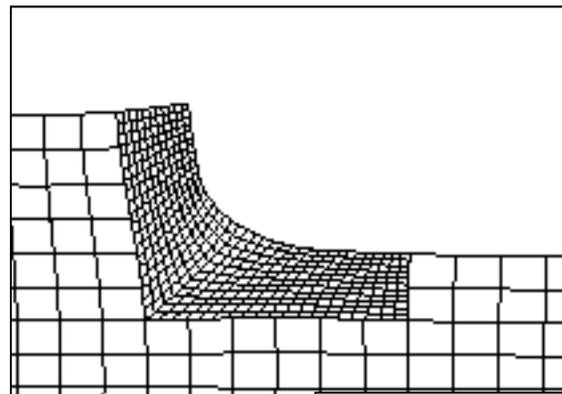


Displacement fringe plot



Un-Averaged stress fringe plot

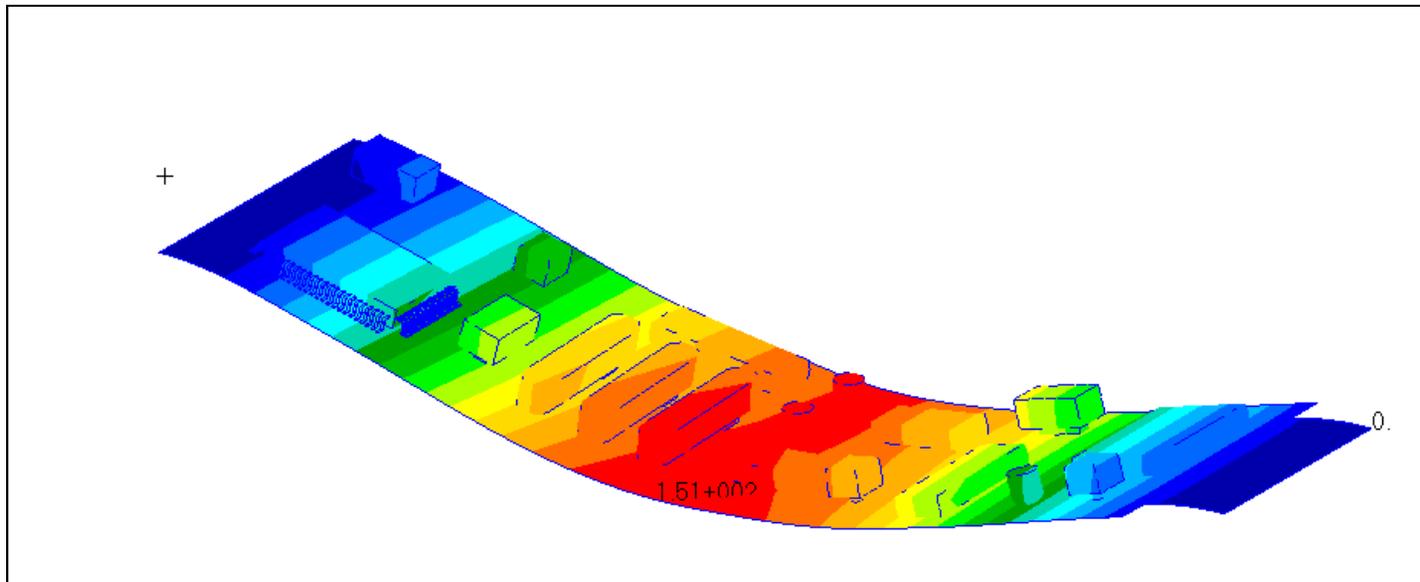
Make additional plots on your own to evaluate the quality of the glued joint.



Exaggerated deformation plot

WORKSHOP 4

NATURAL FREQUENCY WITH GLUED CONTACT



- **Workshop Objectives**

- Perform a SOL103 (Natural Frequency) analysis with glued contacts on a model of a Printed Circuit Board (PCB).
- Use of the ERROR parameter on a contact body.

- **Software Version**

- MSC Nastran 2013.1
- Patran 2013

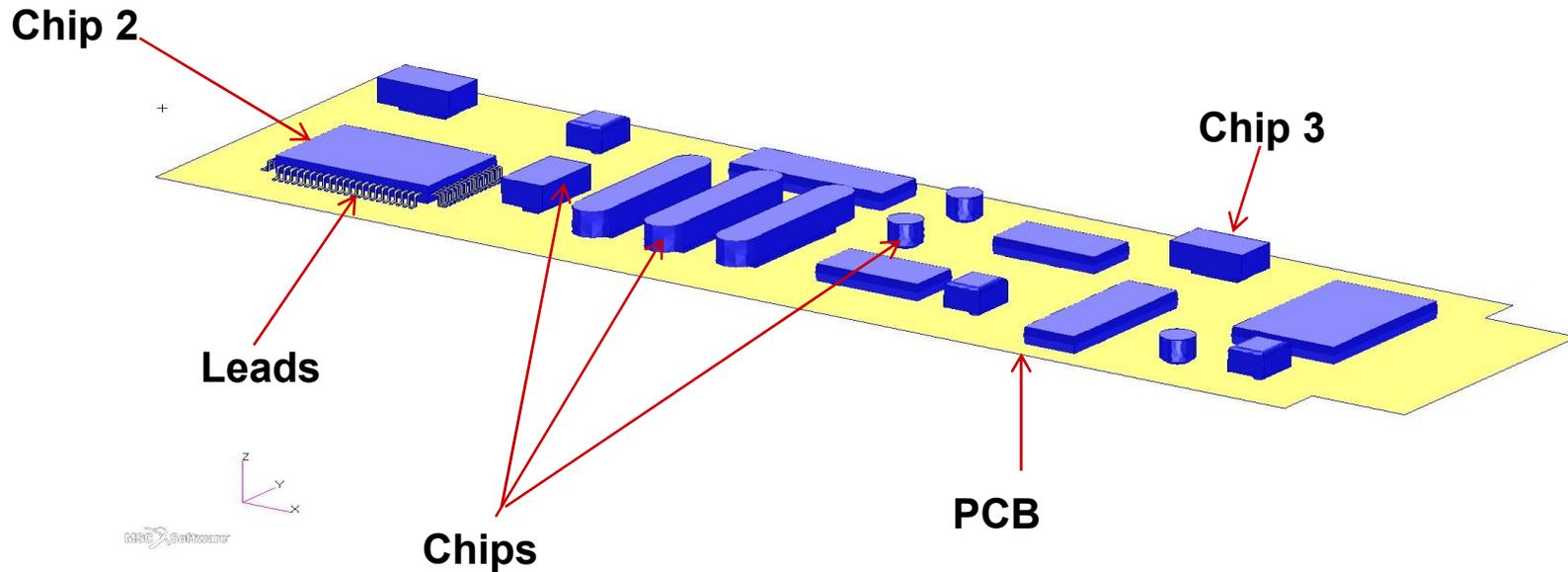
- **Files Required**

- **PCB_model.bdf**

- **Problem Description**

1. The model is a PCB with components soldered onto the board.
2. Glued contact will be used to model the connections between the board and its chips. The contact bodies for this model are shown in the diagram on the following page. Most of the chips will be combined into one contact body except for Chips 2 and 3. Chip 2 and its leads are in their own contact bodies because Chip 2 is not connected directly to the PCB. Instead, the leads from this chip are connected to the PCB.
3. Chip 3 is defined as a separate contact body because it has a slightly larger clearance than the other chips. This becomes important later since the contact distance tolerance needs to be increased for this component in order for glued contact to work.

- **Problem Description (Cont.)**



Component	PCB	Chips	Leads
Young's Modulus (E) ksi	2980	27000	18000
Poisson's Ratio (nu)	0.34	0.30	0.30
Modulus of Rigidity (G) ksi	1111.94	10384.6	6923.08
Density (rho) lb/cu. in	0.065	0.2	-----
Coefficient of thermal expansion (alpha) in/in-deg (F)	-----	2e -6	2e -6

• Suggested Steps

1. Create a new Patran database called **PCB_Natural_Freq.db**.
2. Import the BDF file, **PCB_model.bdf**.
3. Rename the contact bodies to match the component names shown on the Problem Description page and color code the contact bodies.
4. Verify the material properties.
5. Verify that the ends of the PCB are fully fixed.
6. Create four new body pairs between chip_3 and PCB, chip_2 and leads, leads and PCB, and chips and PCB.
 - Check Glued Contact and Stress free InitCont
 - Select automatic contact detection
7. Set up and run a normal modes analysis
8. Review the eigenvalues in the result file.
9. Attach the result file.
10. View the deformed shape.
11. Measure the distance between chip 3 and the PCB.
12. Modify the chip_3_pcb contact body pair and add **0.016** as the Distance
13. Re-run the analysis.



- **Suggested Steps (Cont.)**

14. Review the eigenvalues in the result file.

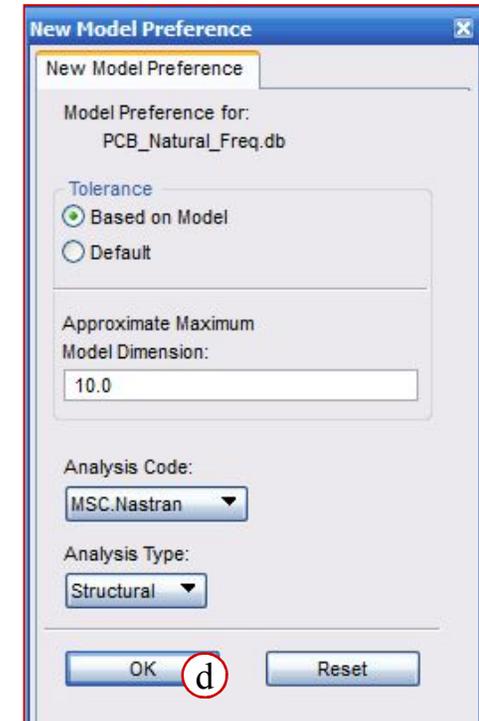
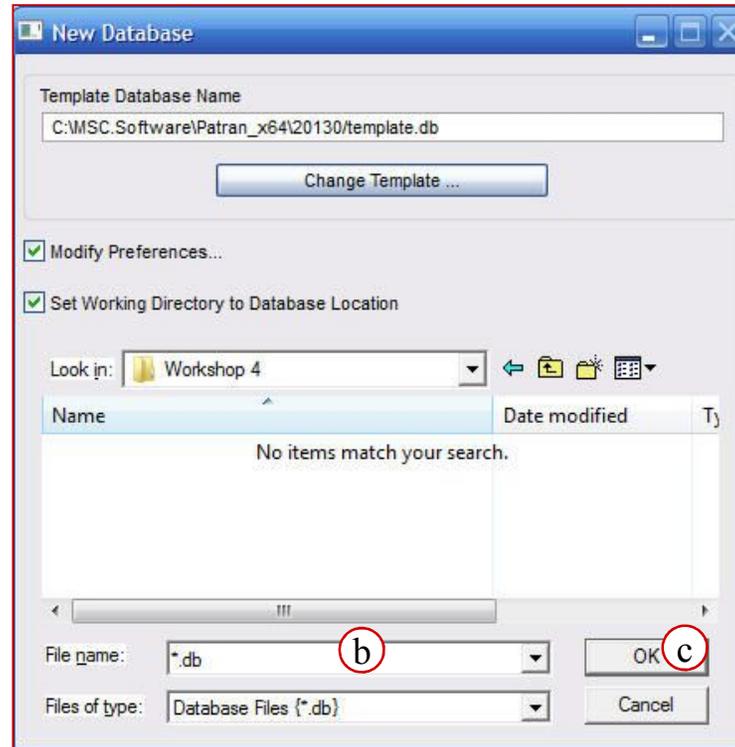
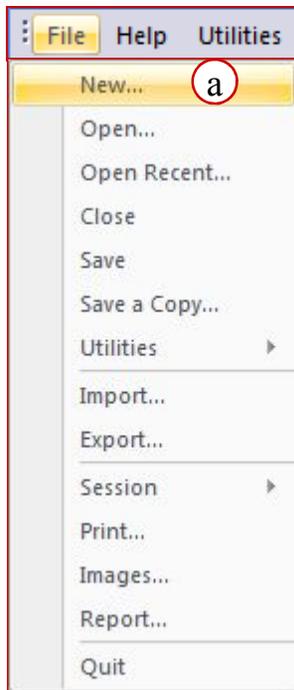
15. Attach the new result file.

16. View the deformed shape.

Step 1. Create a Database

Open a new database.

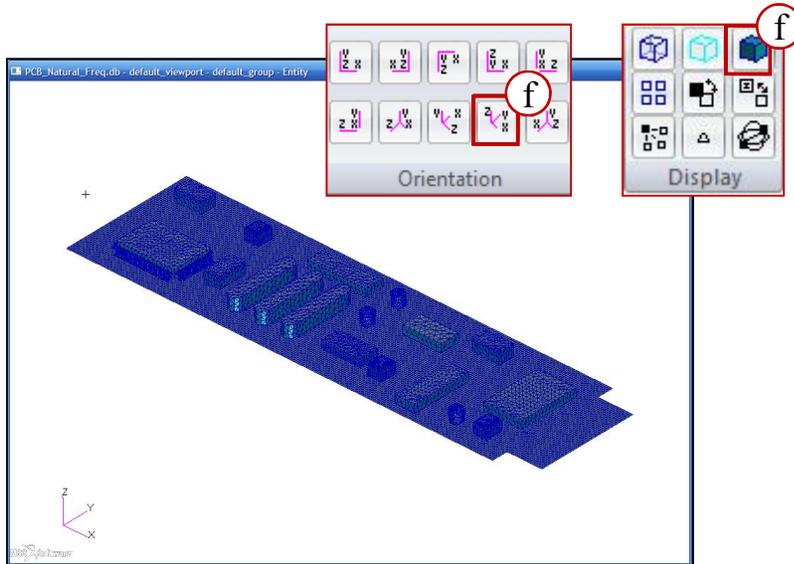
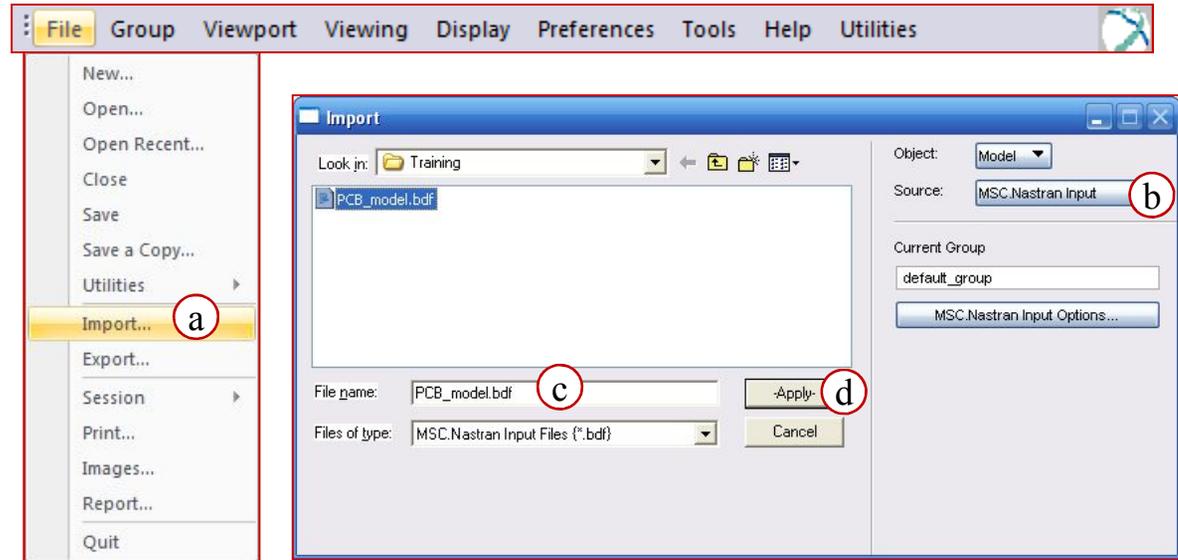
- a. Pull down **File > New**
- b. Enter **PCB_Natural_Freq** for *File Name*.
- c. Click **OK**.
- d. Click **OK** on the *New Model Preference* form to retain the default tolerance.



Step 2. Import the Model

Import the model.

- a. Pull down **File > Import**.
- b. Pull down **MSC.Nastran Input** for *Source*.
- c. Select **PCB_model.bdf** as the file to import.
- d. Click **Apply**.
- e. Check the *MSC Nastran Input File Import Summary* and make sure there are 29212 Nodes, 47352 Elements. Click **OK**.
- f. Rotate and shade the model.



The screenshot shows the 'Nastran Input File Import Summary' dialog box. The table below shows the import results:

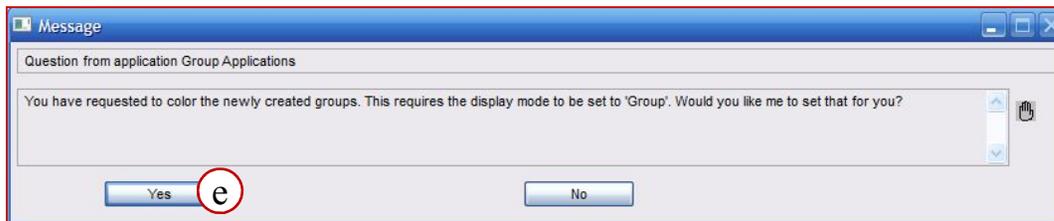
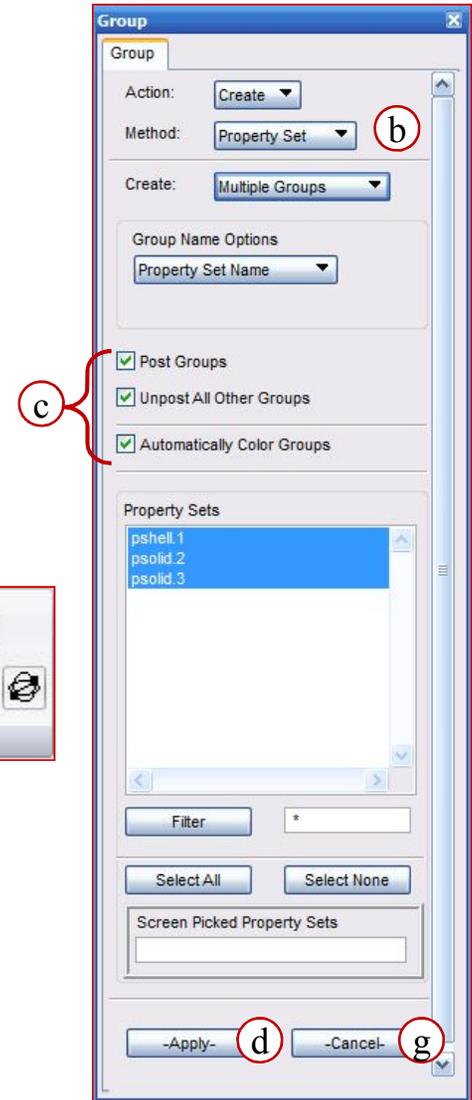
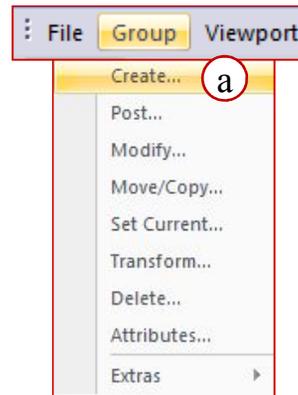
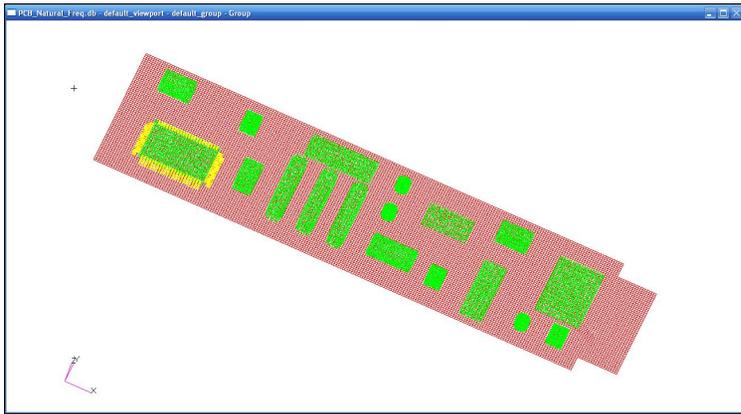
	Imported	Imported with Warning	Not Imported
Nodes	29212	0	0
Elements	47352	0	0
Coordinate Frames	0	0	0
Material Propertie	3	0	0
Element Properties	3	0	0
Load Sets	1	0	0
Subcases	0	0	0
MPC Data	0	0	0
Comment Lines	17	0	0

The 'Elements' row is circled in red and labeled with a red 'e'. The 'OK' button is visible at the bottom of the dialog box.

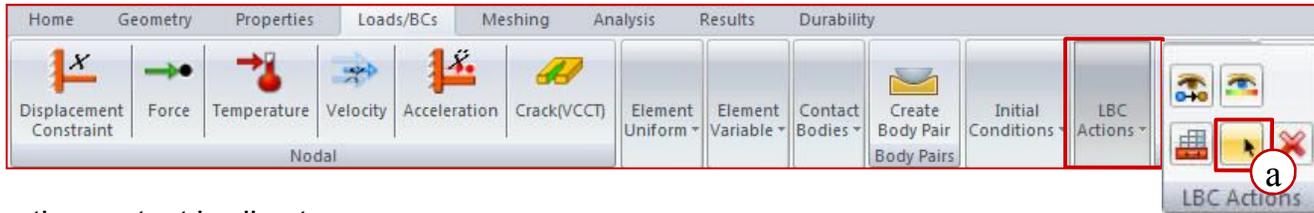
Step 2. Import the Model (Cont.)

Create groups of elements based on their associated properties. Color code the groups.

- a. Pull down **Group > Create**.
- b. Pull down **Property Set** and **Multiple Groups** for *Method* and *Create* respectively.
- c. Check **Post Groups**, **Unpost All Other Groups** and **Automatically Color Groups**.
- d. Click **Apply**.
- e. Click **Yes** to the message “You have requested to color the newly created groups. This requires the display mode to be set to ‘Group.’ Would you like me to set that for you?”
- f. Shade the model.
- g. Click **Cancel**

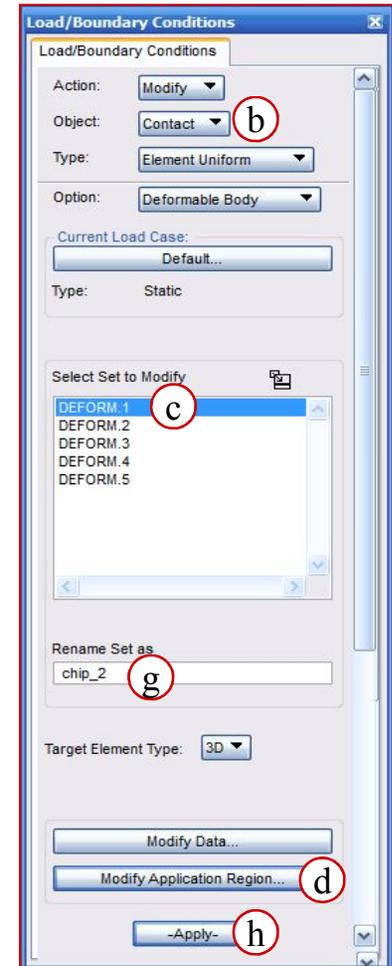
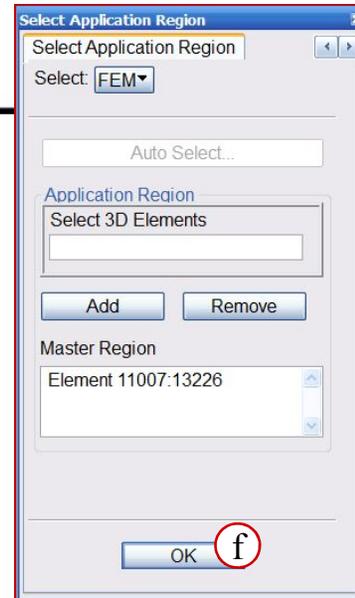
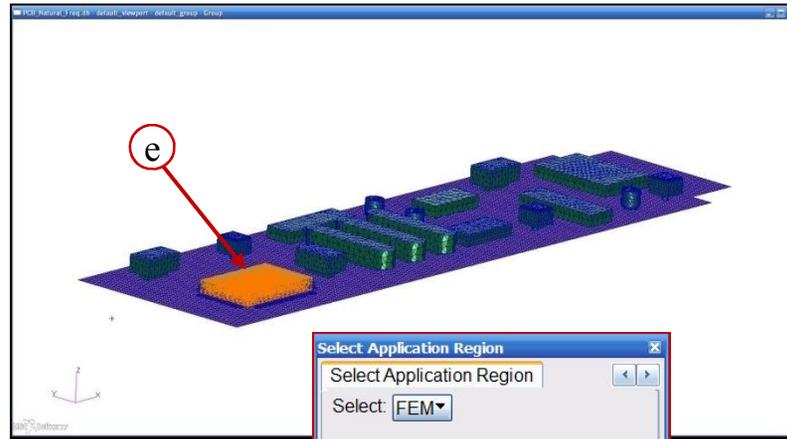


Step 3. Rename and Color Code Contact Bodies



Rename the contact bodies to match the component names shown in the

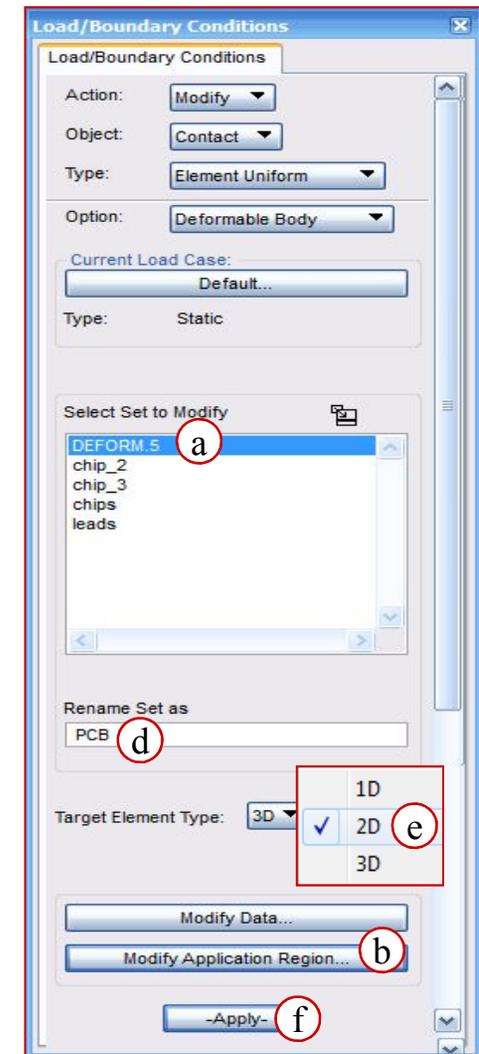
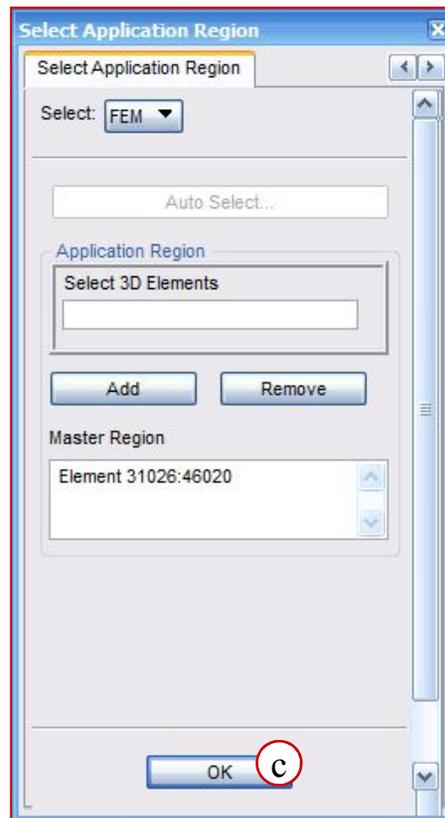
- Under the *Loads/BCs* tab, click **Modify** in the *LBC Actions* group.
- Pull down *Object* to **Contact**.
- Highlight **DEFORM.1** under *Select Set to Modify*.
- Click **Modify Application Region**
- Observe which component is highlighted
- Click **OK**
- Enter **chip_2** for *Rename Set as* to match the Model Description.
- Click **Apply**
- Repeat for:
 - DEFORM.2** - chip_3
 - DEFORM.3** - chips
 - DEFORM.4** - leads



Step 3. Rename and Color Code Contact Bodies (Cont.)

Rename the contact bodies to match the component names shown in the

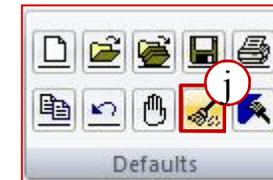
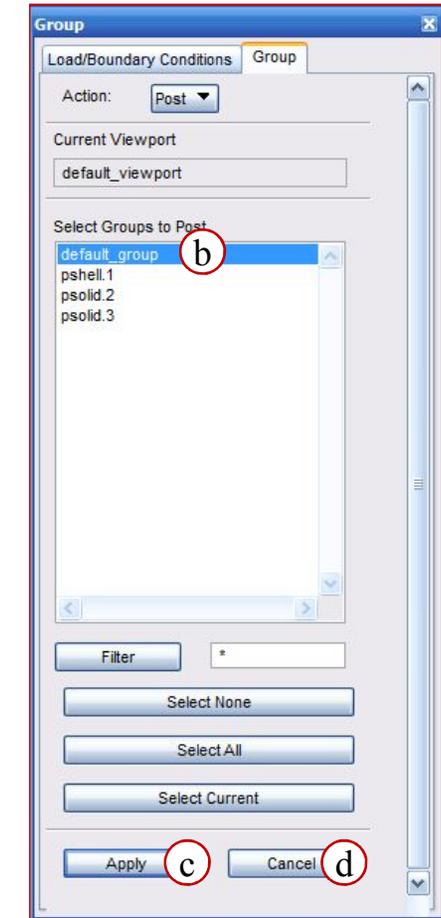
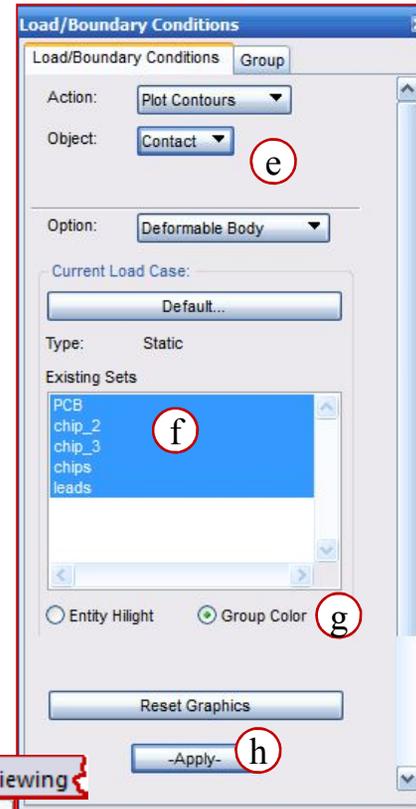
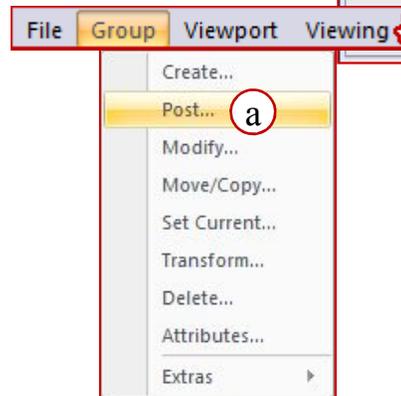
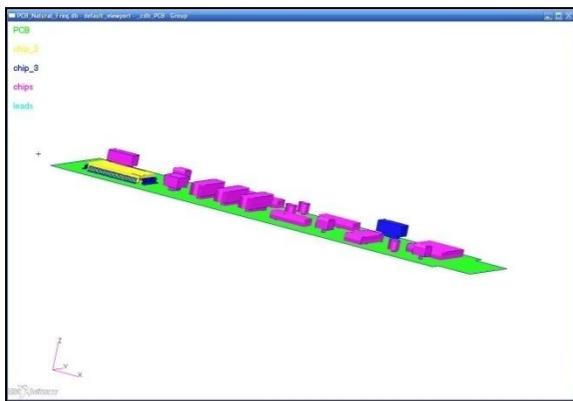
- a. Highlight **DEFORM.5** under *Select Set to Modify*.
- b. Click **Modify Application Region**.
- c. Click **OK**.
- d. Enter **PCB** for *Rename Set as* to match the name in the Problem Description.
- e. Change *Target Element Type* to **2D**.
- f. Click **Apply**.



Step 3. Rename and Color Code Contact Bodies (Cont.)

Color code the contact bodies.

- a. Pull down **Group > Post**.
- b. Highlight **default_group** under *Select Groups to Post*.
- c. Click **Apply**.
- d. Click **Cancel**.
- e. Pull down **Plot Contours / Contact / Deformable Body** for *Action*, *Object* and *Option* respectively.
- f. Highlight all *Existing Sets* using shift-click.
- g. Select **Group Color**.
- h. Click **Apply**.
- i. Shade the model.
- j. When you have finished verifying the contact bodies, click **Reset Graphics**.

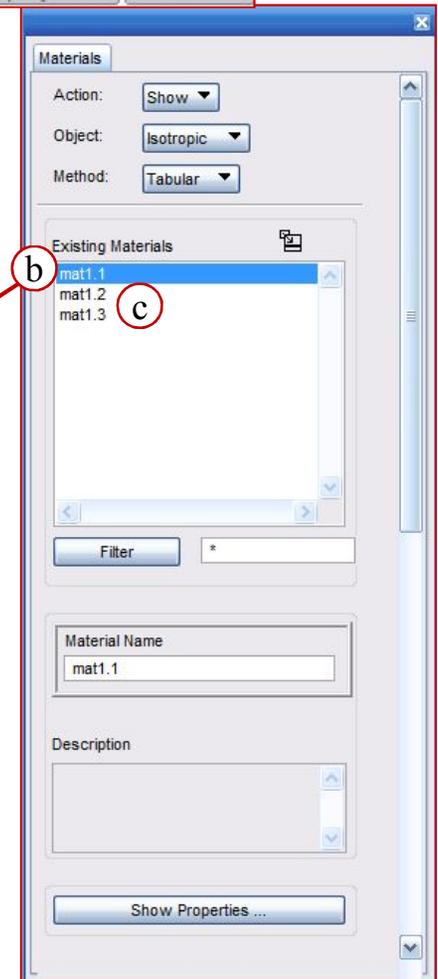
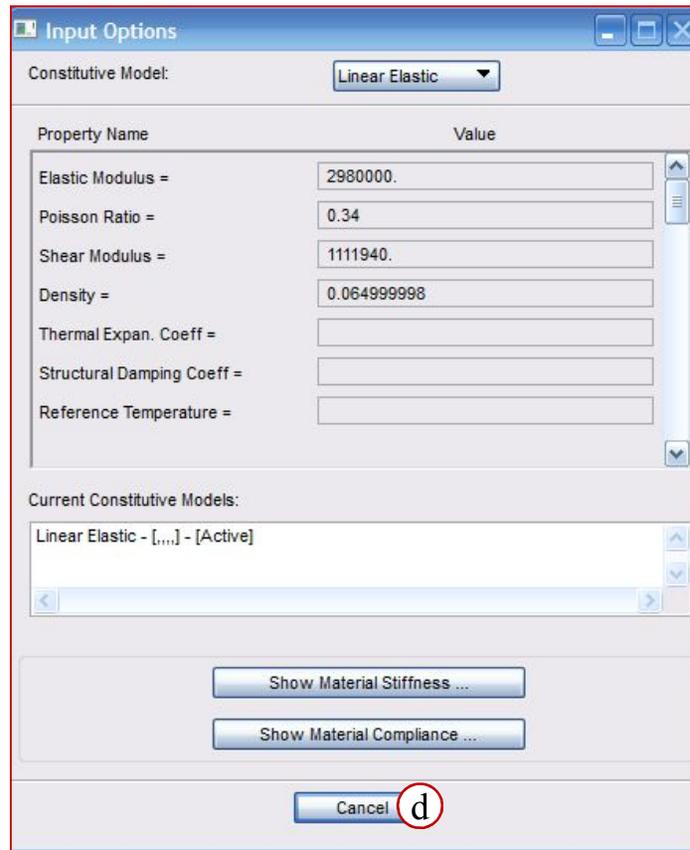


Step 4. Verify the Material Properties

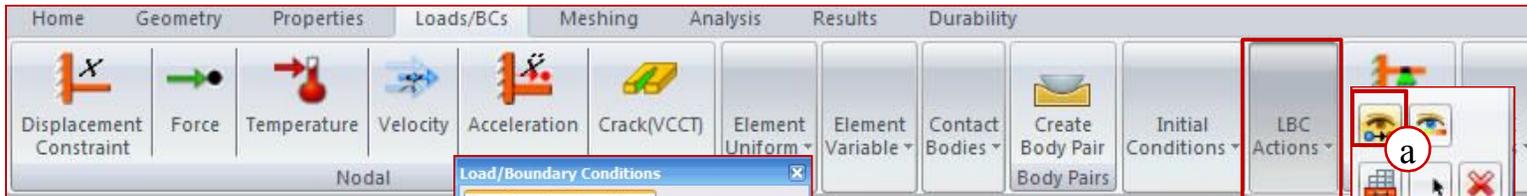


Check the materials.

- Under the *Properties* tab, click **Show Material** in the *Property Actions* group.
- Highlight **mat1.1** under *Existing Materials* ensure that the material properties are identical to the properties given in the Model Description.
- Repeat for **mat1.2** and **mat1.3**.
- Click **Cancel**

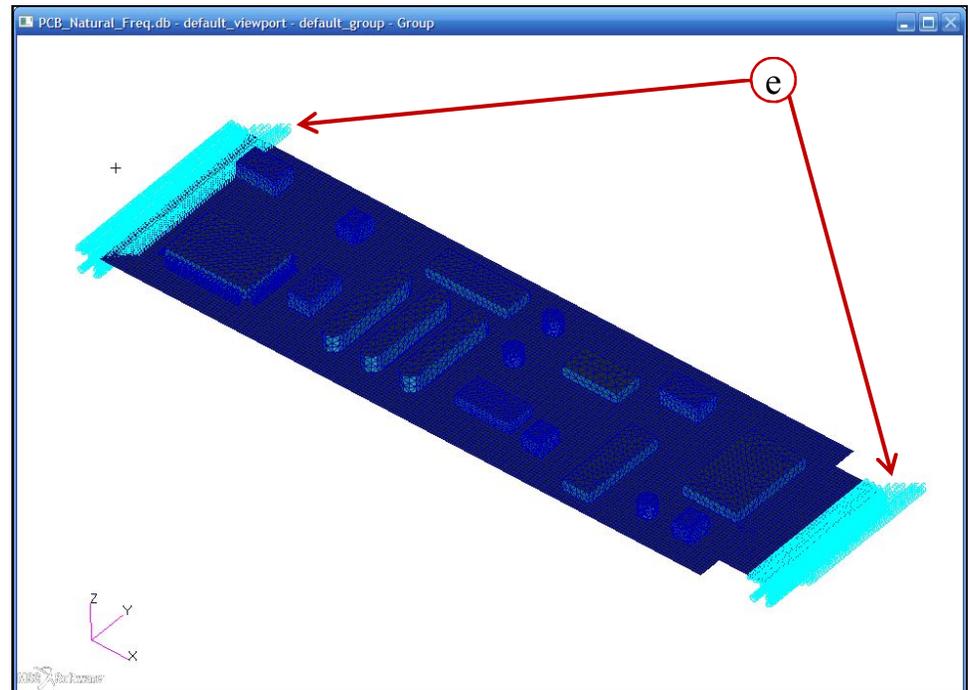
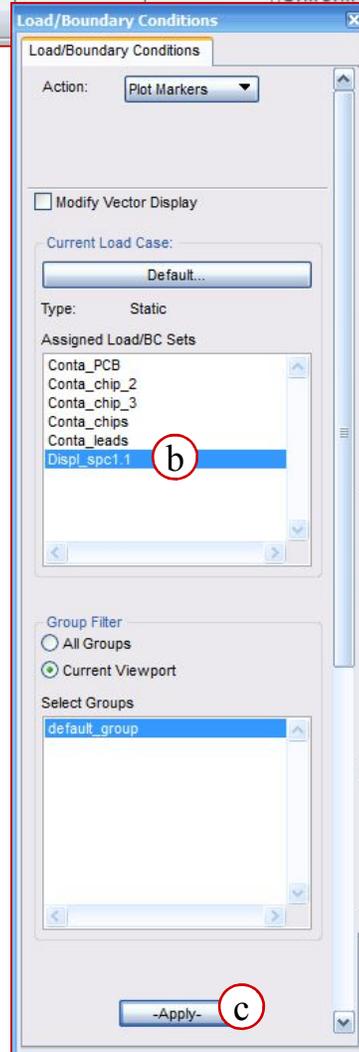


Step 5. Verify the Boundary Conditions



Check the constraints.

- a. Under the *Loads/BCs* tab, click **Plot Markers** in the *LBC Actions* group.
- b. Highlight **Displ_spc1.1** under *Assigned Load/BC Sets*.
- c. Click **Apply**.
- d. Shade the model.
- e. Observe the BC markers showing the ends of the PCB are fully fixed.
- f. Click **Reset Graphics**.

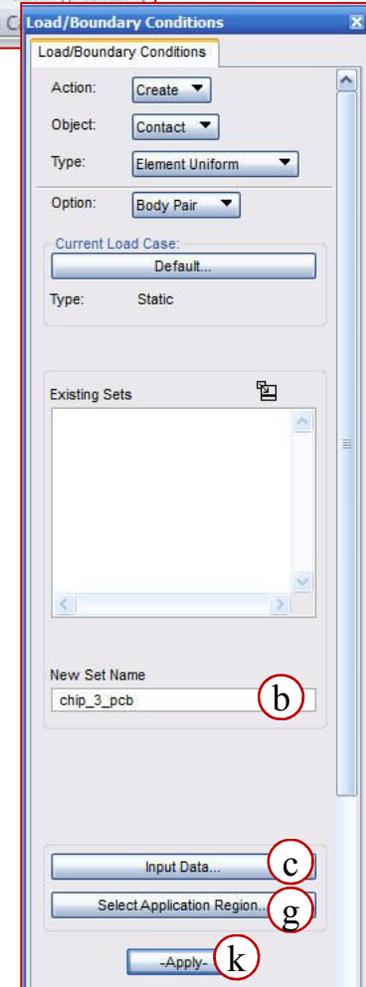
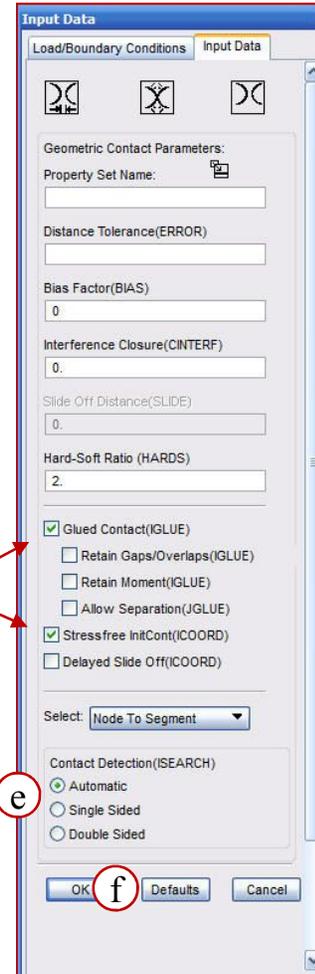
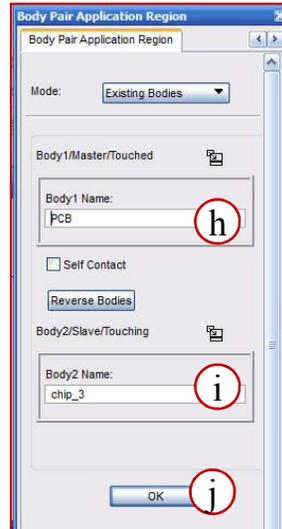


Step 6. Create Body Pairs



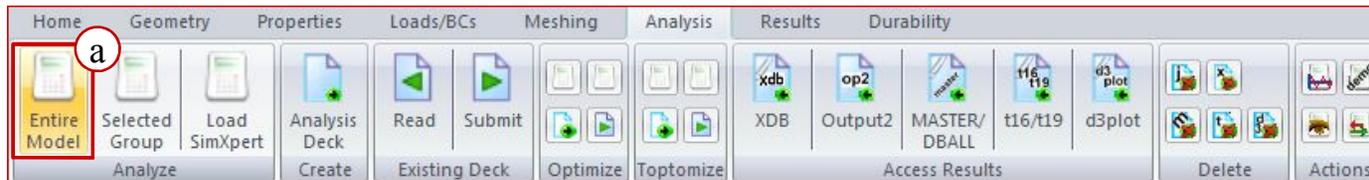
Create Body Pairs:

- a. Under the *Load/BCs* tab click **Create Body Pair** in the *Body Pairs* group.
- b. Enter **chip_3_pcb** for *New Set Name*.
- c. Click **Input Data...**
- d. Check **Glued Contact(IGLUE)** and **Stressfree InitCont(ICOORD)**
- e. Select **Automatic** for *Contact Detection (ISEARCH)*.
- f. Click **OK**
- g. Click **Select Application Region**
- h. Select **PCB** for *Body1/Master/Touched*.
- i. Select **chip_3** for *Body2/Slave/Touching*.
- j. Click **OK**
- k. Click **Apply**
- l. Repeat steps b-k for the bodies indicated in the table on the right (follow the same naming convention)



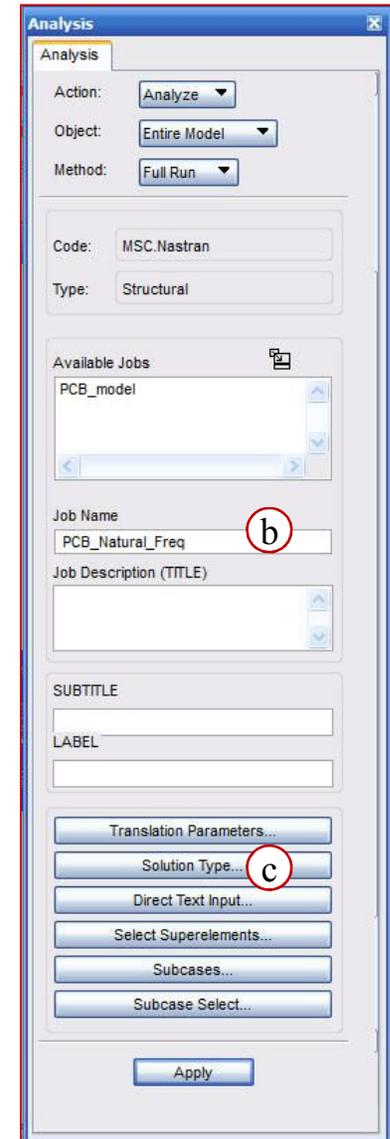
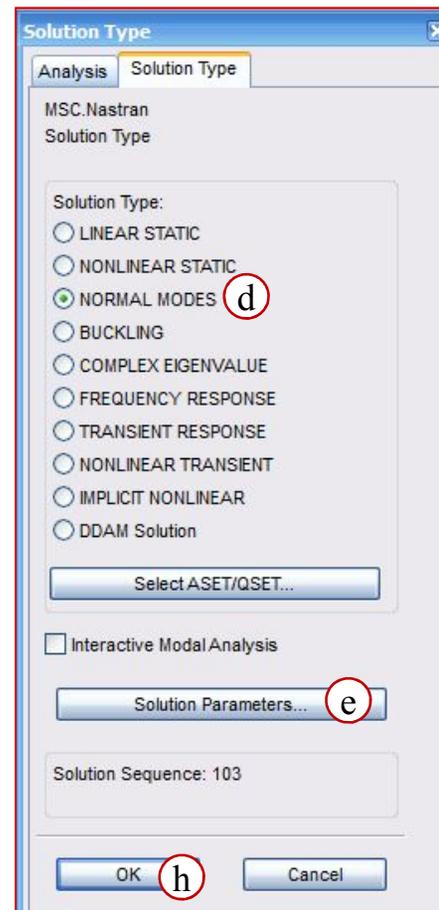
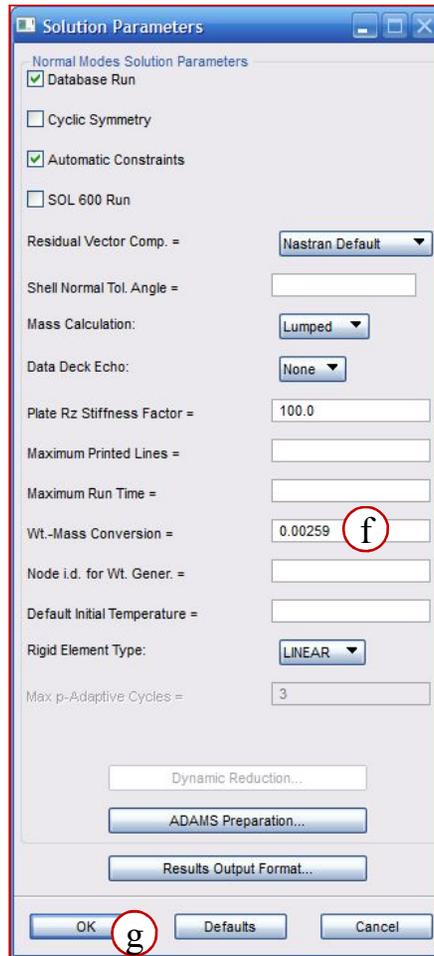
Touching Body	Touched Body
chip_2	leads
leads	PCB
chips	PCB

Step 7. Set Up a Normal Modes Analysis



Set the Solution Type to SOL103, Normal Modes.

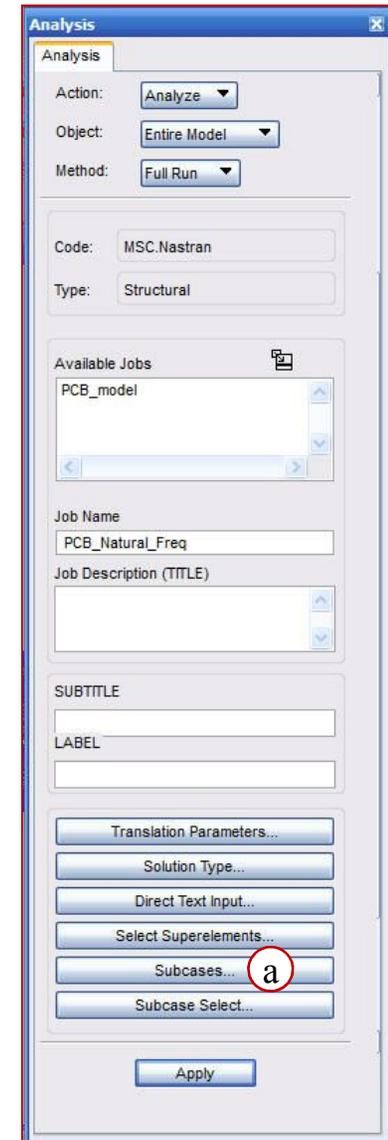
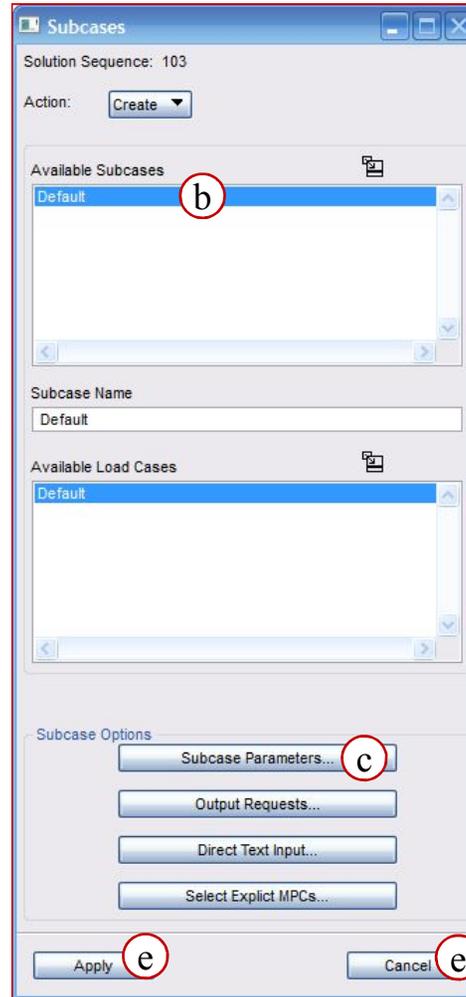
- a. Under the *Analysis* tab, click **Entire Model** in the *Analyze* group.
- b. Enter **PCB_Natural_Freq** for *Job Name*.
- c. Click **Solution Type**.
- d. Select **Normal Modes** for *Solution Type*.
- e. Click **Solution Parameters**.
- f. Enter **0.00259** for *Wt.-Mass Conversion*.
- g. Click **OK**.
- h. Click **OK**.



Step 7. Set Up a Normal Modes Analysis (Cont.)

Enter the number of desired roots.

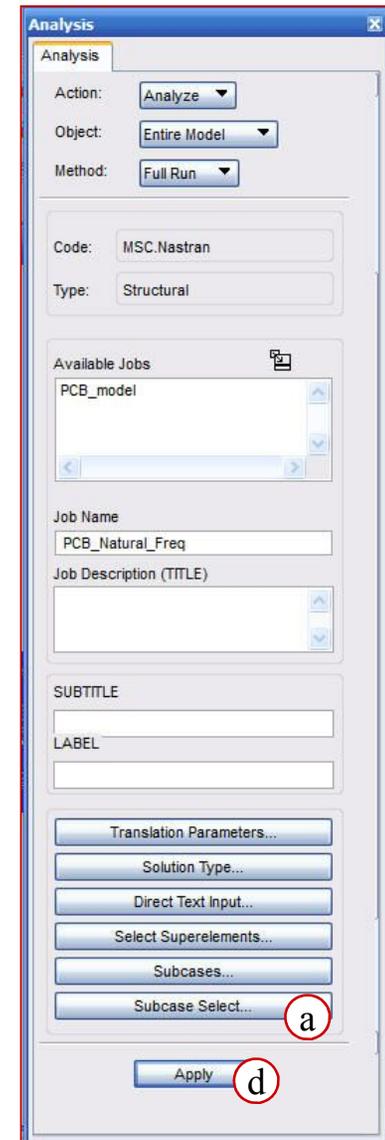
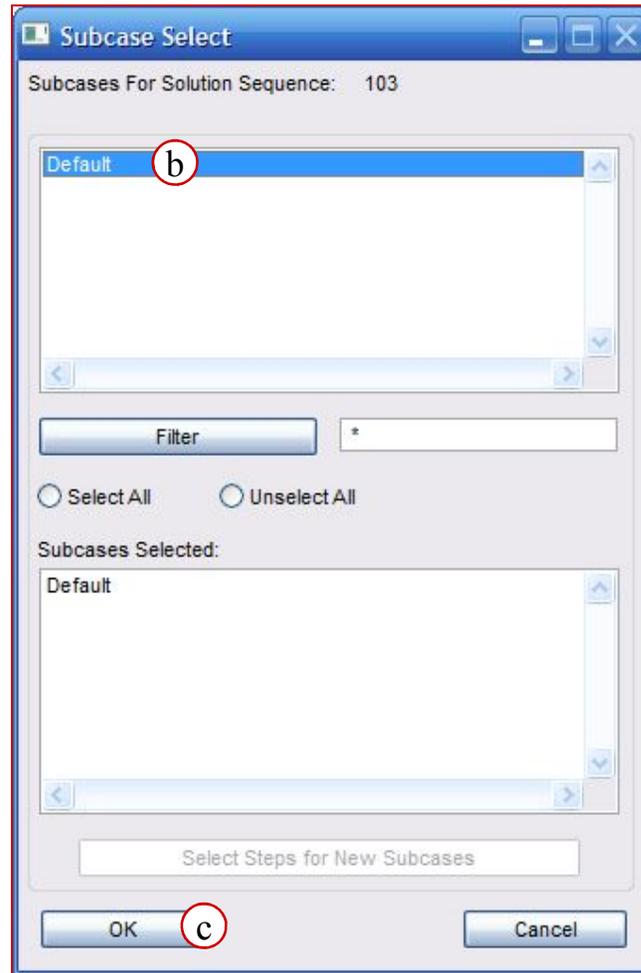
- a. Click **Subcases**.
- b. Highlight **Default** under *Available Subcases*.
- c. Click **Subcase Parameters**.
- d. Make sure *Number of Desired Roots* is **10**.
- e. Click **OK**
- f. Click **Apply**, then **Cancel**.



Step 7. Set Up a Normal Modes Analysis (Cont.)

Select the *Default* subcase and run the job.

- a. Click **Subcase Select**.
- b. Select **Default**
- c. Click **OK**.
- d. Click **Apply**.

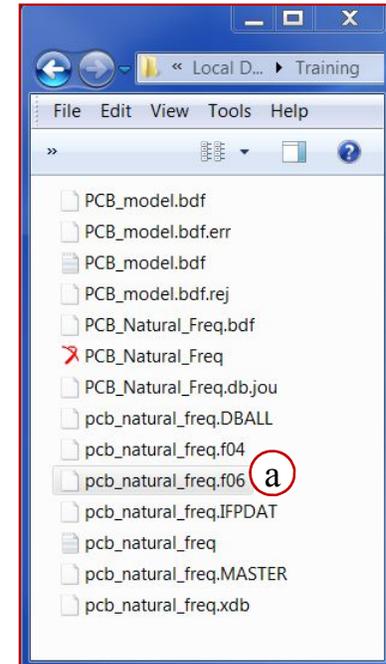
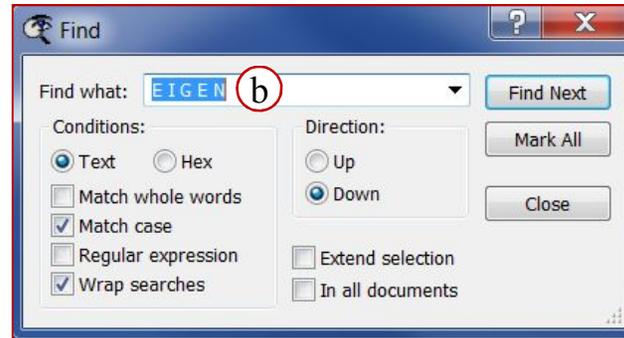


Step 8. Review the Results File

Review the eigenvalues in the .f06 file

- Navigate to and open the file **pcb_natural_freq.f06** in a text editor.
- Search on the string **"E I G E N"**
- Check the natural frequencies.

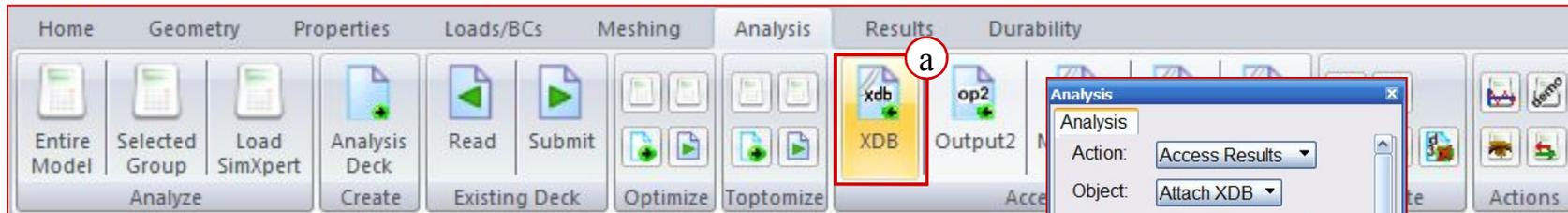
There are six near-zero natural frequencies present. This indicates that there are rigid body modes in the model.



c

MODE NO.	EXTRACTION ORDER	EIGENVALUE	R E A L E I G E N V A L U E S		GENERALIZED MASS	GENERALIZED STIFFNESS
			RADIANS	CYCLES		
1	1	-6.196849E-03	7.872006E-02	1.252869E-02	1.000000E+00	-6.196849E-03
2	2	-9.603342E-04	3.098926E-02	4.932094E-03	1.000000E+00	-9.603342E-04
3	3	3.291345E-03	5.737024E-02	9.130757E-03	1.000000E+00	3.291345E-03
4	4	4.171615E-03	6.458805E-02	1.027951E-02	1.000000E+00	4.171615E-03
5	5	9.478791E-03	9.735908E-02	1.549518E-02	1.000000E+00	9.478791E-03
6	6	1.068012E-02	1.033447E-01	1.644782E-02	1.000000E+00	1.068012E-02
7	7	1.140721E+06	1.068045E+03	1.699847E+02	1.000000E+00	1.140721E+06
8	8	8.925751E+06	2.987600E+03	4.754912E+02	1.000000E+00	8.925751E+06
9	9	1.343186E+07	3.664950E+03	5.832949E+02	1.000000E+00	1.343186E+07
10	10	3.623969E+07	6.019941E+03	9.581033E+02	1.000000E+00	3.623969E+07

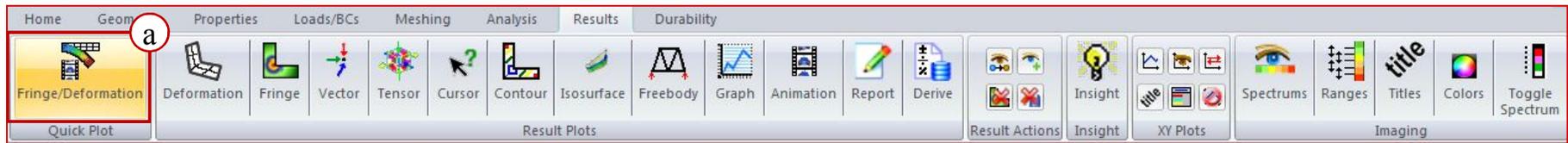
Step 9. Attach the Results File



Import the results.

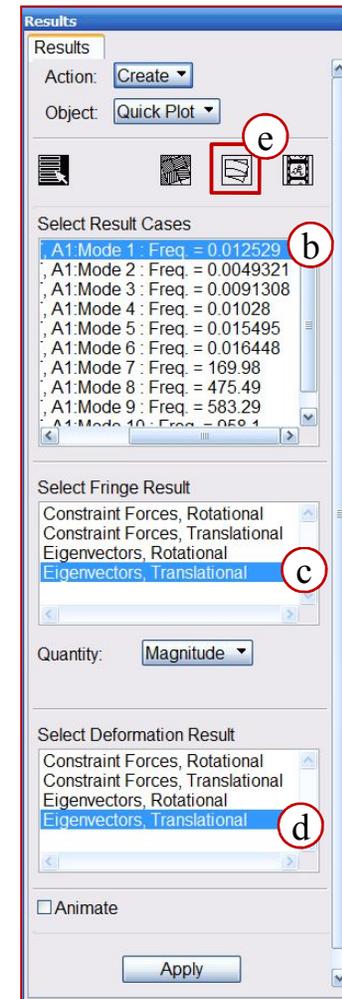
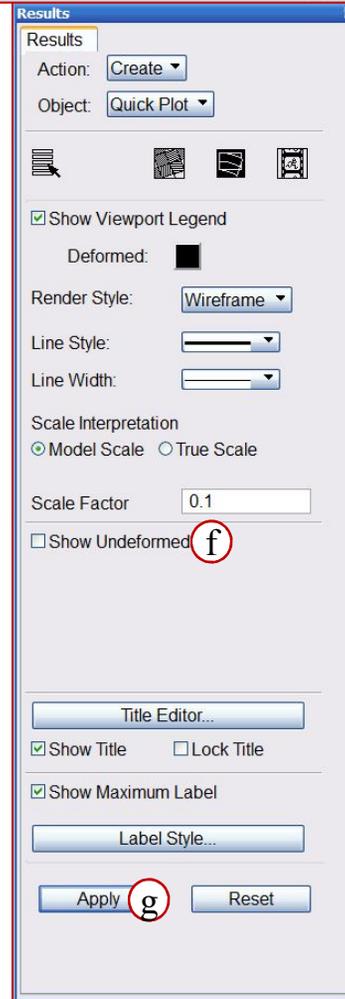
- a. Click **XDB** in the *Access Results* group.
- b. Select **PCB_Natural_Freq** in the *Available Jobs* list.
- c. Click **Apply**.

Step 10. View the Deformed Shape

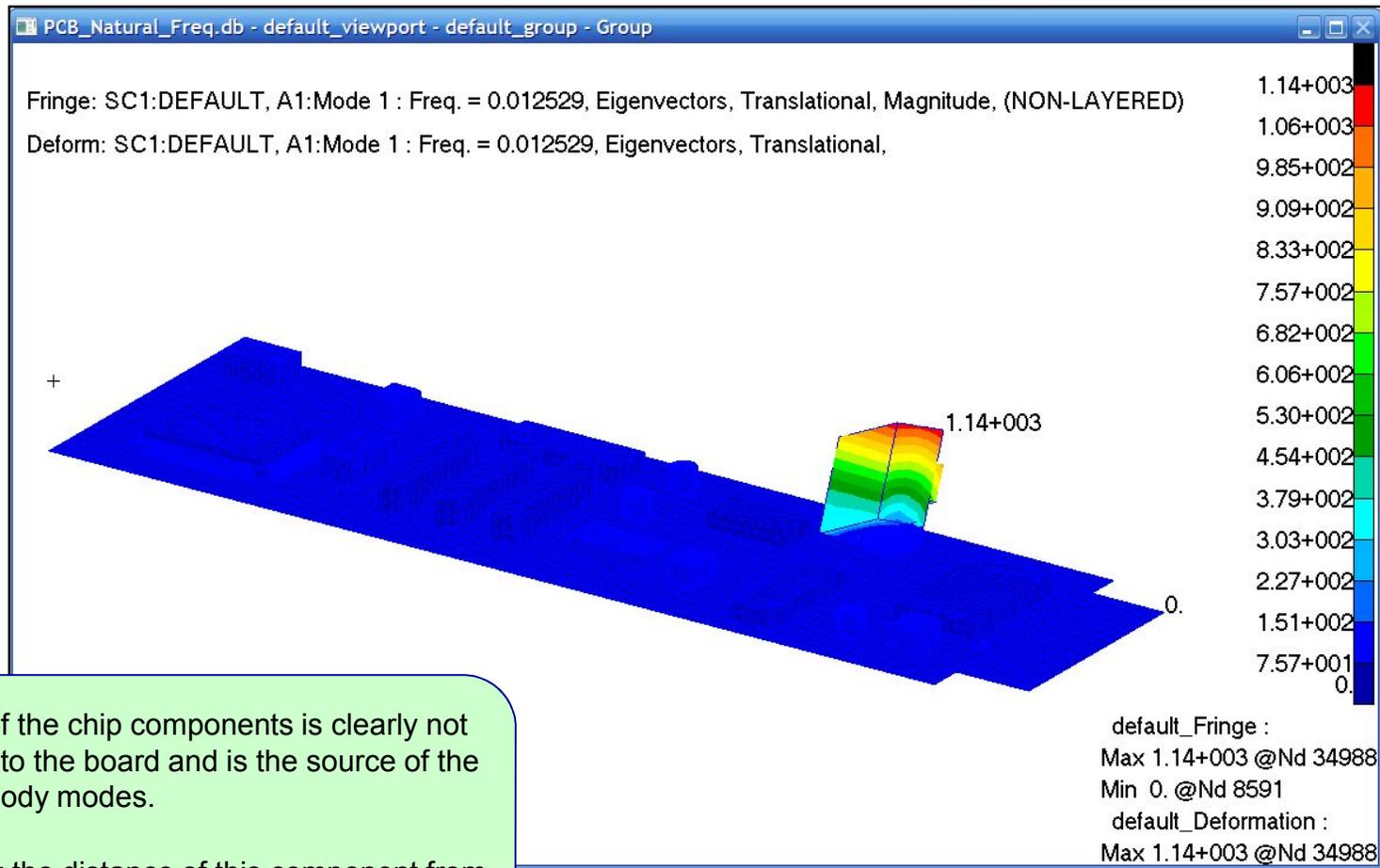


Plot the deformed shape.

- a. Under the *Results* tab, click **Fringe/Deformation** in the *Quick Plot* group.
- b. Highlight **Default, A1:Mode 1 : Freq = 0.012529** under *Select Result Cases*.
- c. Highlight **Eigenvectors, Translational** under *Select Fringe Result*.
- d. Highlight **Eigenvectors, Translational** under *Select Deformation Result*.
- e. Click the **Deform Attributes** icon.
- f. Uncheck *Show Undeformed*.
- g. Click **Apply**.



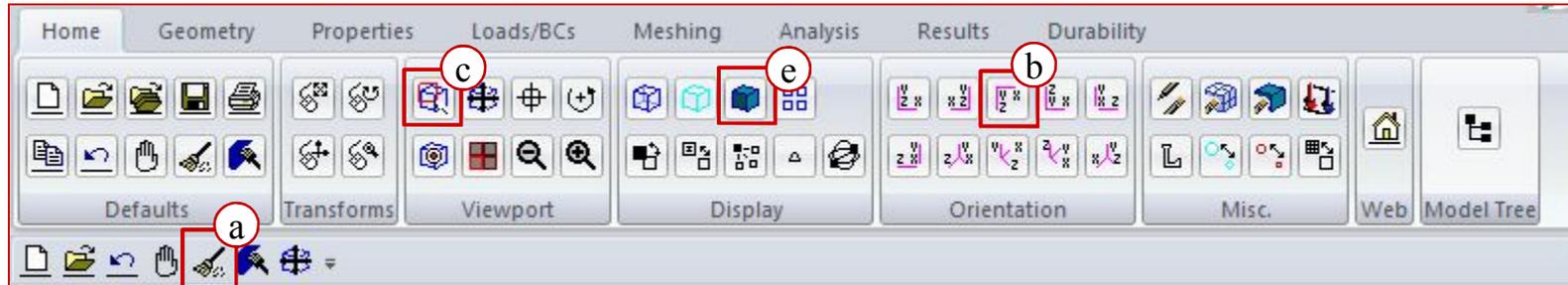
Step 10. View the Deformed Shape (Cont.)



One of the chip components is clearly not glued to the board and is the source of the rigid body modes.

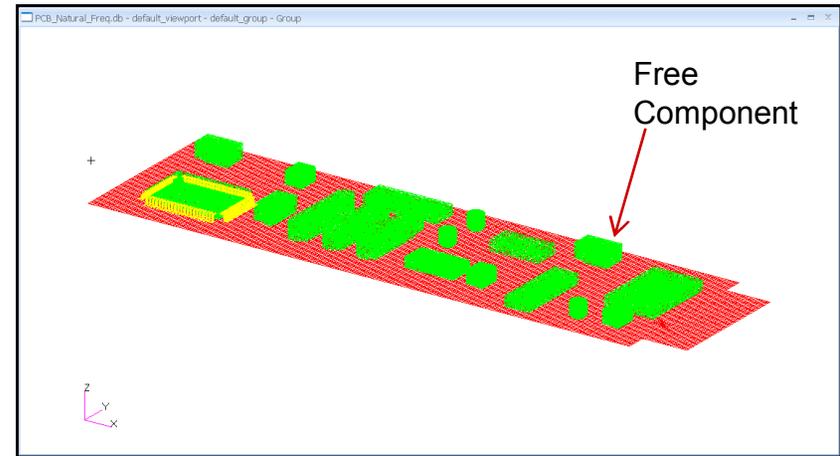
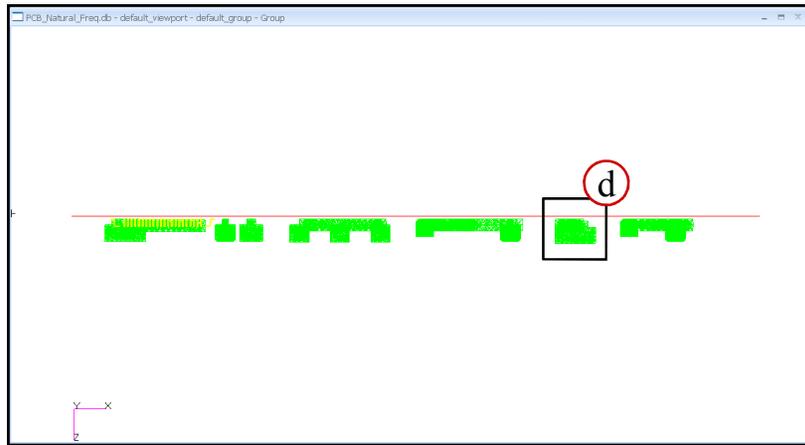
Check the distance of this component from the board and adjust the contact tolerance (ERROR) parameter.

Step 11. Measure the Distance Between Chip 3 and PCB

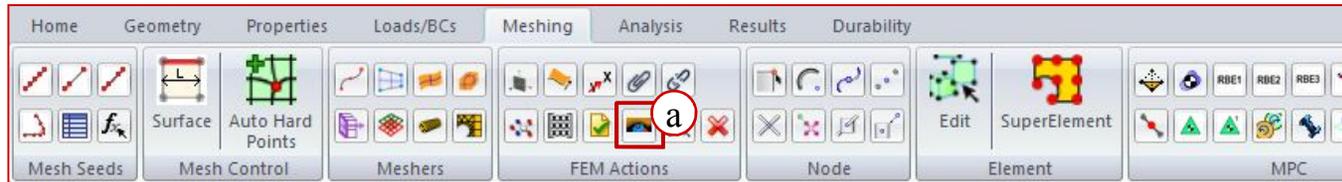


Set the view to *Top View* for easy measuring.

- a. Click **Reset Graphics**.
- b. Under the *Home* tab, click **Top View**.
- c. Click **View Corners**
- d. Drag a rectangle around the component in question.
- e. Click **Smooth Shaded**.

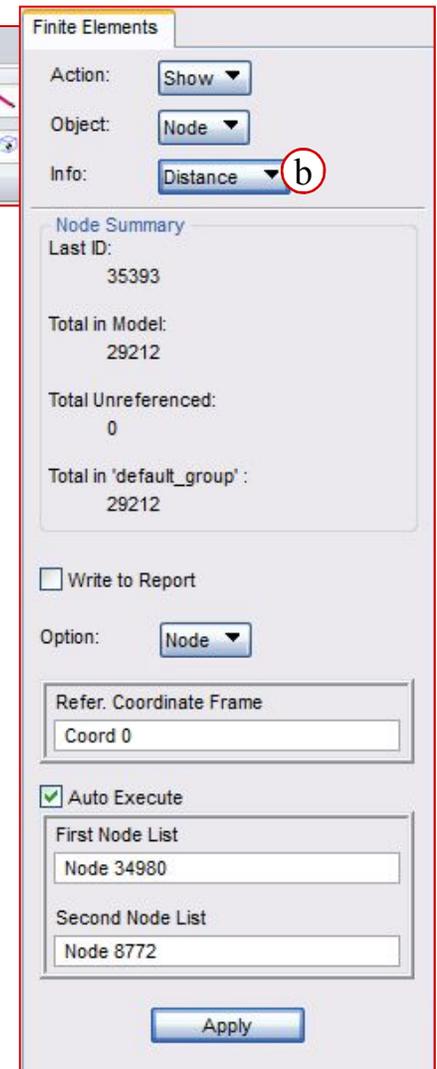
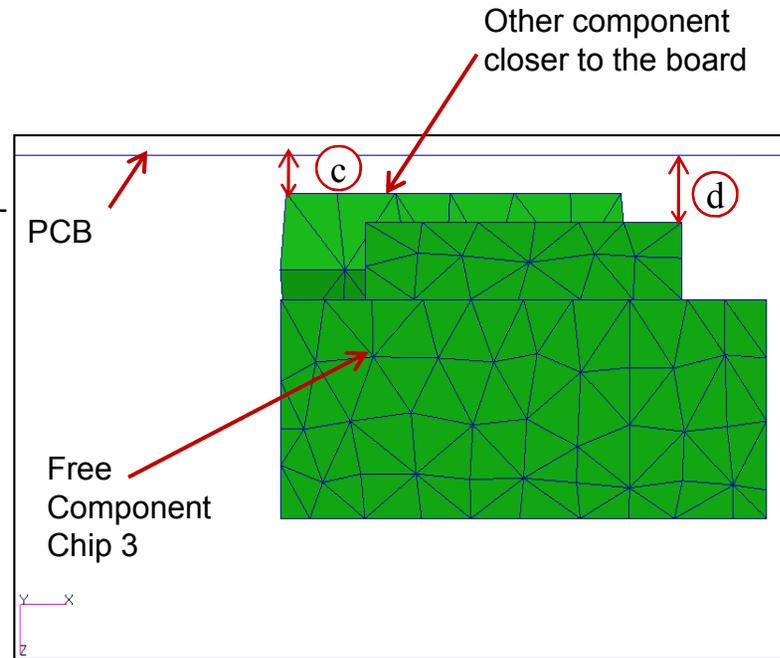


Step 11. Measure the Distance Between Chip 3 and PCB (Cont.)



Check the component distance from the board.

- Under the *Meshing* tab, click **Show** in the *FEM Actions* group
- Pull down **Distance** for *Info* to determine the distance in the Z-direction from the PCB to the components.
- First for comparison, check the distance of the component that is closer to the PCB. Choose a node on the upper edge of the component and any node on the PCB
- Check the distance of the free component, Chip 3. Choose a node on the upper edge of the component and any node on the PCB.



Step 11. Measure the Distance Between Chip 3 and PCB (Cont.)

Check the component distance from the board continue.

- a. Note the reported Z distances.
- b. Click **Cancel** to close the table.

From Node ID	To Node ID	Distance	Delta $\langle x y z \rangle$	Reference CID
1141	8761	1.2395231	$\langle -0.00115 1.24 -0.0194 \rangle$	(Global) Rectangular
34980	8772	0.30074114	$\langle -0.00609 0.299 -0.0344 \rangle$	(Global) Rectangular

Page 1 of 3 Distance

Reset Cancel

The standard distance of the components from the PCB is 0.0194". The component in question is 0.0344" away, and is not falling within the default contact tolerance.

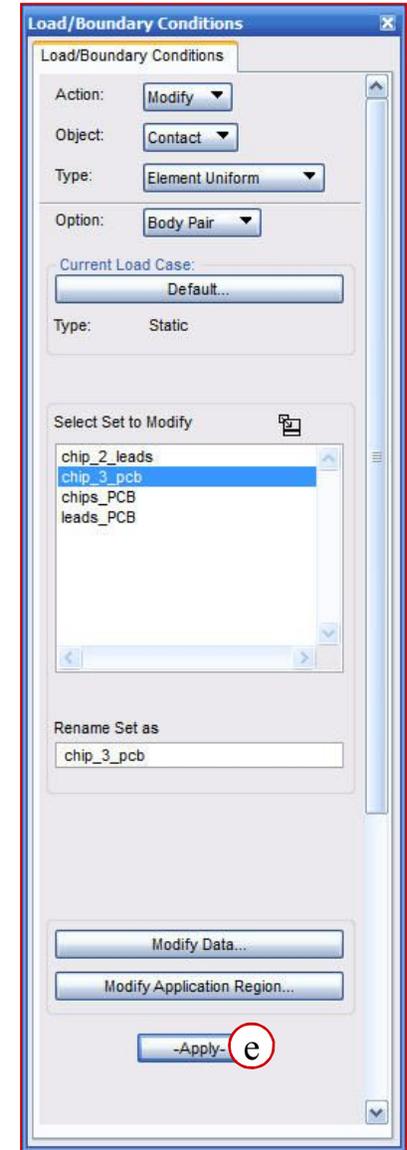
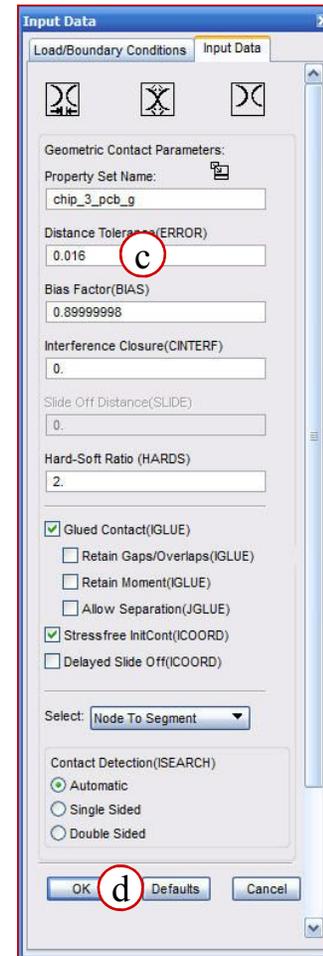
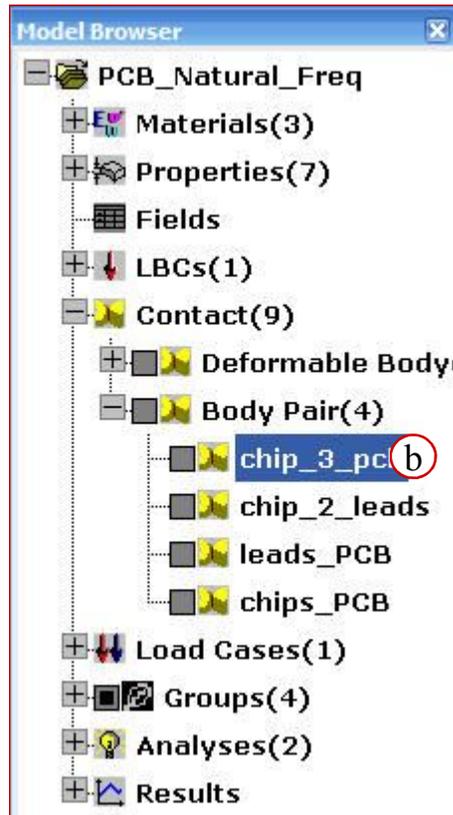
PCB thickness = 0.0389"
 $0.0344" - 0.0389" / 2 = 0.015"$
Try a Distance Tolerance of 0.016"

Step 12. Modify Body Pair

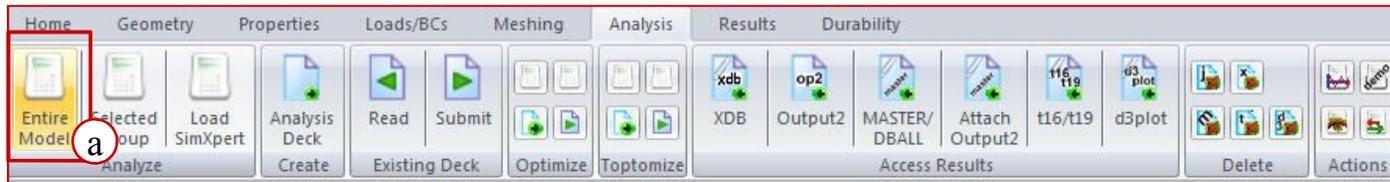


Modify the chip_3_pcb body pair:

- Under the *Home* tab, click **Show/Hide Model Browser Tree** in the *Model Tree* Group.
- Double click on **chip_3_pcb** under **Contact > Body Pair**
- Enter **0.016** for *Distance Tolerance(ERROR)*.
- Click **OK**
- Click **Apply**

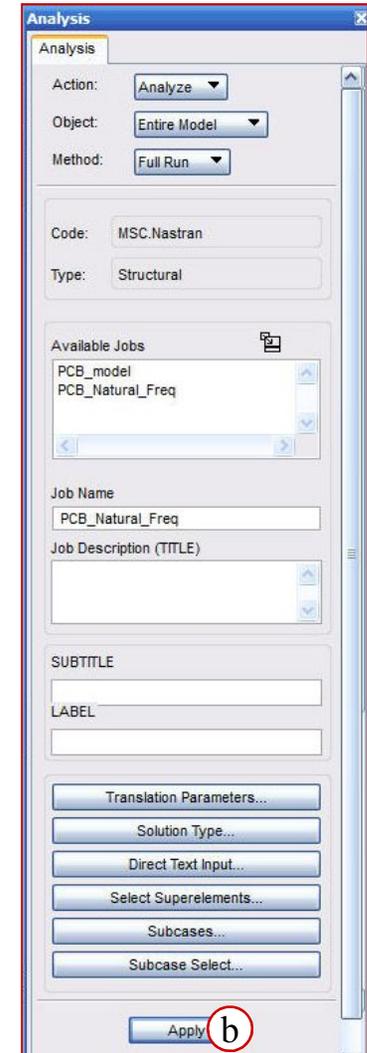
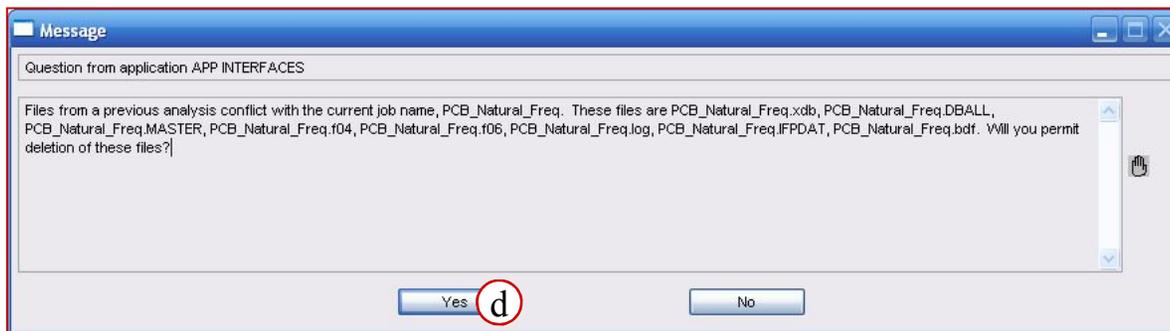
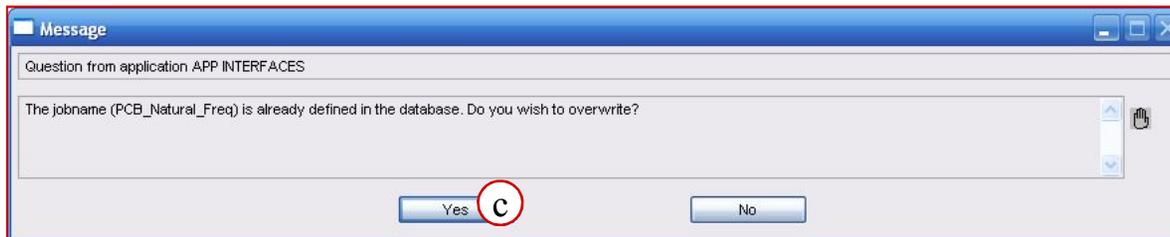


Step 13. Rerun the Analysis



Return to the *Analysis* form and rerun the job.

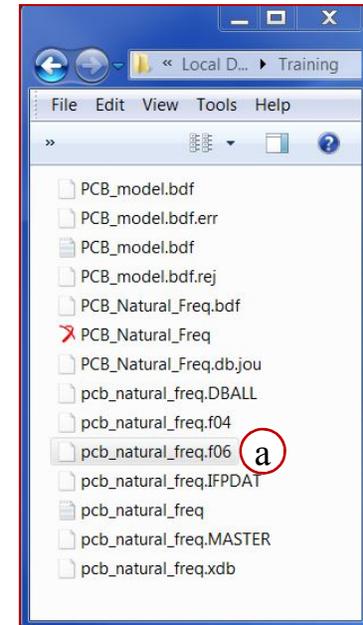
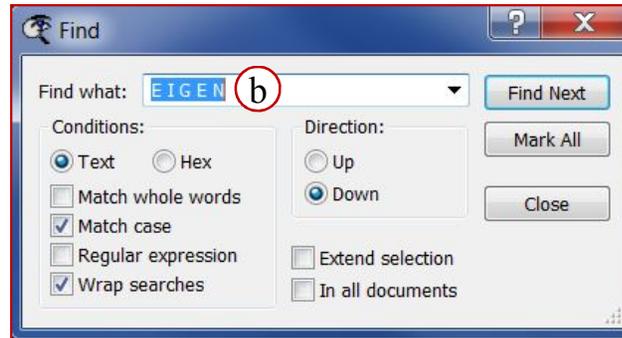
- Under the *Analysis* Tab, click **Entire Model** in the *Analyze* Group.
- Click **Apply**
- Click **Yes** to overwrite the old job.
- Click **Yes** to overwrite the old files.



Step 14. Review the New Results File

Review the f06 file.

- Navigate to and open the file **pcb_natural_freq.f06**
- Search on the string **"E I G E N"** and check the natural frequencies.



MODE NO.	EXTRACTION ORDER	EIGENVALUE	R E A L E I G E N V A L U E S		GENERALIZED MASS	GENERALIZED STIFFNESS
			RADIANS	CYCLES		
1	1	1.114580E+06	1.055737E+03	1.680257E+02	1.000000E+00	1.114580E+06
2	2	8.405630E+06	2.899246E+03	4.614294E+02	1.000000E+00	8.405630E+06
3	3	1.330572E+07	3.647701E+03	5.805497E+02	1.000000E+00	1.330572E+07
4	4	3.533292E+07	5.944150E+03	9.460408E+02	1.000000E+00	3.533292E+07
5	5	6.221519E+07	7.887661E+03	1.255360E+03	1.000000E+00	6.221519E+07
6	6	1.110568E+08	1.053835E+04	1.677230E+03	1.000000E+00	1.110568E+08
7	7	1.540715E+08	1.241255E+04	1.975519E+03	1.000000E+00	1.540715E+08
8	8	2.194564E+08	1.481406E+04	2.357731E+03	1.000000E+00	2.194564E+08
9	9	2.701275E+08	1.643556E+04	2.615800E+03	1.000000E+00	2.701275E+08
10	10	3.505131E+08	1.872200E+04	2.979698E+03	1.000000E+00	3.505131E+08

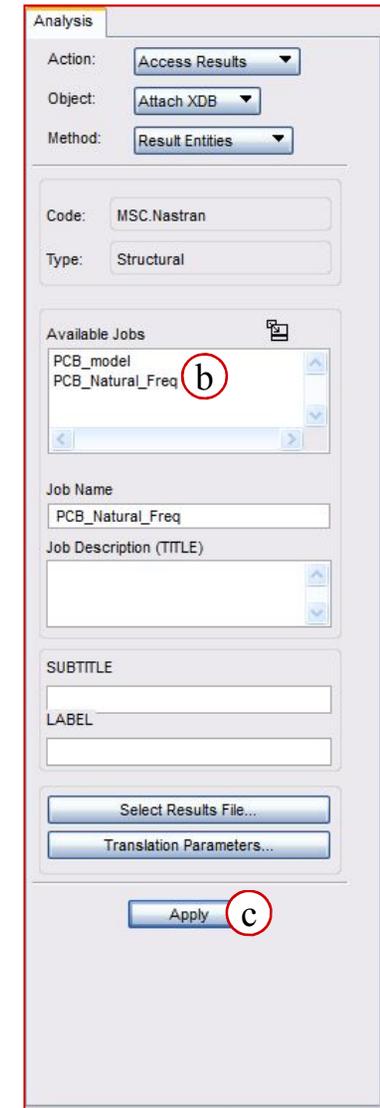
This time there are now no rigid body modes

Step 15. Attach the New Results File

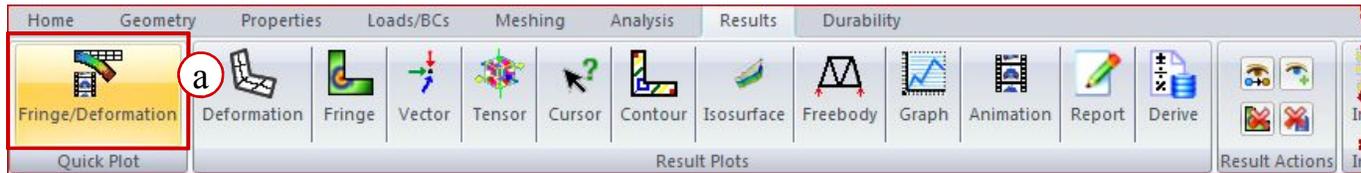


Import the results.

- a. Click **XDB** in the *Access Results* group.
- b. Select **PCB_Natural_Freq**
- c. Click **Apply**.

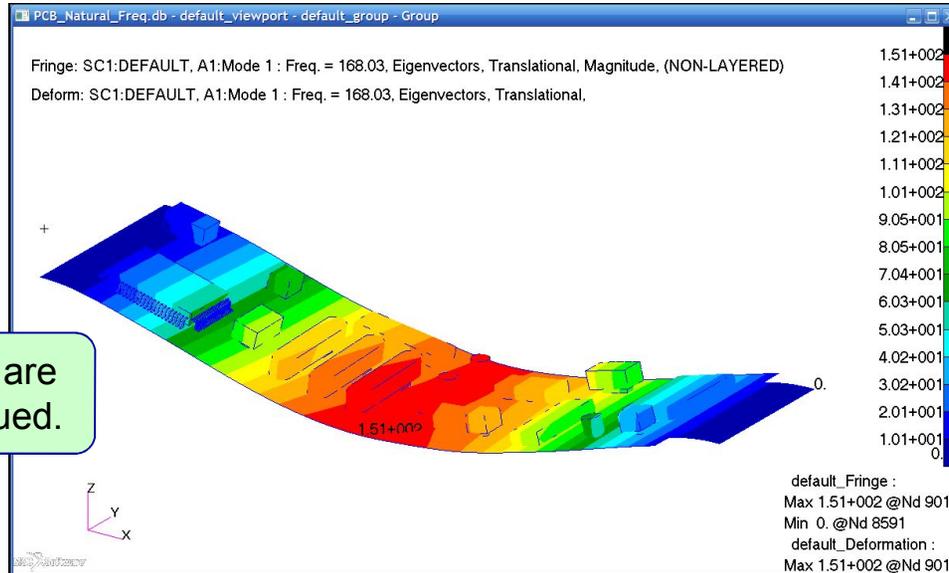
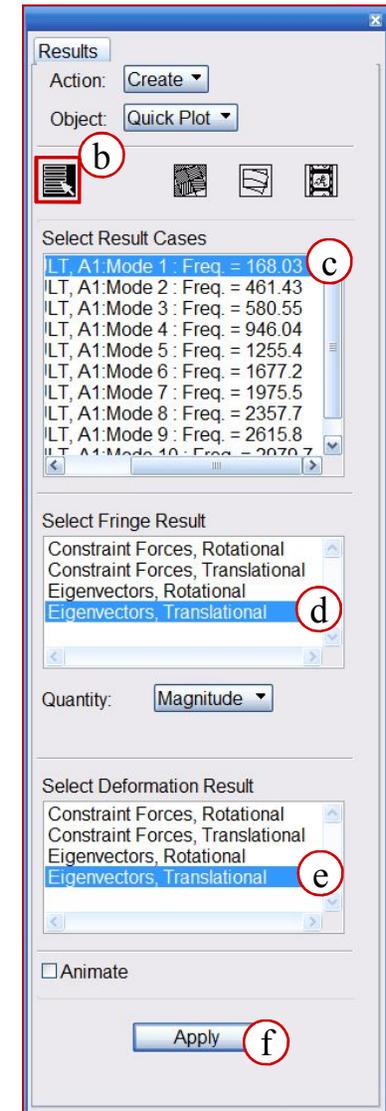


Step 16. View the Deformed Shape



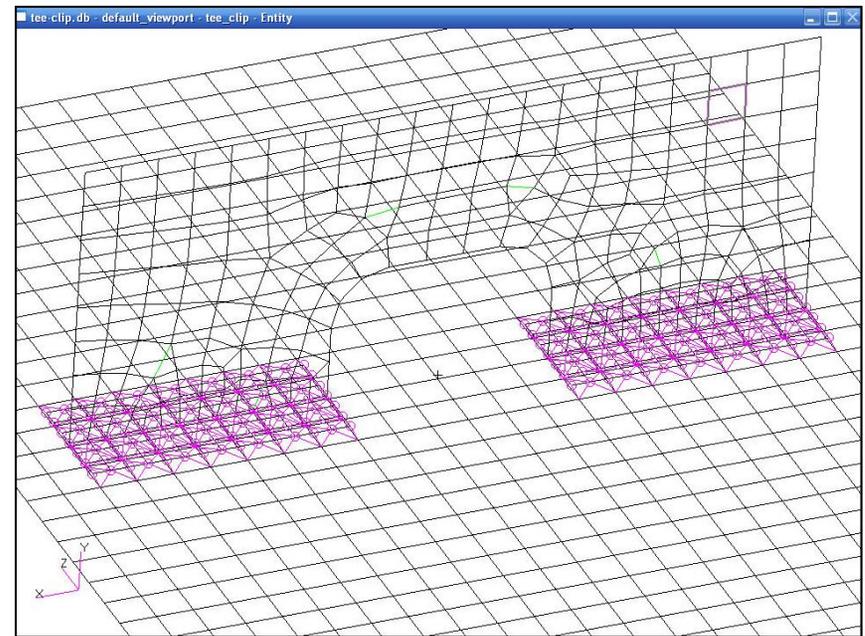
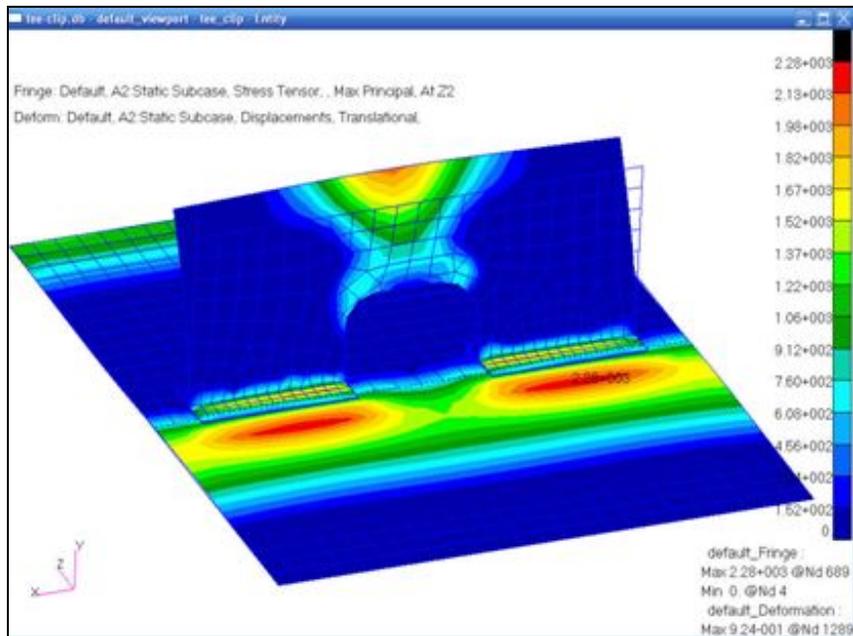
Plot the deformed shape.

- a. Under the *Results* tab, click **Fringe/Deformation** in the *Quick Plot* group.
- b. Click **Select Results**
- c. Highlight **Default, A1:Mode 1 :Freq = 168.03** under *Select Result Cases*.
- d. Highlight **Eigenvectors, Translational** under *Select Fringe Result*.
- e. Highlight **Eigenvectors, Translational** under *Select Deformation Result*.
- f. Click **Apply**.



WORKSHOP 5

SHELL FACE TO FACE CONTACT





- **Workshop Objectives**

- Create glued contact between shell faces.
- Create and view the MPCs which model the glued contact.
- See the difference in contact order on the quality of the glued contact.

- **Software Version**

- MSC Nastran 2013.1
- Patran 2013

- **Files Required**

- **tee-clip_fe.bdf**

• Suggested Steps

1. Build the starting model by importing the MSC Nastran input file **tee-clip_fe.bdf**.
2. Define two deformable contact bodies using the Contact Body/Pair Tool, one for the plate and one for the tee. Color code the bodies.
3. Create a body pair between the plate and tee
 - Check Glued Contact and Stress free InitCont
 - Select double sided contact detection
4. Set up and run a linear static analysis:
 - Set Solution Type to Linear Static
 - Change the Bias Tolerance to **0.0**
 - Specify glued contact between the contact bodies
 - Request a punch file that contains the MPCs used to model the glued contact be saved
5. Attach the xdb results file.
6. Post-Process Results in Patran:
 - Review the deformation results
 - Review max Principal stress results.

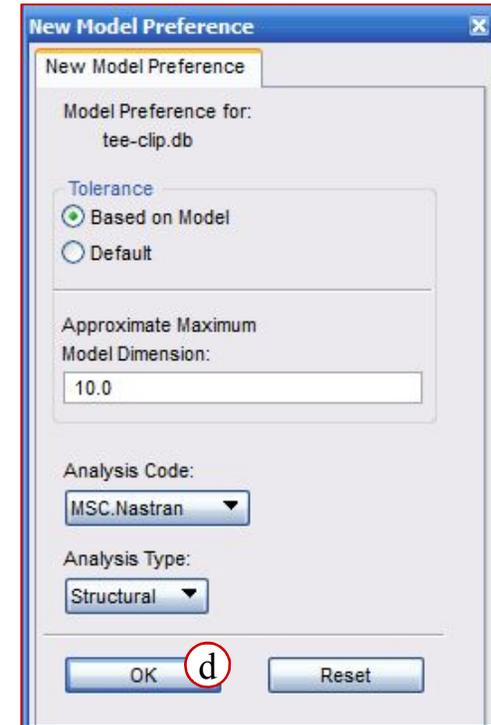
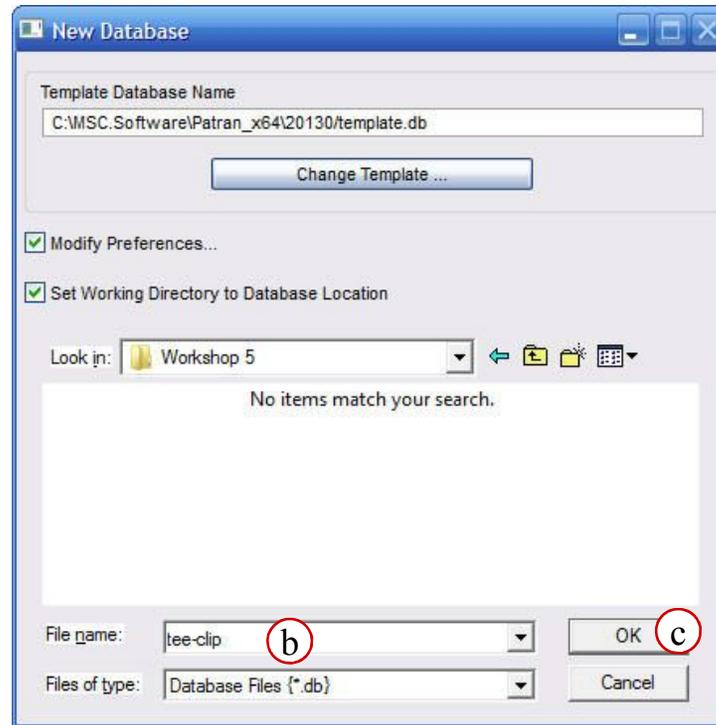
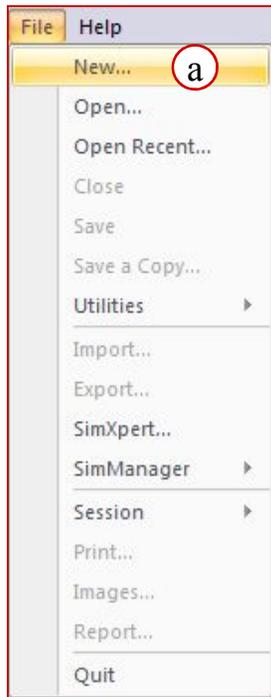
- **Suggested Steps (Cont.)**

7. Import the punch file that contains the MPCs. Review the pattern of contact.
8. Modify the Body Pair to select automatic contact detection.
9. Attach the new xdb result file.
10. Create a fringe plot of maximum principal stress in the contact region. Compare the plot to that obtained in step 6.
11. Import the punch file for this analysis. Review the pattern of contact.

Step 1. Import the Model

Open a new database.

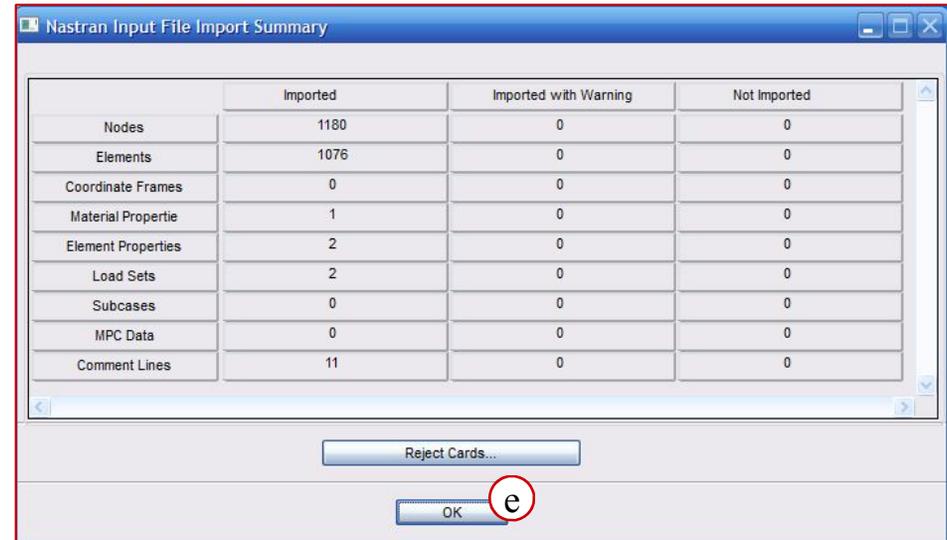
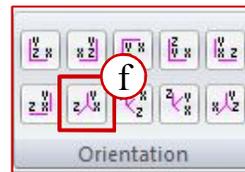
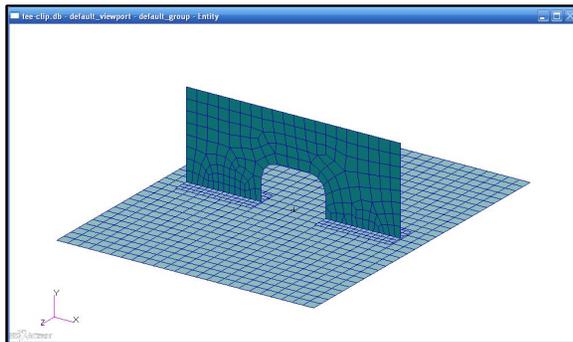
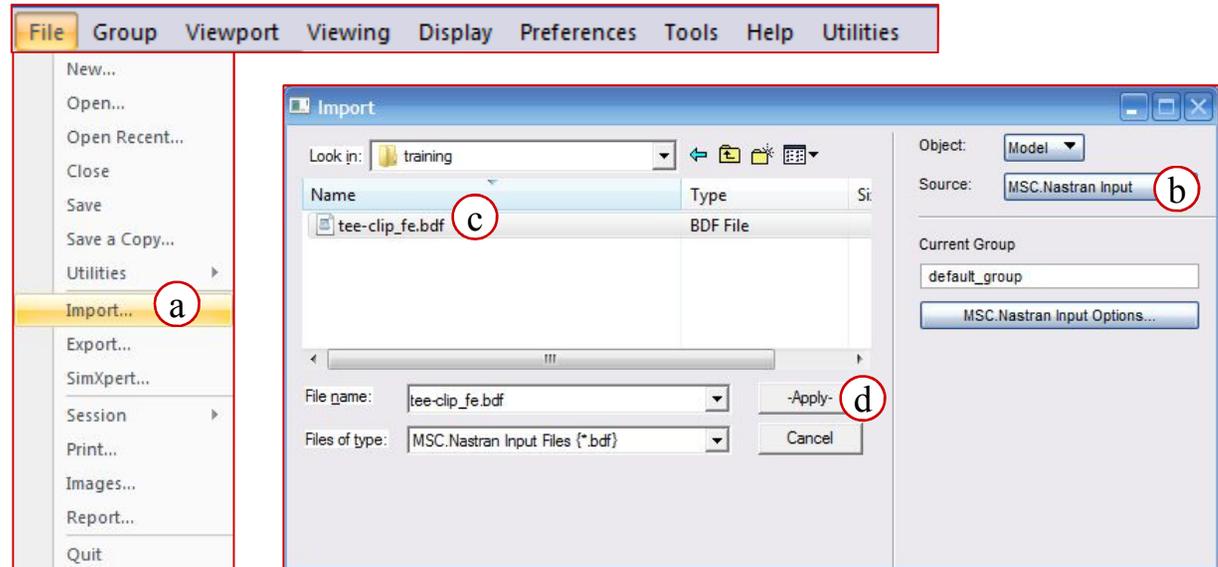
- a. Pull down **File > New**
- b. Enter **tee-clip.db** for *File name*
- c. Click **OK**.
- d. Click **OK** on the *New Model Preference* form.



Step 1. Import the Model (Cont.)

Import the model.

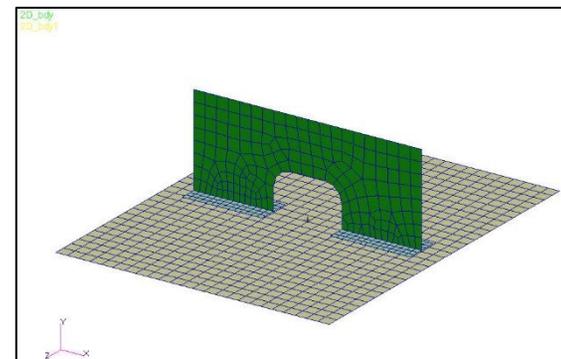
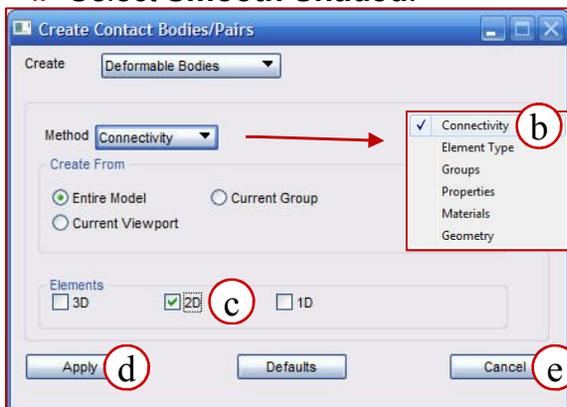
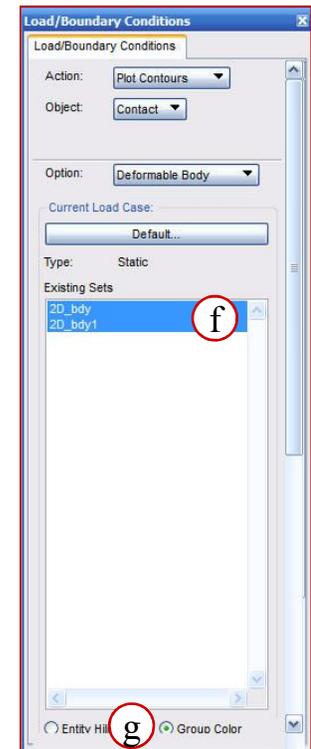
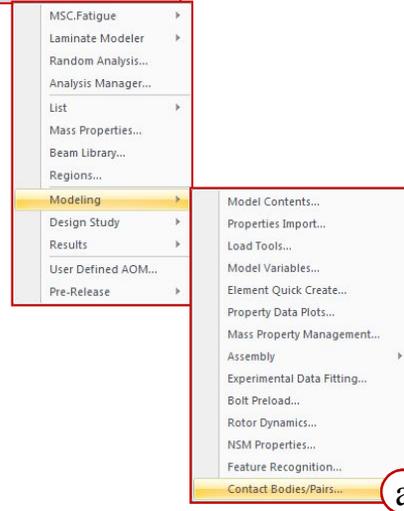
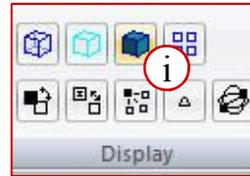
- a. Pull down **File > Import**.
- b. Pull down **MSC Nastran Input** for *Source*.
- c. Select **tee-clip_fe.bdf** as the file to import.
- d. Click **Apply**.
- e. Review the *MSC Nastran Input File Import Summary* then click **OK**.
- f. Orient the model to **Iso 1 view**
- g. Click **Smooth shaded** in the *Display* group
- h. Click **Fit view**.



Step 2. Define Two Deformable Contact Bodies

Define Contact Bodies using the Create Contact Body/Pair Tool:

- Pull down **Tools > Modeling > Contact Bodies/Pair**
- Pull down *Method* to **Connectivity**.
- Select **2D** for *Elements*
- Click **Apply** and **Cancel**.
- Under the *Loads* Tab, select **Color Code Bodies** in the *Contact Bodies* Group
- Highlight *Existing Sets* **2D_bdy** and **2D_bdy1**.
- Select **Group Color**
- Click **Apply** (not shown)
- Select **Smooth Shaded**.

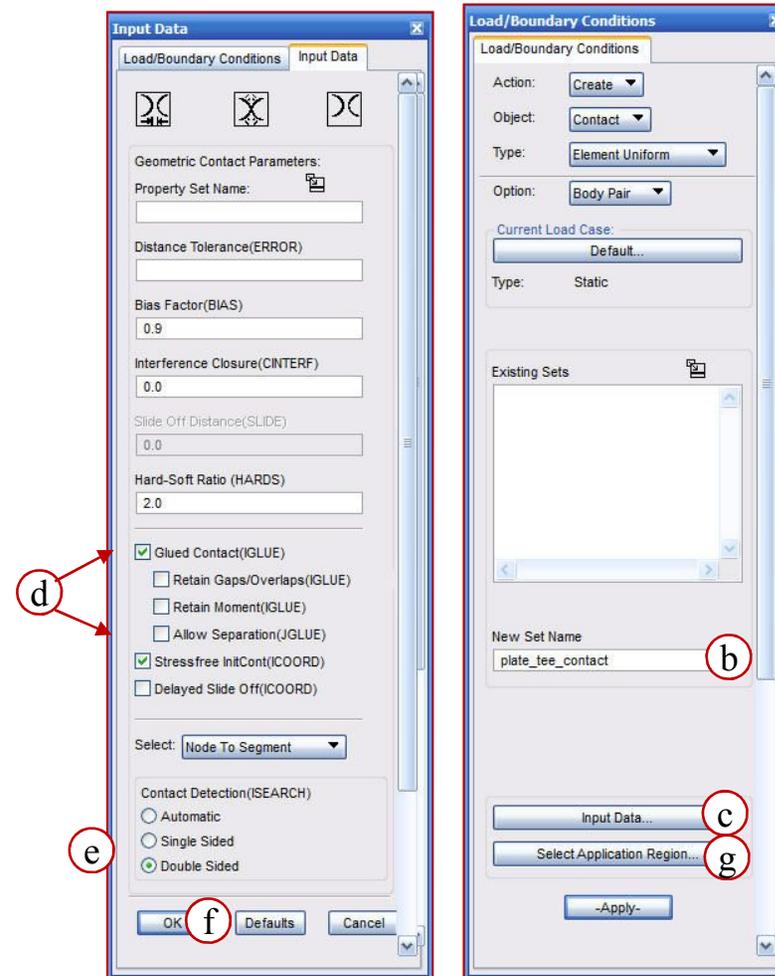


Step 3. Create Body Pair



Create a New Body Pair:

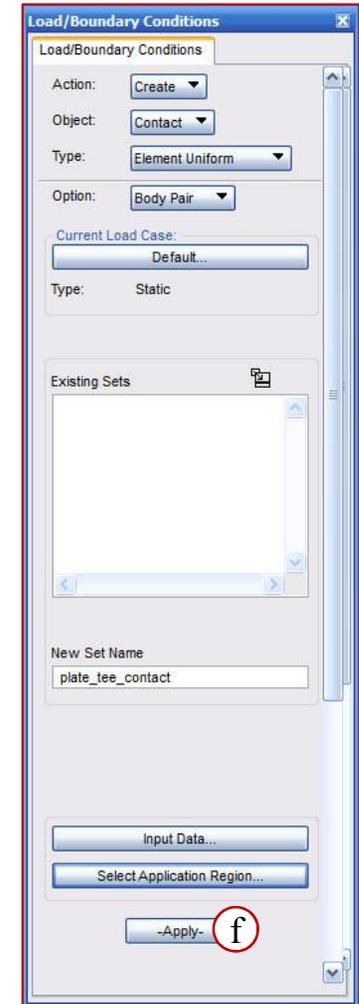
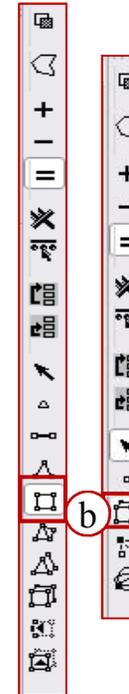
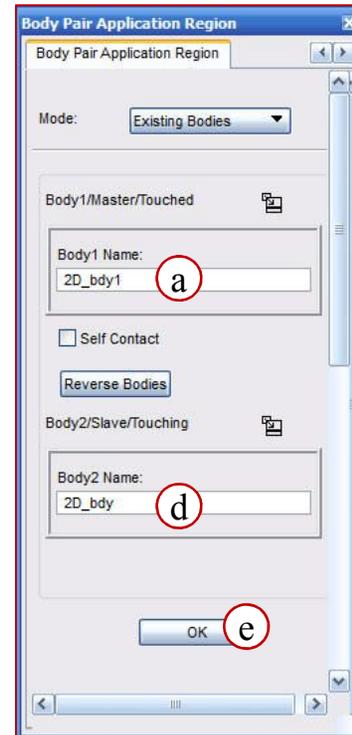
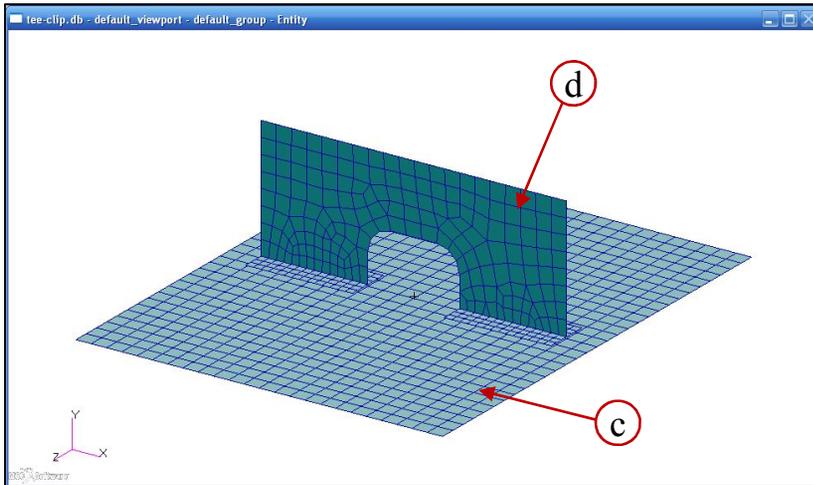
- a. Under the *Load/BCs* tab click **Create Body Pair** in the *Body Pairs* group.
- b. Enter **plate_tee_contact** for *New Set Name*.
- c. Click **Input Data...**
- d. Check **Glued Contact(IGLUE)** and **Stress free InitCont(ICoord)**.
- e. Select **Double Sided** for *Contact Detection (ISEARCH)*.
- f. Click **OK**
- g. Click **Select Application Region**



Step 3. Create Body Pair (Cont.)

Select Application Region:

- Click in the **Body1 Name** text box
- Select **FEM** and then **QUAD Elements** in the *Picking Toolbar*
- Select any element in **2D_bdy1** (the plate).
- Repeat steps b and c with **Body2 Name** and in **2D_bdy** (the tee).
- Click **OK**
- Click **Apply**



Step 4. Set Up and Run the Analysis

The image shows a screenshot of the MSC Nastran software interface. The top ribbon has tabs for Home, Geometry, Properties, Loads/BCs, Meshing, Analysis, Results, and Durability. The Analysis tab is active, and the 'Entire Model' button in the 'Analyze' group is highlighted with a red circle 'a'. Below the ribbon, the 'Solution Type' dialog box is open, showing the 'Solution Type' tab. The 'MSC.Nastran' section is selected, and the 'Solution Type' list has 'LINEAR STATIC' selected with a radio button. The 'Solution Parameters...' button is highlighted with a red circle 'd'. Below the dialog box, the 'Analysis' dialog box is open, showing the 'Action' dropdown set to 'Analyze', 'Object' set to 'Entire Model', and 'Method' set to 'Full Run'. The 'Job Name' field is filled with 'tee-clip_contact' and highlighted with a red circle 'b'. The 'Solution Type...' button in the 'Available Jobs' section is highlighted with a red circle 'c'.

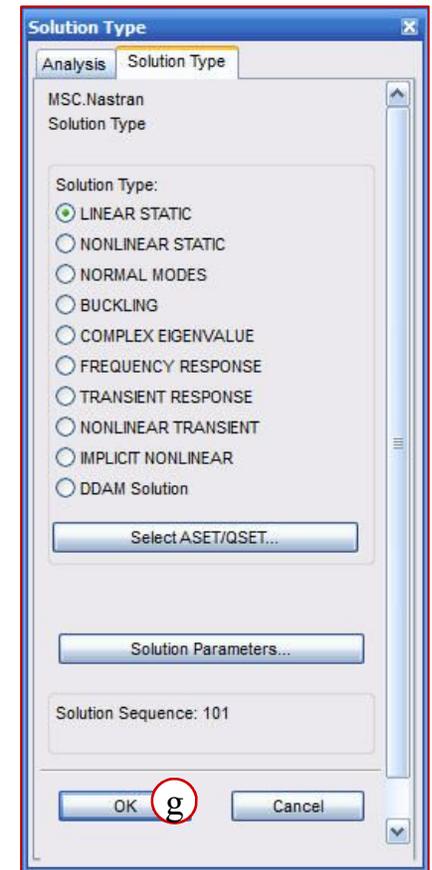
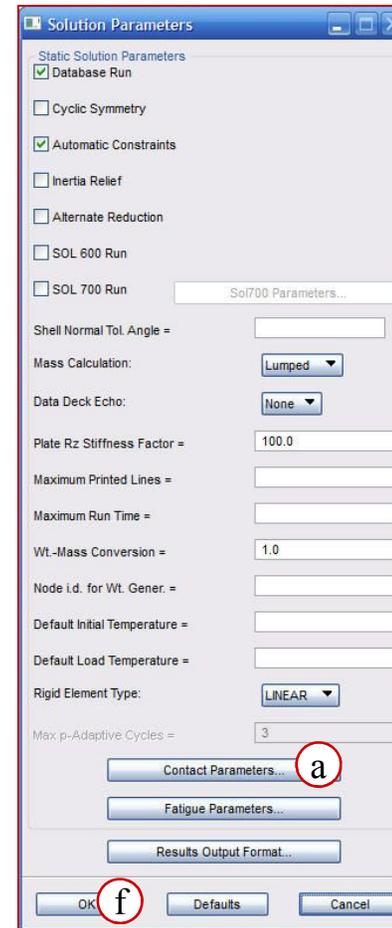
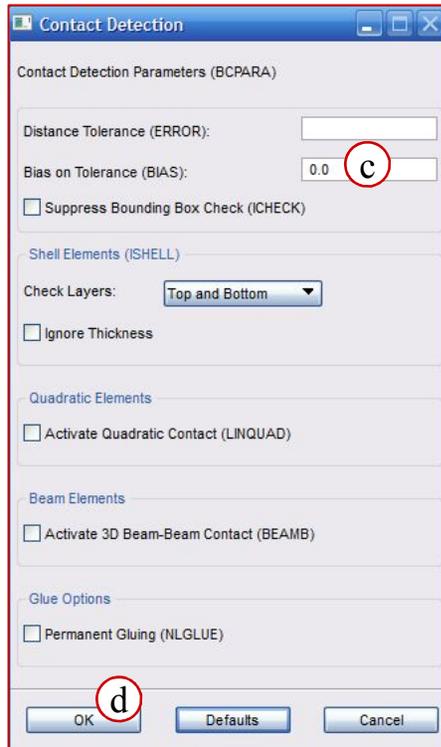
Set Solution Type to 101 Linear Static.

- Under the *Analysis* tab click **Entire Model** in the *Analyze* group.
- Enter **tee-clip_contact** for *Job Name* .
- Click **Solution Type**.
- Click **Solution Parameters**.

Step 4. Set Up and Run the Analysis (Cont.)

Change the Bias on Tolerance.

- a. Click **Contact Parameters**.
- b. Click **Contact Detection**.
- c. Change *Bias on Tolerance (BIAS)* to **0.0**.
- d. Click **OK**.
- e. Click **OK**.
- f. Click **OK**.
- g. Click **OK**.

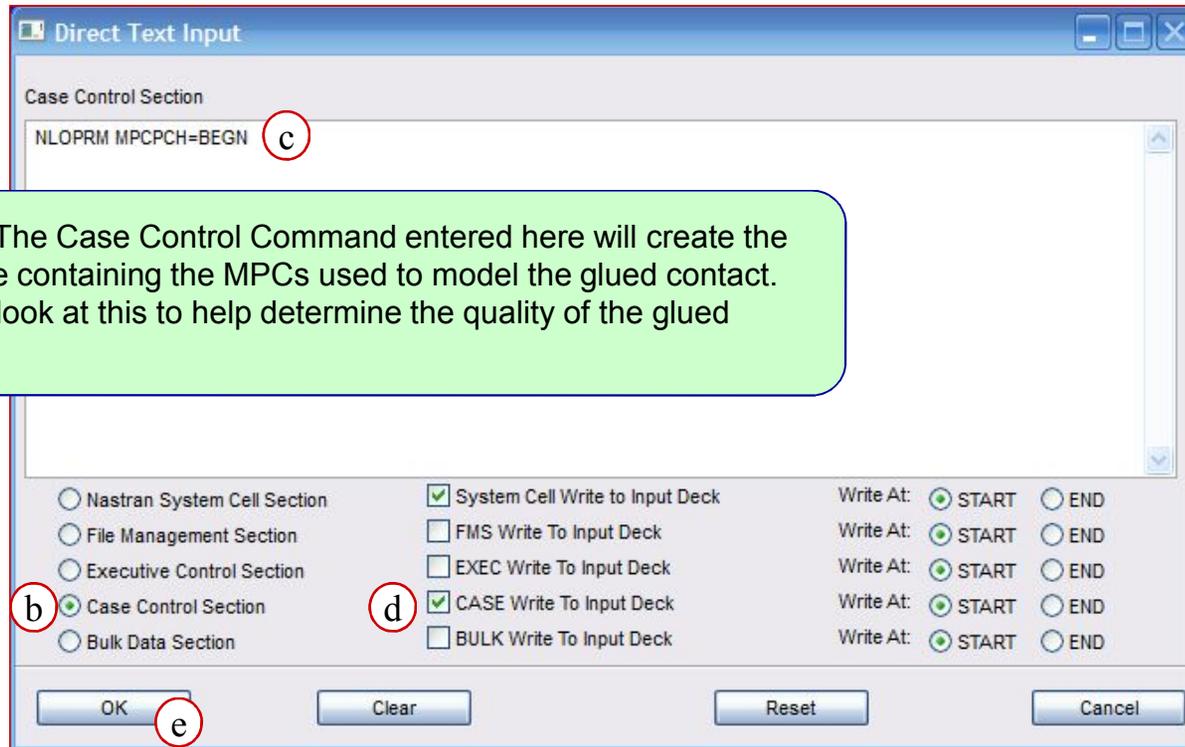


Step 4. Set Up and Run the Analysis (Cont.)

Request a punch file that contains the MPCs used to model the glued contact.

- a. Click **Direct Text Input**.
- b. Select **Case Control Section**.
- c. Enter **NLOPRM MPCPCH=BEGN** in the *Case Control Section* textbox.
- d. Check **CASE Write To Input Deck**.
- e. Click **OK**.
- f. Click **Apply**.

NOTE: The Case Control Command entered here will create the *.pch file containing the MPCs used to model the glued contact. We will look at this to help determine the quality of the glued contact.

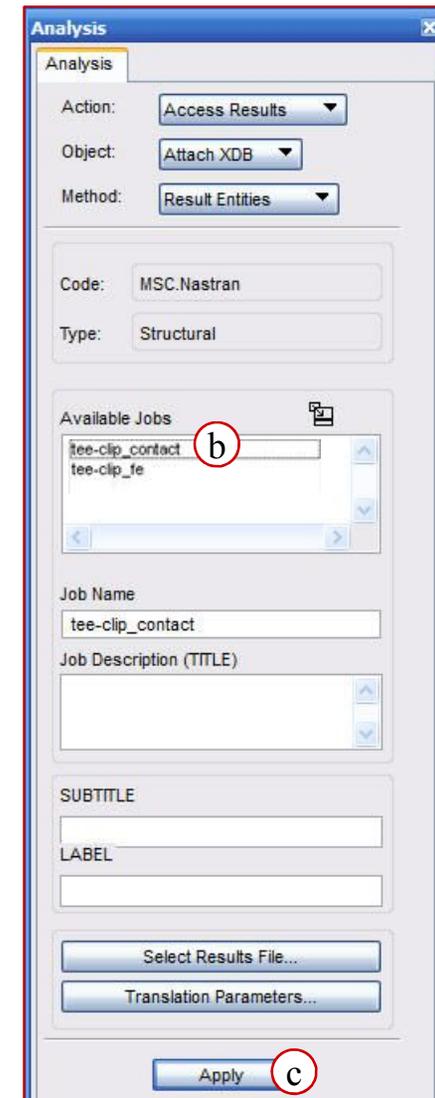


Step 5. Attach the Results File

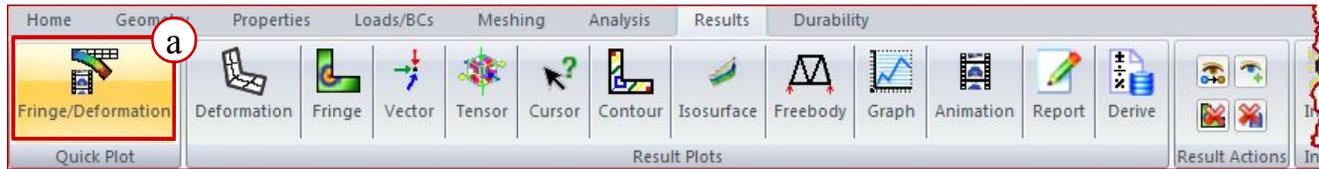


Attach the XDB analysis result file.

- a. Click **XDB** in the *Access Results* group
- b. Click **tee-clip_contact** under *Available Jobs*.
- c. Click **Apply**.

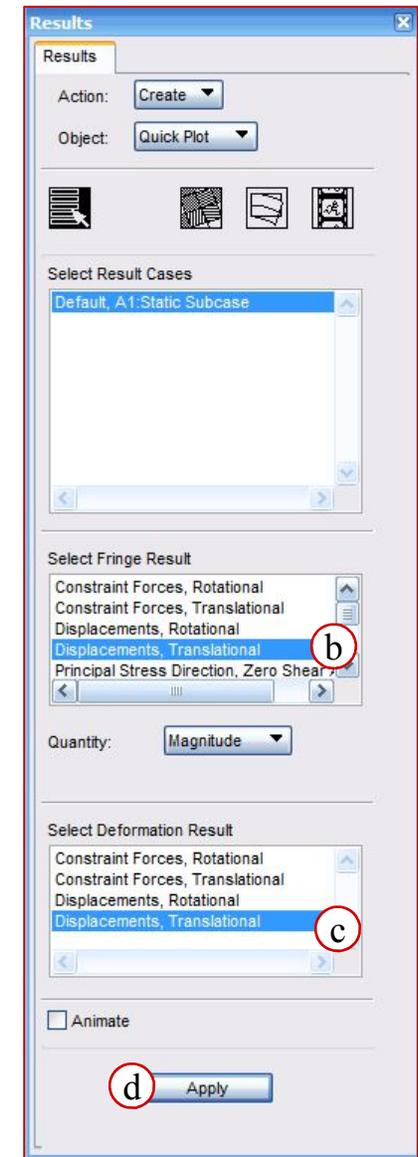
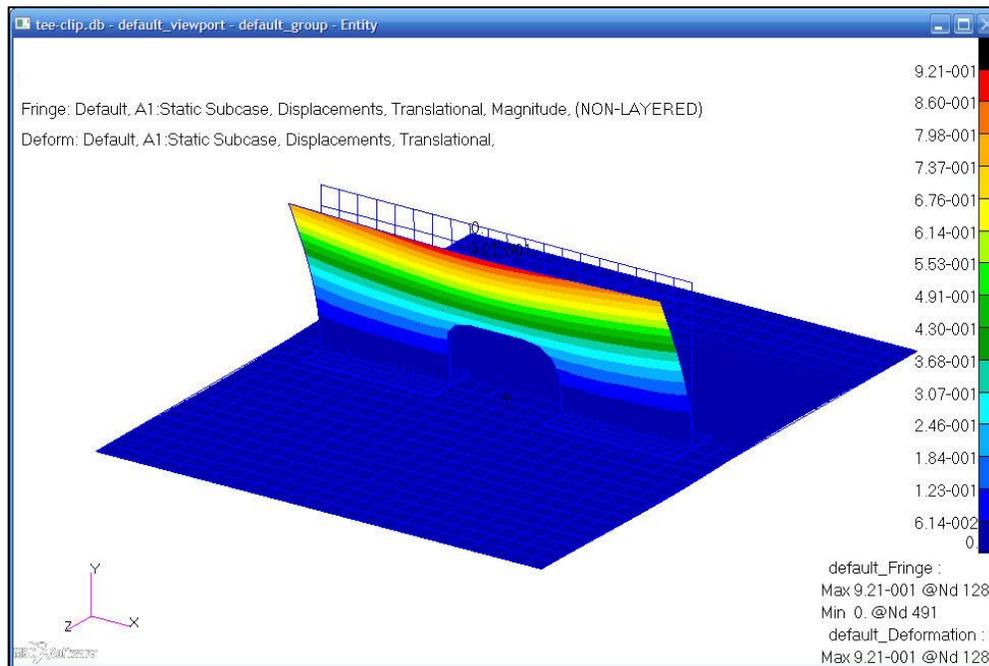


Step 6. Post-Process Results in Patran



Review the deformation results

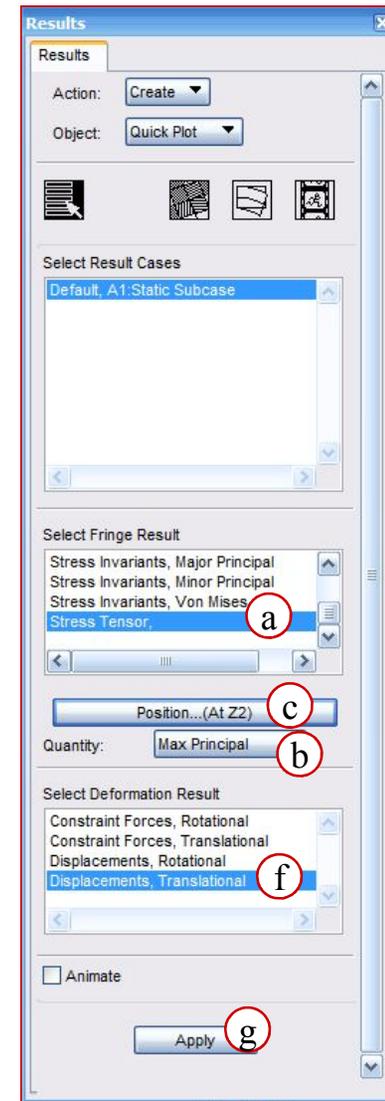
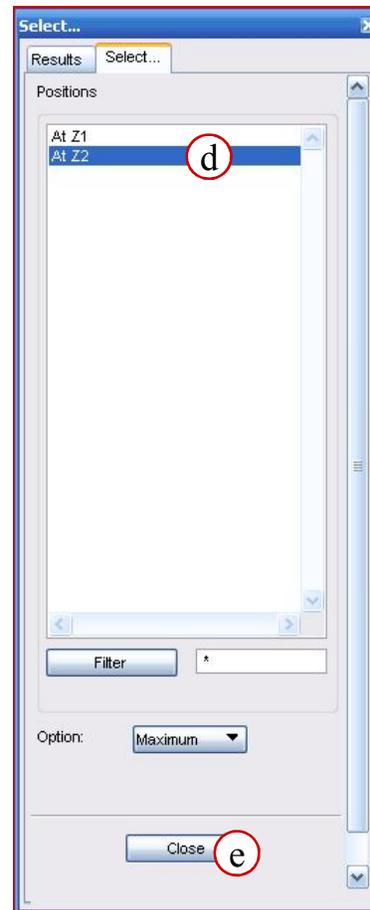
- Under the *Results* tab click **Fringe/Deformation** in the *Quick Plot* group.
- Highlight **Displacements, Translational** under *Select Fringe Result*.
- Highlight **Displacements, Translational** under *Select Deformation Result*.
- Click **Apply**.



Step 6. Post-Process Results in Patran (Cont.)

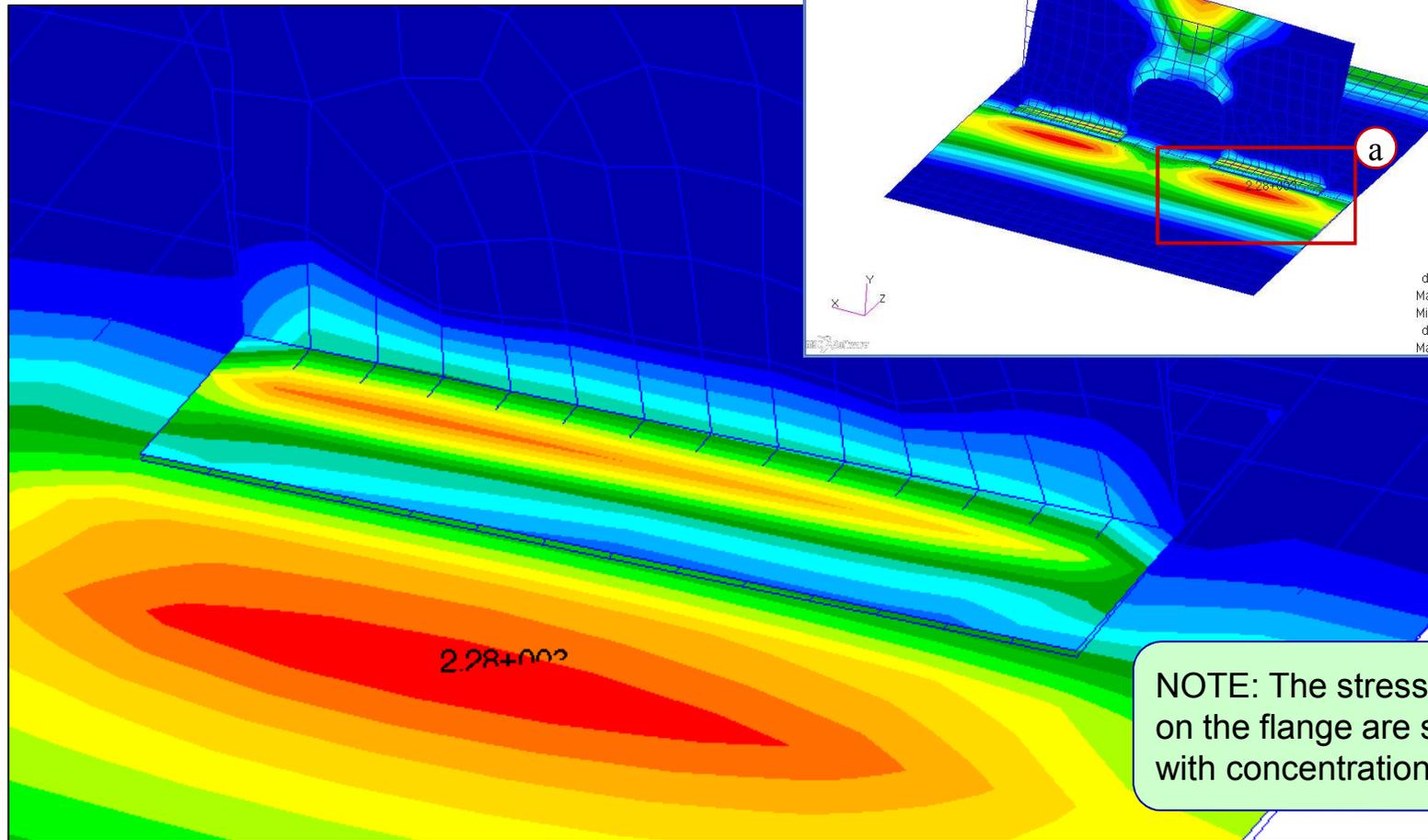
Review max principal stress.

- a. Highlight **Stress Tensor** under *Select Fringe Result*.
- b. Pull down **Max Principal** for *Quantity*.
- c. Click **Position ... (At Z1)**
- d. Highlight **At Z2**
- e. Click **Close**
- f. Highlight **Displacements, Translational** under *Select Deformation Result*.
- g. Click **Apply**.



Step 6. Post-Process Results in Patran (Cont.)

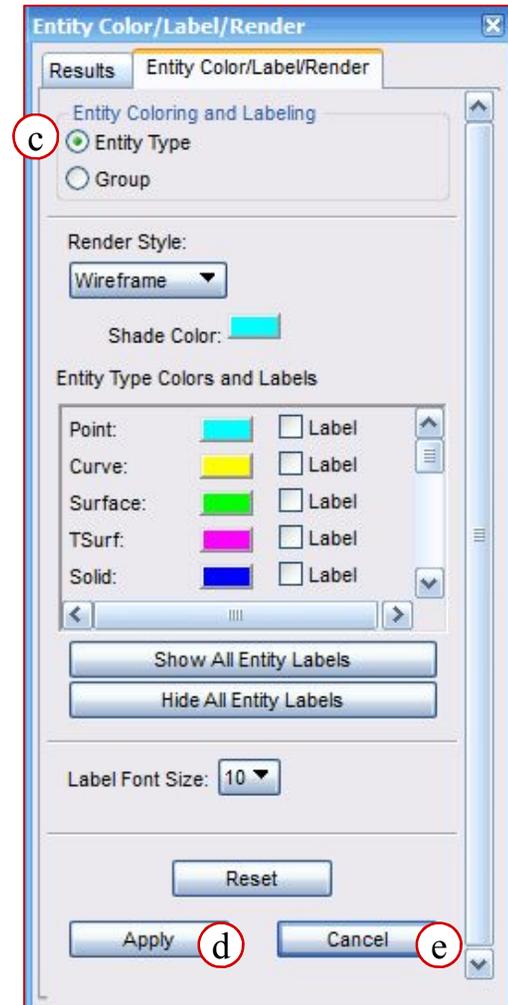
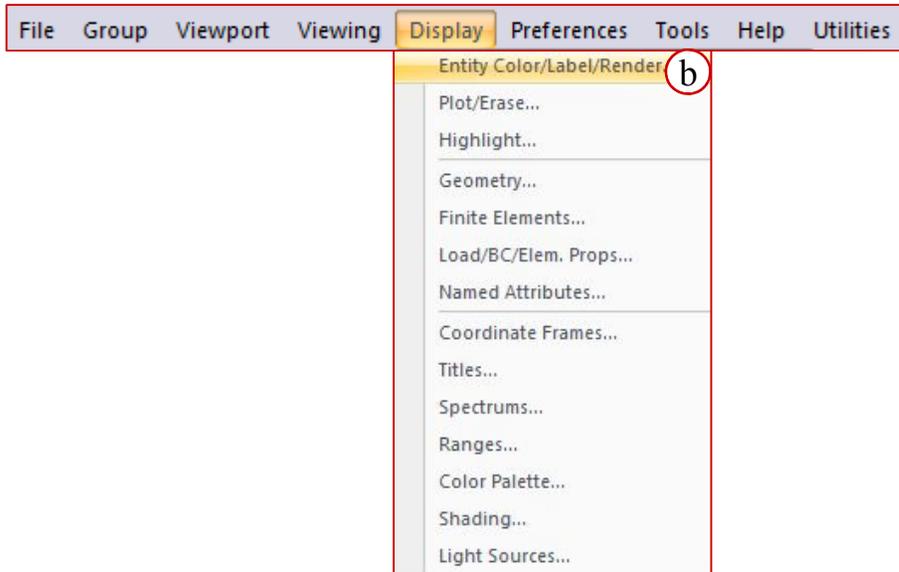
- a. Rotate the model and zoom in to focus on the areas of interest



Step 7. Import the Punch File

Import the *.pch file that contains the MPCs used to model the glued contact. This will help to check the contact status.

- Click **Reset Graphics**.
- Pull down **Display > Entity Color/Label/Render**.
- Select **Entity Type** under *Entity Coloring and Labeling*.
- Click **Apply**.
- Click **Cancel**.



Step 7. Import the Punch File (Cont.)

Specify to keep the original node numbering when importing the .pch file.

- a. Pull down **File > Import**.
- b. Make sure *Object* and *Source* are set to **Model** and **MSC.Nastran Input** respectively.
- c. Click **MSC Nastran Input Options**.
- d. Click **Define Offsets**.
- e. Click in the cell in the *Nodes* row and *Offset/Maximum* column.
- f. Enter **0** for the *Input Offset Value* and press **Enter**.
- g. Click **OK**.
- h. Click **OK**.

The screenshot shows the software interface with several dialog boxes open. The **File** menu is open, and the **Import...** option is highlighted with a red circle 'a'. The **Import** dialog box is open, showing the **Object** set to **Model** (circled 'b') and the **Source** set to **MSC.Nastran Input**. The **MSC.Nastran Input Options...** button is circled 'c'. The **Entity Selection** dialog box is open, showing the **Define Offsets...** button circled 'd'. The **Entity Label Offset Definition** dialog box is open, showing the **Input Offset Value** field set to **0** (circled 'f') and the **Offset/Maximum** cell in the **Nodes** row circled 'e'. The **OK** button is circled 'g'.

Entity	Existing ID Range in Db	New ID	
	Minimum	Maximum	Offset/Maximum
Nodes	1	1460	Auto
Elements	1	1286	Auto
Material Properties			Auto
Element Properties			Auto
Coordinate Frames			Auto
Load Sets		2	Auto
Subcases			Auto
MPC Data			Auto
Comment Lines	12	12	0

Step 7. Import the Punch File (Cont.)

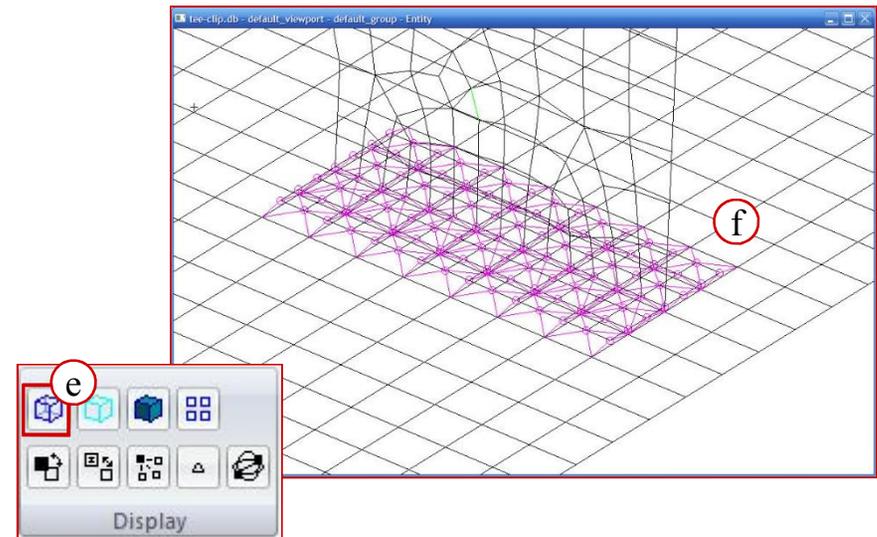
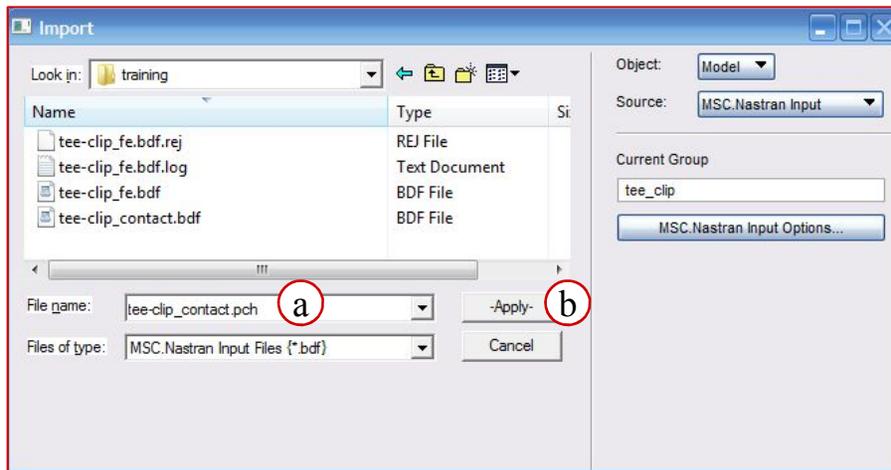
Look at the pattern of contact

- Enter **tee-clip_contact.pch** for *File name*.
- Click **Apply**.
- Review the *MSC Nastran Input File Import Summary* to verify that the MPCs have been imported.
- Click **OK**.
- Change the display to **Wireframe**.
- Observe the MPC pattern between the tee-clip and the plate. Note that single MPCs connect the two components and the edges of the flange are unsupported.

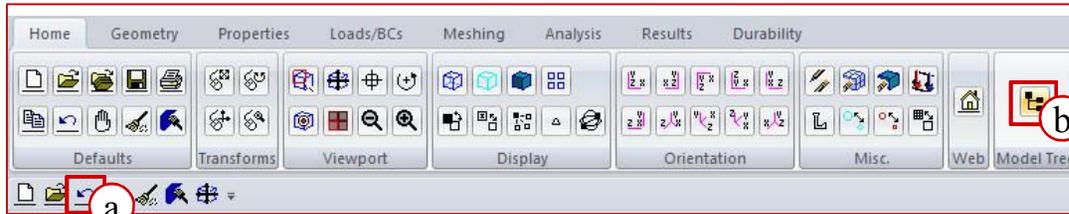
Nastran Input File Import Summary

	Imported	Imported with Warning	Not Imported
Nodes	0	0	0
Elements	0	0	0
Coordinate Frames	0	0	0
Material Propertie	0	0	0
Element Properties	0	0	0
Load Sets	0	0	0
Subcases	0	0	0
MPC Data	629	0	0
Comment Lines	9	0	0

Buttons: Reject Cards..., OK

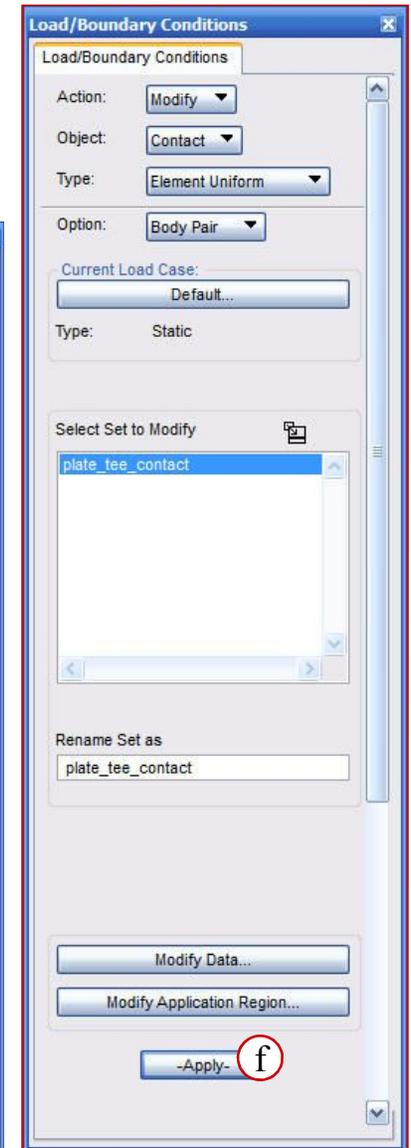
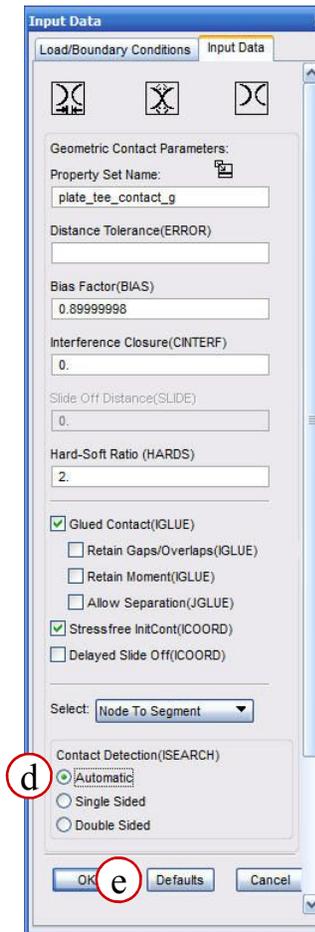
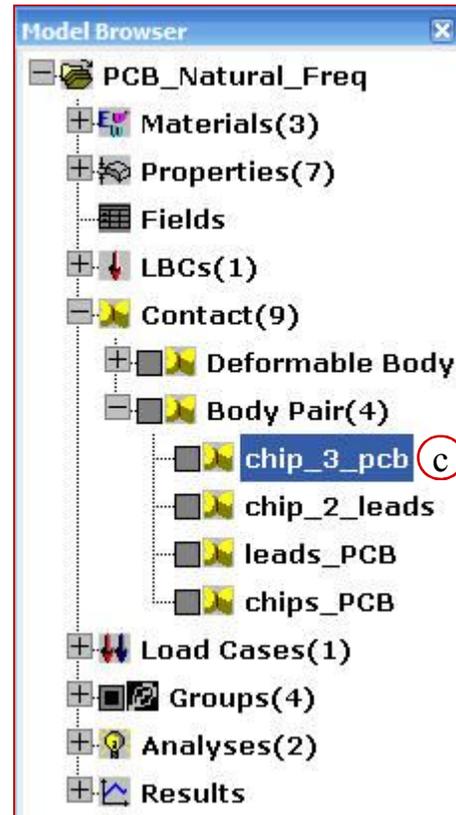


Step 8. Modify Body Pair



Modify the chip_3_pcb body pair:

- a. Click **Undo** to remove the MPCs
- b. Under the *Home* tab, click **Show/Hide Model Browser Tree** in the *Model Tree* Group.
- c. Double click on **plate_tee_contact** under **Contact > Body Pair**
- d. Set the *Contact Detection(ISEARCH)* to **Automatic**.
- e. Click **OK**
- f. Click **Apply**

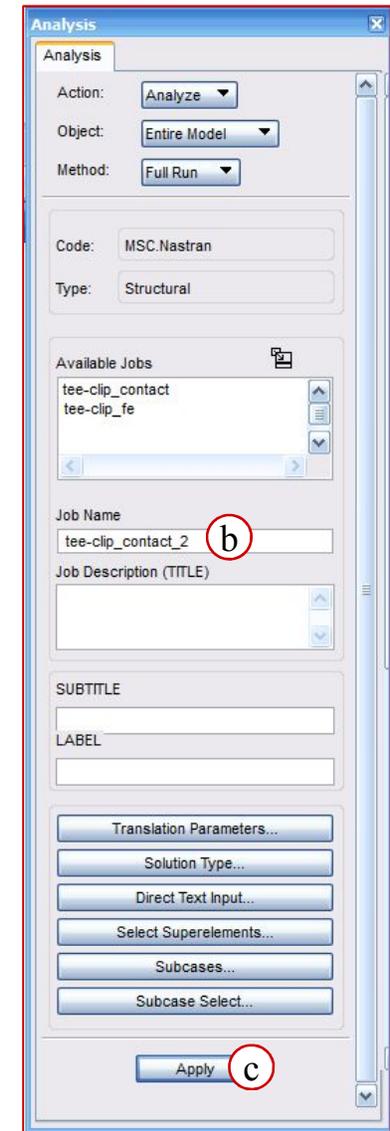


Step 9. Reverse the Search Order and Resubmit the Analysis



Change the Contact Table to reverse the contact search order.

- a. Under the *Analysis* tab click **Entire Model** in *Analyze* group.
- b. Enter **tee-clip_contact_2** for *Job Name*.
- c. Click **Apply**.

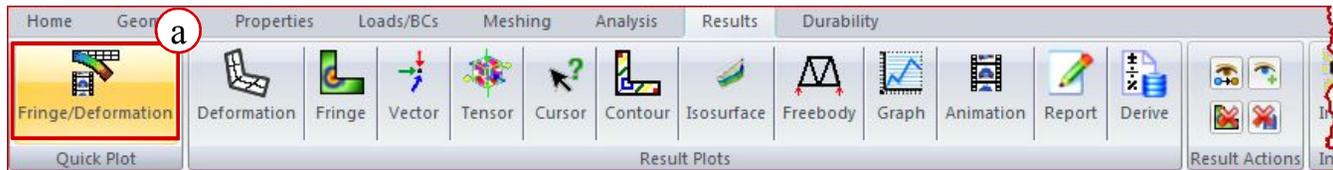


Step 10. Attach New Results File

Post-process the analysis results.

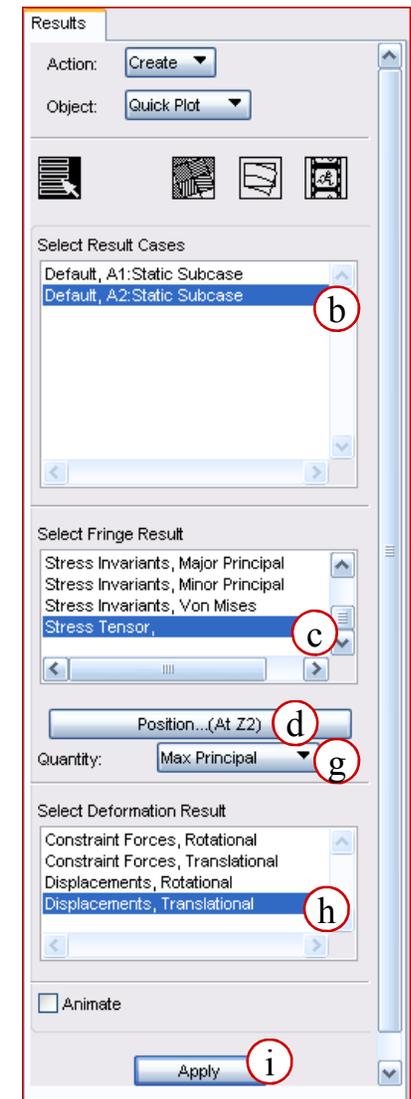
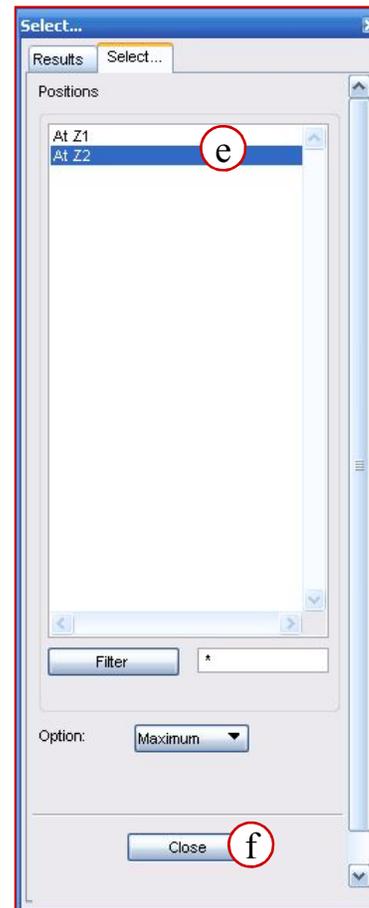
- Click **XDB** in *Access Results* group.
- Select **tee-clip_contact_2** under *Available Jobs*.
- Click **Apply**.

Step 11. Post-Process Results in Patran

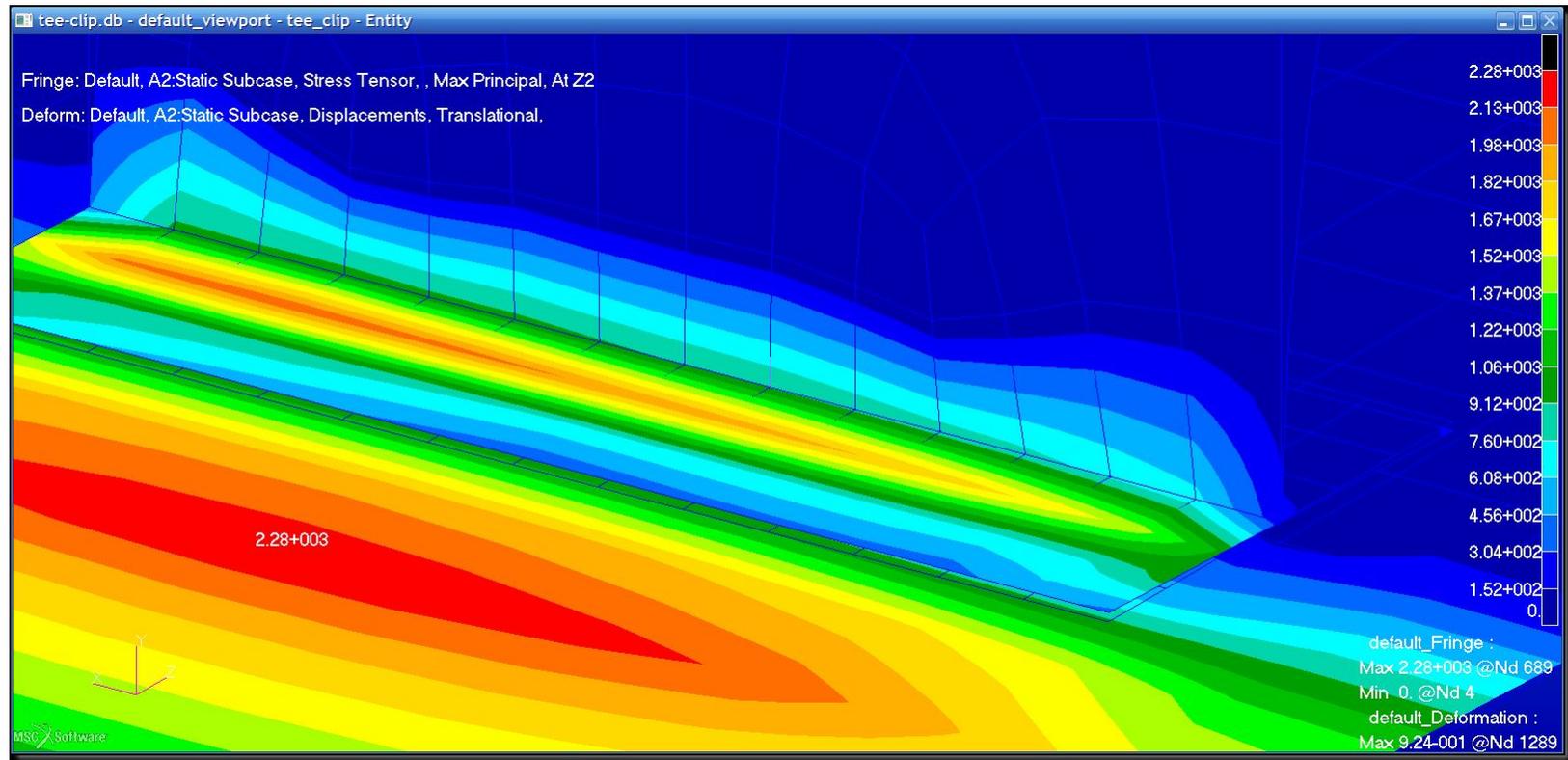


Review the max principal stresses.

- a. Under the *Results* tab, click **Fringe/Deformation** in the *Quick Plot* group.
- b. Highlight **Default, A2: Static Subcase** under *Select Result Cases*.
- c. Highlight **Stress Tensor** under *Select Fringe Result*.
- d. Click **Position ... (At Z1)**
- e. Highlight **At Z2**.
- f. Click **Close**.
- g. Pull down **Max Principal** for *Quantity*.
- h. Highlight **Displacements, Translational** under *Select Deformation Result*.
- i. Click **Apply**.



Step 11. Post-Process Results in Patran (Cont.)

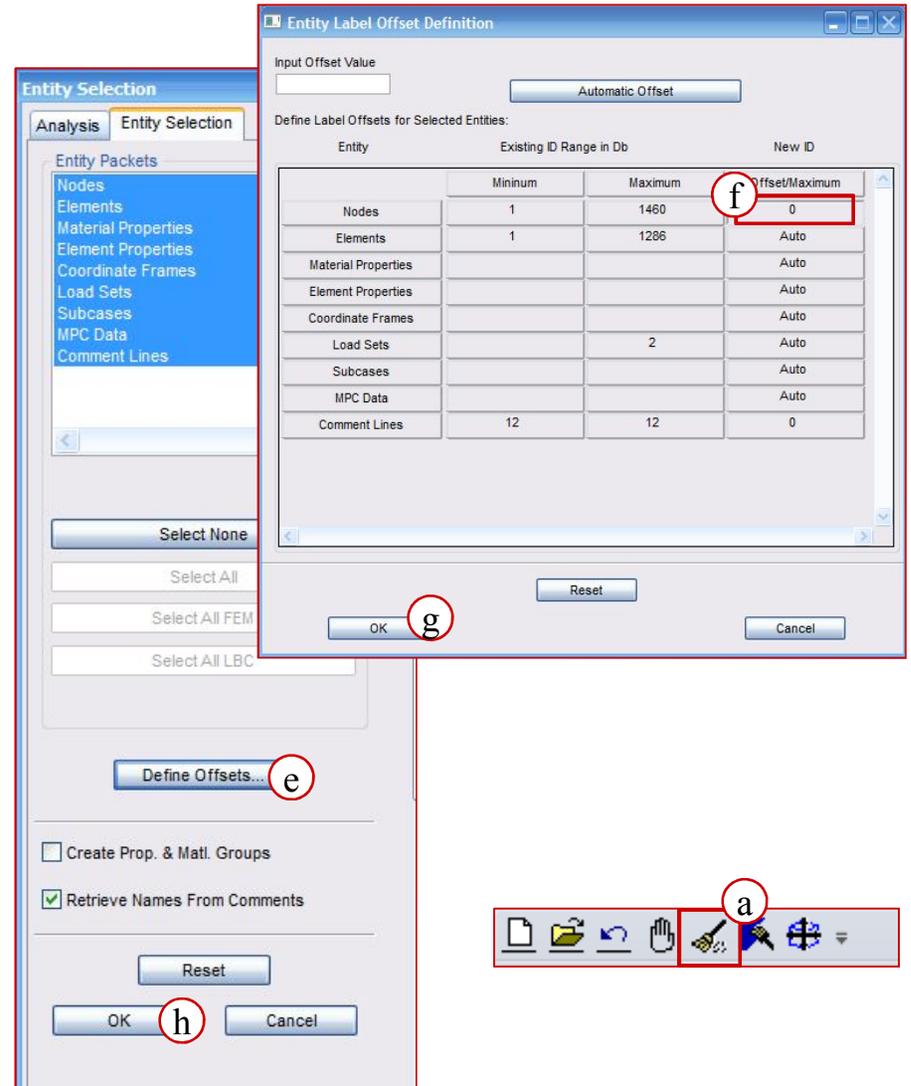
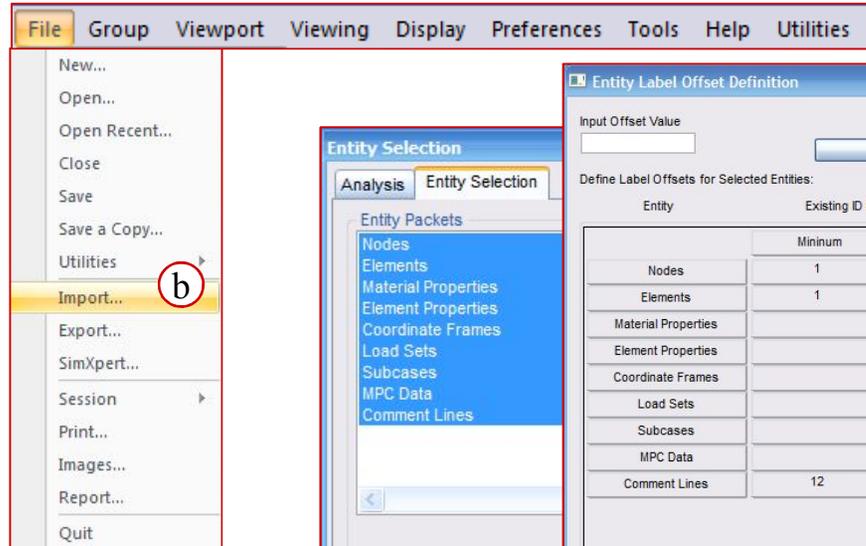
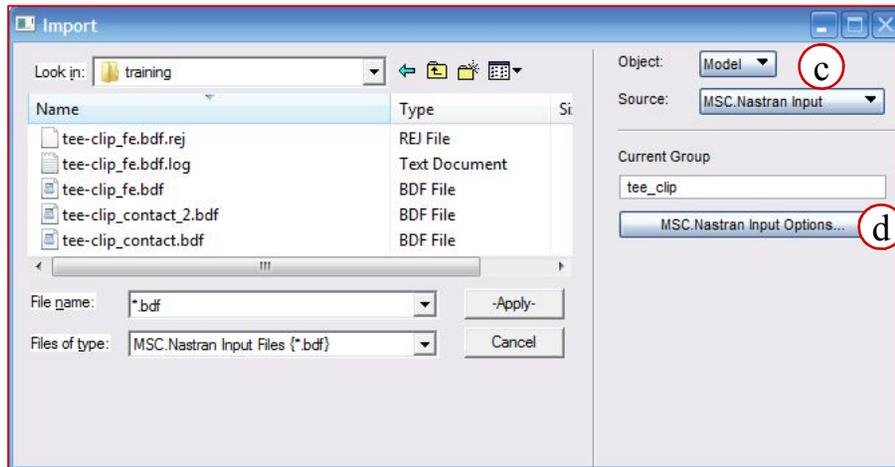


NOTE: The stress pattern on the flange is much smoother and the peak stress has fallen by nearly 50%.

Step 12. Import the Punch File

Look at the updated MPC pattern.

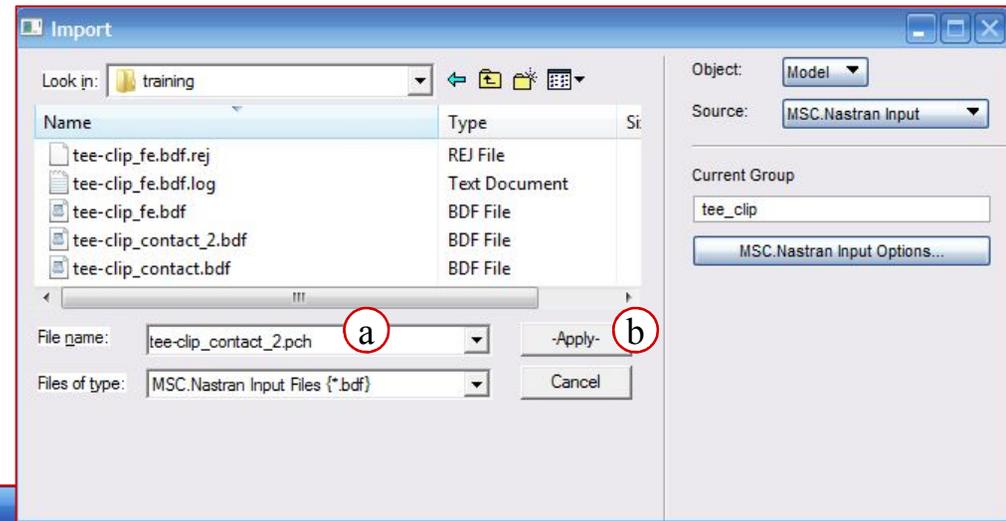
- a. Click **Reset Graphics**.
- b. Pull down **File > Import**.
- c. Make sure *Object* and *Source* are set to **Model** and **MSC.Nastran Input** respectively.
- d. Click **MSC Nastran Input Options**.
- e. Click **Define Offsets**
- f. Verify that *Input Offset Value* is **0**
- g. Click **OK**.
- h. Click **OK**.



Step 12. Import the Punch File (Cont.)

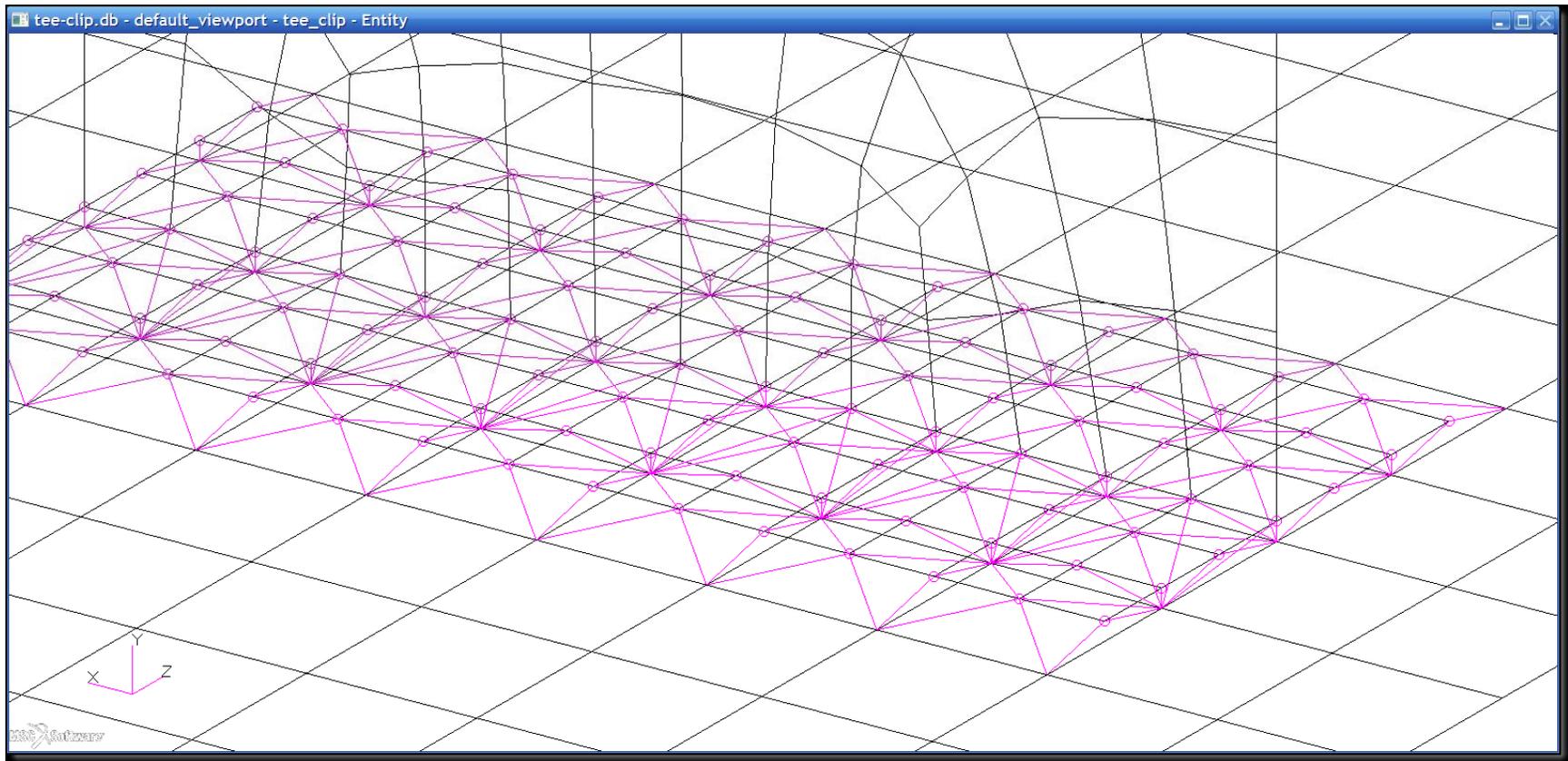
Look at the updated MPC pattern (cont.)

- a. Enter **tee-clip_contact_2.pch** for *File name*.
- b. Click **Apply**.
- c. Review the *MSC Nastran Input File Import Summary* to verify that the MPCs have been imported.
- d. Click **OK**.



	Imported	Imported with Warning	Not Imported
Nodes	0	0	0
Elements	0	0	0
Coordinate Frames	0	0	0
Material Propertie	0	0	0
Element Properties	0	0	0
Load Sets	0	0	0
Subcases	0	0	0
MPC Data	629	0	0
Comment Lines	9	0	0

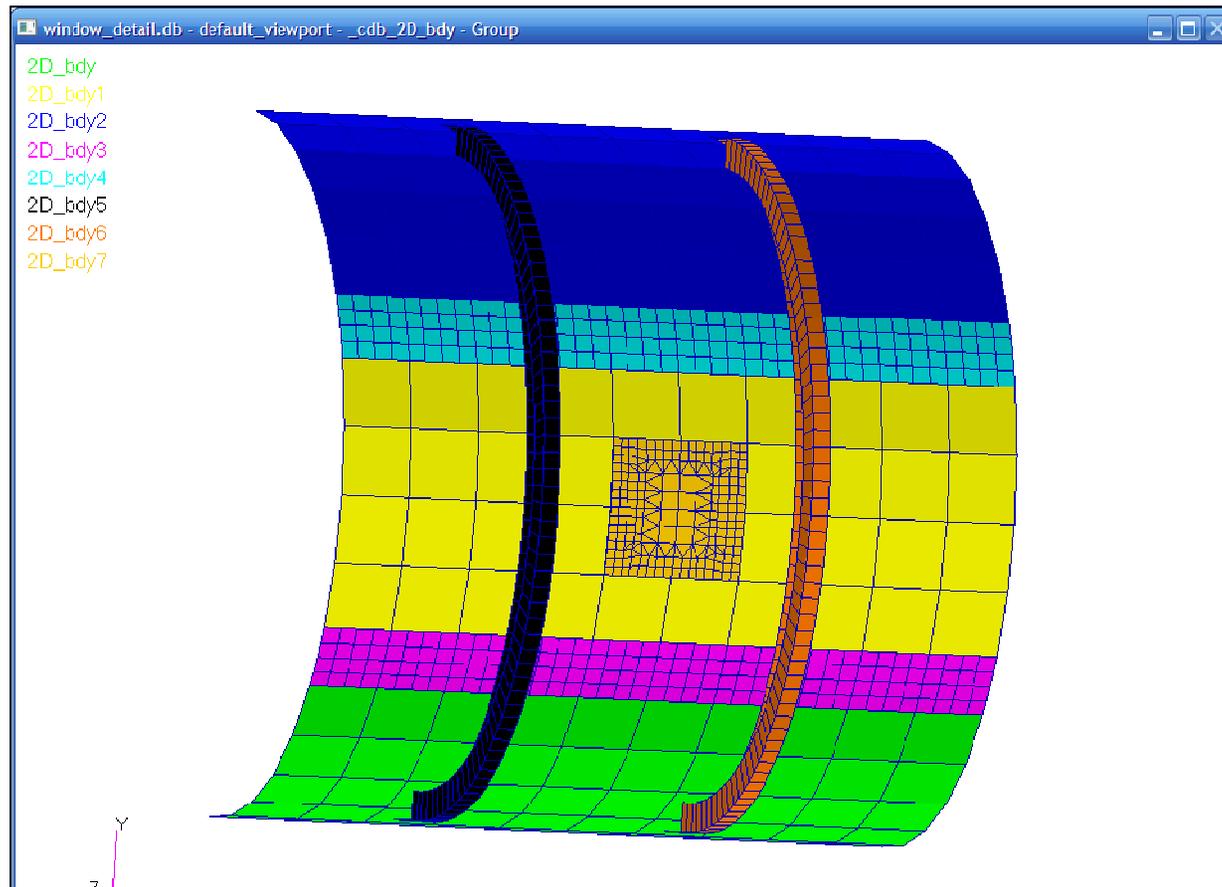
Step 12. Import the Punch File (Cont.)



NOTE: The MPC pattern is much more extensive. The edges of the flange are now fully supported.

WORKSHOP 6

CURVED SURFACE CONTACT





- **Workshop Objectives**

- Demonstrate how fine-meshed parts can be included in coarser-meshed models.
- Demonstrate Patran Automatic Contact Body and Body pair detection that can detect multiple contact bodies with one click.
- Use the Distance Tolerance (ERROR) to achieve glued contact between surfaces with gaps.

- **Files Required**

- **wingskin_fe.bdf**

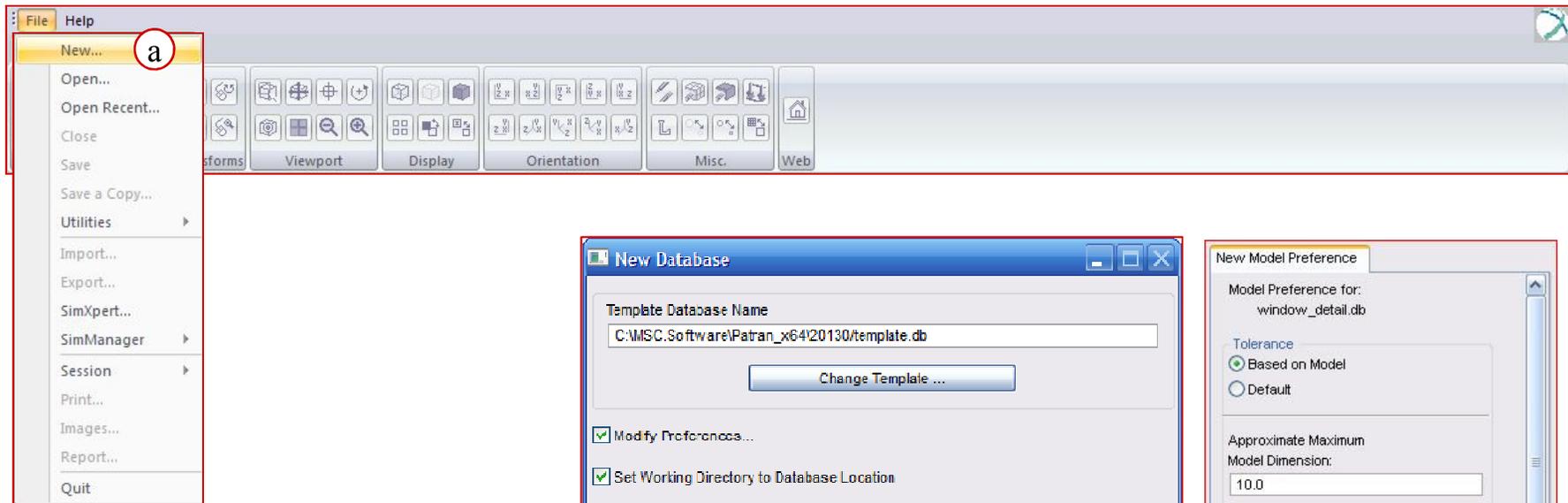
- **Software Version**

- Nastran 2013.1
- Patran 2013

- **Suggested Steps:**

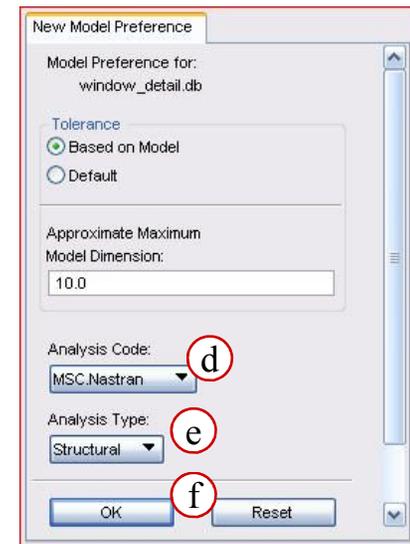
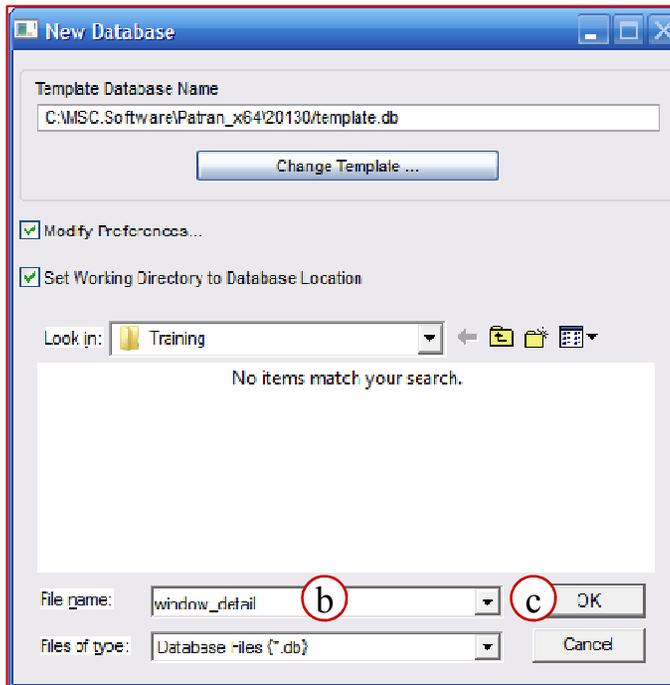
1. Create a new Patran database.
2. Import the Nastran Input file, **wingskin_fe.bdf**.
3. Define Contact Bodies using Patran auto detect tool.
4. Define Contact Pairs
 - Contact Pair between the Panels with tolerance of **0.003** and glued contact.
 - Contact Pair between the Rib and panels with tolerance of **0.02** and glued contact.
5. Verify the Boundary Conditions
6. Set up a linear static analysis with Contact results. Run the analysis.
7. Attach the MASTER/DBALL results
8. Review the Deformation shape.
9. Verify contact status.

Step 1. Create a New Database

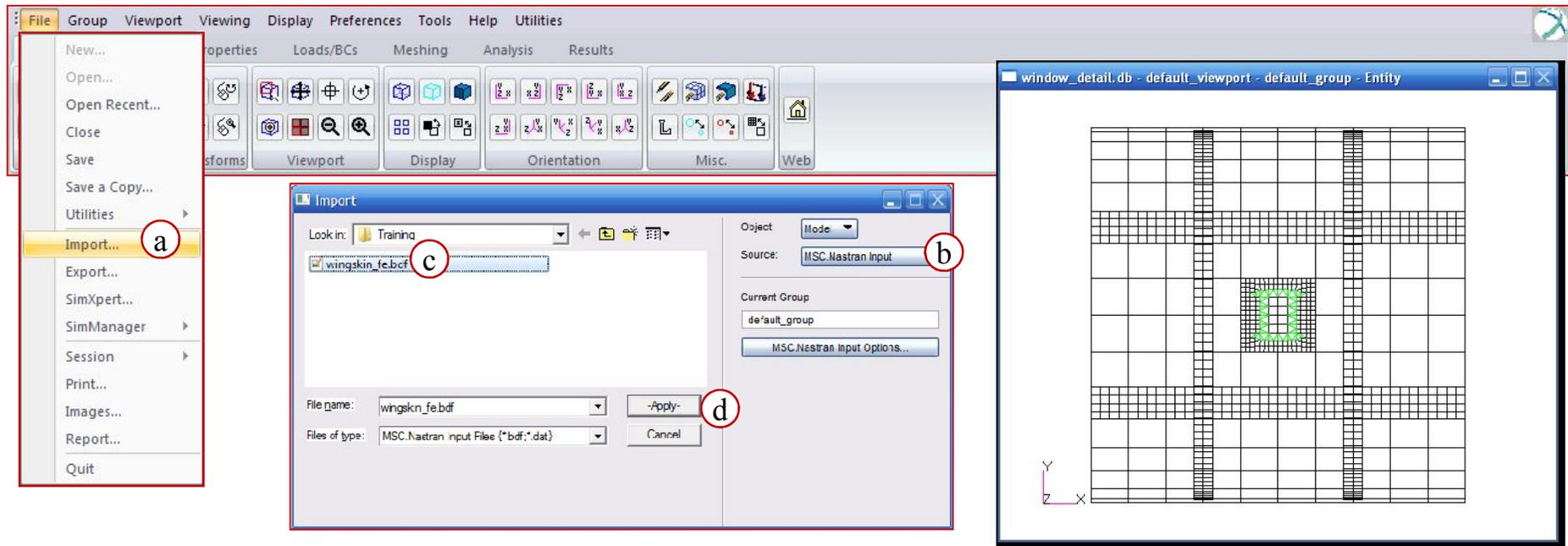


Create a new database. Name it window_detail.db

- 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 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**

The screenshot shows the 'Nastran Input File Import Summary' dialog box. It contains a table with the following data:

	Imported	Imported with Warning	Not Imported
Nodes	1330	0	0
Elements	1064	0	0
Coordinate Frames	0	0	0
Material Properties	1	0	0
Element Properties	4	0	0
Load Sets	2	0	0
Subcases	1	0	0
MPC Data	0	0	0
Comment Lines	15	0	0

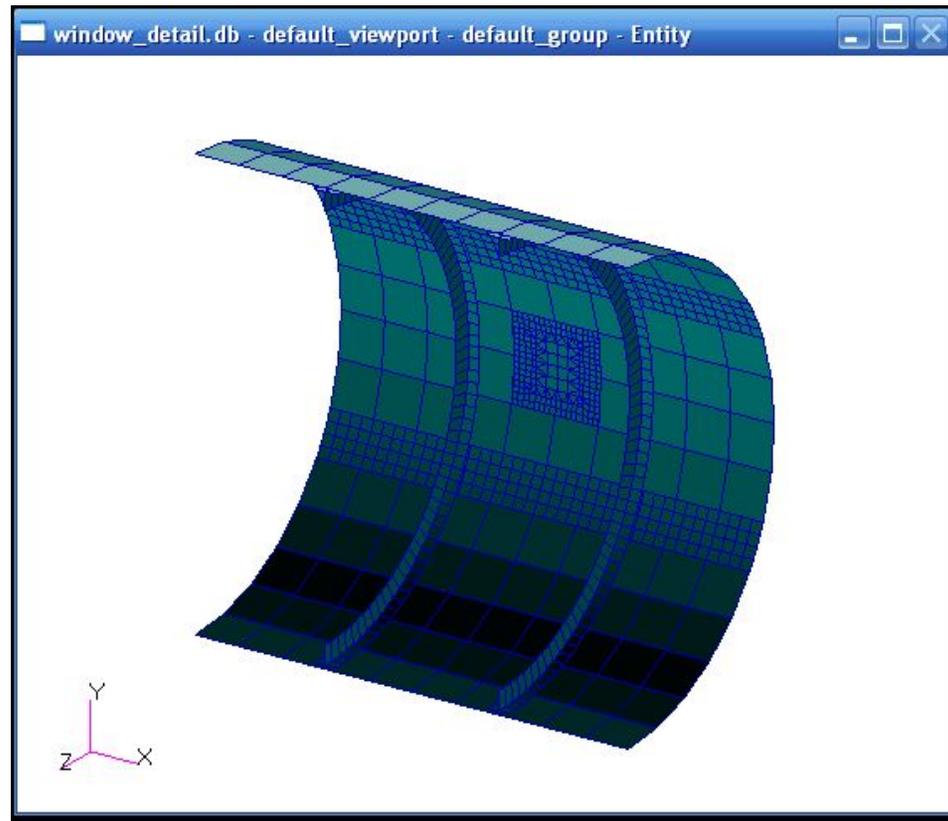
At the bottom of the dialog, there is a 'Reject Cards...' button and an 'OK' button (marked with a red circle 'e').

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.

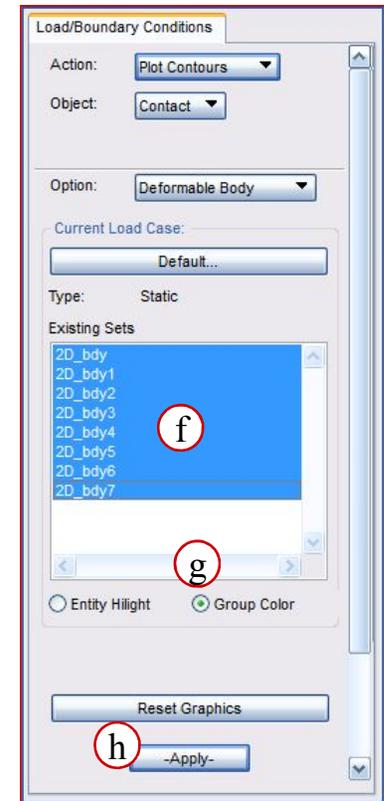
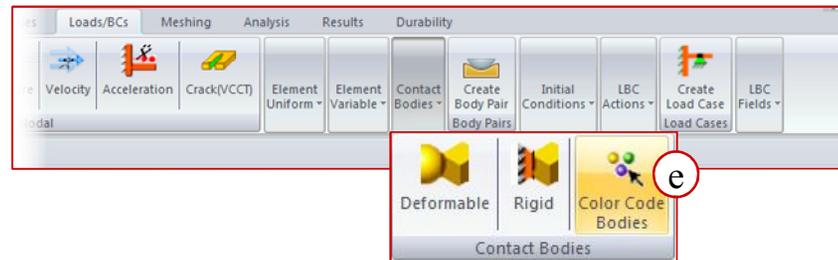
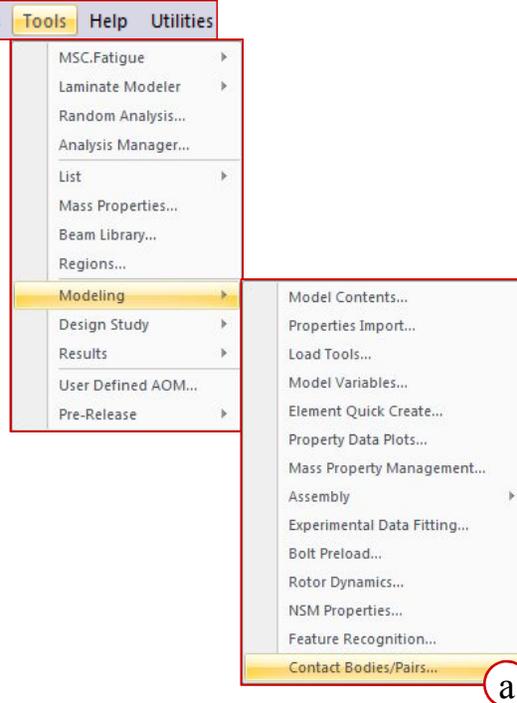
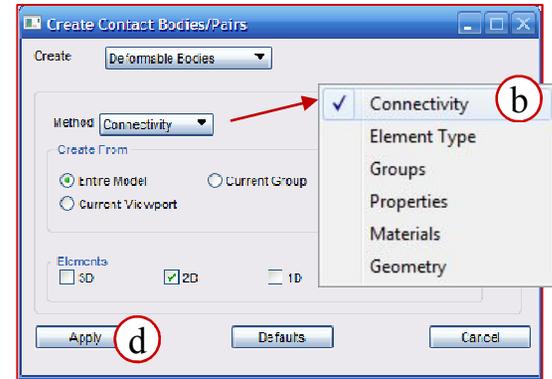
NOTE: Notice that here we have an airframe model that has a very fine mesh in the window and some of its panels. Other panels have a comparatively coarser mesh.



Step 3. Define Contact Bodies

Define Contact bodies using Create Contact Body/Pair tool.

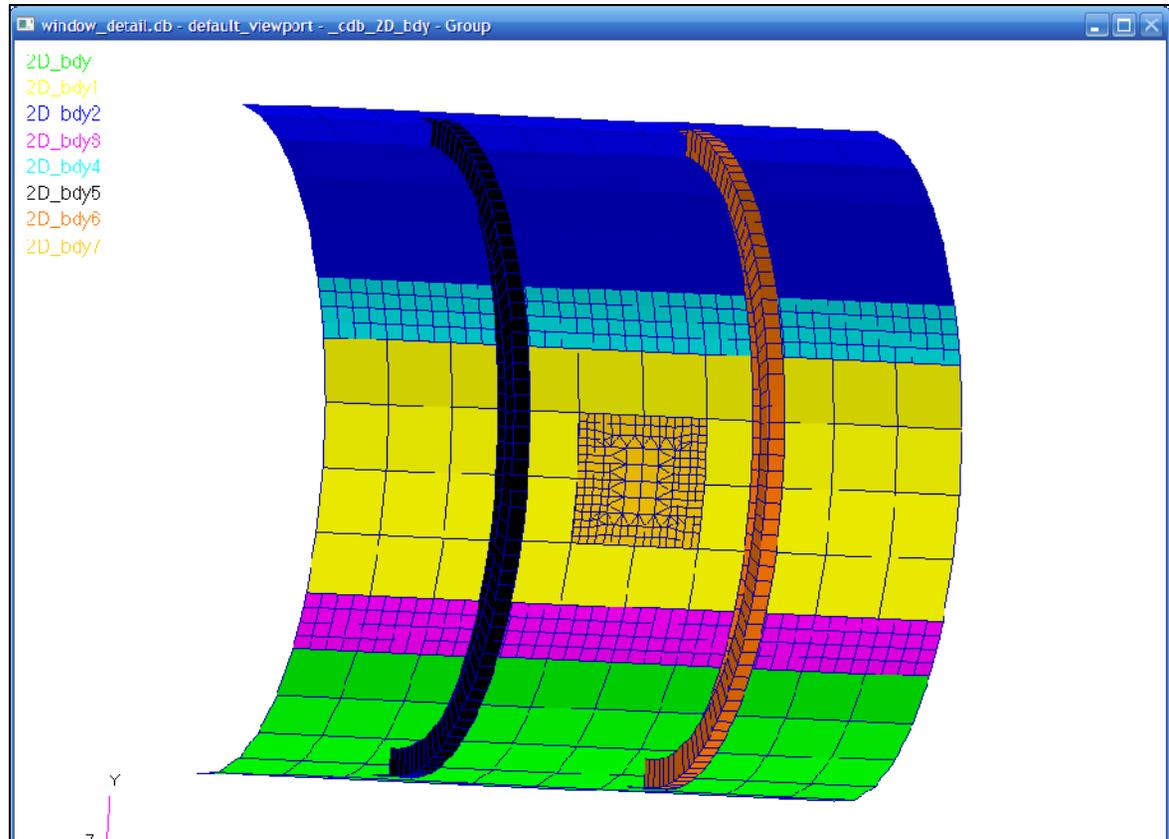
- Pull down *Tools > Modeling > Contact Bodies/Pair*
- Pull down *Method > Connectivity*
- Check **2D** for Elements.
- Click **Apply**
- Under the *Loads/BCs* tab, click **Color Code Bodies**, in the *Contact Bodies* group.
- Highlight all three sets for *Existing Sets*.
- Select **Group Color**
- Click **Apply**



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

Step 3. Define Contact Bodies (Cont.)

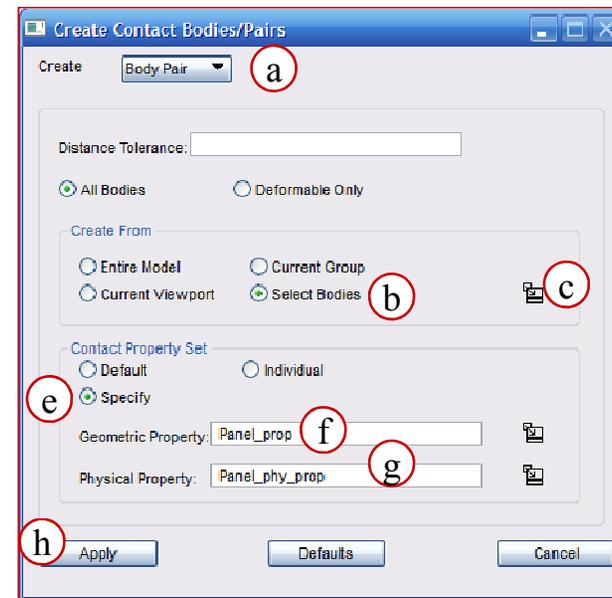
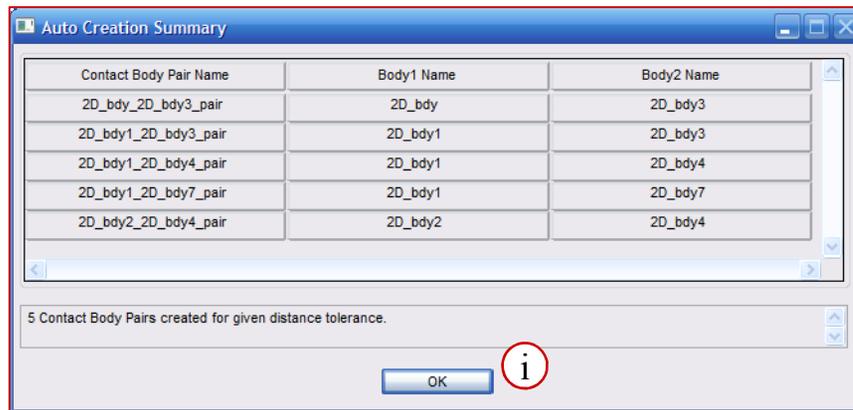
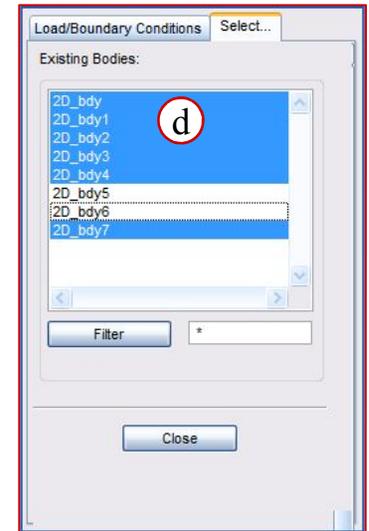
You can modify the light on the view by Pulling down **Display > Light Sources** and select more Light Sources and click Apply



Step 4. Define Contact Pairs

Define Contact Pairs using Create Contact Body/Pair tool.

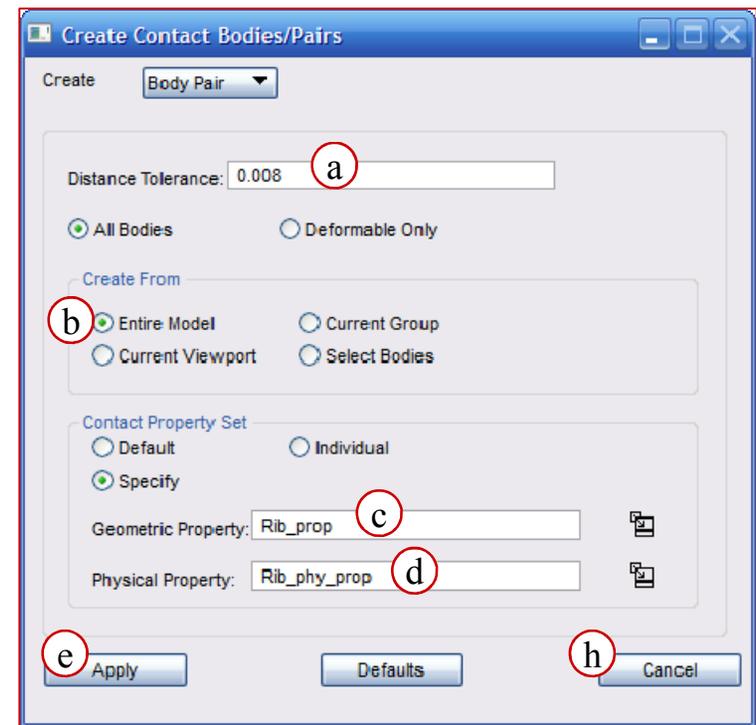
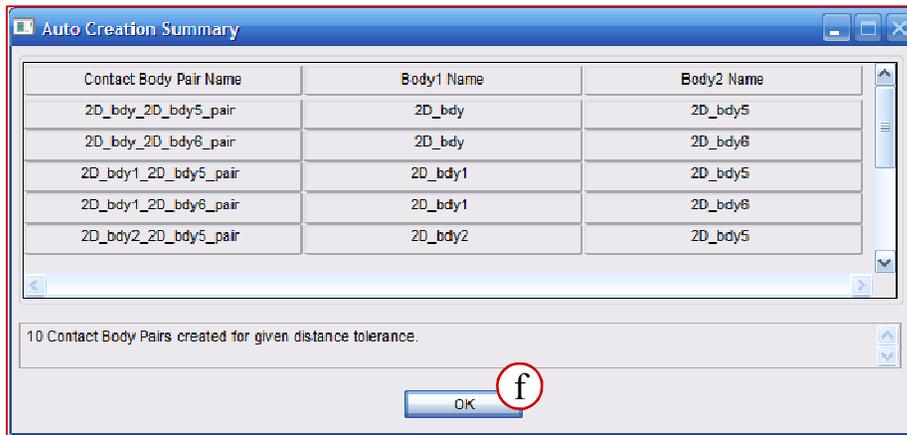
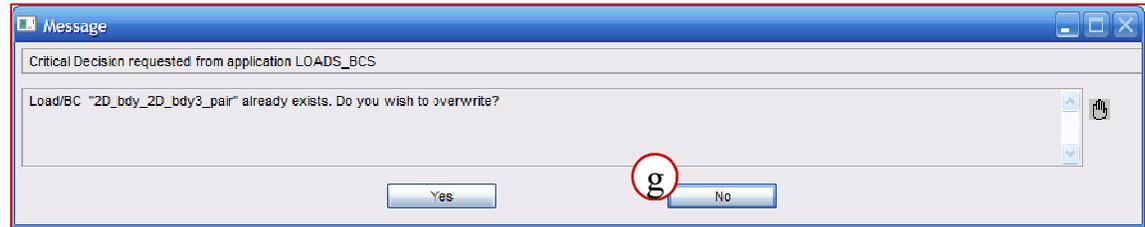
- Pull down *Create* > **Body Pair**.
- Select **Select Bodies** in *Create From*.
- Click *Select Existing Bodies* icon.
- Highlight **all Sets** in *Existing Bodies*, except *2D_bdy5* and *2D_bdy6* (*The Ribs*).
- Select **Specify** in *Contact Properties Sets*
- Enter **Panel_prop** in *Geometric Property*.
- Enter **Panel_phy_prop** in *Physical Property*.
- Click **Apply**.
- Click **OK** to close the Auto Creation Summary.



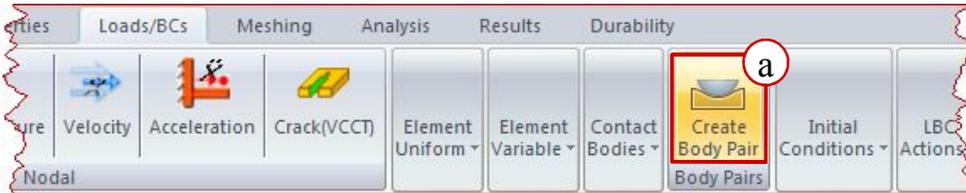
Step 4. Define Contact Pairs (Cont.)

Defining 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** in *Geometric Property*.
- d. Enter **Rib_phy_prop** in *Physical Property*.
- e. Click **Apply**.
- f. Click **OK**.
- g. Click **No** on all the Messages for duplicate pairs.
- h. Click **Cancel** to close the window.

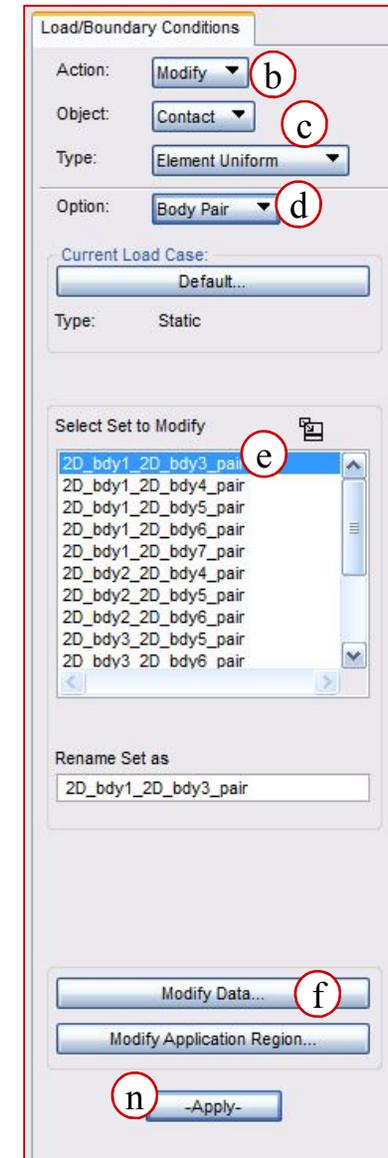
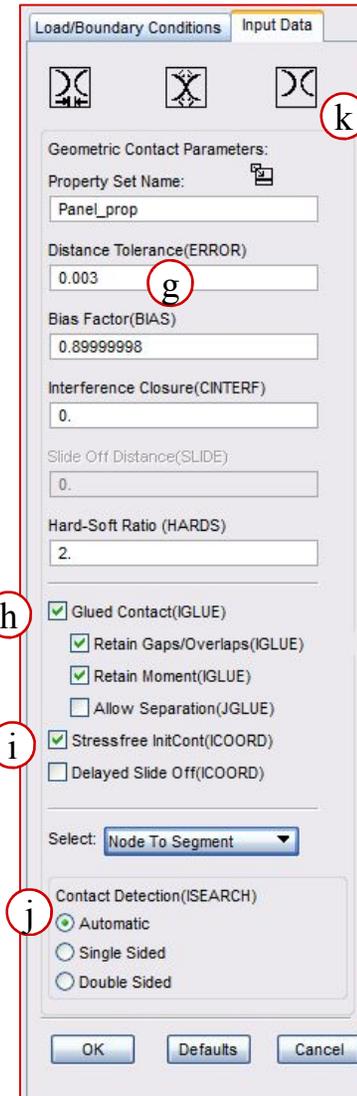
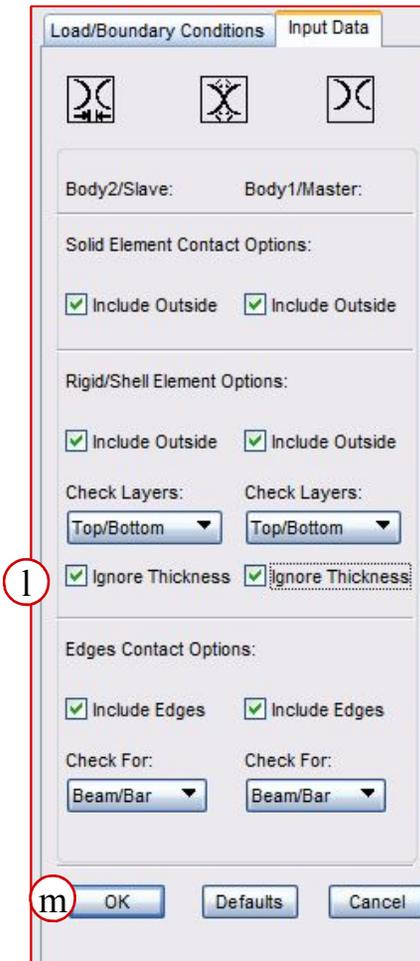


Step 4. Define Contact Pairs (Cont.)



Modify the contact pair parameters

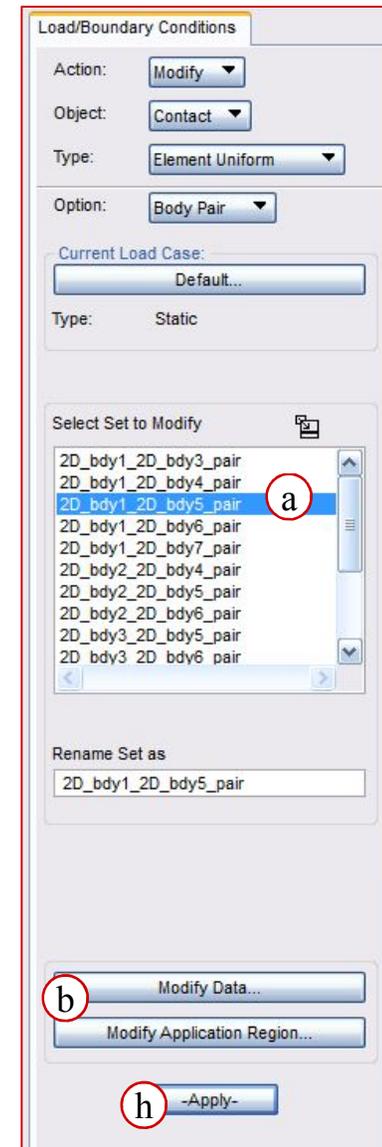
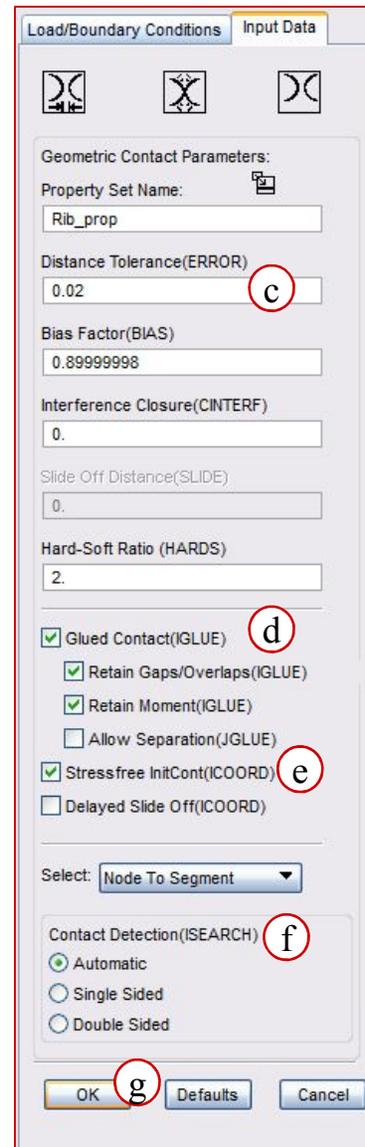
- a. Click Create Body Pair
- b. Pull down Action > **Modify**
- c. Pull down Object > **Contact**
- d. Pull down Option > **Body Pair**
- e. Select **2D_bdy1_2D_bd3_pair**
- f. Select **Modify Data...**
- g. Enter **0.003** for **Distance Tolerance**
- h. Check **Glue Contact, Retain Gaps/Overlaps,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



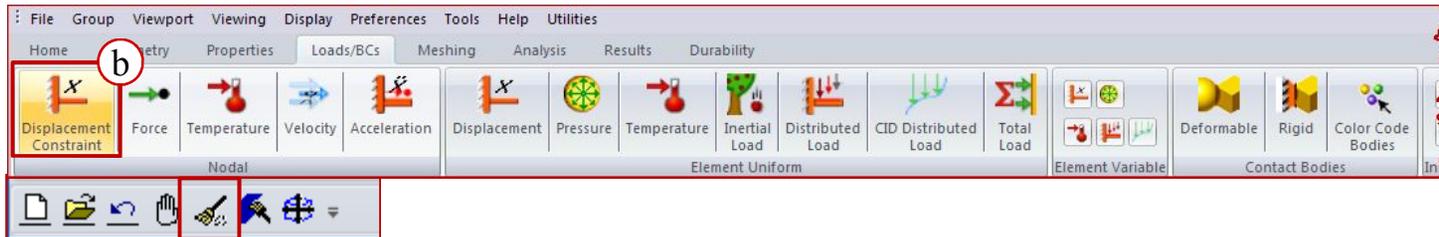
Step 4. Define Contact Pairs (Cont.)

Modify the contact pair parameters

- a. Select **2D_bdy1_2D_bd5_pair**
- b. Select **Modify Data...**
- c. Enter **0.02** for **Distance Tolerance**
- d. Check **Glue Contact, Retain Gaps/Overlaps, Retain Moment.**
- e. Check **Stress Free InitCont**
- f. Select **Automatic** in **Contact Detection**
- g. Click **OK**
- h. Click **Apply**

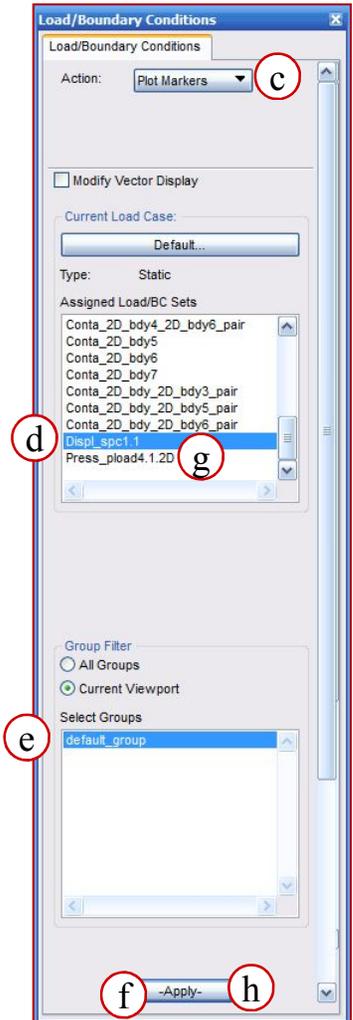
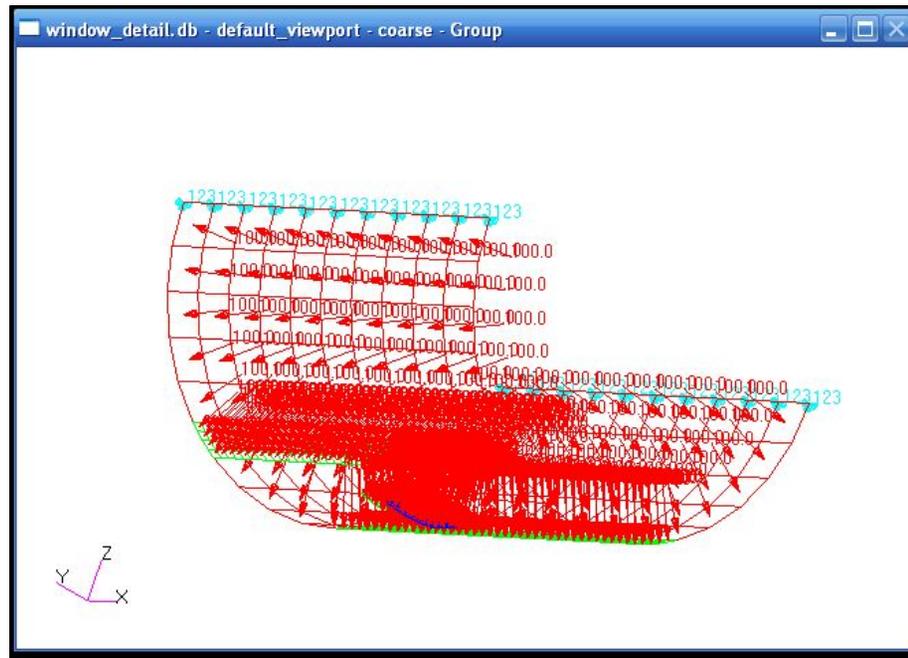


Step 5. View Boundary Conditions



Create a pinned constraint on the axial edges:

- a. Click **Reset Graphics**.
- b. Click **Displacement Constraint** in the *Nodal* group
- c. Pull down Action > **Plot Markers**
- d. Select **Displ_spec1.1**
- e. Select **Default_group**
- f. Click **Apply**
- g. Select **Press_pload4.1.2D**
- h. Click **Apply**

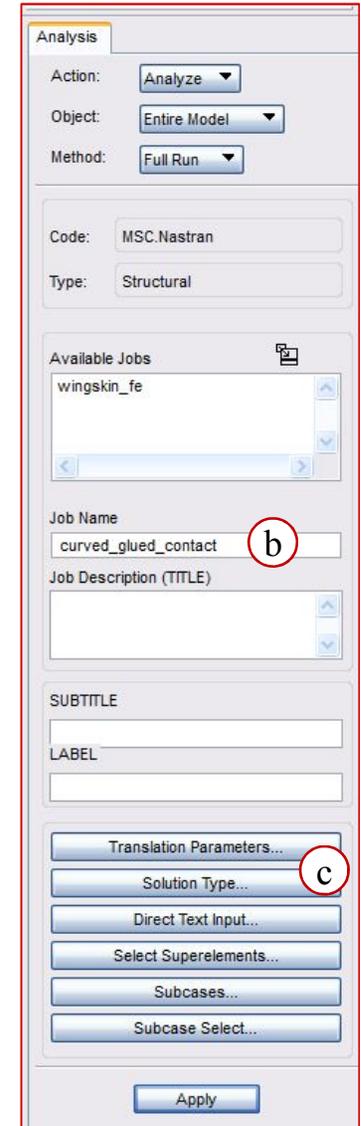
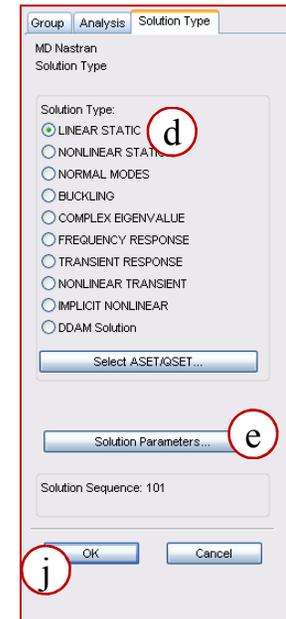
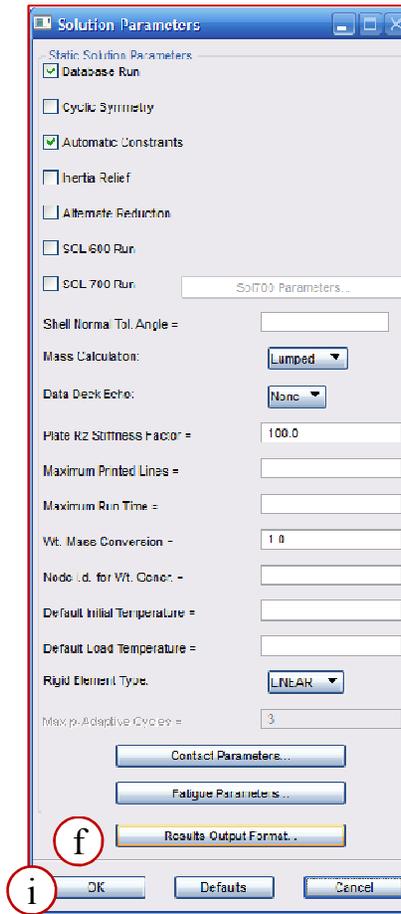
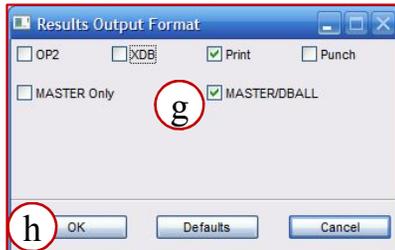


Step 6. Set up Linear Static Analysis



Set up the linear static analysis.

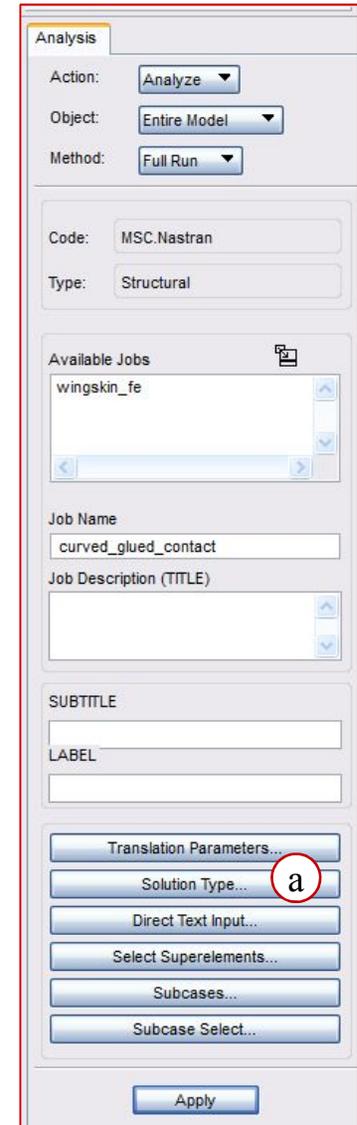
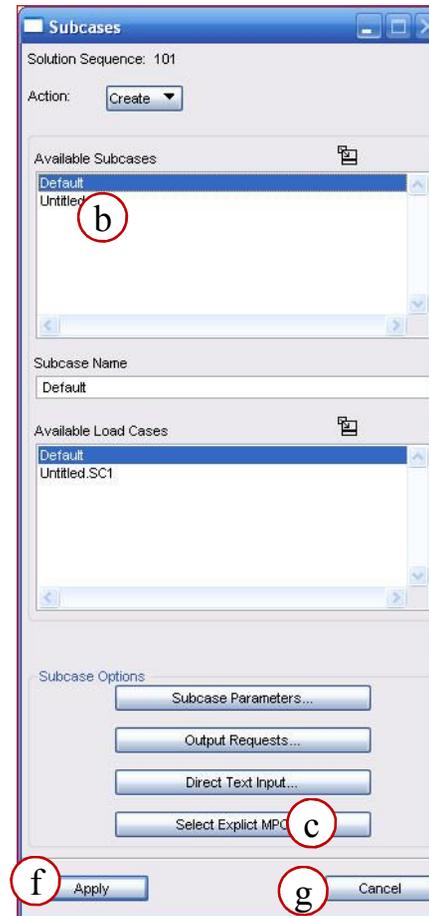
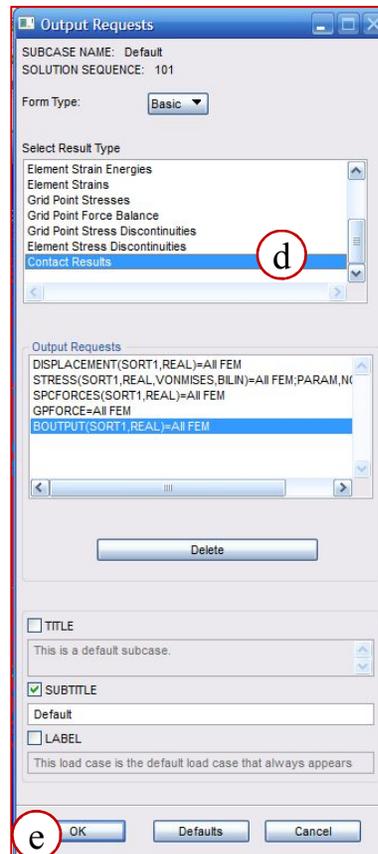
- a. Under the *Analysis* tab, click **Entire Model** in the *Analyze* group.
- b. Enter **curved_glued_contact** for *Job Name*.
- c. Click **Solution Type**.
- d. Select **Linear Static** for *Solution Type*.
- e. Select **Solution Parameters**.
- f. Select **Results Output Format**.
- g. Uncheck XDB and select **MASTER/DBALL**.
- h. Select **OK**
- i. Select **OK**
- j. Select **OK**



Step 6. Set up Linear Static Analysis (Cont.)

Define glued contact between all contact bodies, adjusting the distance tolerance as required.

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

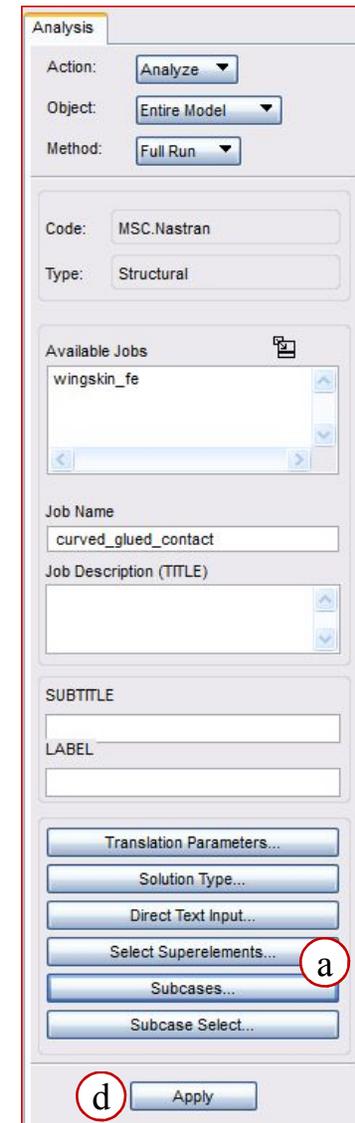
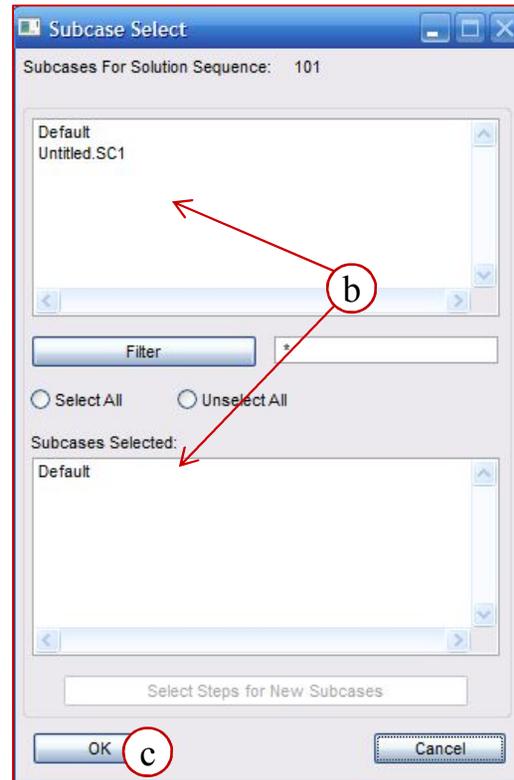


Step 6. Set up Linear Static Analysis (Cont.)

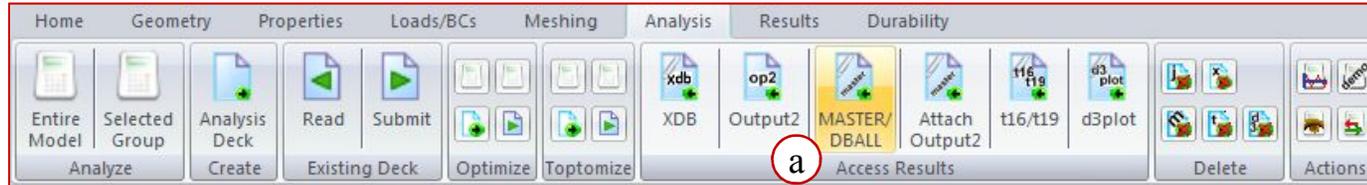
Run the analysis.

- a. Click **Subcase Select**.
- b. Highlight **Default** in the top pane. Ensure *Default* is the only subcase present under *Subcases Selected*.
- c. Click **OK**.
- d. Click **Apply**.

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

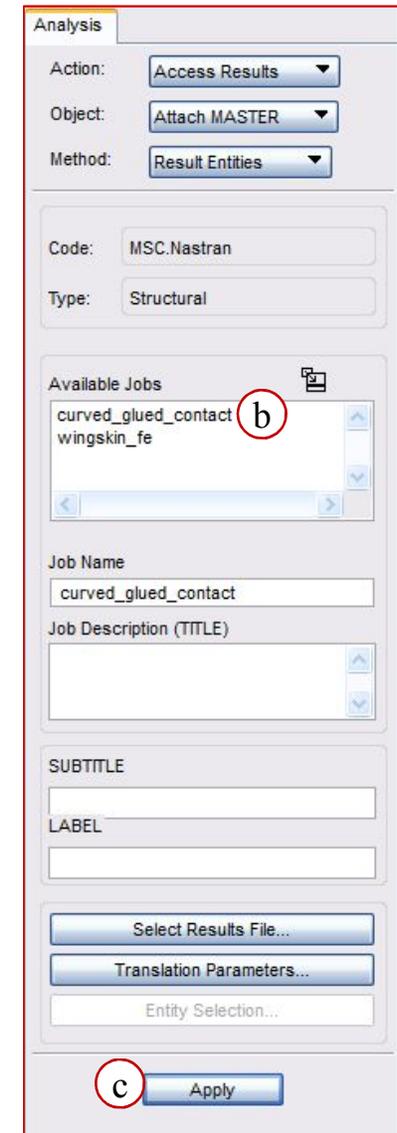


Step 7. Attach the MASTER/DBALL Results

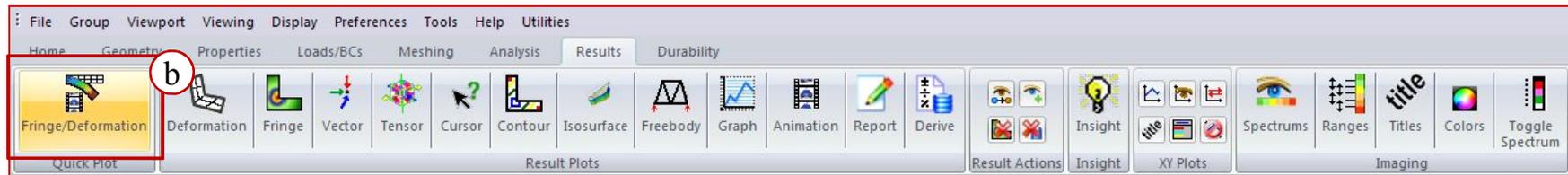


Attach the MASTER/DBALL results:

- a. Click **MASTER/DBALL** in the *Access Results* group
- b. Select `curved_glued_contact` from the list of Available Jobs
- c. Click **Apply**. Patran will look for the results file with a name that matches the Job Name.



Step 8. Post Process with Patran

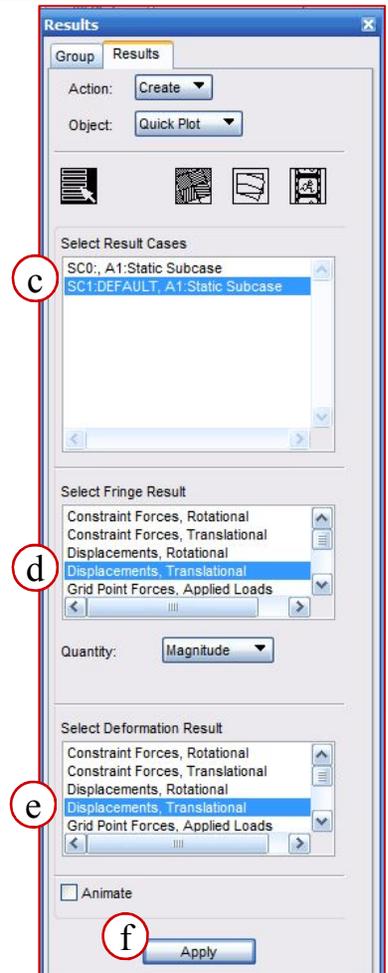
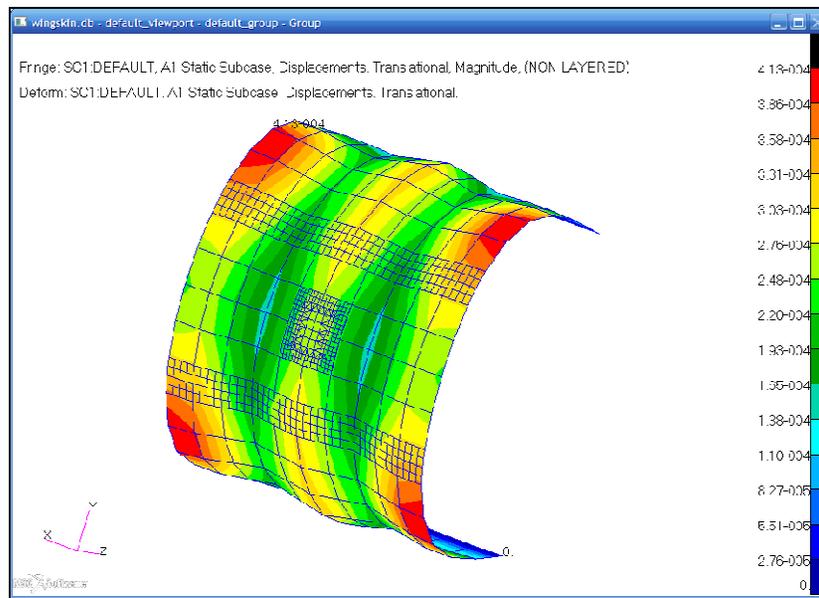


Review the deformation results:



- Click **Reset Graphics**.
- Under the *Results* tab, click **Fringe/Deformation** in the *Quick Plot* group.
- Select the second subcase **SC1:DEFAULT, A1 Static Subcase**
- Highlight **Displacements, Translational** for *Fringe Result*.
- Highlight **Displacements, Translational** for *Deformation Result*.
- Click **Apply**.

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



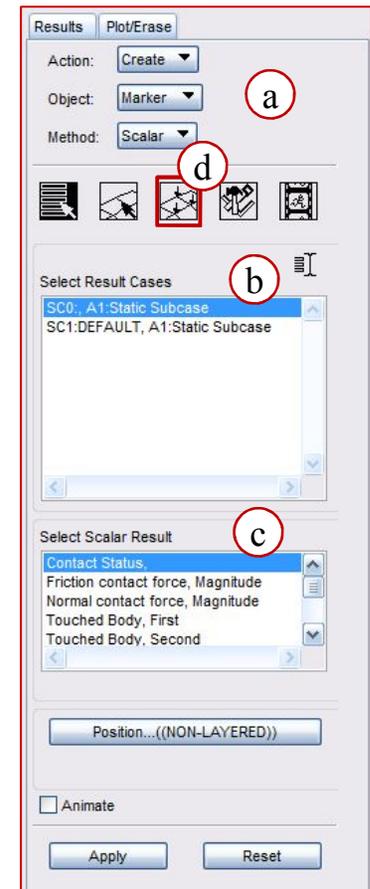
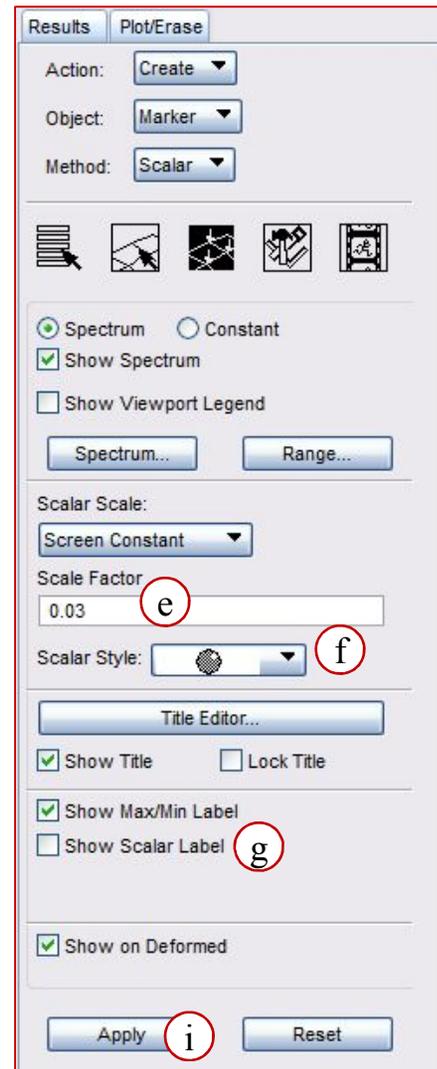
Step 9. Verify Contact Status



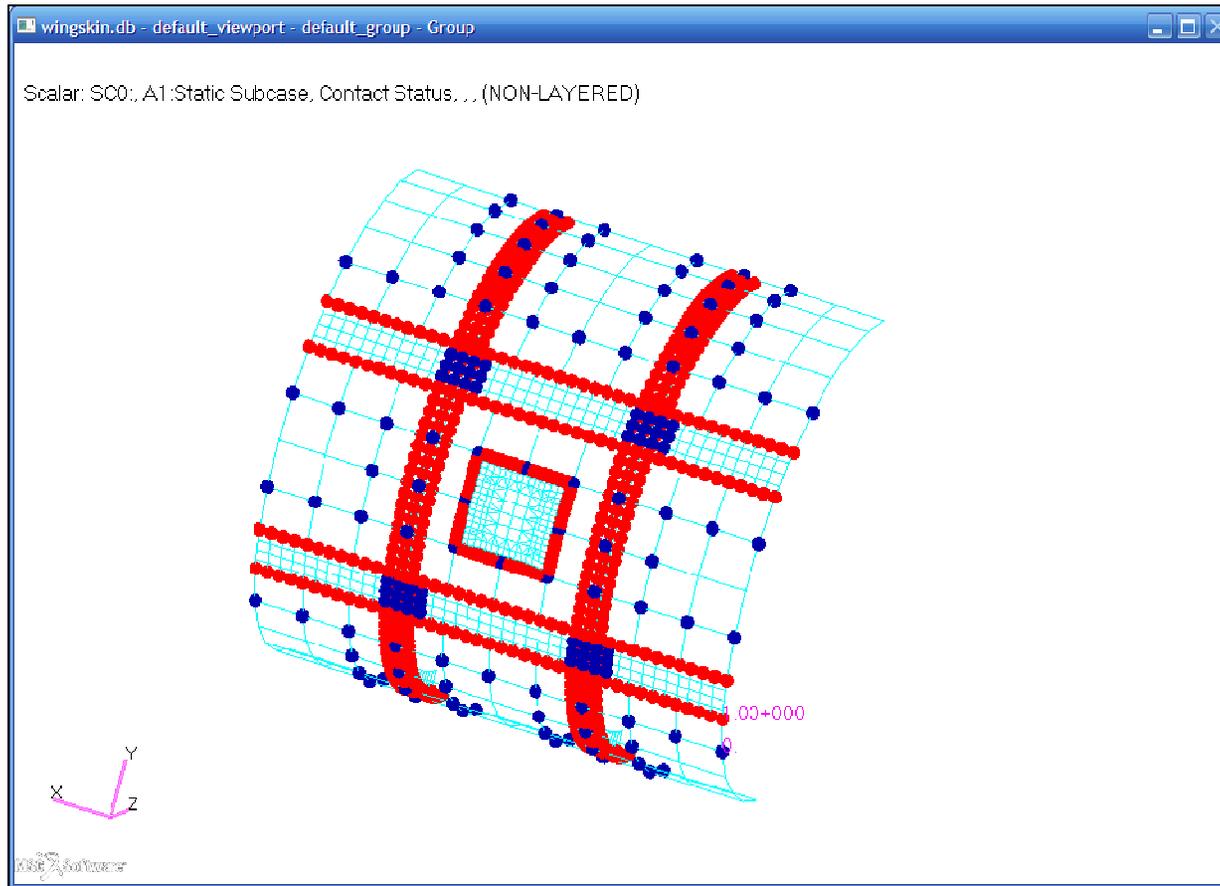
Verify the contact status for each of the contact bodies

- a. Select **Create/Marker/Scalar** from the Results menu.
- b. Select 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. Set the **Scale Factor** to 0.03.
- f. Set the **Scalar Style** to your preference.
- g. Turn off **Show Scalar Label**.
- h. Click **Reset Graphics**.
- i. Select **Apply**.

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



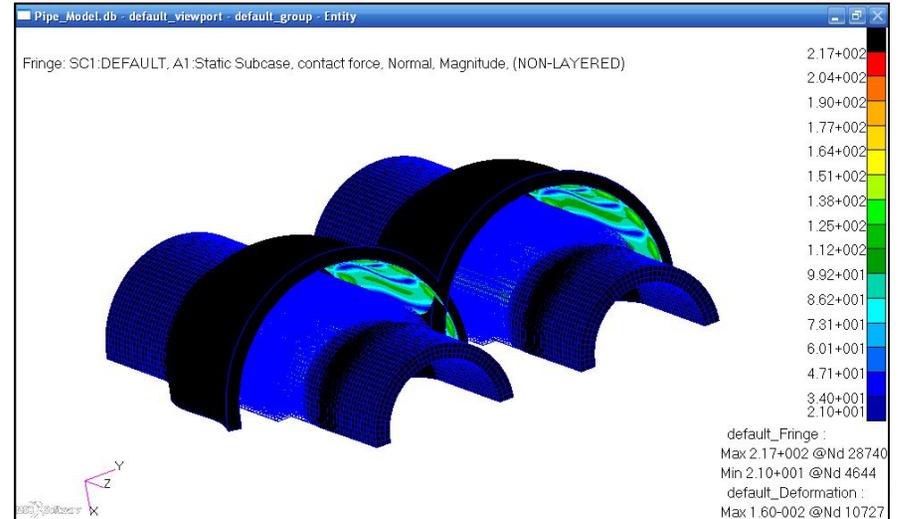
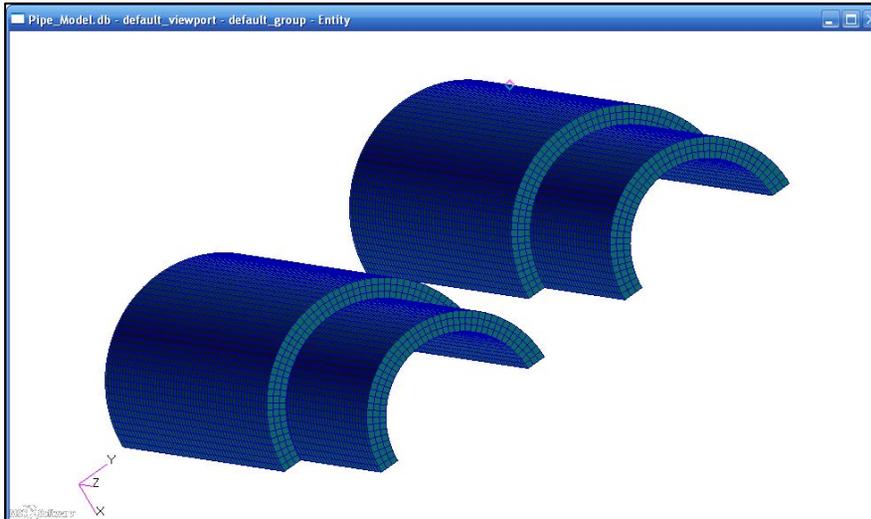
Step 9. Verify Contact Status (Cont.)



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

WORKSHOP 7

INTERFERENCE FIT



- **Workshop Objectives**

- Perform an interference fit using both a fitted closure and geometric overlap

- **Software Version**

- MSC Nastran 2013.1
- Patran 2013

- **Files Required**

- **Pipe_Model_Input.bdf**

• Suggested Steps

1. Create a new database.
2. Import the MSC Nastran input file **Pipe_Model_Input.bdf**.
3. Apply following boundary conditions:
 - Constrain the plane of symmetry
 - Fix both ends of the cylinders
4. Create deformable contact bodies for each of the following property set:
 - Psolid_Fitted_Inner
 - Psolid_Fitted_Outer
 - Psolid_Overlap_Inner
 - Psolid_Overlap_Outer
5. Define a contact pair such that the fitted inner and outer cylinders are touching and the overlapping inner and outer cylinders are touching.
6. Run a Linear Static analysis:
 - Set solution type to 101
 - Select output format to be Master/Dball
 - Request contact results
 - Select subcase and create analysis deck
 - Run the analysis using MSC Nastran

- **Suggested Steps Continued**

7. Attach Master/Dball results file.

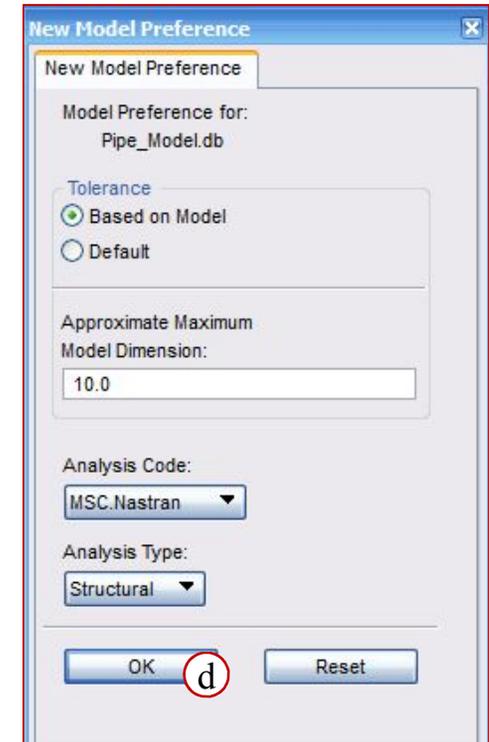
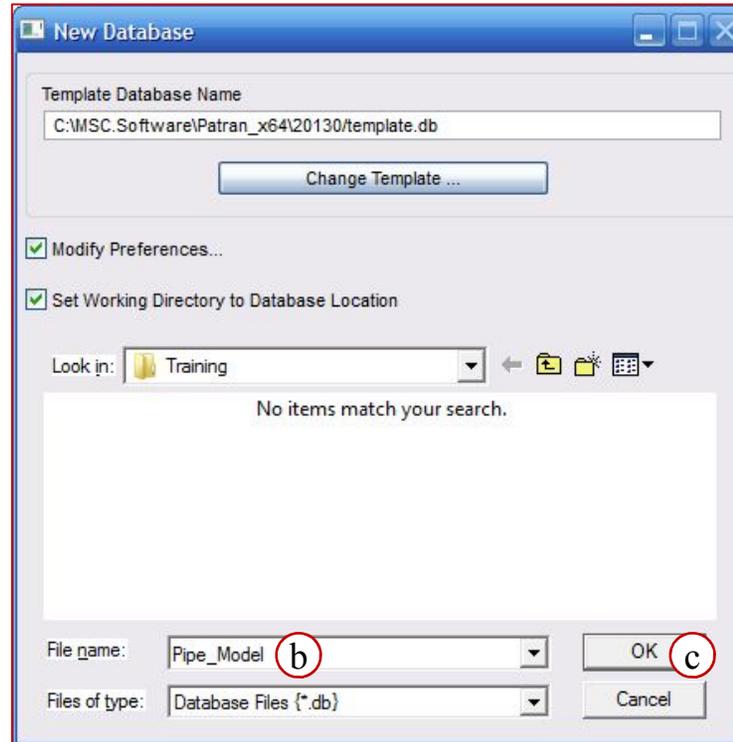
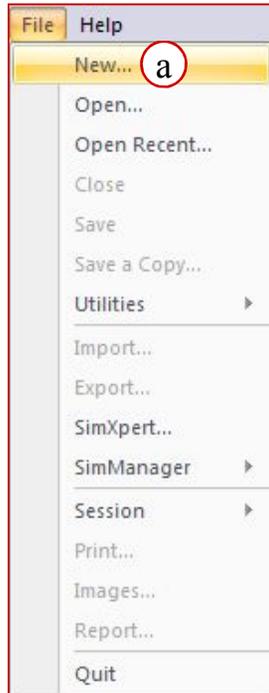
8. Post-Process using Patran:

- Create a fringe plot of von Mises Stress plot superimposed on the deformed shape.
- Create a fringe plot of Contact Force

Step 1. Create a New Database

Create a new database:

- a. Pull down **File > New**
- b. Enter **Pipe_Model** for *File Name*
- c. Click **OK**.
- d. Click **OK** on the *New Model Preference* form to retain the default tolerance.

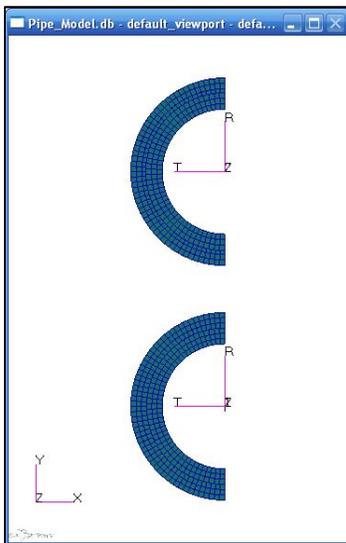
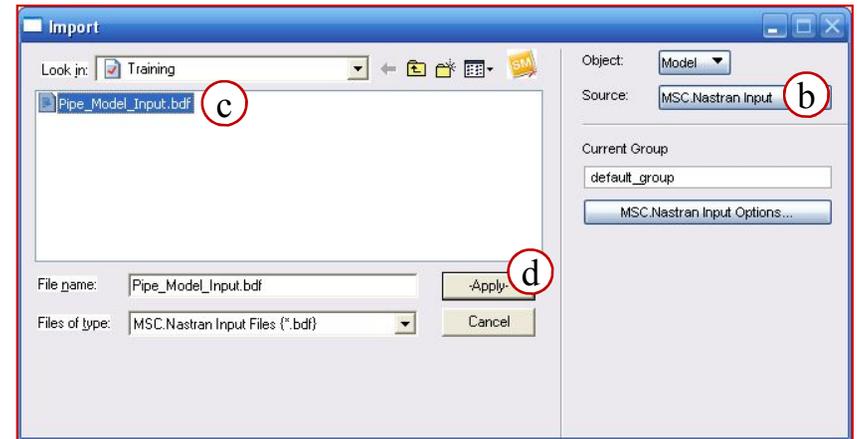


Step 2. Import the Model



Import the MSC Nastran Input file
Pipe_Model_input.bdf:

- a. Pull down **File > Import...**
- b. Pull down **MSC.Nastran Input** for *Source*.
- c. Select **Pipe_Model_Input.bdf** as the file to import.
- d. Click **Apply**.
- e. Check the *Nastran Input File Import Summary* and make sure there are 46864 Nodes, 33858 Elements.
- f. Click **OK**.
- g. Shade the model by clicking **Smooth shaded** in the *Display* group.



	Imported	Imported with Warning	Not Imported
Nodes	46864	0	0
Elements	33858	0	0
Coordinate Frames	2	0	0
Material Propertie	1	0	0
Element Properties	4	0	0
Load Sets	0	0	0
Subcases	1	0	0
MPC Data	0	0	0
Comment Lines	12	0	0

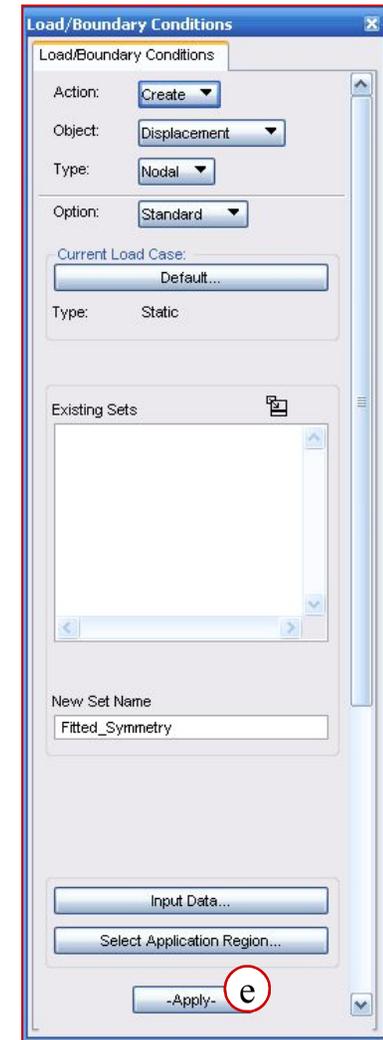
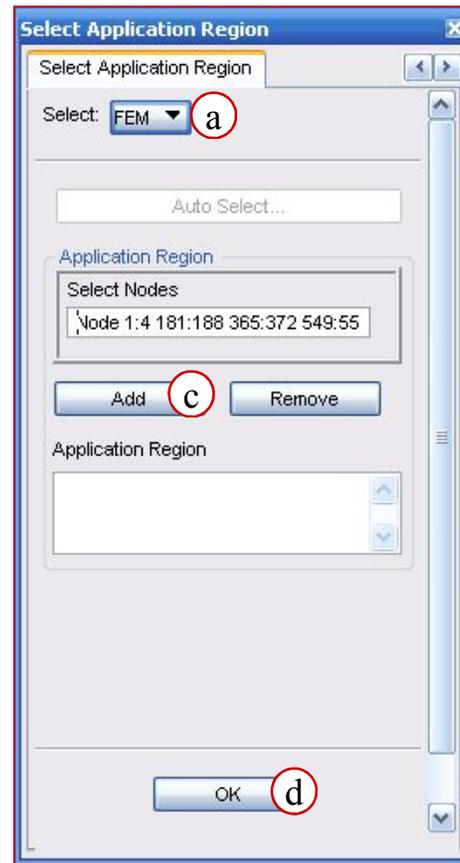
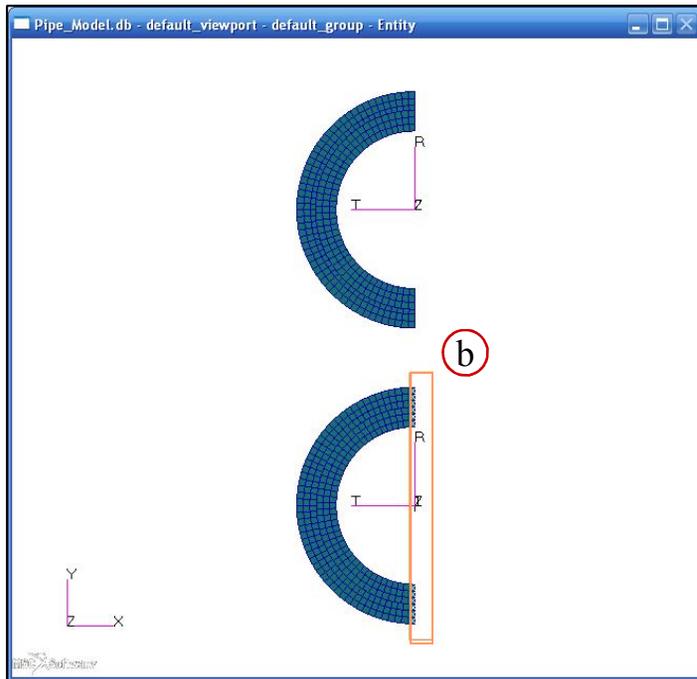
Step 3. Apply Boundary Conditions

Create displacement constraint to constrain the plane of symmetry

- Under the *Loads/BCs* tab, click **Displacement Constraint** in the *Nodal* group
- Enter *New Set Name*: **Fitted_Symmetry**
- Click **Input Data**
- Enter *Translations* $\langle ,0, \rangle$
- Click in the *Analysis Coordinate Frame* list box
- Select the coordinate system at the center of the bottom cylinders.
- Click **OK**
- Click **Select Application Region**

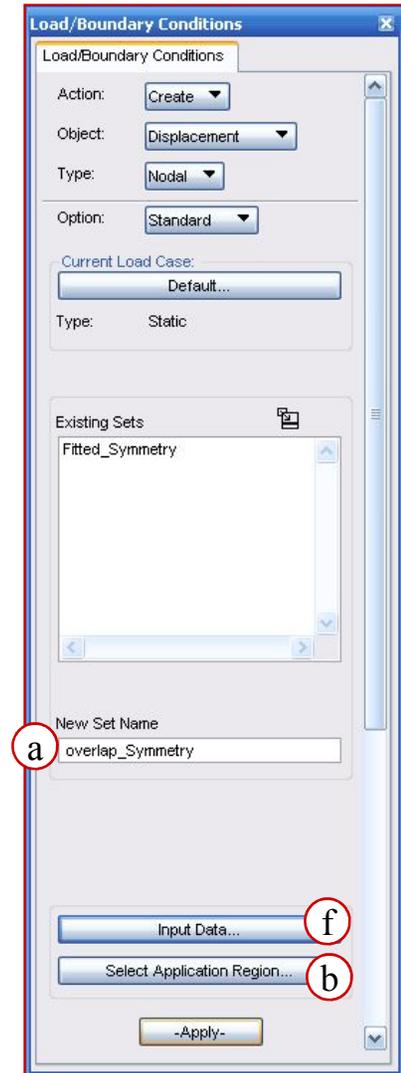
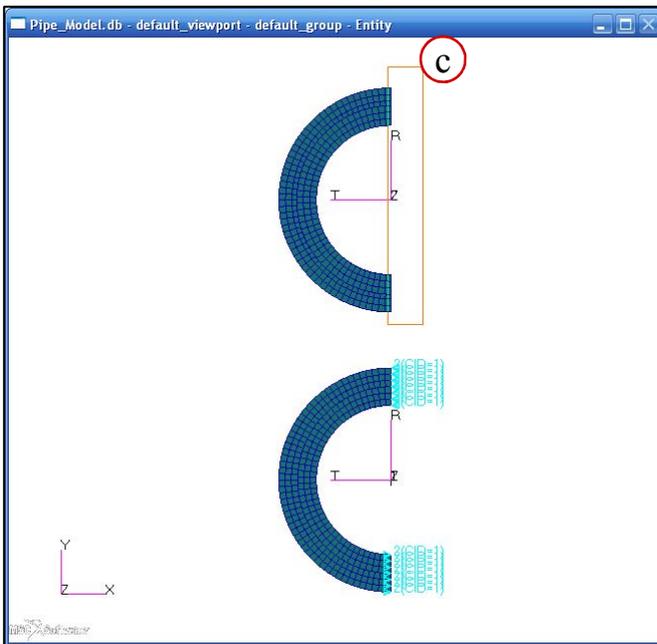
Step 3. Apply Boundary Conditions (Cont.)

- a. Pull down *Select* to **FEM**
- b. Screen Select the **Nodes** as shown
- c. Click **Add**
- d. Click **OK**
- e. Click **Apply**



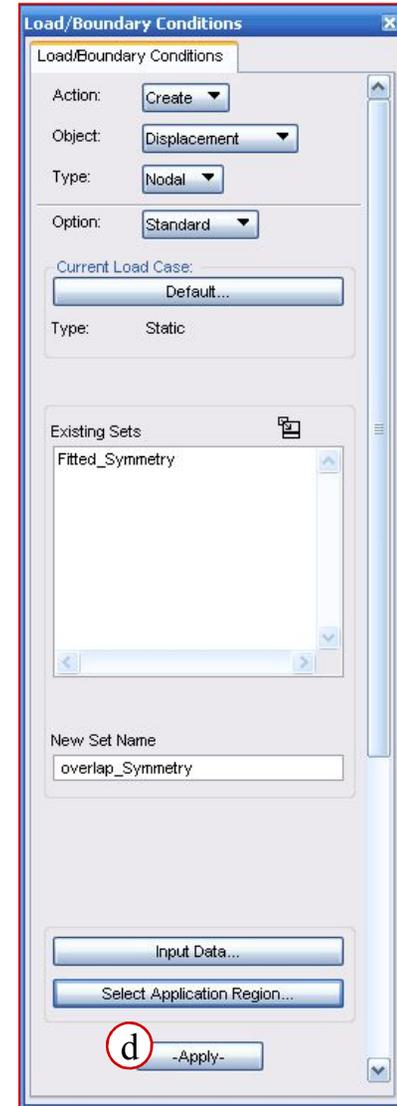
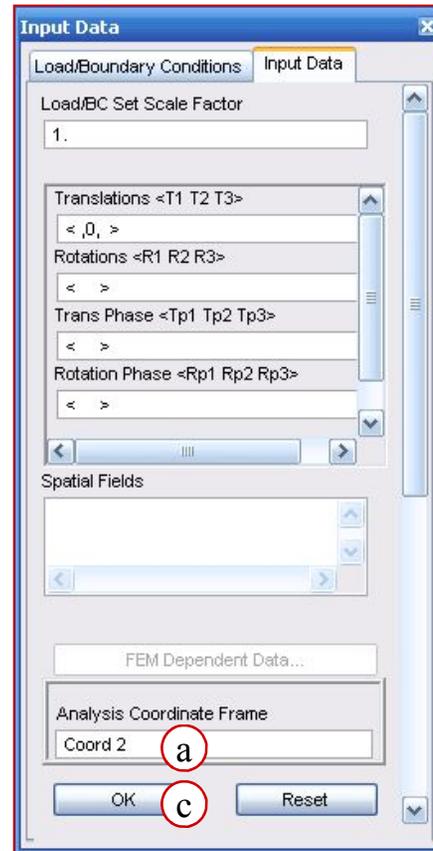
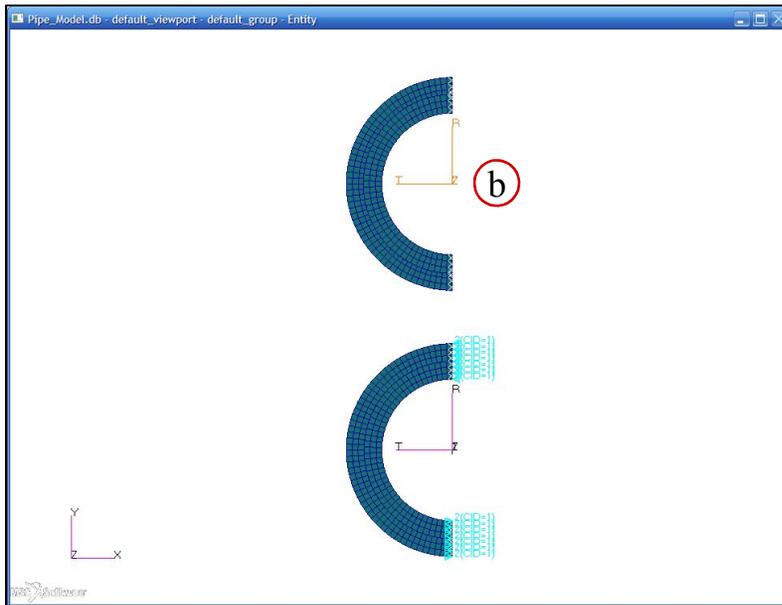
Step 3. Apply Boundary Conditions (Cont.)

- a. Enter New Set Name:
Overlap_Symmetry
- b. Click **Select Application Region**
- c. Screen Select the **Nodes** as shown
- d. Click **Add**
- e. Click **OK**
- f. Click **Input Data**

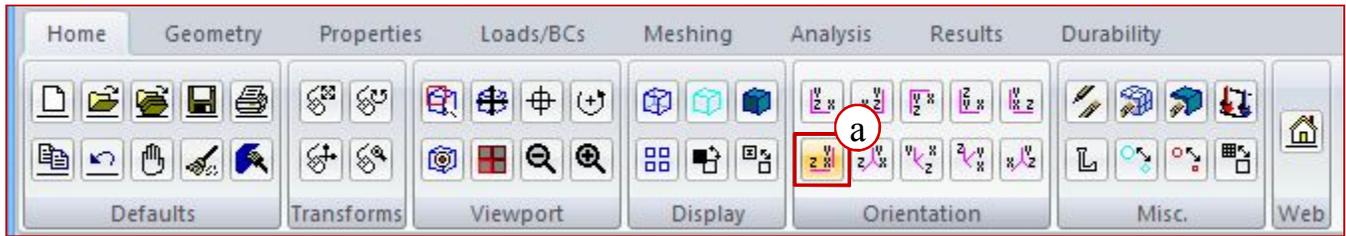


Step 3. Apply Boundary Conditions (Cont.)

- a. Click in the *Analysis Coordinate Frame* list box
- b. Select the coordinate system at the center of the top cylinders.
- c. Click **OK**
- d. Click **Apply**

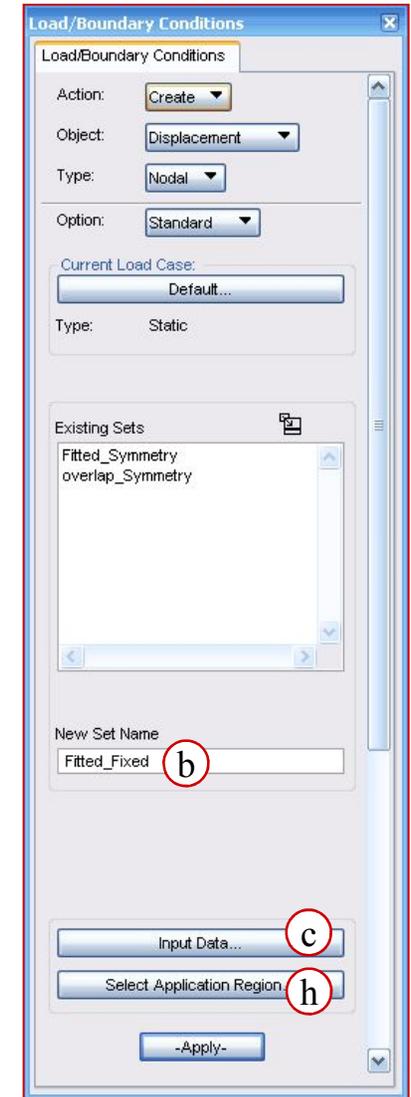
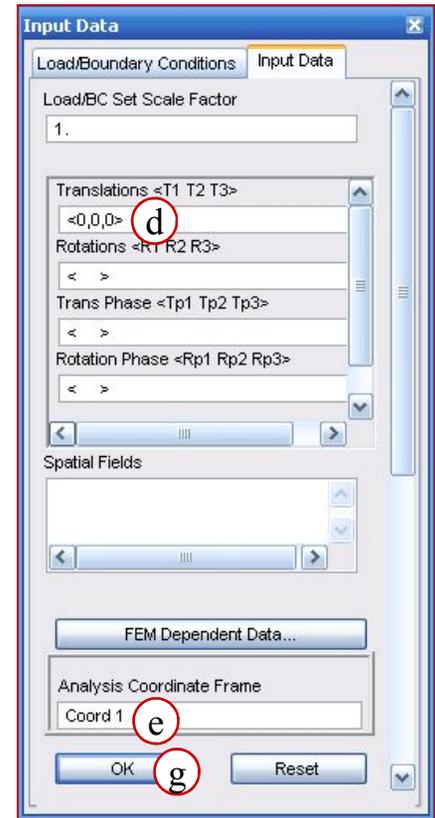
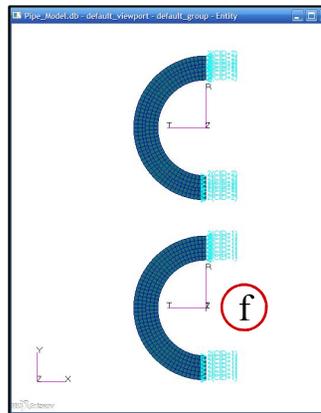


Step 3. Apply Boundary Conditions (Cont.)



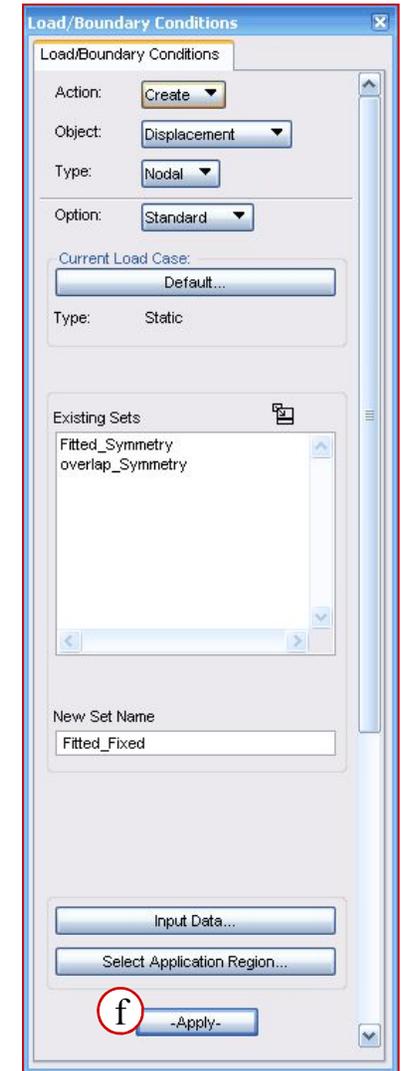
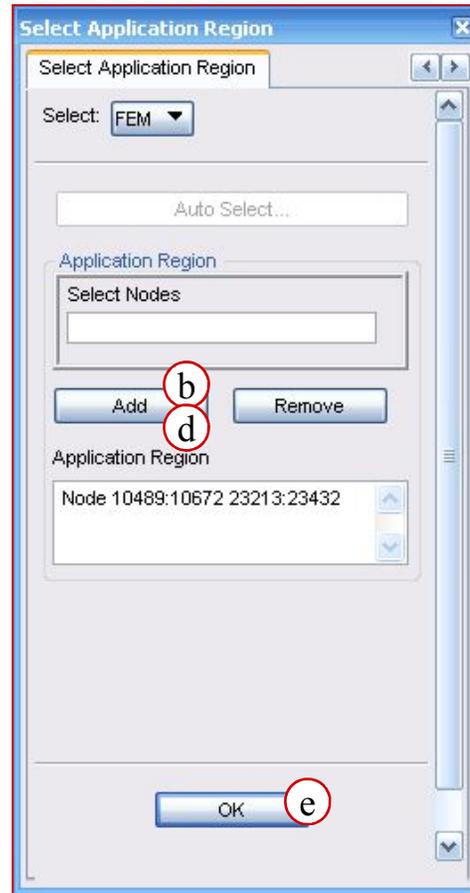
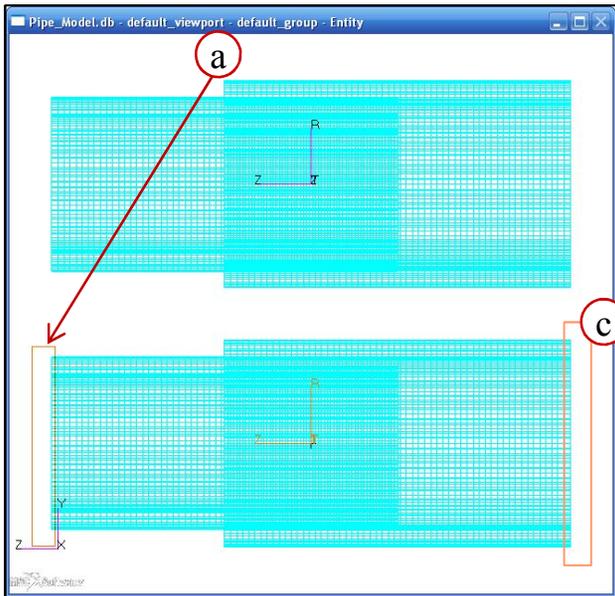
Create a Displacement Constraint to fix the ends of the cylinders

- a. Under the *Home* tab, click **Right side view** in the *Orientation* group
- b. Enter *New Set Name*: **Fitted_Fixed**
- c. Click **Input Data**
- d. Enter *Translations* **< 0 , 0 , 0 >**
- e. Click in the *Analysis Coordinate Frame* list box
- f. Select the coordinate system at the center of the bottom cylinder.
- g. Click **OK**
- h. Click **Select Application Region**



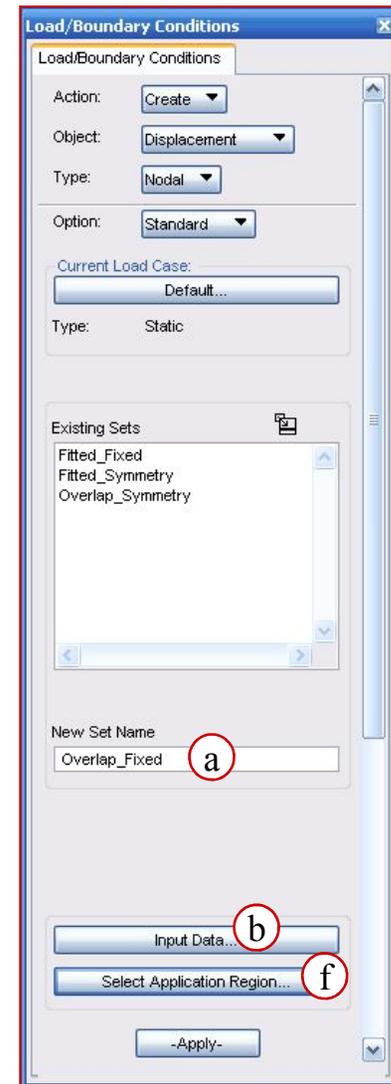
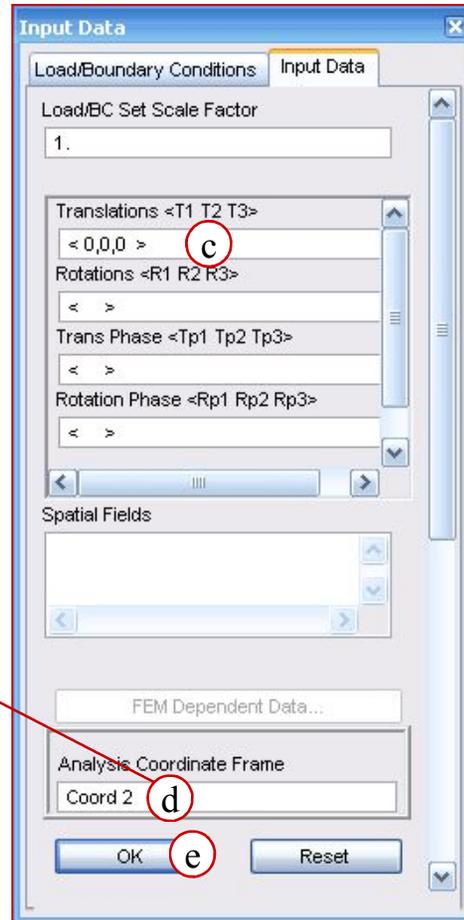
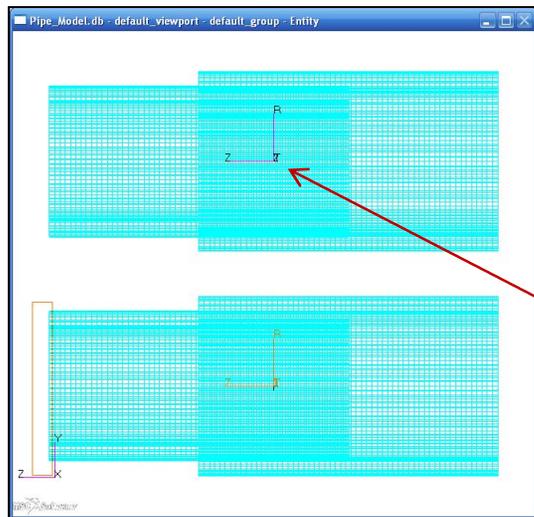
Step 3. Apply Boundary Conditions (Cont.)

- a. Screen select the Nodes on the left end of the lower cylinders as shown.
- b. Click **Add**
- c. Screen select Nodes on the right end of the lower cylinders as shown.
- d. Click **Add**
- e. Click **OK**
- f. Click **Apply**



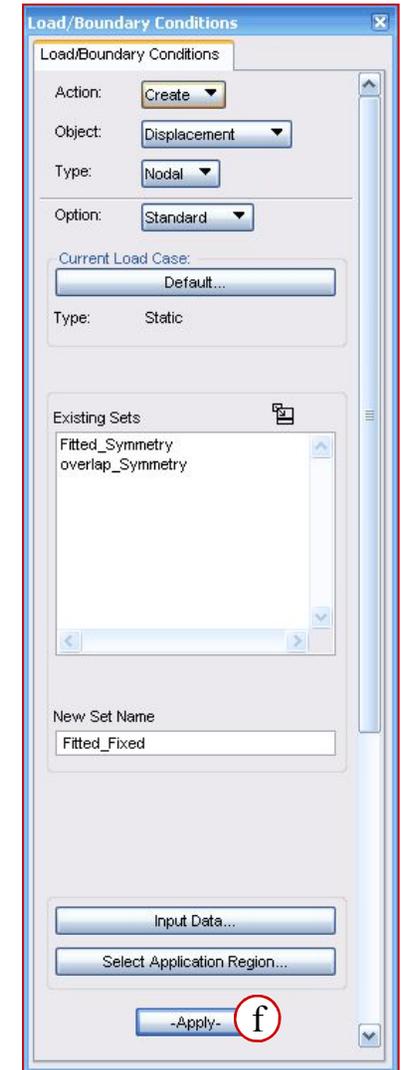
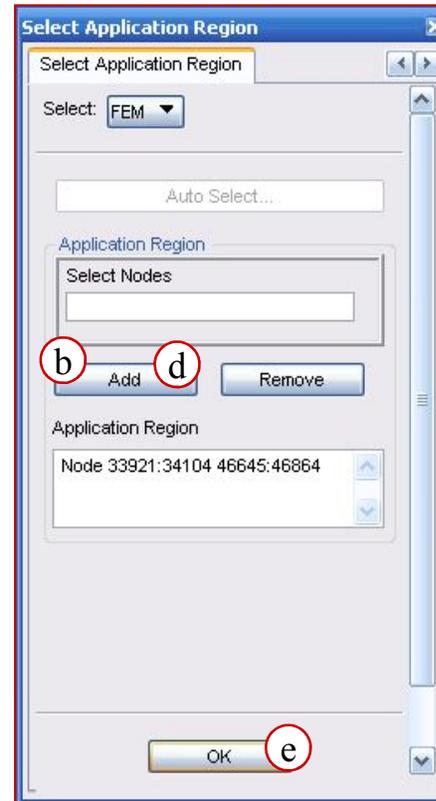
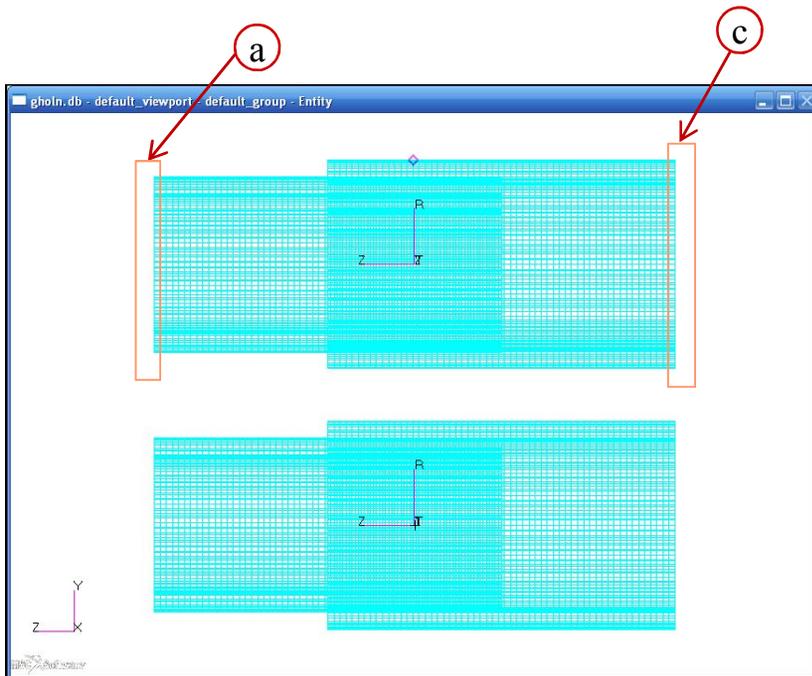
Step 3. Apply Boundary Conditions (Cont.)

- a. Enter New Set Name: **Overlap_Fixed**
- b. Click **Input Data**
- c. Verify the *Translations* are $\langle 0, 0, 0 \rangle$
- d. Click in the *Analysis Coordinate Frame* list box and select the coordinate system at the center of the top cylinders.
- e. Click **OK**
- f. Click **Select Application Region**



Step 3. Apply Boundary Conditions (Cont.)

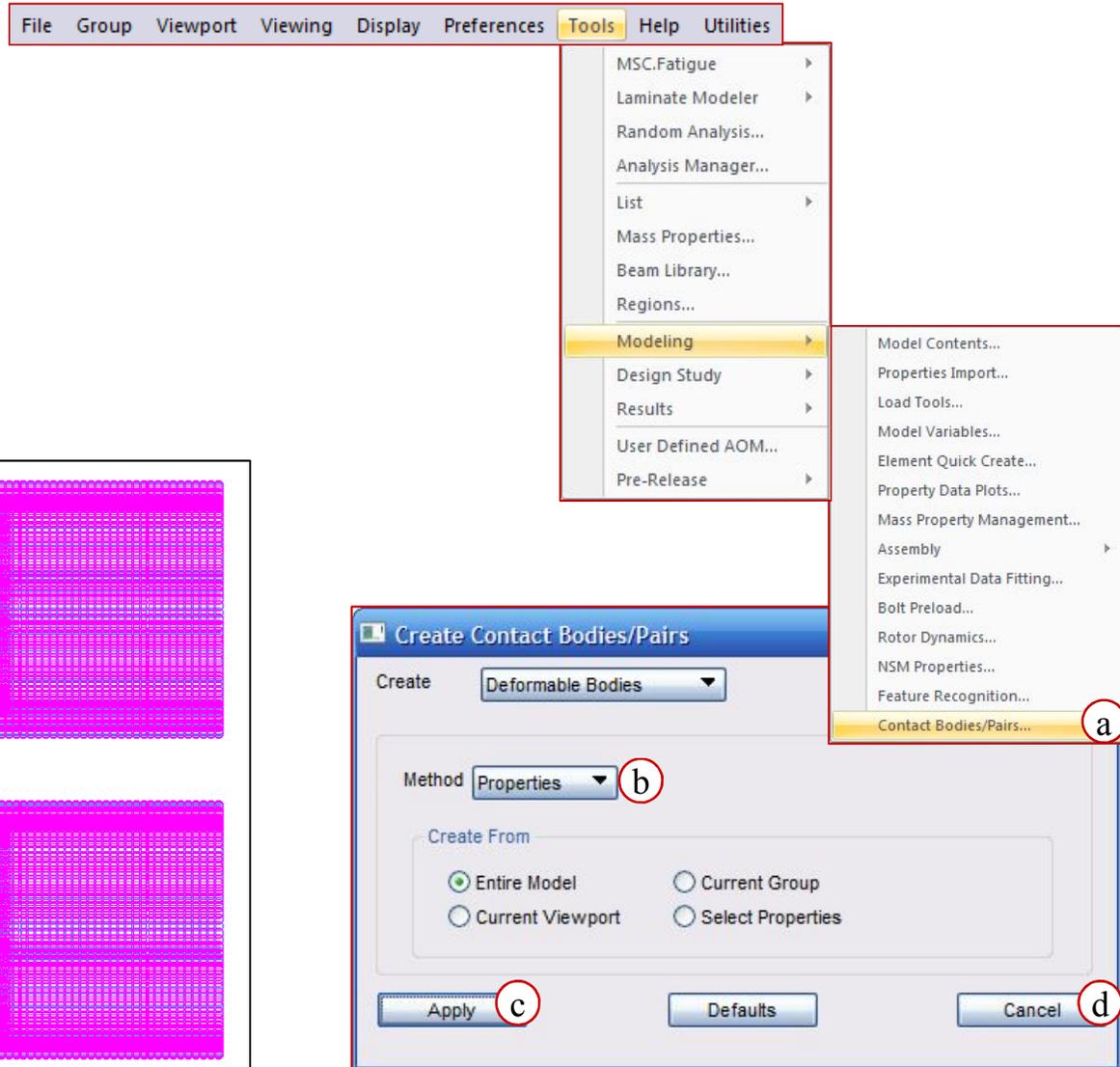
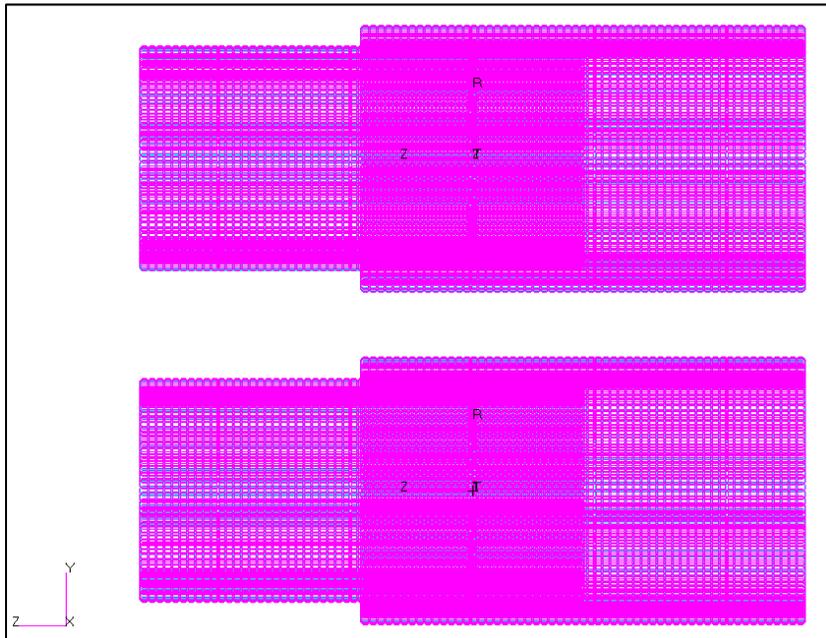
- a. Screen select the Nodes on the left end of the upper cylinders as shown.
- b. Click **Add**
- c. Screen select Nodes on the right end of the upper cylinders as shown.
- d. Click **Add**
- e. Click **OK**
- f. Click **Apply**



Step 4. Create Contact Bodies

Define four deformable bodies for elements associated with each of the four property sets.

- Pull down *Tools > Modeling* and select **Contact Bodies/Pairs...**
- Pull down *Method* to **Properties**
- Click **Apply**
- Click **Cancel**



Step 5. Create Contact Pairs

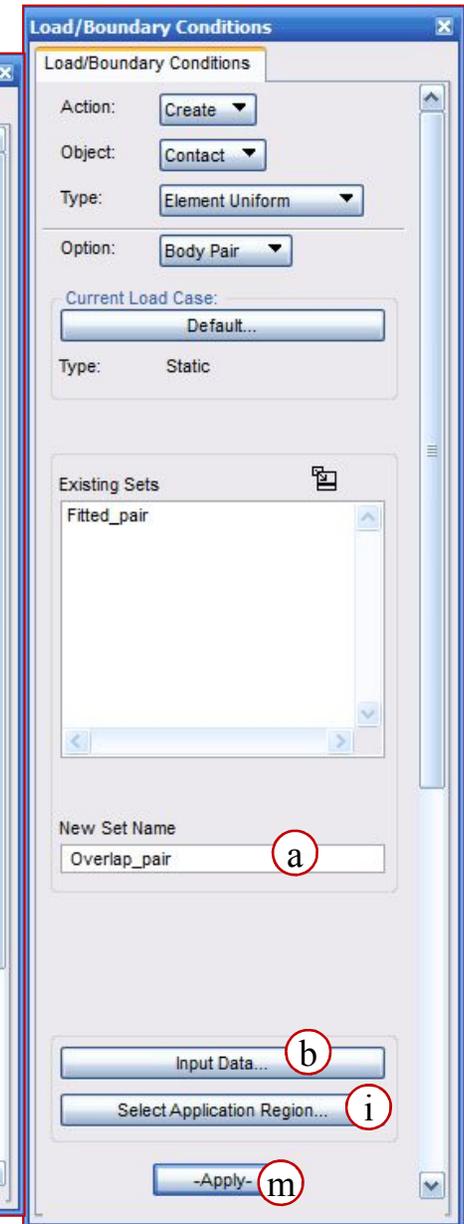
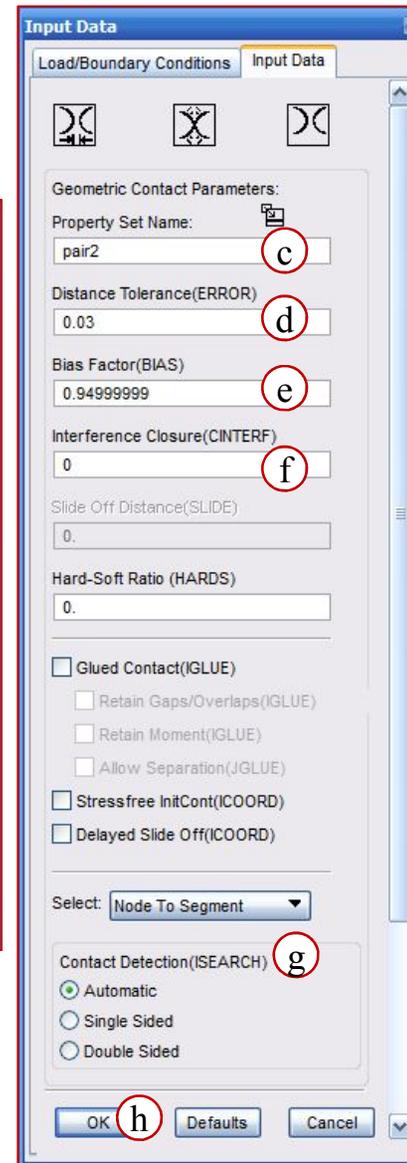
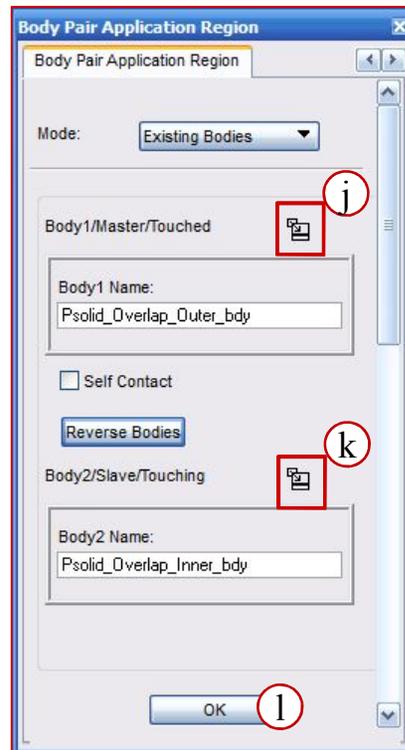
Create touching Contact Pair between Fitted Inner and Outer cylinders

- Under the *Loads/BCs* tab click **Create Body Pair** in the *Body Pairs* group
- Enter **Fitted_pair** for *New Set Name*
- Click **Input Data**.
- Enter **pair1** for *Property Set Name*
- Enter **0.95** for the *Bias Factor (BIAS)*
- Enter **0.02** for the *Interference Closure*
- Select **Automatic** under *Contact Detection*
- Click **OK**
- Click **Select Application Region**
- Click *Body1/Master/Touched* icon and select **Psolid_Fitted_Outer_bdy**
- Click *Body2/Slave/Touching* icon and select **Psolid_Fitted_Inner_bdy**
- Click **OK**
- Click **Apply**

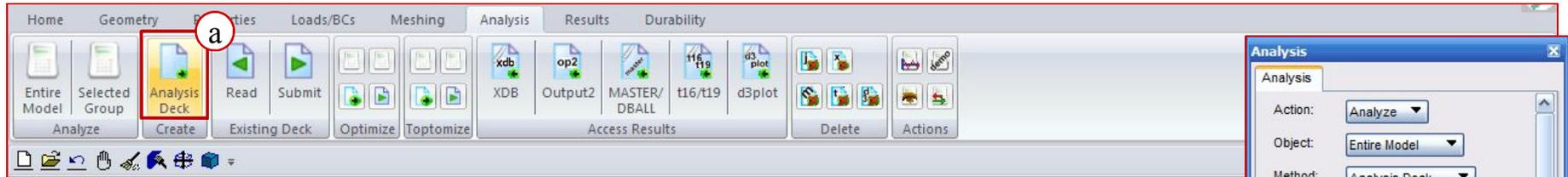
Step 5. Create Contact Pairs (Cont.)

Create touching Contact Pair between Overlapping Inner and Outer cylinders

- Enter **Overlap_pair** for *New Set Name*
- Click **Input Data**.
- Enter **pair2** for *Property Set Name*
- Enter **0.03** for the *Distance Tolerance*
- Enter **0.95** for the *Bias Factor (BIAS)*
- Enter **0** for the *Interference Closure*
- Select **Automatic** under *Contact Detection*
- Click **OK**
- Click **Select Application Region**
- Click *Body1/Master/Touched* icon and select **Psolid_Overlap_Outer_bdy**
- Click *Body2/Slave/Touching* icon and select **Psolid_Overlap_Inner_bdy**
- Click **OK**
- Click **Apply**

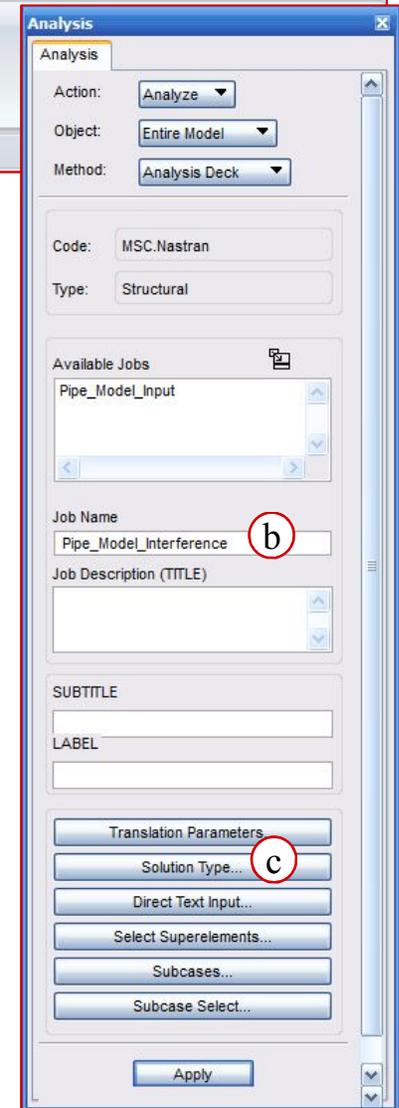
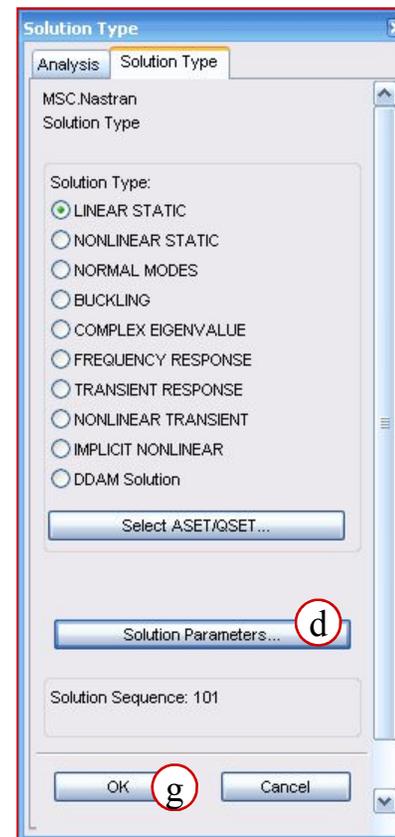
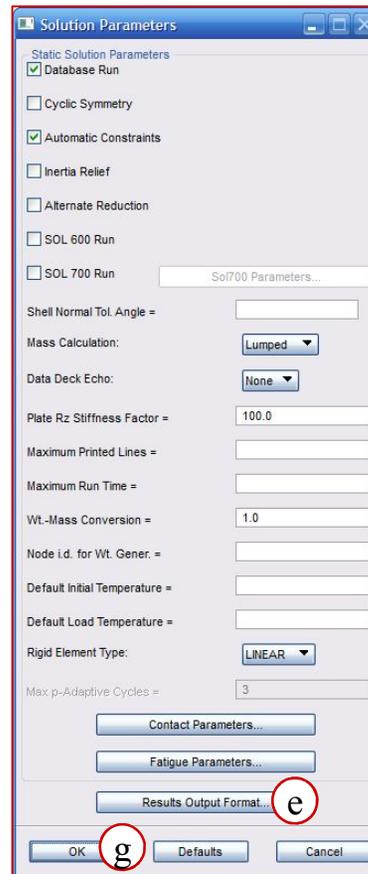
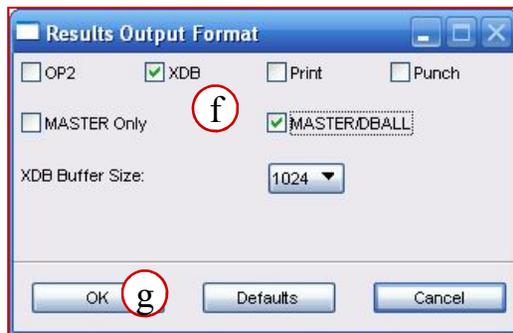


Step 5. Set Up and Run the Analysis



Set Solution Type to 101 and request output format as MASTER/DBALL.

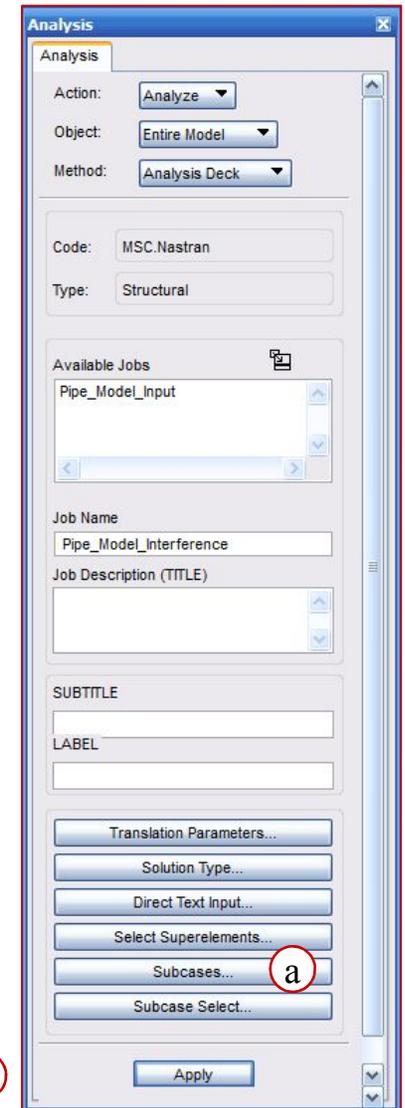
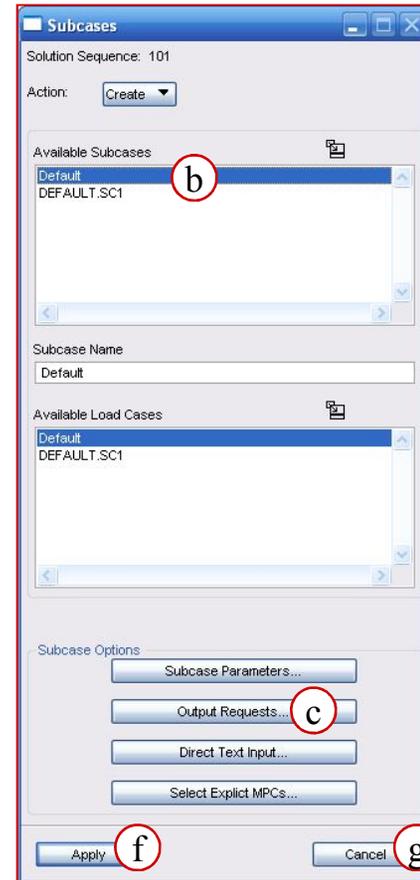
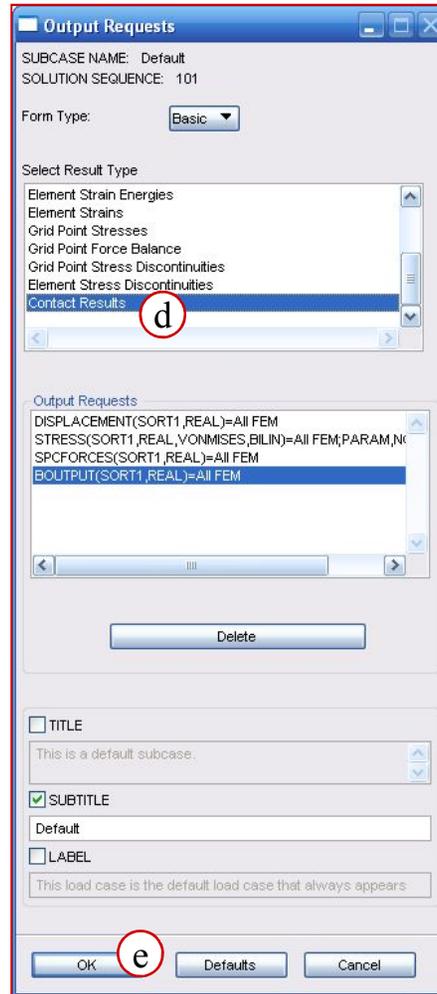
- a. Under the *Analysis* tab, click **Analysis Deck** in the *Create* group
- b. Enter *Job Name*: **Pipe_Model_Interference**
- c. Click **Solution Type...**
- d. Click **Solution Parameters...**
- e. Click **Results Output Format**
- f. Uncheck **Print** and check **XDB** and **Master/DBALL**
- g. Click **OK** on all forms



Step 5. Set Up and Run the Analysis (Cont.)

Modify the Default subcase to include contact results.

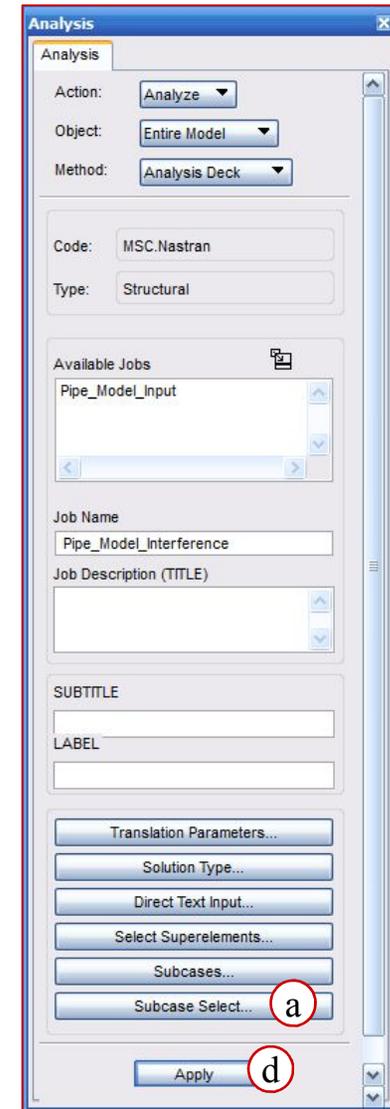
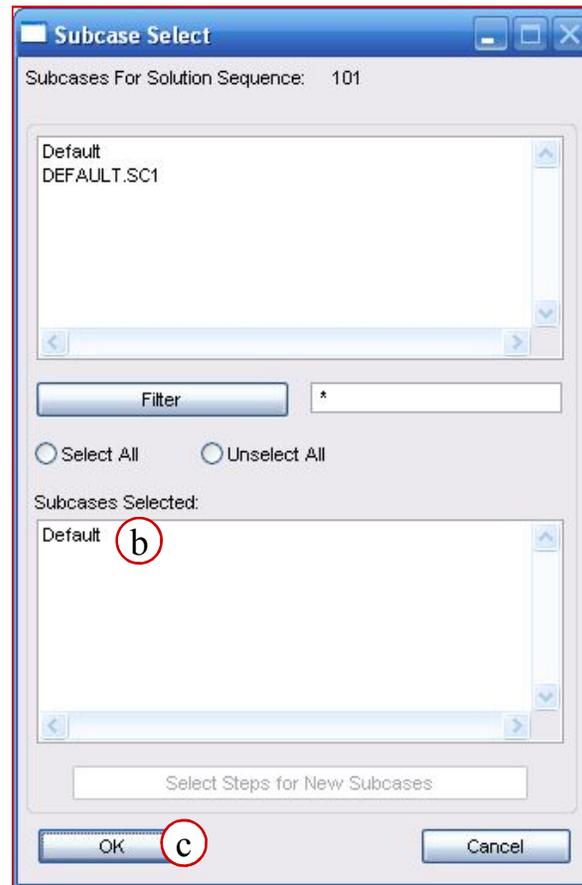
- a. Click **Subcases**.
- b. Under *Available Subcases* highlight **Default**
- c. Click **Output Requests**
- d. Highlight **Contact Results**
- e. Click **OK**
- f. Click **Apply**
- g. Click **Cancel**



Step 5. Set Up and Run the Analysis (Cont.)

Select the Default subcase and create an analysis deck.

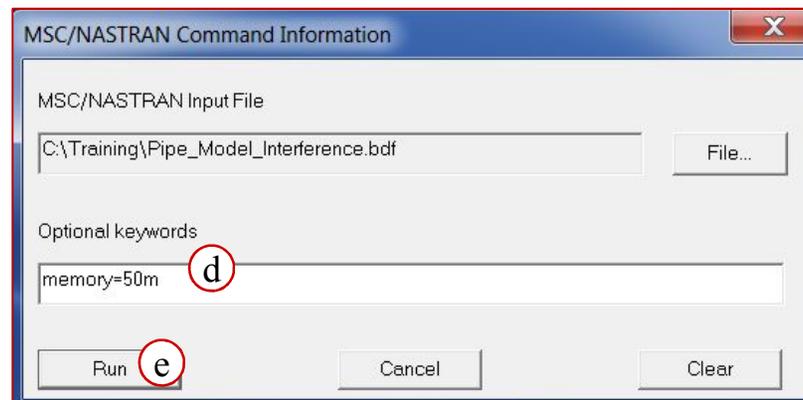
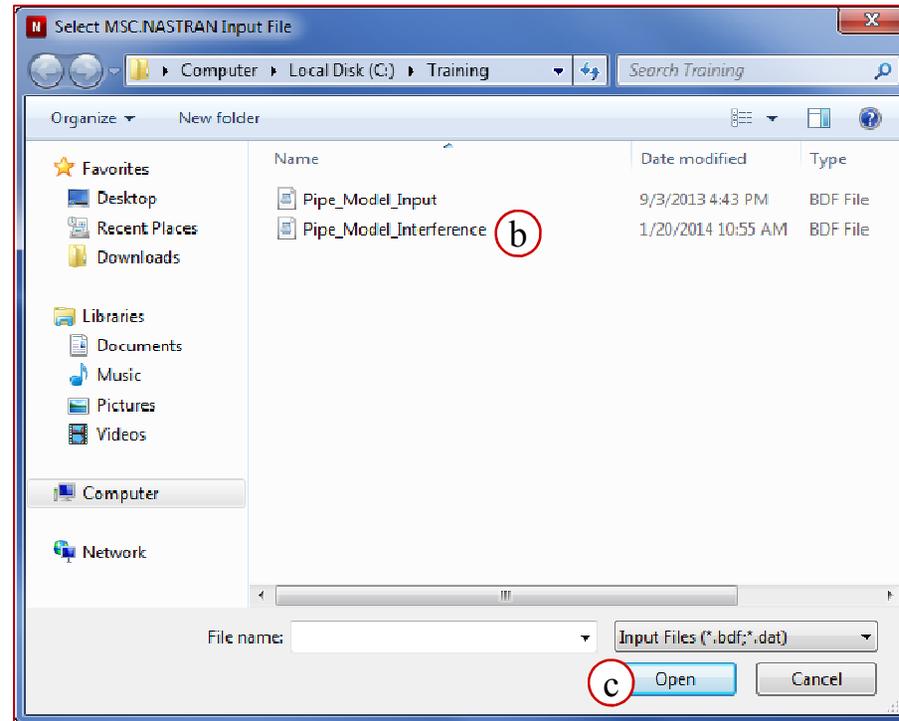
- a. Click **Subcase Select**.
- b. Make sure **Default** is selected
- c. Click **OK**
- d. Click **Apply**



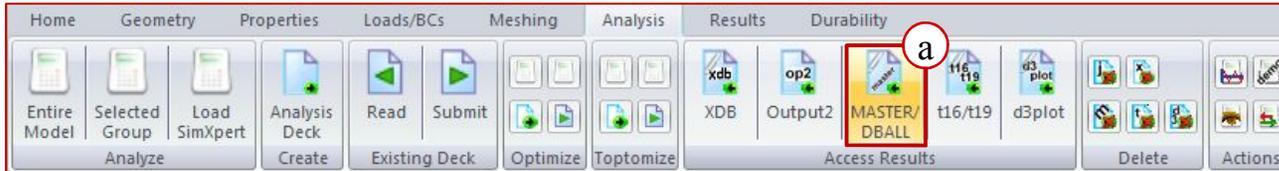
Step 5. Set Up and Run the Analysis (Cont.)

Run the **Pipe_Model_Interference.bdf** file using MSC Nastran.

- a. Click the **MSC Nastran 2013.1** icon on your Desktop.
- b. Browse to and select **Pipe_Model_Interference.bdf**.
- c. Click **Open**.
- d. Under *Optional keywords* enter: **memory=50m** to allocate more memory for running this analysis.
- e. Click **Run**.

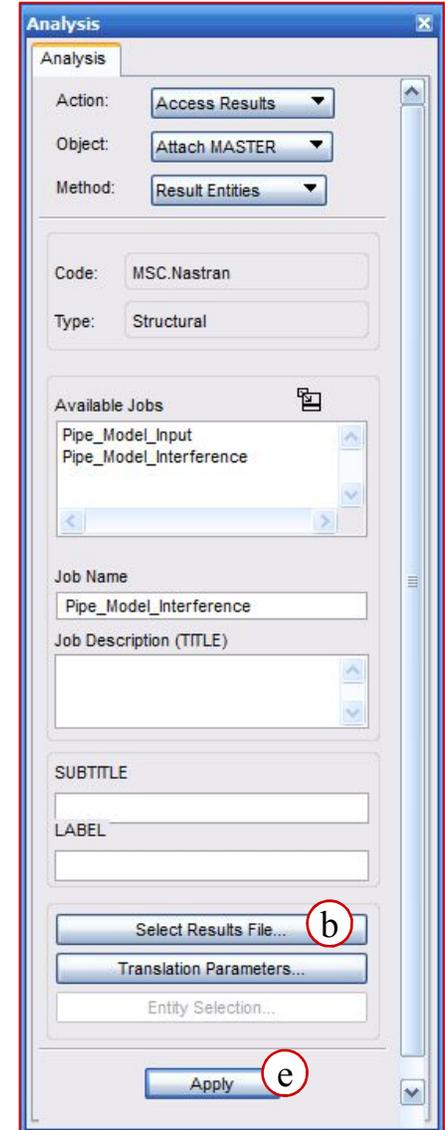
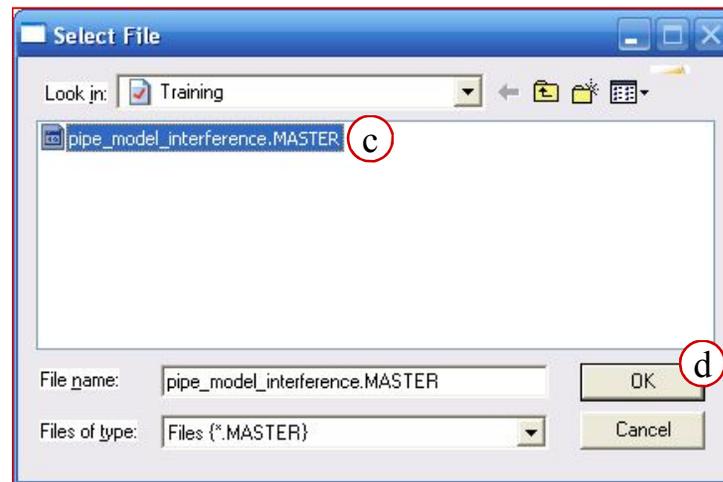


Step 6. Attach Results File

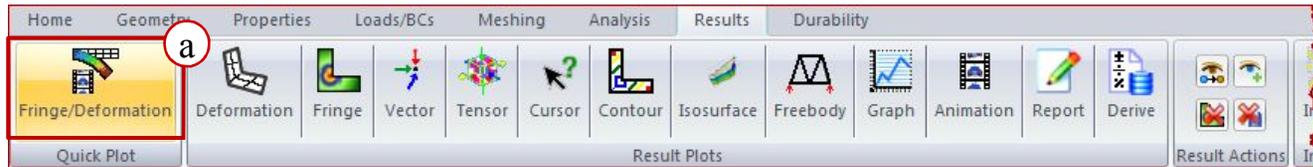


Attach result file:

- a. Under the *Analysis* tab, click **MASTER/DBALL** in the *Access Results* Group.
- b. Click **Select Results File**.
- c. Browse to and select **pipe_model_interference.MASTER**.
- d. Click **OK**
- e. Click **Apply**

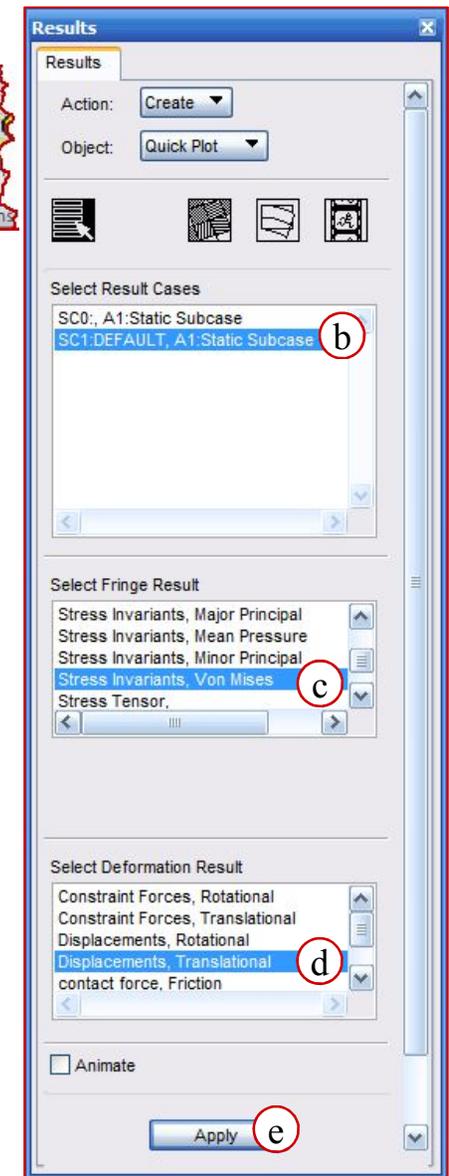


Step 7. Post-Process with Patran

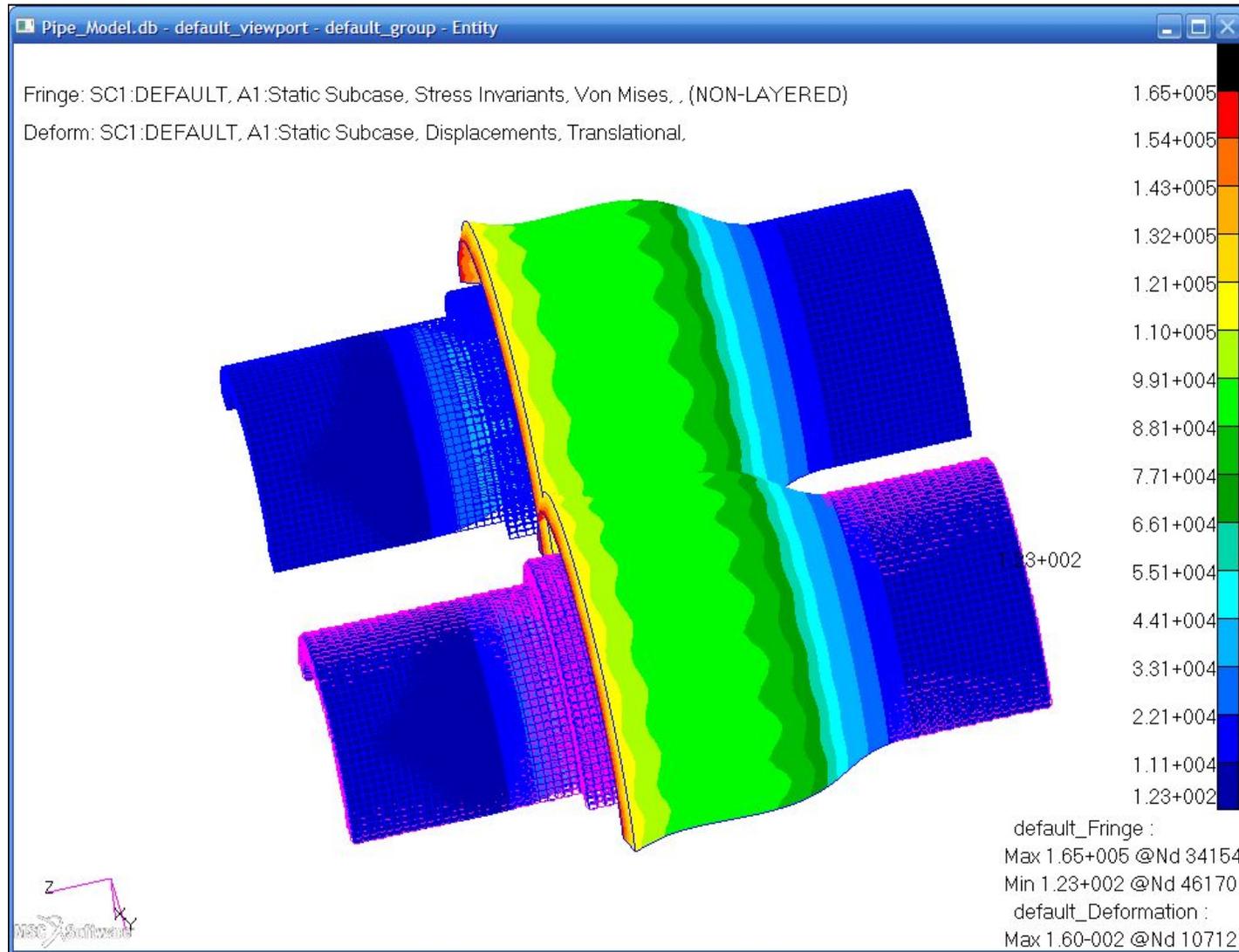


Create a fringe plot of von Mises Stress plot superimposed on the deformed shape.

- a. Under *Results* tab, click **Fringe/Deformation** in the *Quick Plot* group.
- b. Highlight **SC1:DEFAULT, A1:Static Subcase**
- c. Under *Select Fringe Result*, highlight **Stress Invariants, von Mises**
- d. Under *Select Deformation Result*, highlight **Displacements, Translational**
- e. Click **Apply**



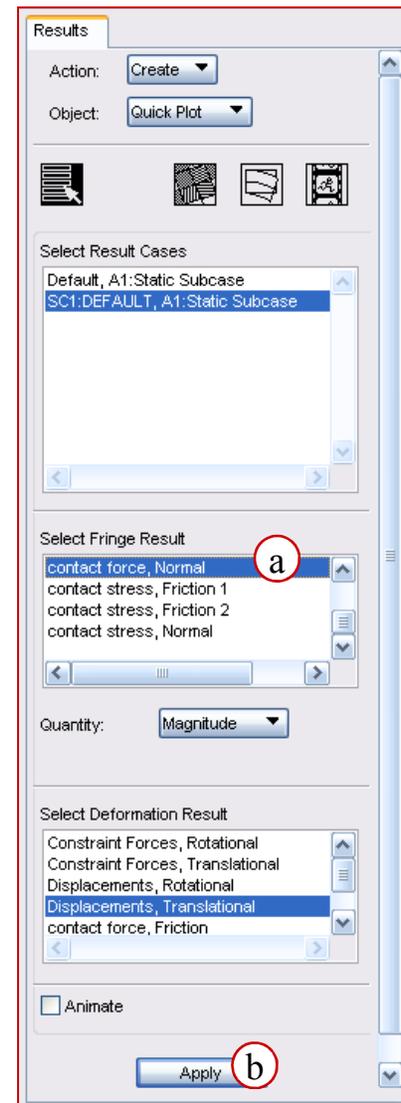
Step 7. Post-Process with Patran (Cont.)



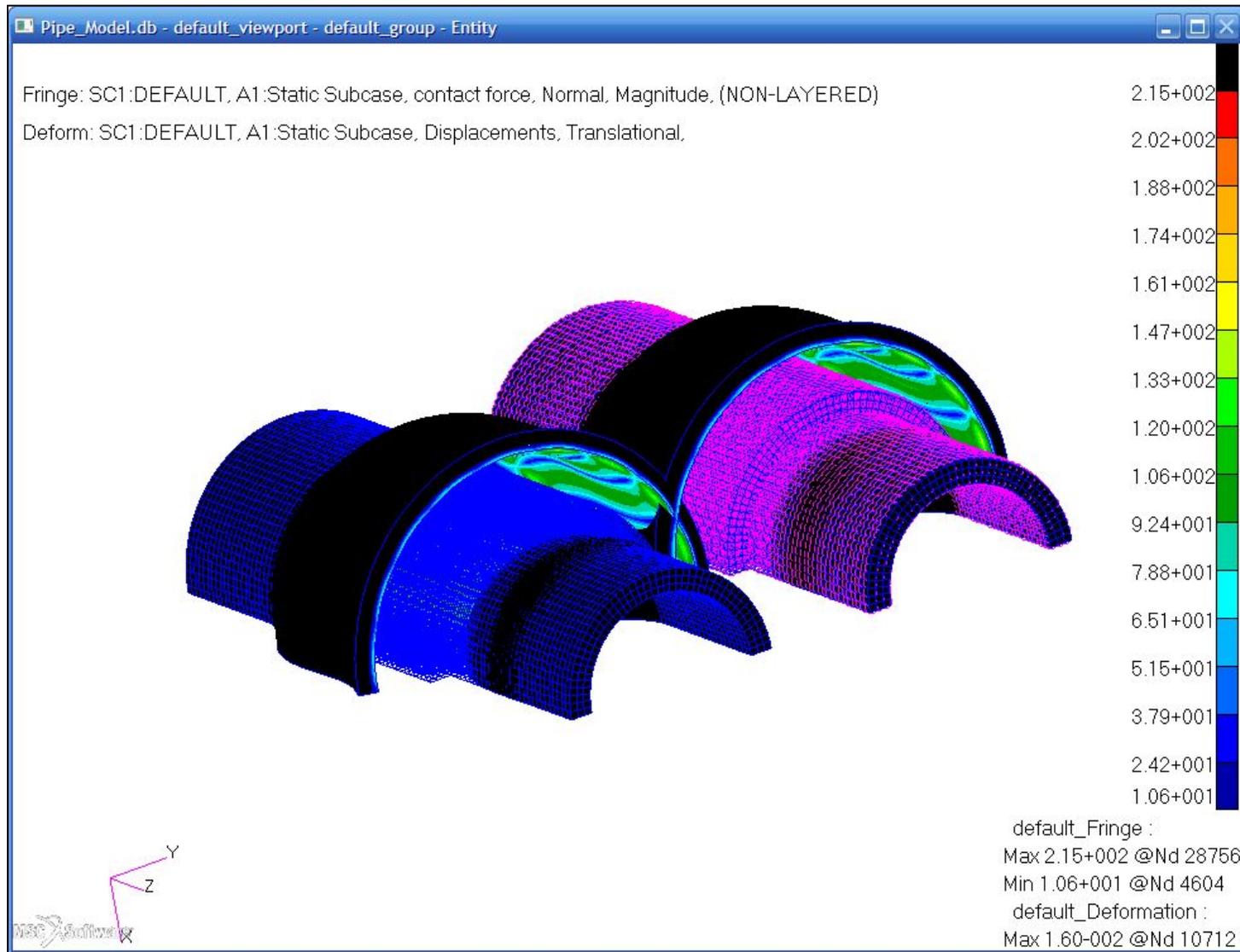
Step 7. Post-Process with Patran (Cont.)

Create a fringe plot of Contact Force

- a. Under *Select Fringe Result*, highlight **Contact force, Normal**
- b. Click **Apply**

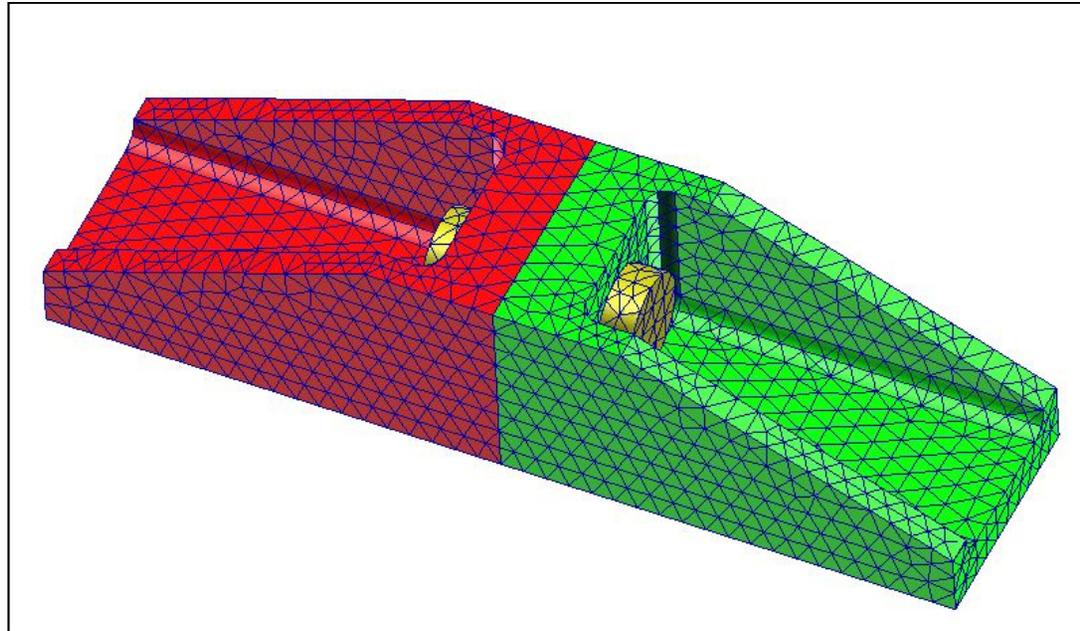


Step 7. Post-Process with Patran (Cont.)



WORKSHOP 8

BOLTED-JOINT ANALYSIS



- **Workshop Objectives**

- Compare SOL 101 and SOL 400 analyses for a bolted joint.
- Note that in SOL 101 bolt preload cannot be applied whereas in SOL 400 bolt preload can be applied.

- **Software Version**

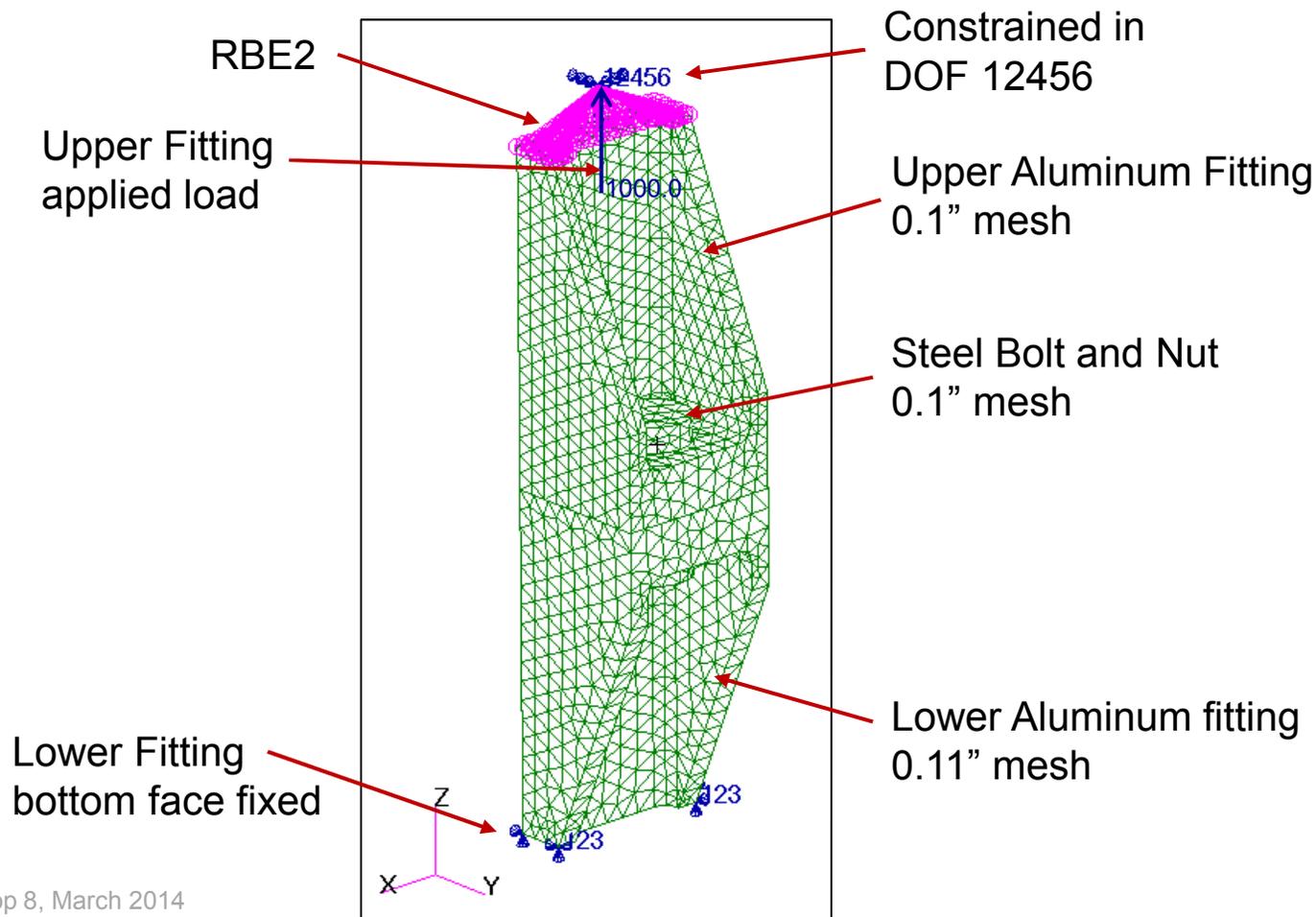
- MSC Nastran 2013.1
- Patran 2013

- **Files Required**

- **bolted_joint.db**

• Problem Description

- Two aluminum fittings are bolted together. The Upper Fitting is constrained to only be able to move in the Z direction and is loaded with 1000 lbs. The Lower Fitting is fixed at the bottom. A steel bolt and nut holds the two fittings together. The bolt is glued to the lower fitting to stabilize the bolt/nut body.



- **Suggested Steps**

- PART 1, SOL 101

1. Open and review starting model.
2. Create three deformable contact bodies using the Contact Body/Pair Tool :
 - **Upper_fitting**
 - **Lower_fitting**
 - **Bolt**
3. Create a glued body pair between Lower_Fitting and Bolt and two touching body pairs between Upper_Fitting and Bolt and Upper_Fitting and Lower_Fitting.
4. Set up one load case, called **loadcase_sol101**, to apply the external load.
5. Set up a SOL 101:
 - Set Solution Type to Linear Static, SOL101
 - Set up Separation Criterion to Stresses and Derivation to Extrapolation
 - Select the Results Output to be XDB
 - Set up Load Increment Parameter to 1 fixed increment
 - Create Output Request for contact results
 - Apply changes to Loadcase_sol101
 - Select Loadcase_sol101 and run the job
6. Review the MSC Nastran input file.
7. Review the MSC Nastran results file.
8. Attach the MSC Nastran results file.
9. Review analysis results, both stresses and deformation.

- **Suggested Steps (Cont.)**

- PART 2, SOL 400

1. Create Bolt Preload:
 - Axial Bolt Load = **2000** lbs.
 - Vector Magnitude = **0.6**
 - Direction Vector = **<0 0 1>**
2. Create SPC for Locking the Bolt Preload:
 - Constrain the Bolt in the Axial Direction
3. Set up three load cases:
 - One to preload the bolt
 - One to lock the bolt in position
 - One to apply the external load
4. Set up a SOL 400. All parameters for the three loadcases are same as SOL101 except the Load Increments as defined below:
 - Preload = **2**
 - Bolt_lock = **1**
 - External_Load = **blank for default**
5. Review the MSC Nastran input file.
6. Review the MSC Nastran results file.
7. Attach MSC Nastran results file.
8. Review analysis results, both the stresses and deformation.

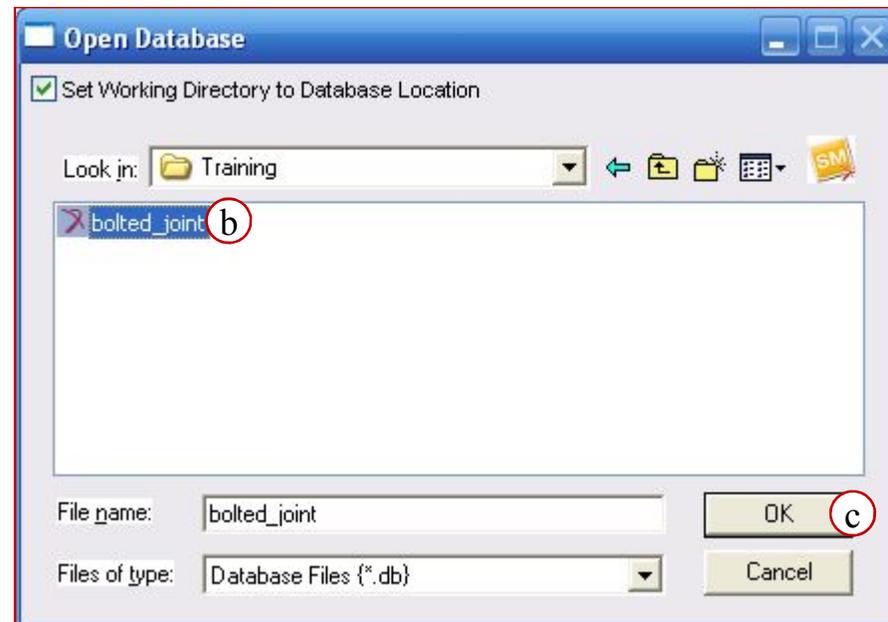
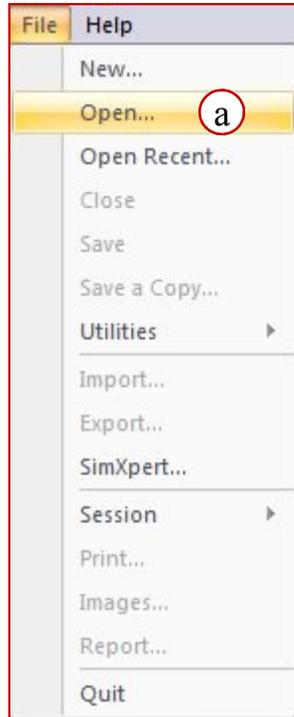
PART 1

SOL 101

Step 1. Open and Review the Starting Model

Open the starting model.

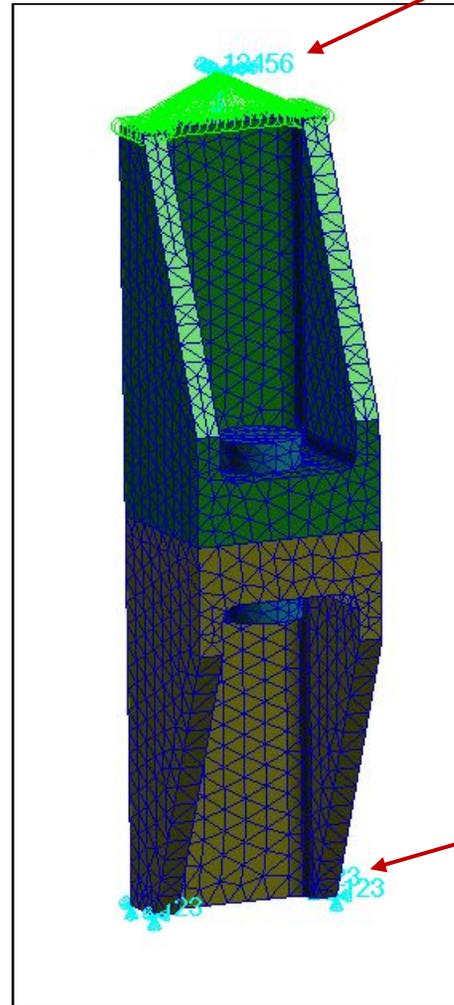
- a. Pull Down **File** > **Open**.
- b. Select **bolted_joint.db**.
- c. Click **OK**.



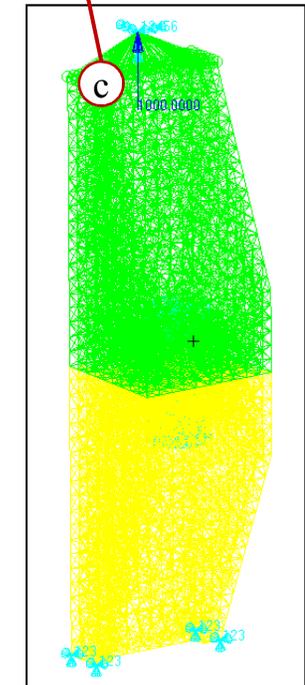
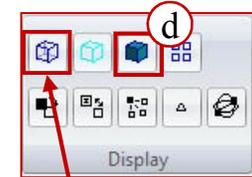
Step 1. Open and Review Starting Model (Cont.)

Review the model.

- a. The Lower Fitting is fixed at the bottom.
- b. The Upper Fitting is constrained in all degrees of freedom except axial translations.
- c. The Upper Fitting has an applied load of 1000.0 lbs. in the Z direction. To see the load value, click the **Wireframe** icon.
- d. Click **Smooth Shaded**.



Constrained (b)
in DOF 12456

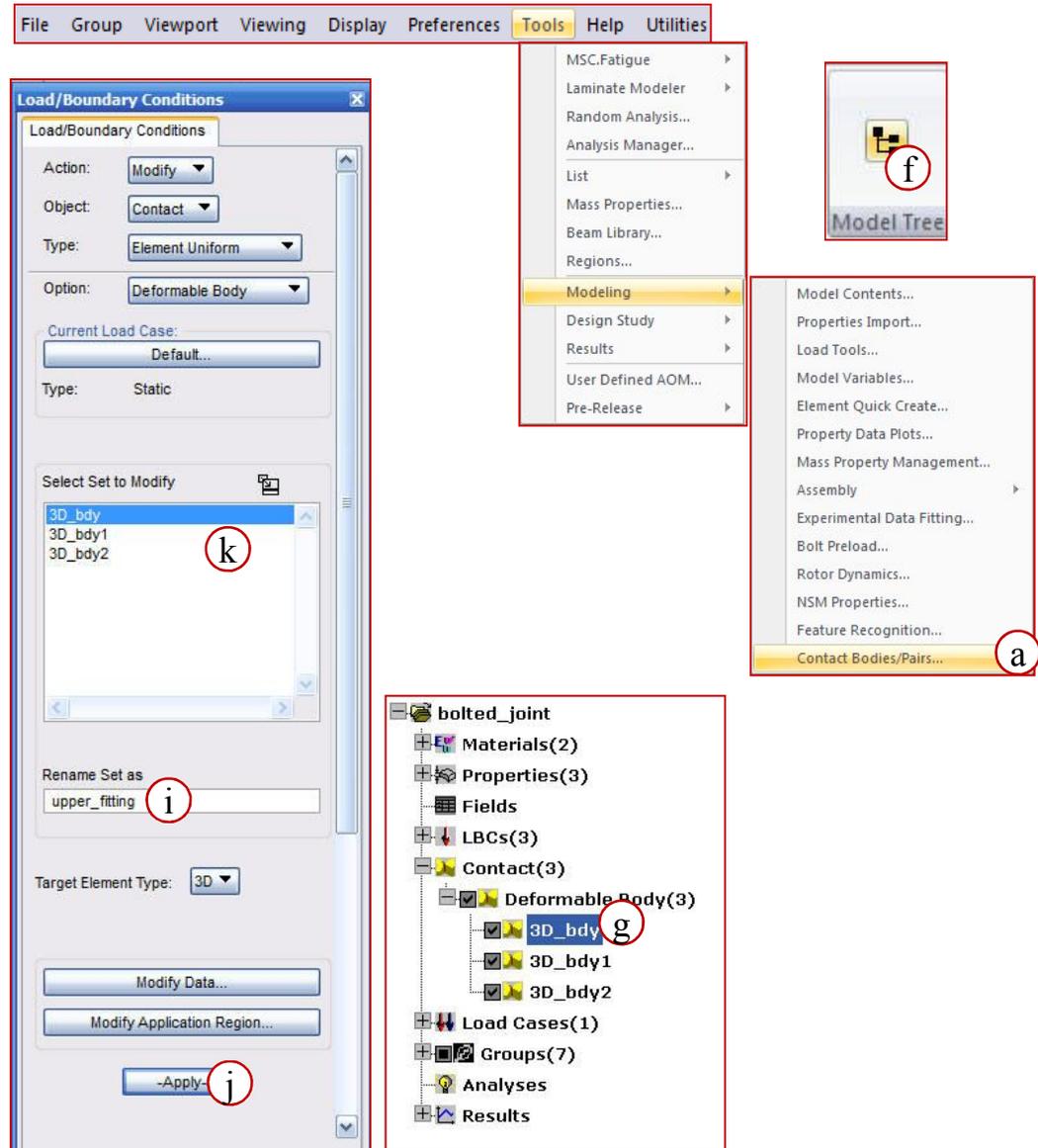
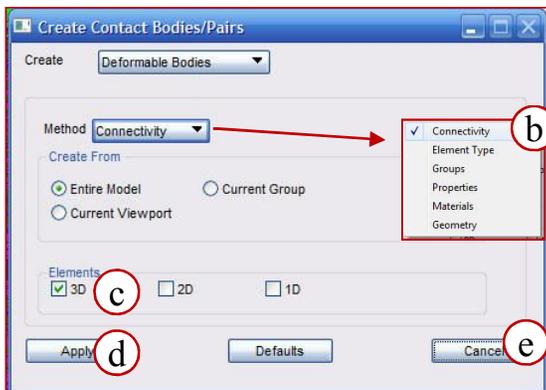


Lower Fitting
fixed bottom (a)

Step 2. Define Three Deformable Contact Bodies

Define Contact Bodies using the Create Contact Body/Pair Tool:

- a. Pull down **Tools** > **Modeling** > **Contact Bodies/Pair**
- b. Pull down *Method* to **Connectivity**
- c. Check **3D** for *Elements*.
- d. Click **Apply**
- e. Click **Cancel**
- f. Turn on the **Model Browser Tree**
- g. Double click on **3D_bdy** under *Contact* > *Deformable Body*
- h. Click **OK** (not shown)
- i. Enter **upper_fitting** for *Rename Set as*.
- j. Click **Apply**
- k. Rename **3D_bdy1** as **lower_fitting** and **3D_bdy2** as **bolt**

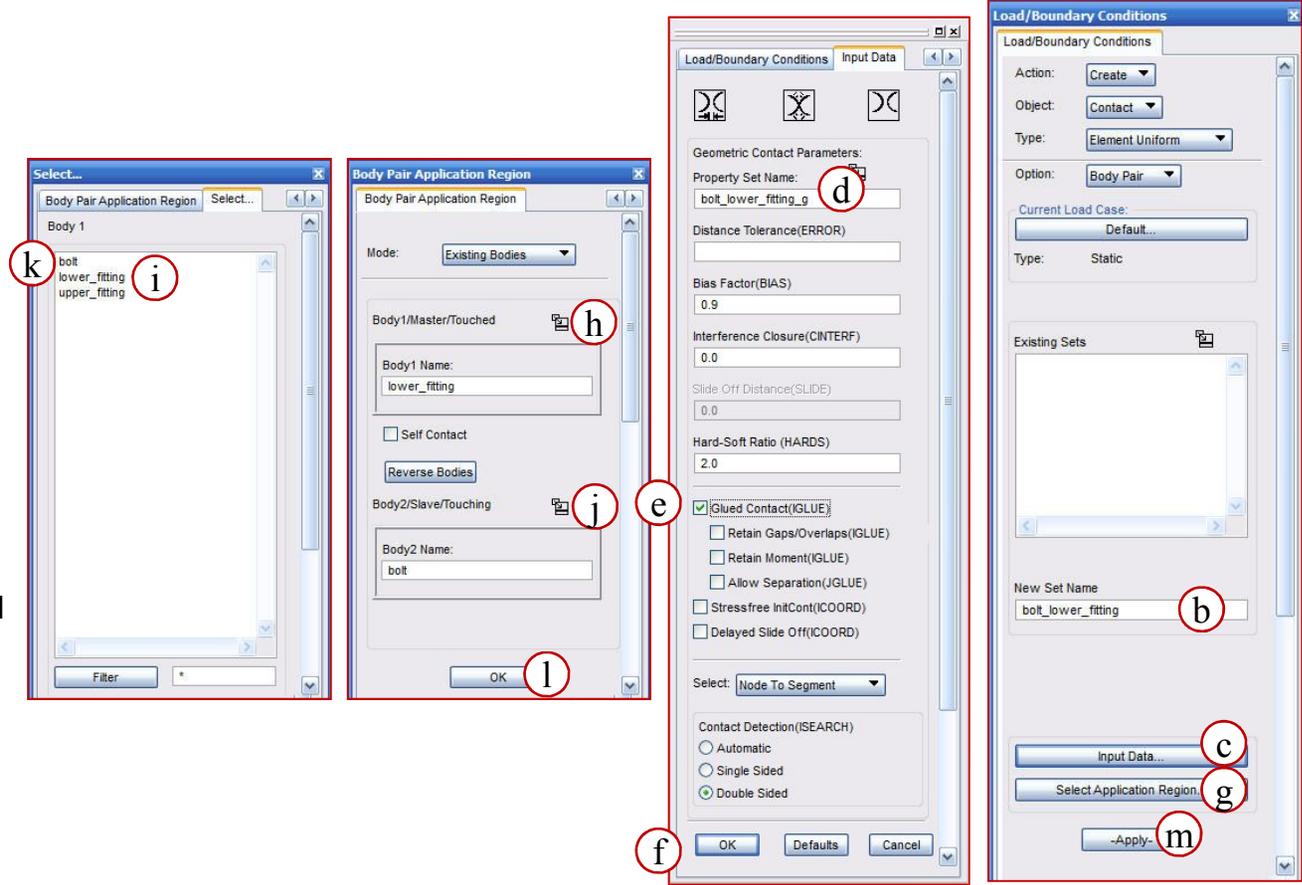


Step 3. Create Body Pairs



Create Three New Body Pairs:

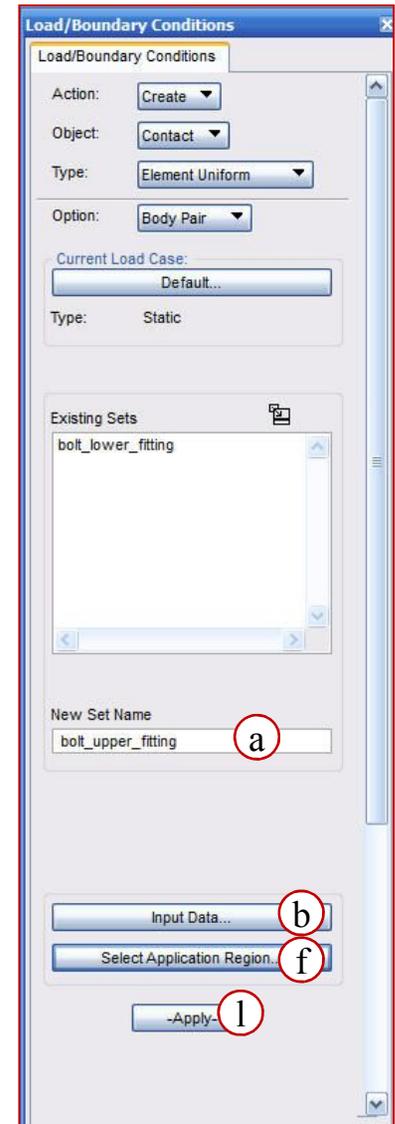
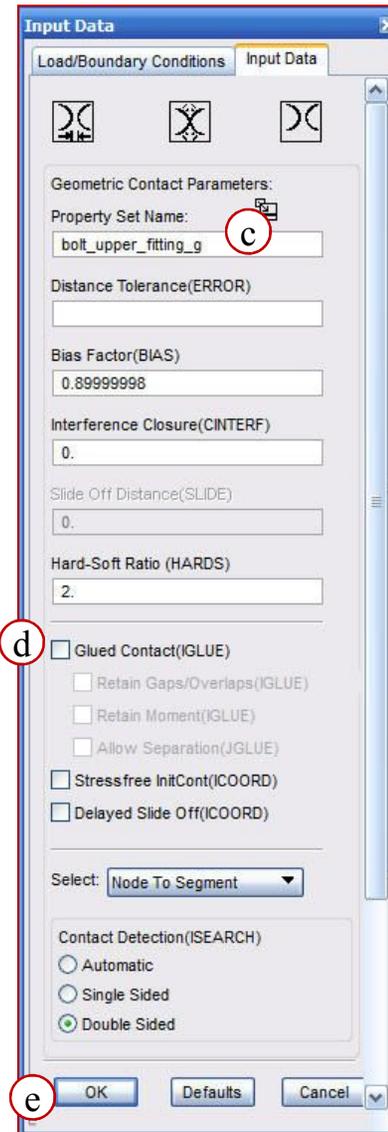
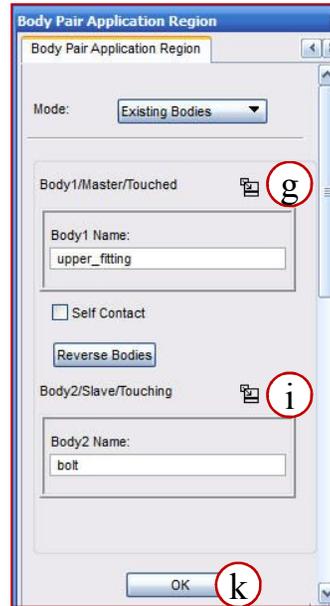
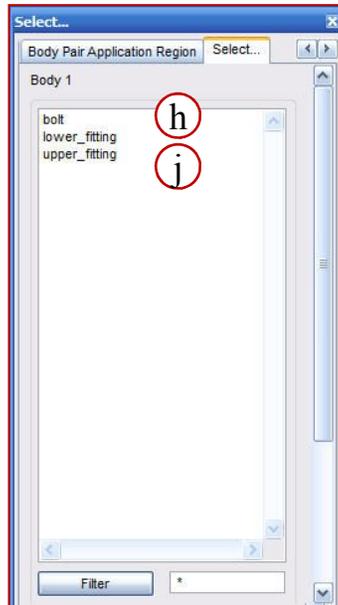
- a. Under the *Load/BCs* tab click **Create Body Pair** in the *Body Pairs* group.
- b. Enter `bolt_lower_fitting` for *New Set Name*.
- c. Click **Input Data...**
- d. Enter `bolt_lower_fitting_g` for Property set name.
- e. Check **Glued Contact(IGLUE)**
- f. Click **OK**
- g. Click **Select Application Region**
- h. Click **Select Existing Touched Body**
- i. Select `lower_fitting`
- j. Click **Select Existing Touching Body**
- k. Select `bolt`
- l. Click **OK**
- m. Click **Apply**



Step 3. Create Body Pairs (Cont.)

Create Three New Body Pairs:

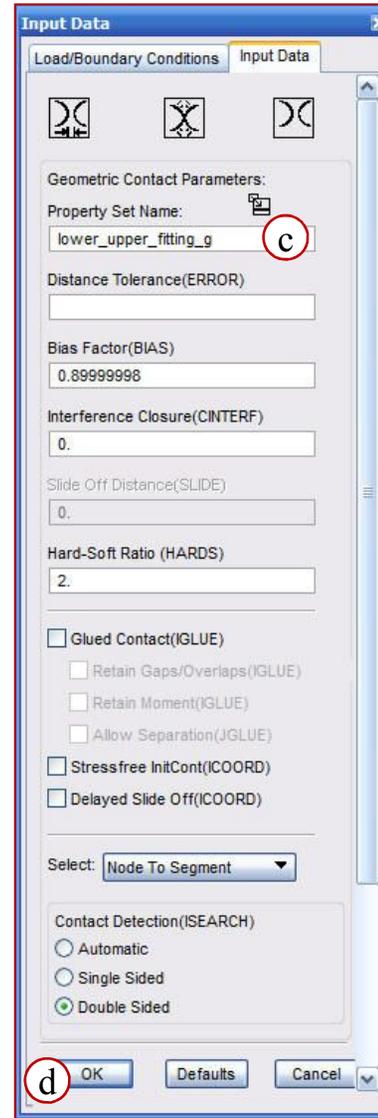
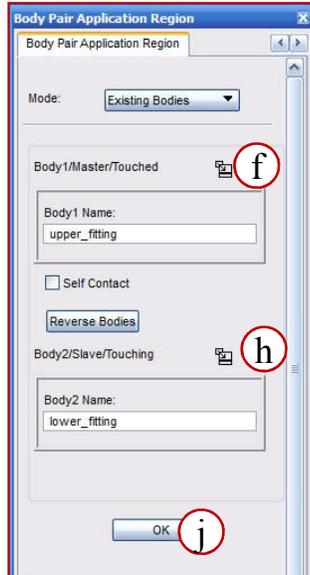
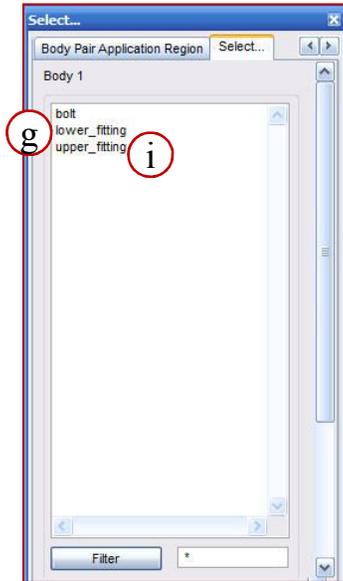
- a. Enter **bolt_upper_fitting** for *New Set Name*.
- b. Click **Input Data...**
- c. Enter **bolt_upper_fitting_g** for Property set name.
- d. Uncheck **Glued Contact(IGLUE)**
- e. Click **OK**
- f. Click **Select Application Region**
- g. Click **Select Existing Touched Body**
- h. Select **upper_fitting**
- i. Click **Select Existing Touching Body**
- j. Select **bolt**
- k. Click **OK**
- l. Click **Apply**



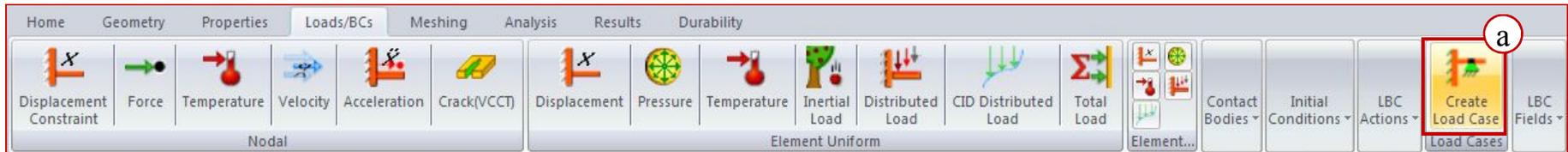
Step 3. Create Body Pairs (Cont.)

Create Three New Body Pairs:

- a. Enter **lower_upper_fitting** for *New Set Name*.
- b. Click **Input Data...**
- c. Enter **lower_upper_fitting_g** for *New Set Name*.
- d. Click **OK**
- e. Click **Select Application Region**
- f. Click **Select Existing Touched Body**
- g. Select **upper_fitting**
- h. Click **Select Existing Touching Body**
- i. Select **lower_fitting**
- j. Click **OK**
- k. Click **Apply**

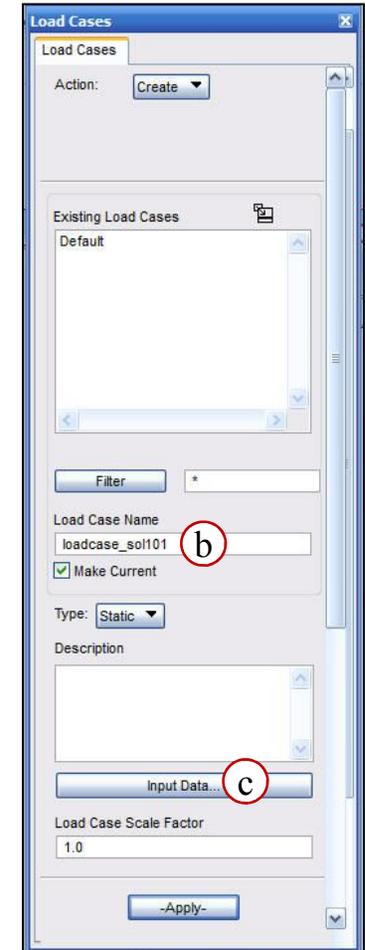
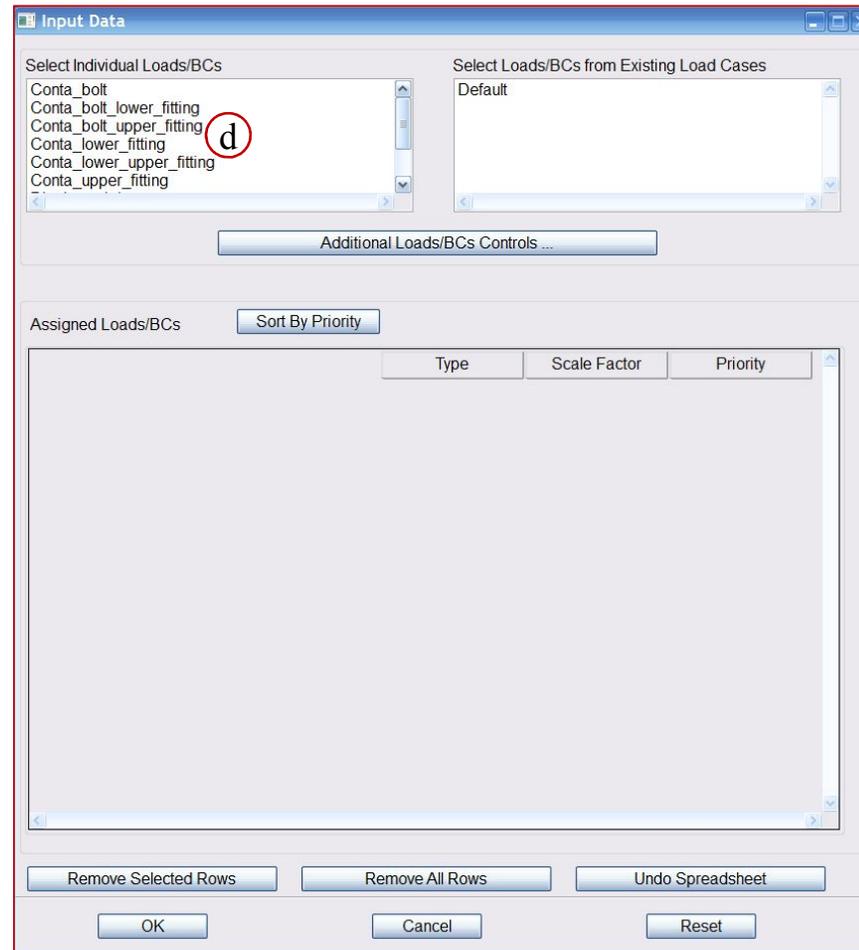


Step 4. Set up Load Case



Set up a second load case to apply the bolt lock.

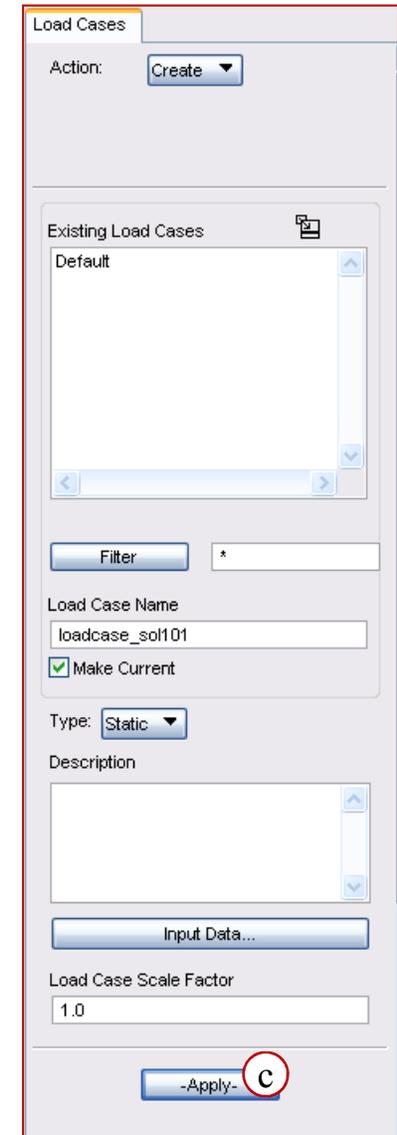
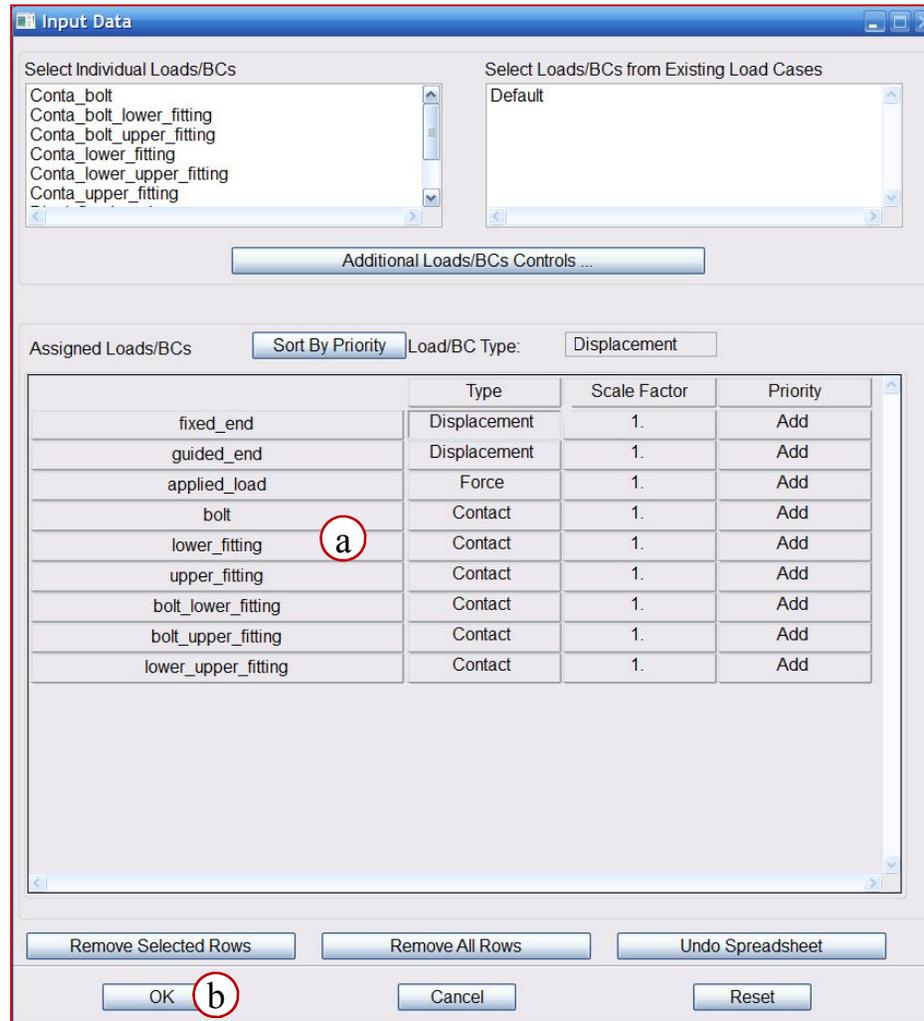
- a. Under the *Loads/BCs* tab, click **Create Load Case** from the *Load Cases* Group.
- b. Enter **loadcase_sol101** as the *Load Case Name*.
- c. Click **Input Data**.
- d. Select all the Loads/BCs list under *Select Individual Loads/BCs*.



Step 4. Set up Load Case (Cont.)

Set up second load case continued.

- The table should now look like the one shown below.
- Click **OK**.
- Click **Apply**.



Step 5. Set up SOL 101 and Run Analysis

The image shows a screenshot of the MSC Nastran software interface. The top ribbon has tabs for Home, Geometry, Properties, Loads/BCs, Meshing, Analysis, Results, and Durability. The Analysis tab is active, and the 'Entire Model' button in the 'Analyze' group is highlighted with a red circle and the letter 'a'. Below the ribbon, the 'Solution Type' dialog box is open, showing the 'Solution Type' tab. The 'LINEAR STATIC' option is selected with a radio button, and this option is also highlighted with a red circle and the letter 'c'. The 'Solution Parameters...' button is highlighted with a red circle and the letter 'd'. To the right of the dialog, the 'Analysis' panel is visible, showing the 'Action' set to 'Analyze', 'Object' set to 'Entire Model', and 'Method' set to 'Full Run'. The 'Code' is 'MSC.Nastran' and the 'Type' is 'Structural'. The 'Job Name' is 'bolted_joint' and the 'Job Description (TITLE)' is empty. The 'SUBTITLE' and 'LABEL' fields are also empty. The 'Solution Type...' button in the 'Available Jobs' list is highlighted with a red circle and the letter 'b'. The 'Apply' button is at the bottom of the panel.

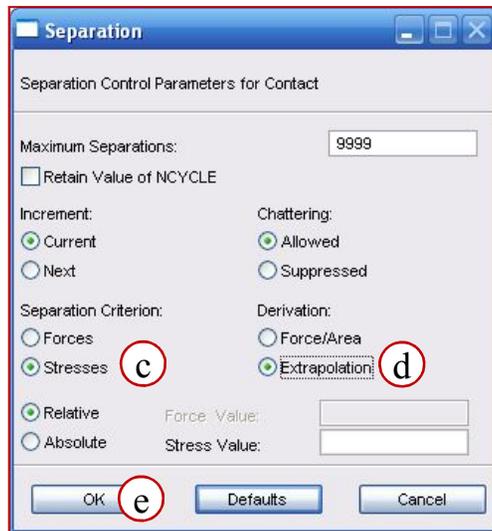
Set the Solution Type to 101, Linear Static.

- Under the *Analysis* tab, click on **Entire Model** in the *Analyze* group.
- Click **Solution Type**.
- Confirm that **LINEAR STATIC** is selected for the *Solution Type*.
- Click **Solution Parameters**.

Step 5. Set up SOL 101 and Run Analysis (Cont.)

Set up Separation Criterion.

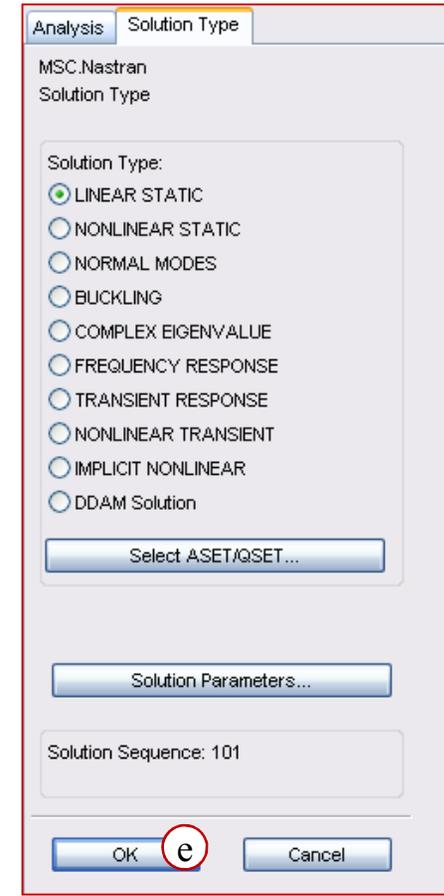
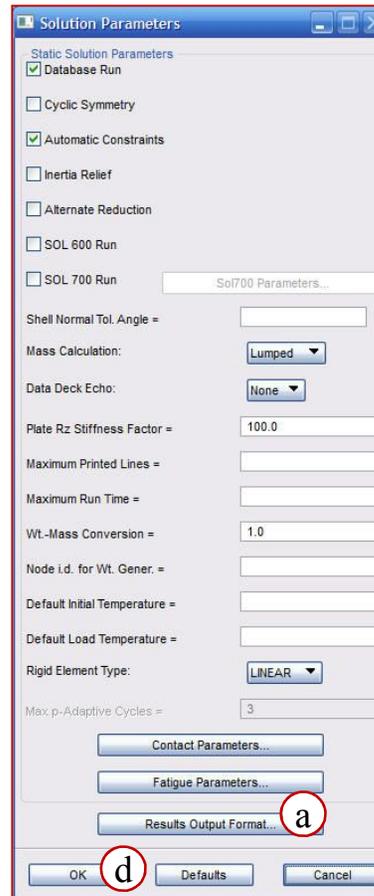
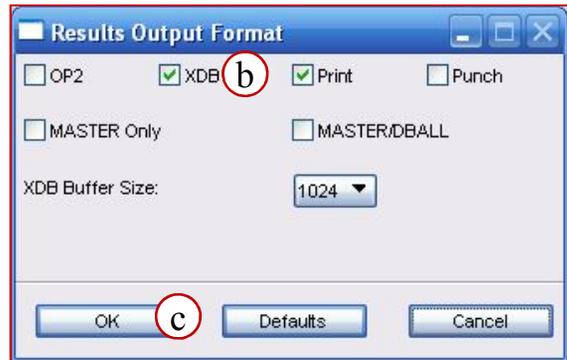
- a. Click **Contact Parameters**.
- b. Select **Separation**.
- c. Select **Stresses** for *Separation Criterion*.
- d. Select **Extrapolation** for *Derivation*.
- e. Click **OK**.
- f. Click **OK**.



Step 5. Set up SOL 101 and Run Analysis (Cont.)

Select the Results Output Format

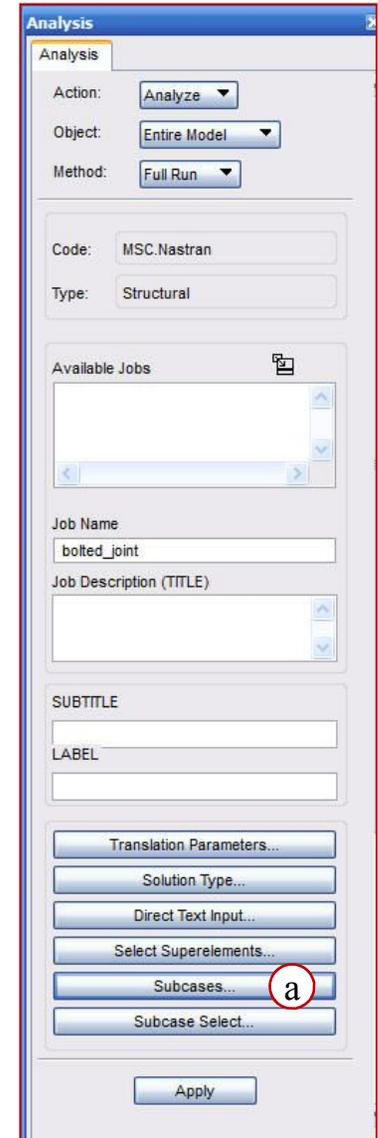
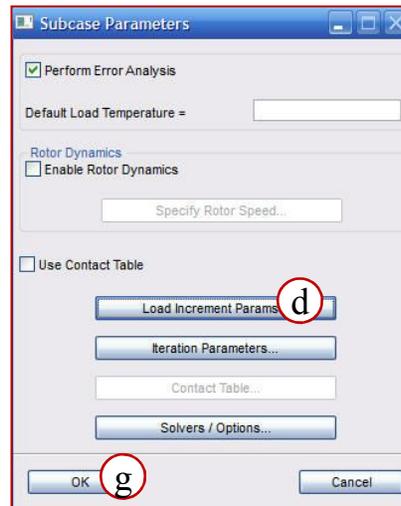
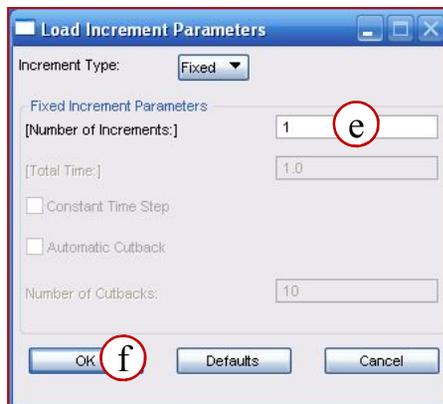
- Select **Results Output Format**.
- Check **XDB**.
- Click **OK**.
- Click **OK**.
- Click **OK**.



Step 5. Set up SOL 101 and Run Analysis (Cont.)

Set up Load Increment

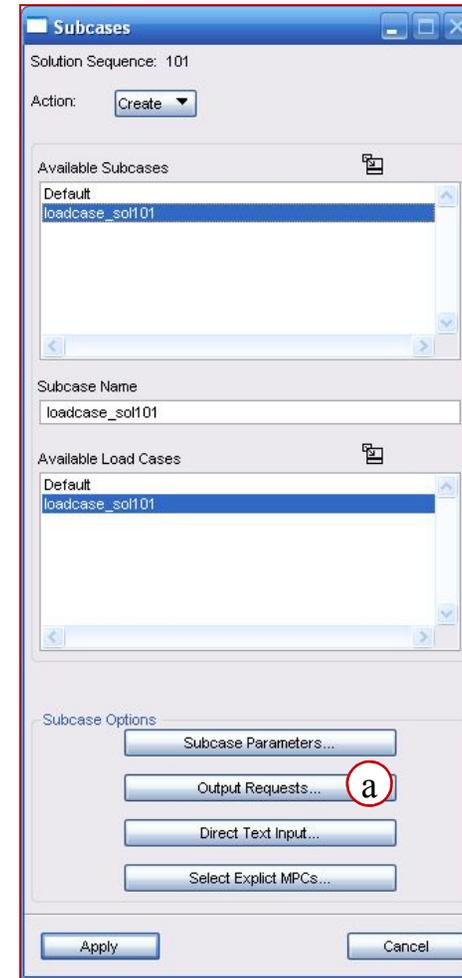
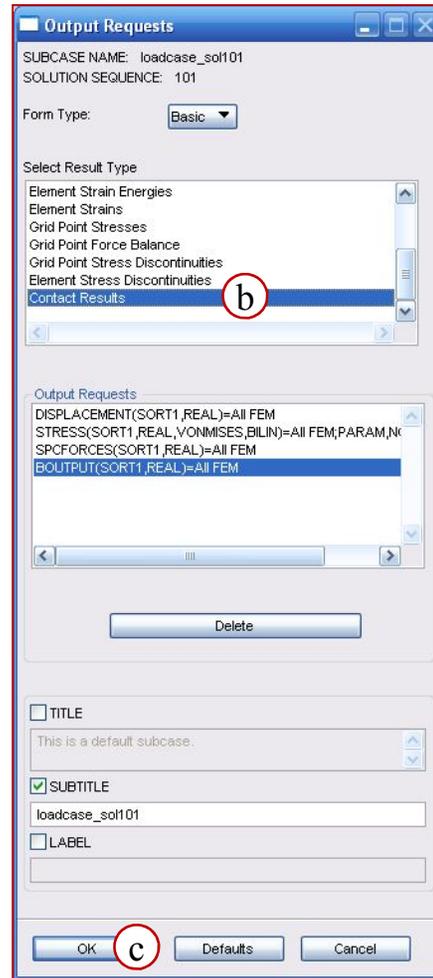
- a. Click **Subcases**.
- b. Click on **loadcase_sol101** under *Available Subcases*.
- c. Click **Subcase Parameters**.
- d. Select **Load Increment Params**.
- e. Enter **1** for *Number of Increments*.
- f. Click **OK**.
- g. Click **OK**



Step 5. Set up SOL 101 and Run Analysis (Cont.)

Create Output Request for contact results.

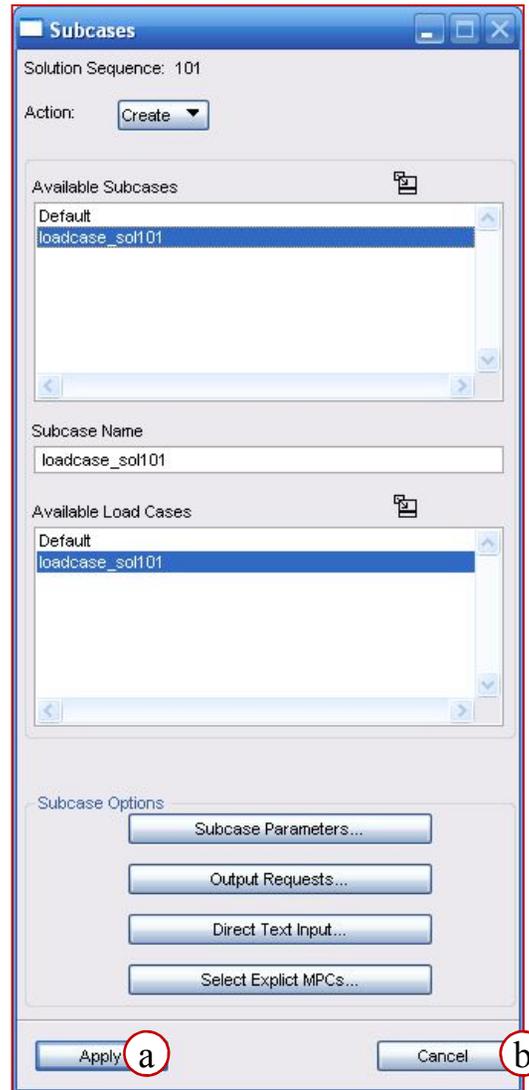
- a. Click **Output Requests**.
- b. Select **Contact Results**.
- c. Click **OK**.



Step 5. Set up SOL 101 and Run Analysis (Cont.)

Apply changes to the subcase.

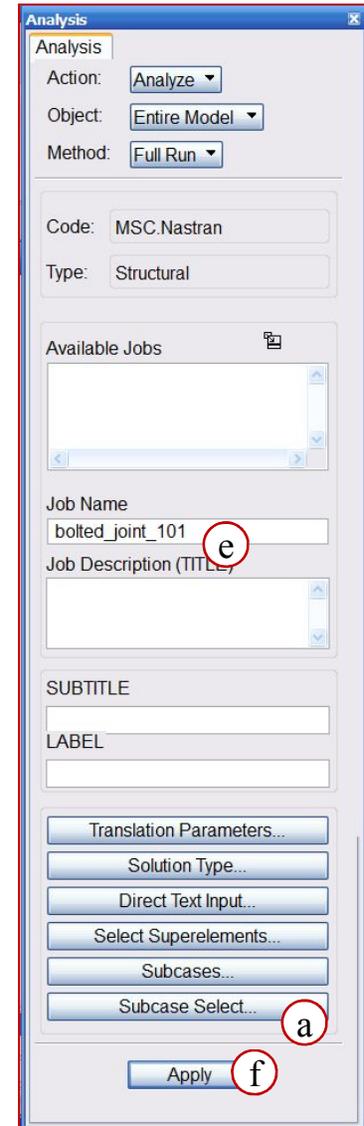
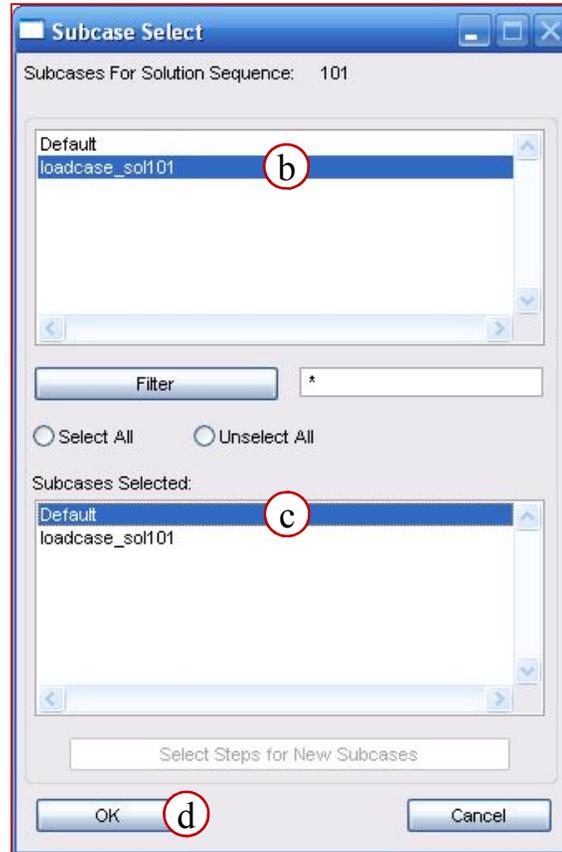
- a. Click **Apply**.
- b. Click **Cancel**.



Step 5. Set up SOL 101 and Run Analysis (Cont.)

Select the subcase and submit the analysis job.

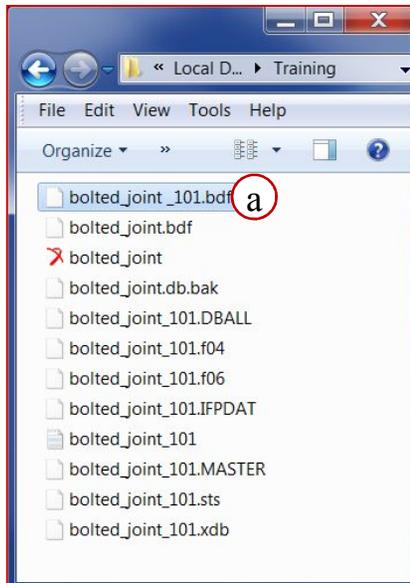
- a. Click **Subcase Select**.
- b. Select **loadcase_sol101**.
- c. Select **Default** from the *Subcases Select* to remove it from the list.
- d. Click **OK**.
- e. Set the *Job Name* to **bolted_joint_101**.
- f. Click **Apply** to submit the job.



Step 6. Review Input File

While the job is running, review the MSC Nastran input file.

- a. Locate the file **bolted_joint_101.bdf**.
- b. Open **bolted_joint_101.bdf** with any text editor.



```
SOL 101
CEND
$ Direct Text Input for Global Case Control Data
ECHO = NONE
  BCONTACT = 0
SUBCASE 1
  SUBTITLE=loadcase_sol101
  NLPARM = 1
  BCONTACT = 1
  SPC = 2
  LOAD = 2
  DISPLACEMENT (SORT1, REAL)=ALL
  SPCFORCES (SORT1, REAL)=ALL
  STRESS (SORT1, REAL, VONMISES, BILIN)=ALL
  BOUTPUT (SORT1, REAL)=ALL
$ Direct Text Input for this Subcase
BEGIN BULK
$ Direct Text Input for Bulk Data
PARAM      POST      0
PARAM      PRTMAXIM YES
BCPARA     0          NLGLUE  0          IBSEP   4
NLPARM     1          1
BCTABL1    0          8007      8008      8009
BCONNECT   8007      3004          1          2
BCONNECT   8008          1          3
BCONNECT   8009          2          3
BCONPRG    3004          IGLUE  1
BCTABL1    1          8007      8008      8009
```

Step 6. Review Input File (Cont.)

Review the MSC Nastran input file continued.

- a. Check the BCPARA Entry.
- b. Check Subcase 1.
- c. Close the input file.

```
PARAM      POST      0
PARAM      PRTMAXIM  YES
a BCPARA    0          NLGLUE  0          IBSEP  4
```

Relative stress based separation criterion

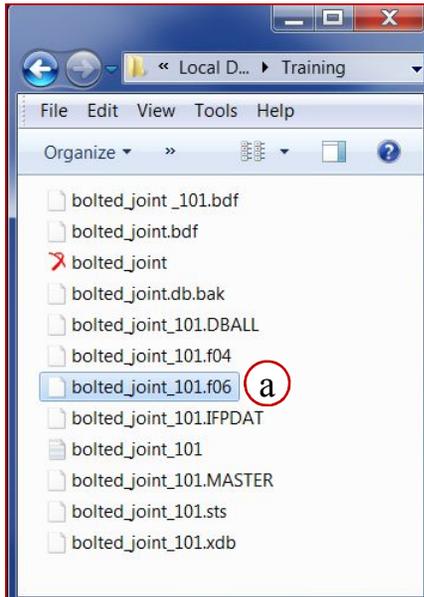
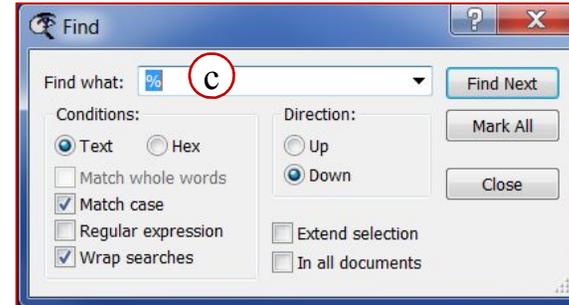
```

SUBCASE 1 b
  SUBTITLE=loadcase_sol101
  NLPARM = 1
  BCONTACT = 1 ← Contact
  SPC = 2 ← Fixed Conditions
  LOAD = 2 ← Applied Load
  DISPLACEMENT (SORT1, REAL) =ALL
  SPCFORCES (SORT1, REAL) =ALL
  STRESS (SORT1, REAL, VONMISES, BILIN) =ALL
  BOUTPUT (SORT1, REAL) =ALL
  $ Direct Text Input for this Subcase
  BEGIN BULK
  $ Direct Text Input for Bulk Data
```

Step 7. Review Results File

After completion of the run, review the .f06 results file.

- Locate the file **bolted_joint_101.f06**.
- Open **bolted_joint_101.f06** with any text editor.
- Search for “%” and scan the **.f06** file.
- Close the **.f06** file.

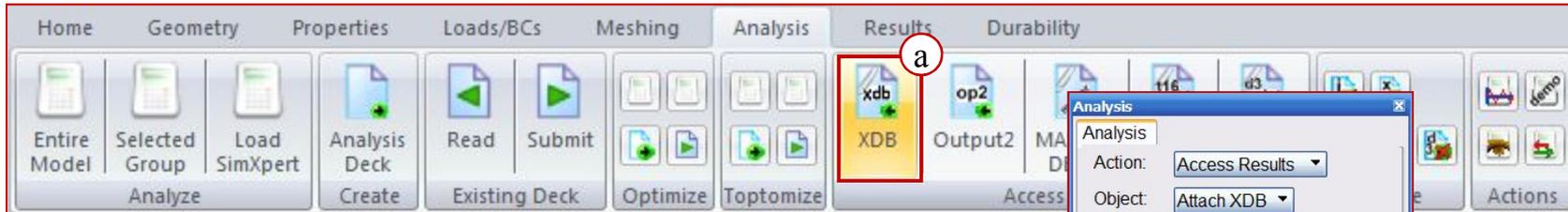


b

```

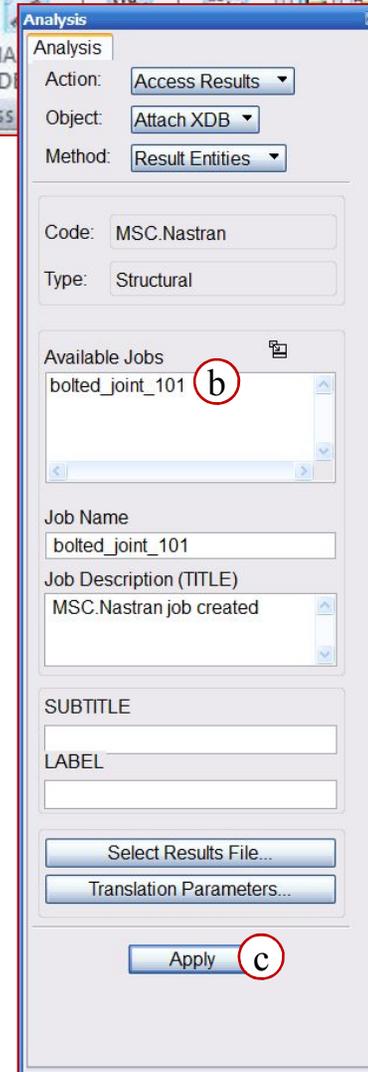
0
0
NON - LINEAR ITERATION MODULE OUTPUT SUBCASE 1
STIFFNESS UPDATE TIME-1892160000.00 SECONDS SUBCASE 1 STEP
0
ITERATION TIME 3.67 SECONDS
LINE_S LOAD NO. -- ERROR FACTORS -- CONV ITR MAT NO. AVG TOTL -- -- DISP -- -- --
STEP INC ITR DISP LOAD WORK RATE DIV DIV BIS R_FORCE WORK AVG MAX AT GRID C FACT
NO QNV KUD ITR
*1.00000E+00 1 1 1.00E+00 1.13E-10 1.00E+00 1.000 0 1 0 7.86E-09 7.562E-01 1.84E-04 1.512E-03 10794 3 1.00
0 0 0 1
*** SYSTEM INFORMATION MESSAGE 8137 (NL3CON)
SEPARATION condition has been detected. Additional iterations will be performed.
*1.00000E+00 1 2 1.41E-03 4.67E-13 3.98E-07 0.004 0 1 0 1.22E-09 7.562E-01 1.84E-04 1.512E-03 10794 3 1.00
0 0 0 2
*** SYSTEM INFORMATION MESSAGE 8137 (NL3CON)
SEPARATION condition has been detected. Additional iterations will be performed.
*1.00000E+00 1 3 8.27E-01 2.49E-11 6.69E-01 0.225 1 1 0 3.20E-08 3.815E+00 6.76E-04 -5.547E-03 11568 2 1.00
0 0 0 3
*** SYSTEM INFORMATION MESSAGE 8137 (NL3CON)
SEPARATION condition has been detected. Additional iterations will be performed.
*1.00000E+00 1 4 2.72E-01 1.30E-12 2.35E-01 0.603 0 1 0 1.87E-07 5.220E+00 8.79E-04 -7.182E-03 11568 2 1.00
0 0 0 4
*** SYSTEM INFORMATION MESSAGE 8137 (NL3CON)
SEPARATION condition has been detected. Additional iterations will be performed.
*1.00000E+00 1 5 8.77E-02 4.45E-13 4.50E-02 0.067 1 1 0 5.24E-06 5.501E+00 9.15E-04 -7.706E-03 11568 2 1.00
0 0 0 5
*** SYSTEM INFORMATION MESSAGE 8137 (NL3CON)
SEPARATION condition has been detected. Additional iterations will be performed.
*1.00000E+00 1 6 2.53E-02 5.68E-13 5.26E-03 0.558 0 1 0 4.08E-07 5.534E+00 9.19E-04 -7.707E-03 11568 2 1.00
0 0 0 6
*1.00000E+00 1 7 6.31E-03 4.67E-13 1.88E-04 0.602 0 1 0 3.36E-09 5.535E+00 9.19E-04 -7.707E-03 11568 2 1.00
0 0 0 7
    
```

Step 8. Attach Results File

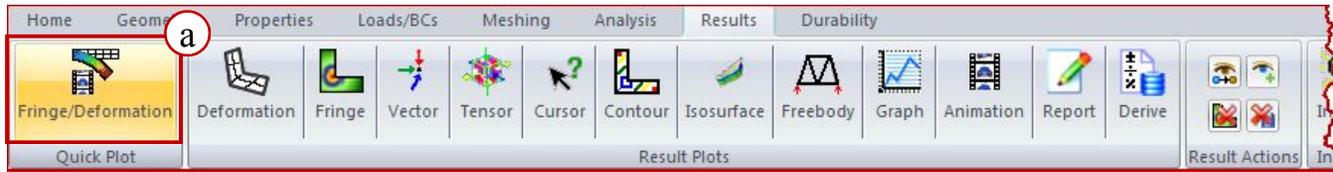


Post process analysis results in Patran.

- a. Under the *Analysis* tab, click **XDB** from the *Access Results* Group.
- b. Select **bolted_joint_101** from the *Available Jobs*.
- c. Click **Apply**.

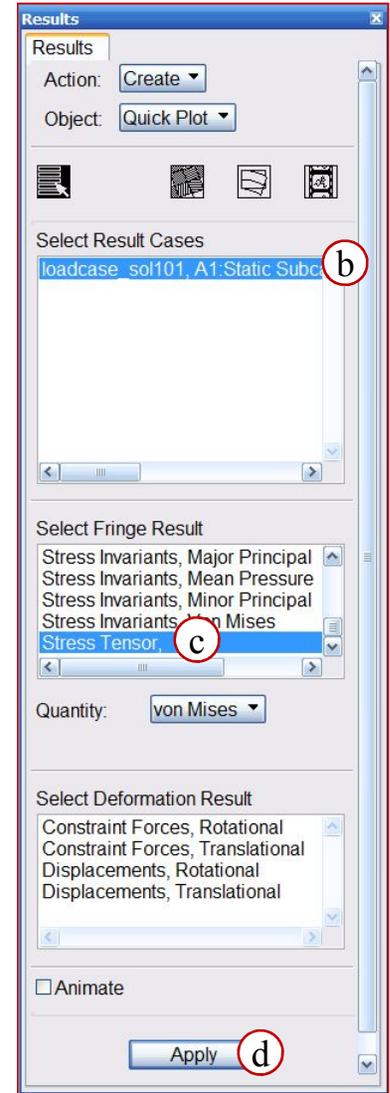


Step 9. Review Analysis Results

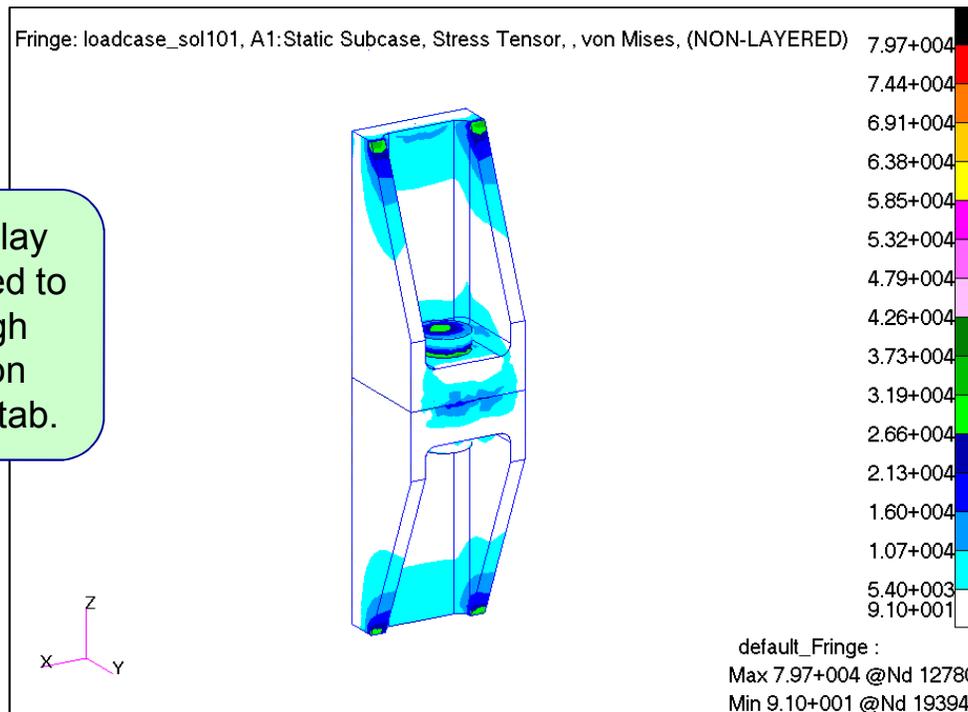


Examine von Mises stresses.

- Under the *Results* tab, click **Fringe/Deformation** from the *Quick Plot* group.
- Select **loadcase_sol101** from *Select Result Cases*.
- Select **Stress Tensor** for *Fringe Result*.
- Click **Apply**.



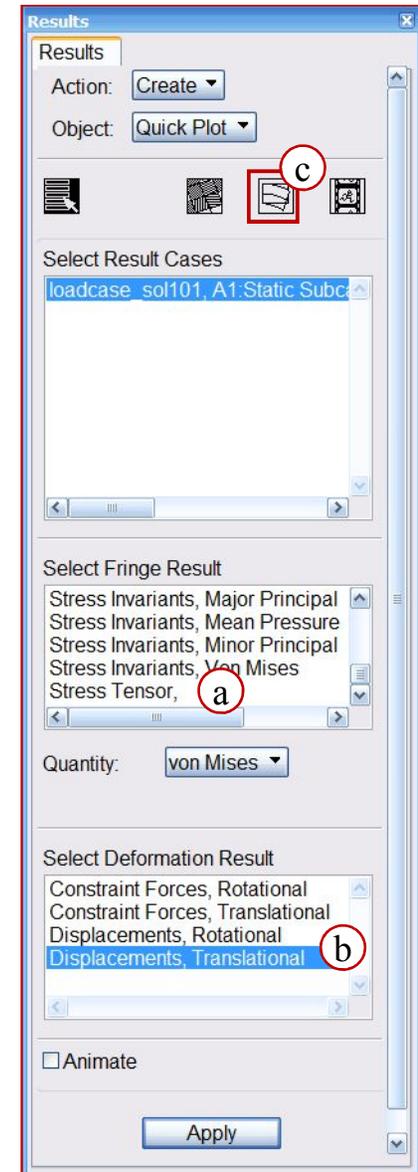
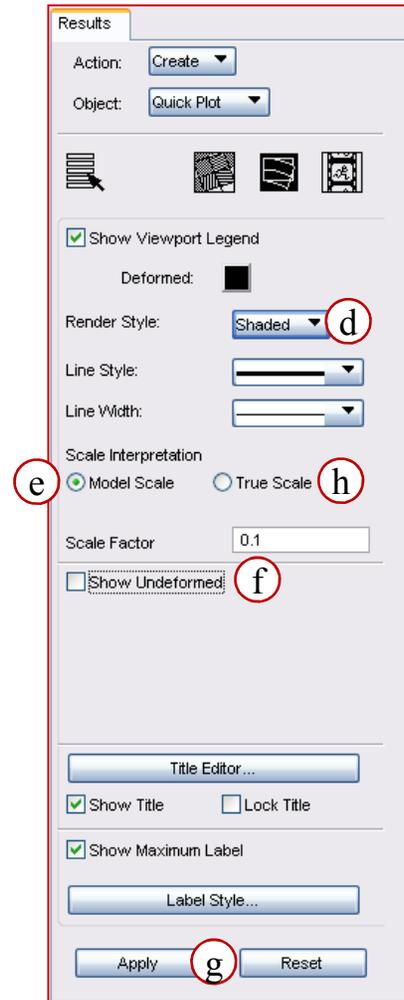
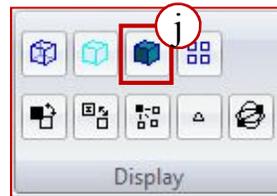
The model's display has been changed to *Wireframe* through the **Display** option under the *Home* tab.



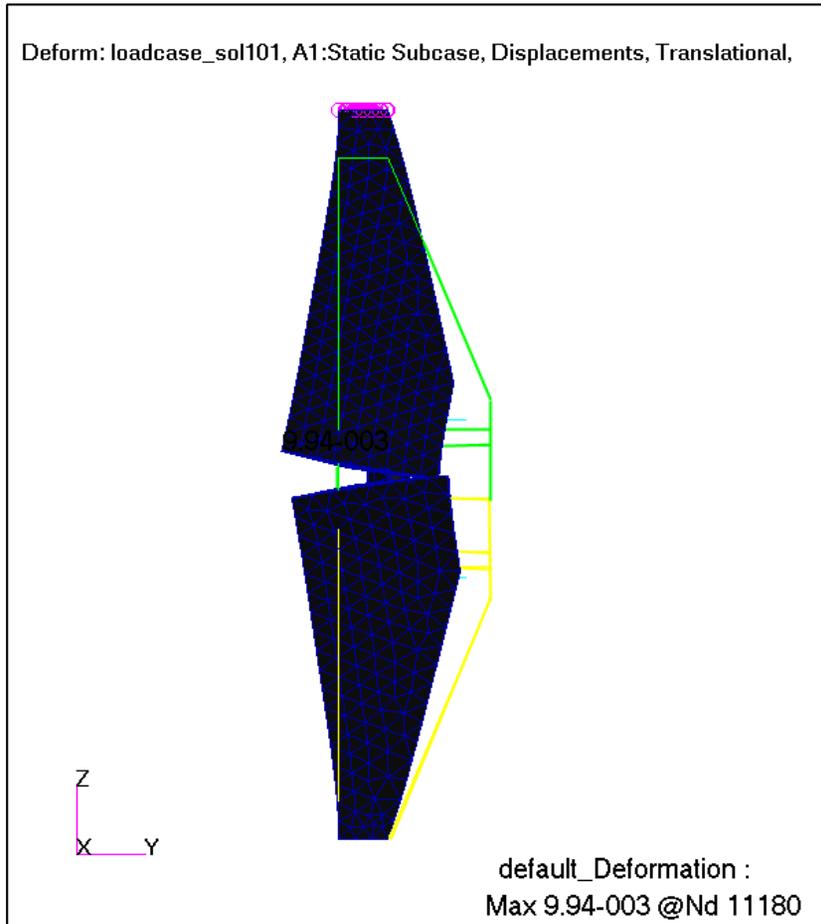
Step 9. Review Analysis Results (Cont.)

Examine deformation.

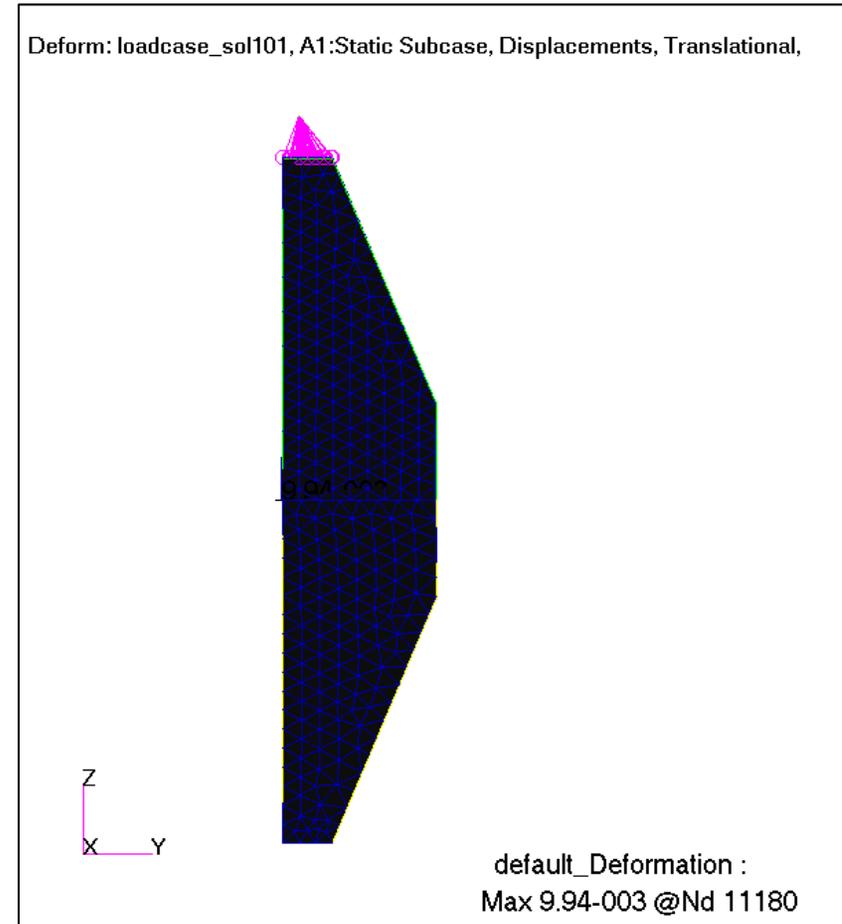
- a. Unselect **Stress Tensor** by double-clicking on it.
- b. Select **Displacement, Translational** for deformation result.
- c. Click the **Deform Attributes** icon.
- d. Select **Shaded** for the *Render Style*.
- e. Make sure that **Model Scale** is selected.
- f. Uncheck **Show Undeformed**.
- g. Click **Apply**.
- h. Plot again, but this time select **True Scale** to see the difference between the two. Both plots are shown on the next page.
- i. Click **Smooth Shaded**.



Step 9. Review Analysis Results (Cont.)



Model Scale



True Scale

PART 2

SOL 400

Step 1. Create Bolt Preload

The image shows a software interface with a menu bar (File, Group, Viewport, Viewing, Display, Preferences, Tools, Help, Utilities) and a toolbar. A 'Group' menu is open, showing options like 'Create...', 'Post...', 'Modify...', etc. A red circle 'a' is around 'Post...'. In the toolbar, a 'Fit View' icon is circled in red with 'e'. A 3D model of a bolt is shown in the center. On the right, a 'Load/Boundary Conditions' dialog box is open, showing 'Action: Post', 'Current Viewport: default_viewport', and a list of 'Select Groups to Post' including 'Bolt', 'Lower_Fitting', 'Upper_Fitting', and 'default_group'. A red circle 'b' is around 'Bolt'. At the bottom of the dialog, 'Apply' is circled in red with 'c' and 'Cancel' is circled in red with 'd'.

First show only the bolt.

- a. Pull down **Group** > **Post**.
- b. Select **Bolt** from the *Select Groups to Post* list.
- c. Click **Apply**.
- d. Click **Cancel**.
- e. Click the **Fit View** icon.

Step 1. Create Bolt Preload (Cont.)

Create bolt preload.

- Pull Down **Tools > Modeling > Bolt Preload...**
- Verify that *MPC Type* is set to **Overclosure**.
- Enter *Axial Bolt Load* of **2000**.
- For *Control Node Offset*, enter **<0 0 1>** for the *Direction Vector* and **0.6** for the *Vector Magnitude*.
- Click in the **Element List** box.
- Rectangular pick all the bolt elements and watch the bolt preload being created.
- Click **Cancel**.

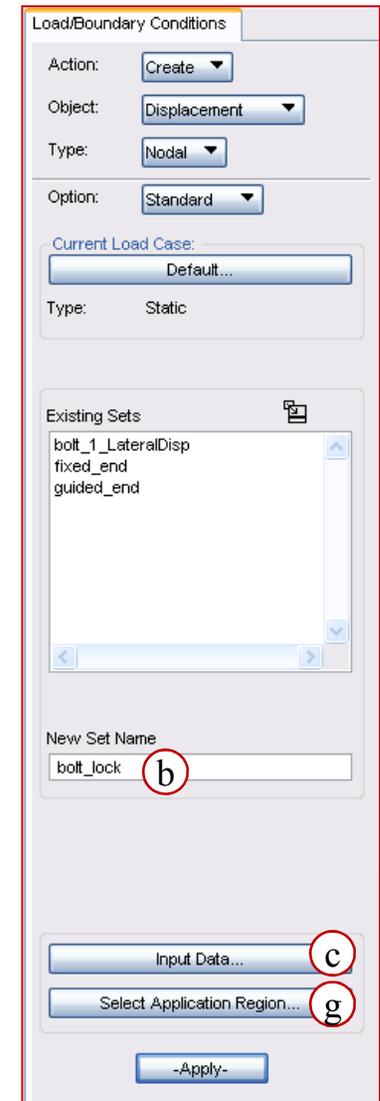
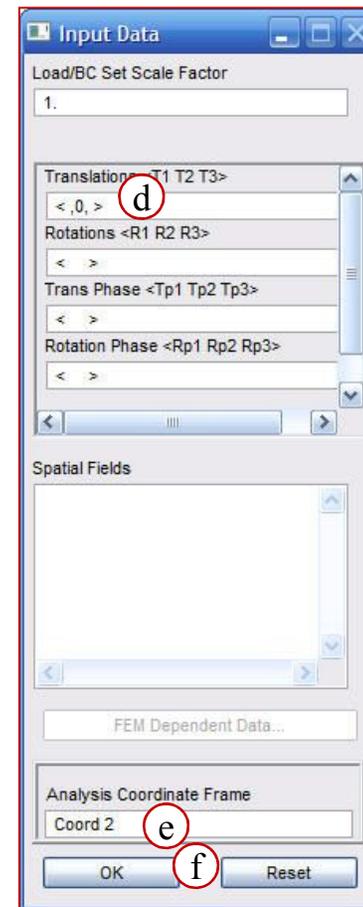
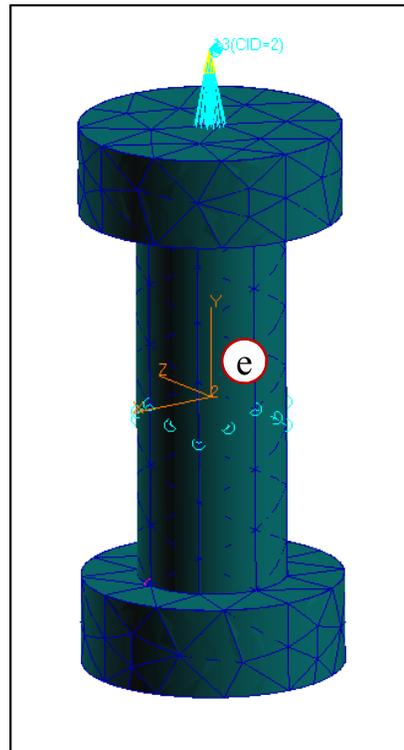
Watch out. The **Auto Execute** box is checked. So don't click on **Apply**.

Step 2. Create SPC to Lock Bolt Preload



Create an SPC to lock the bolt in the second load step.

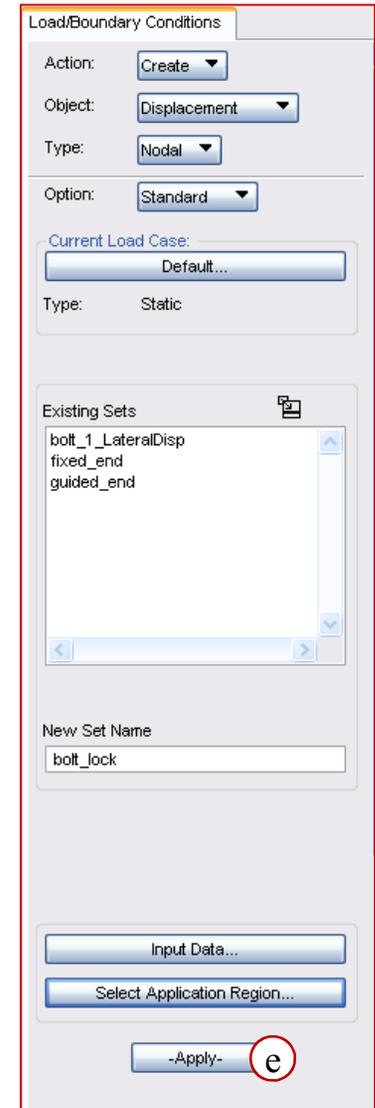
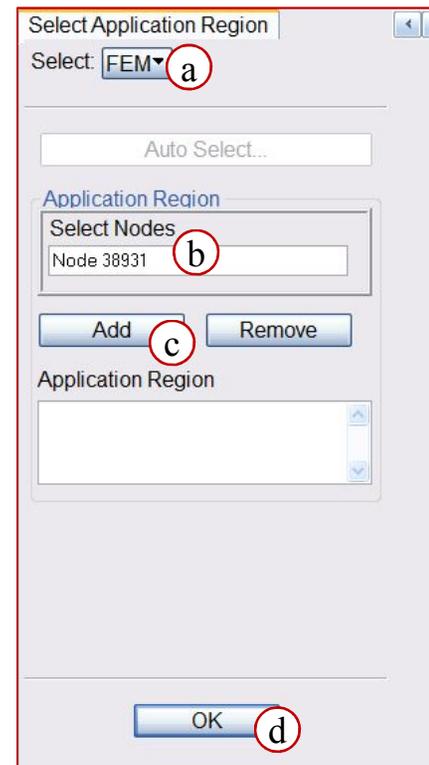
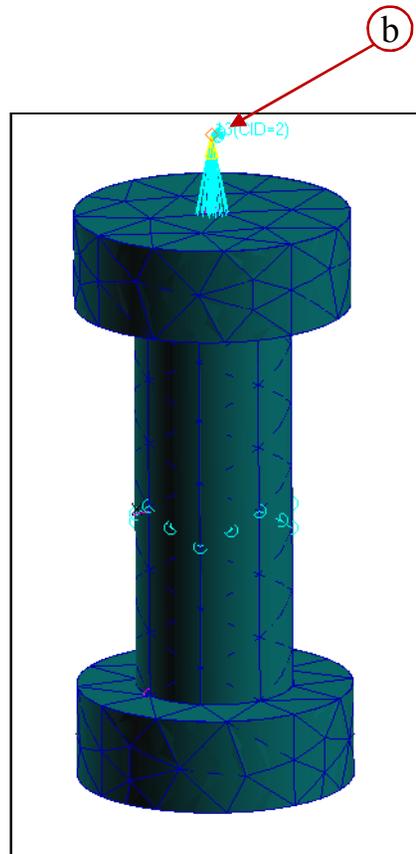
- a. Under the *Loads/BCs* tab, click on **Displacement Constraint** in the *Nodal* group.
- b. Enter **bolt_lock** as name.
- c. Click **Input Data...**
- d. Enter **< ,0, >** to constrain the Y translation in the local bolt coordinate system.
- e. Click in the *Analysis Coordinate Frame* box, then screen pick **Coord 2**, the coordinate frame in the bolt.
- f. Click **OK**.
- g. Click **Select Application Region**.



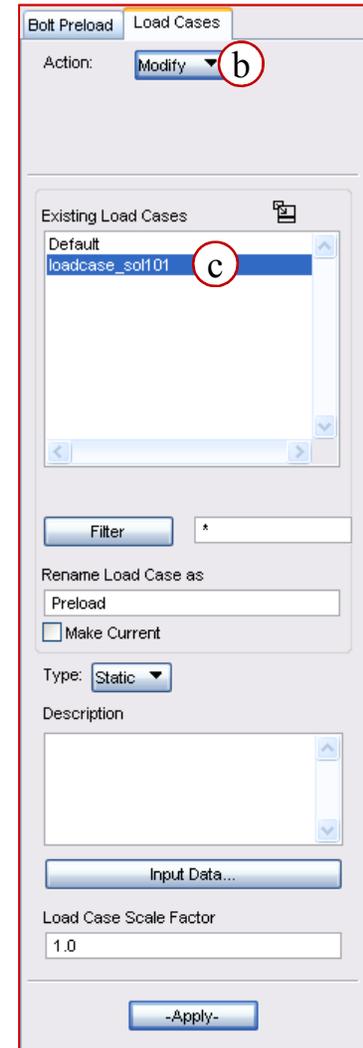
Step 2. Create SPC to Lock Bolt Preload (Cont.)

Create an SPC to lock the bolt in the second load step continued.

- a. Set *Select* to **FEM**.
- b. Screen pick the bolt preload control node.
- c. Click **Add**.
- d. Click **OK**.
- e. Click **Apply**.

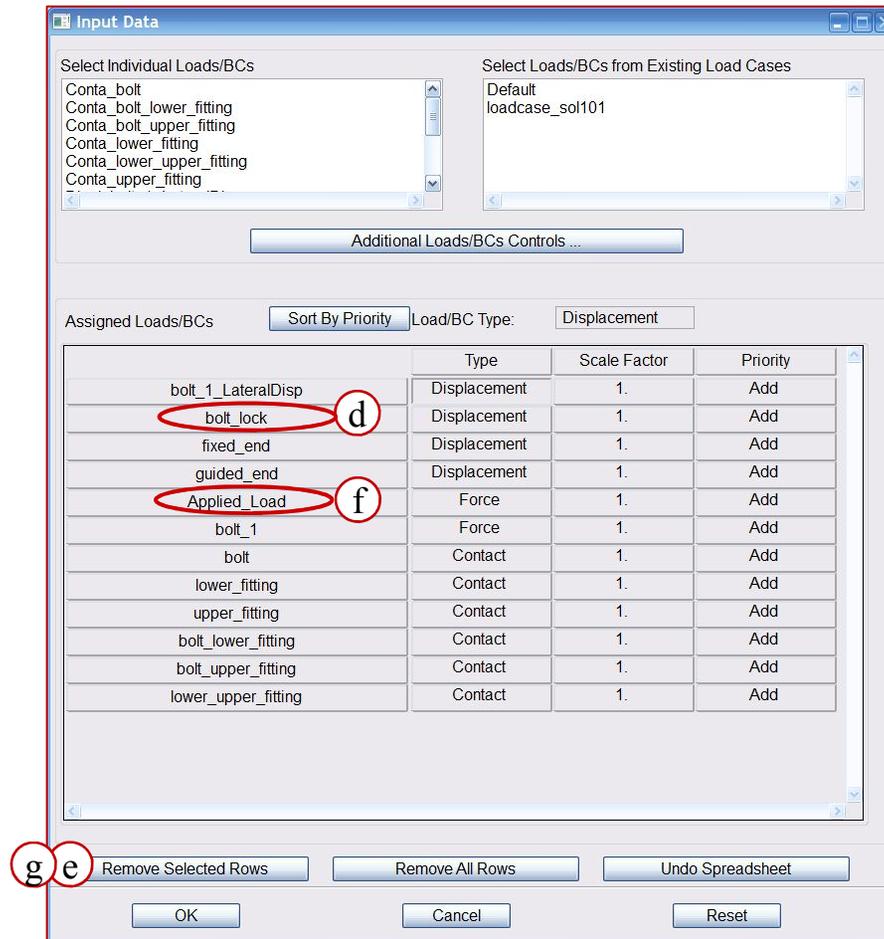


Step 3. Set Up Three Load Cases



Set up a load case to preload the bolt

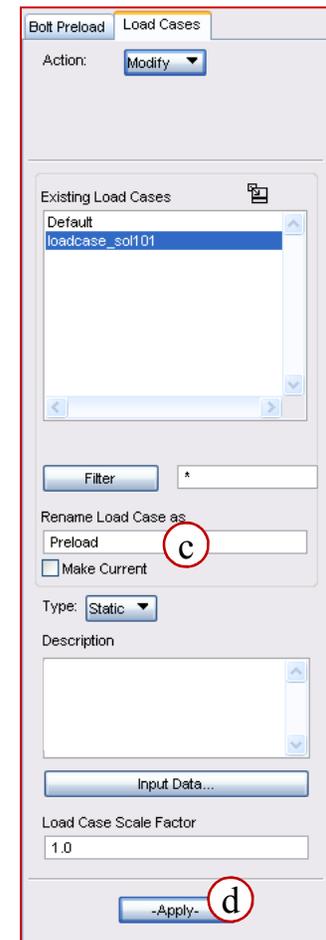
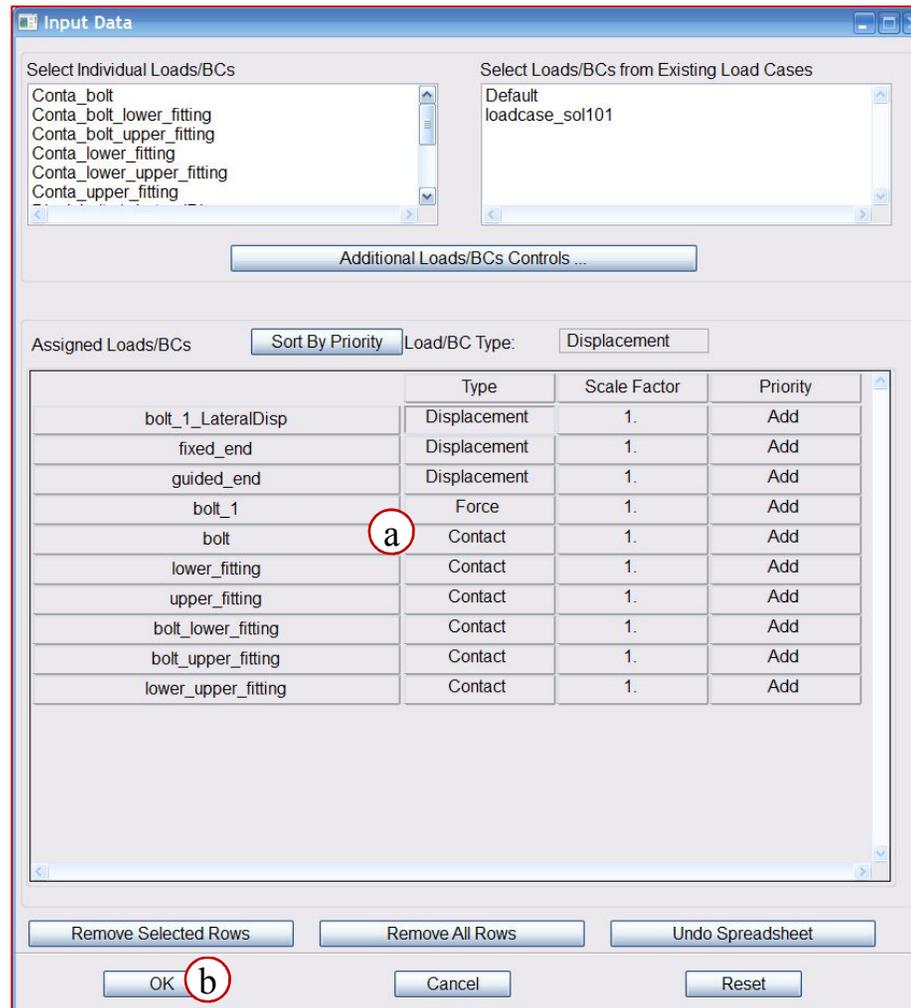
- a. Under the *Loads/BCs* tab, click **Create Load Case** from the *Load Cases* Group.
- b. Set *Action* to **Modify**.
- c. Select **loadcase_sol101** from the *Existing Load Cases* list
- d. Click **bolt_lock** to select this row.
- e. Click **Remove Selected Rows**.
- f. Click **Applied_Load** to select this row.
- g. Click **Remove Selected Rows**.



Step 3. Set Up Three Load Cases (Cont.)

- a. The table should look like the one shown on the right.
- b. Click **OK**.
- c. Enter **Preload** in the *Rename Loadcase as* textbox.
- d. Click **Apply**.

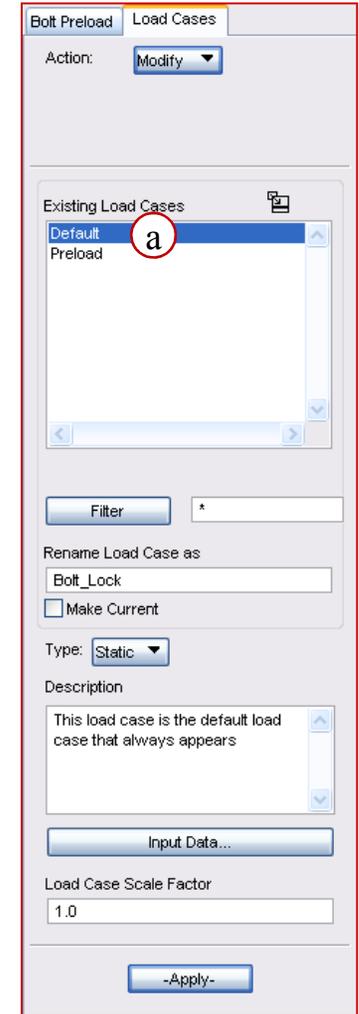
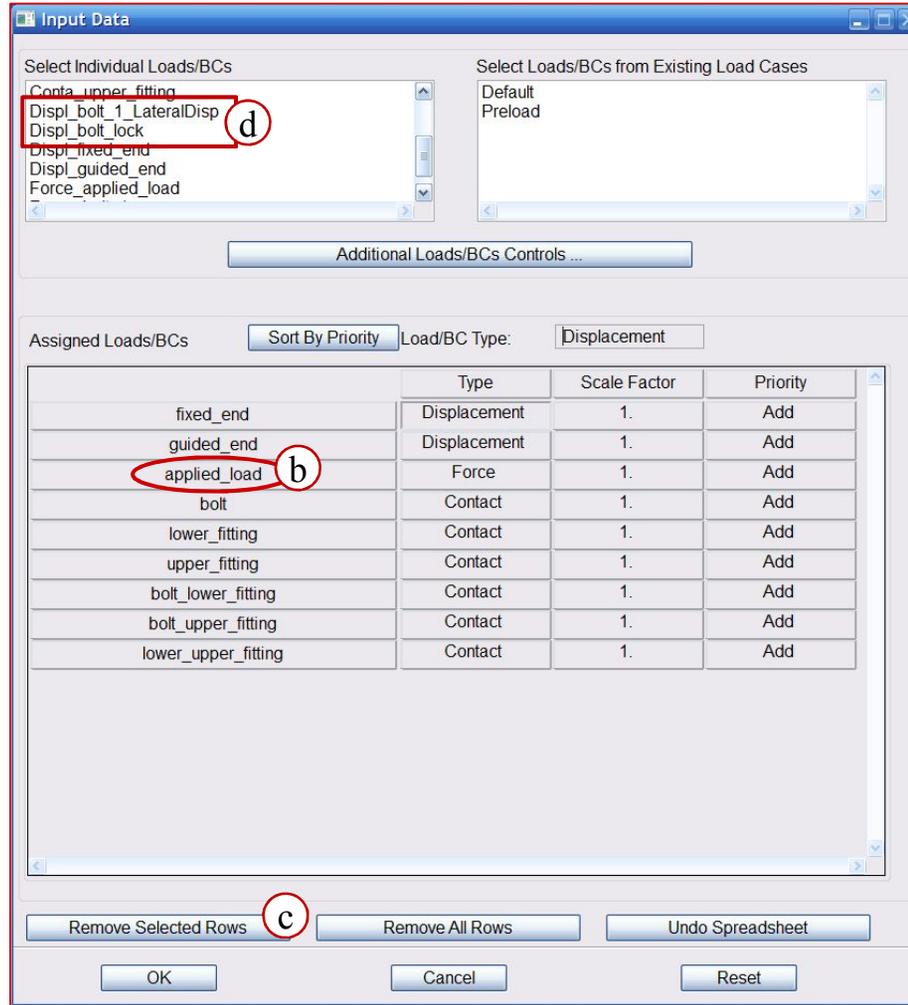
This first load step is used to preload the bolt. No external load is applied.



Step 3. Set Up Three Load Cases (Cont.)

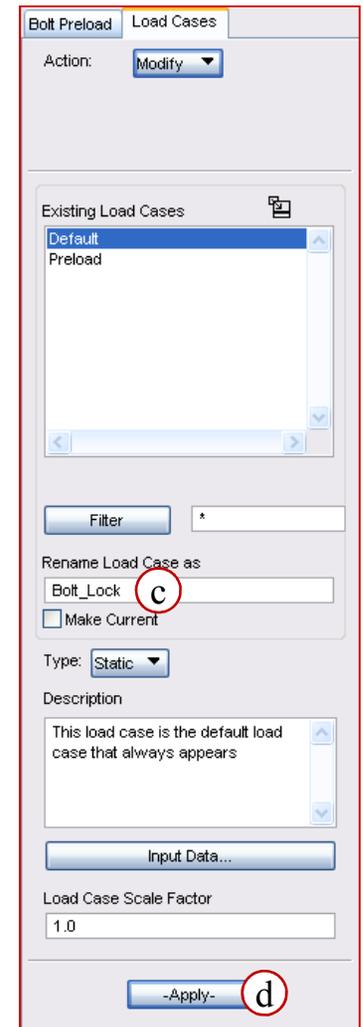
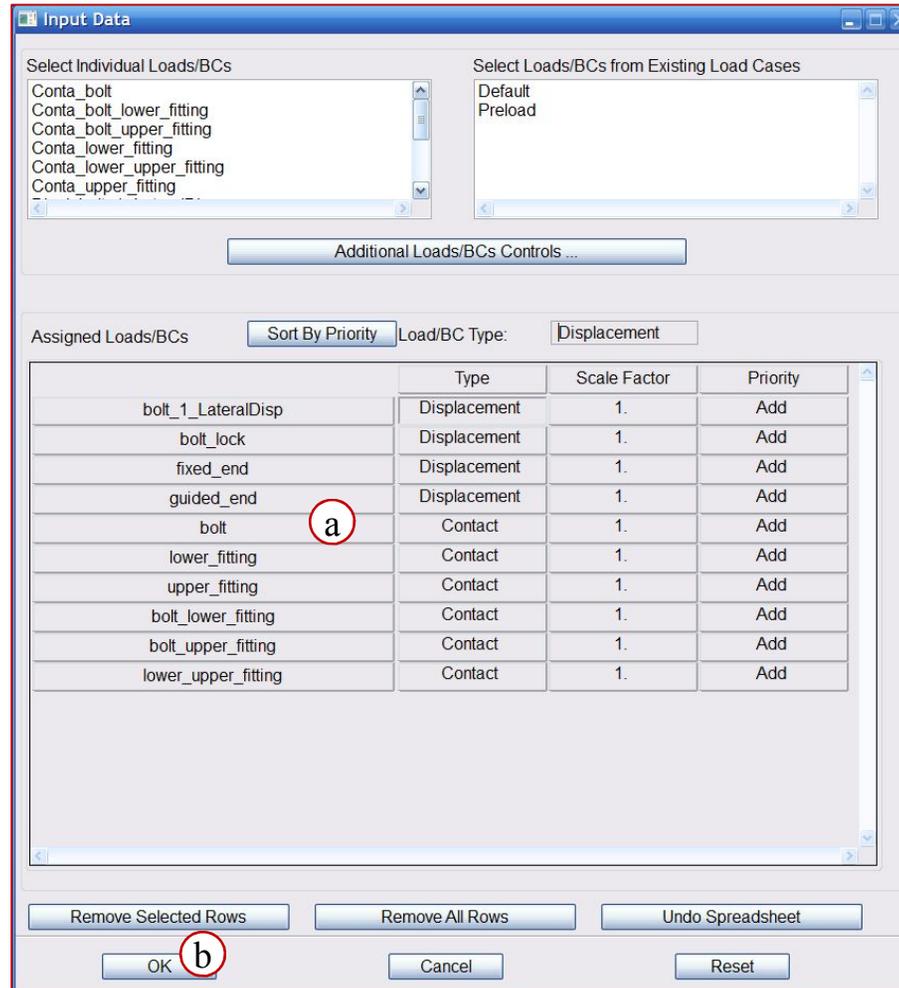
Set up a second load case to apply the bolt lock.

- a. Select **Default** from the *Existing Load Cases* list.
- b. Click **Applied_Load** to select this row.
- c. Click **Remove Selected Rows**.
- d. Click **Displ_bolt_1_LateralDisp** and **Displ_bolt_lock** to add them to the table below.



Step 3. Set Up Three Load Cases (Cont.)

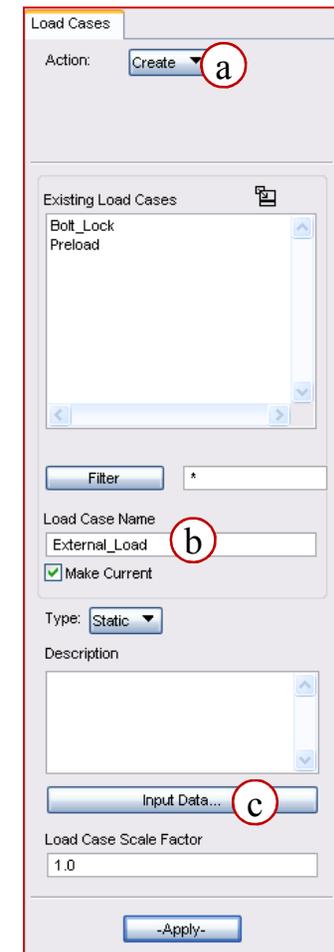
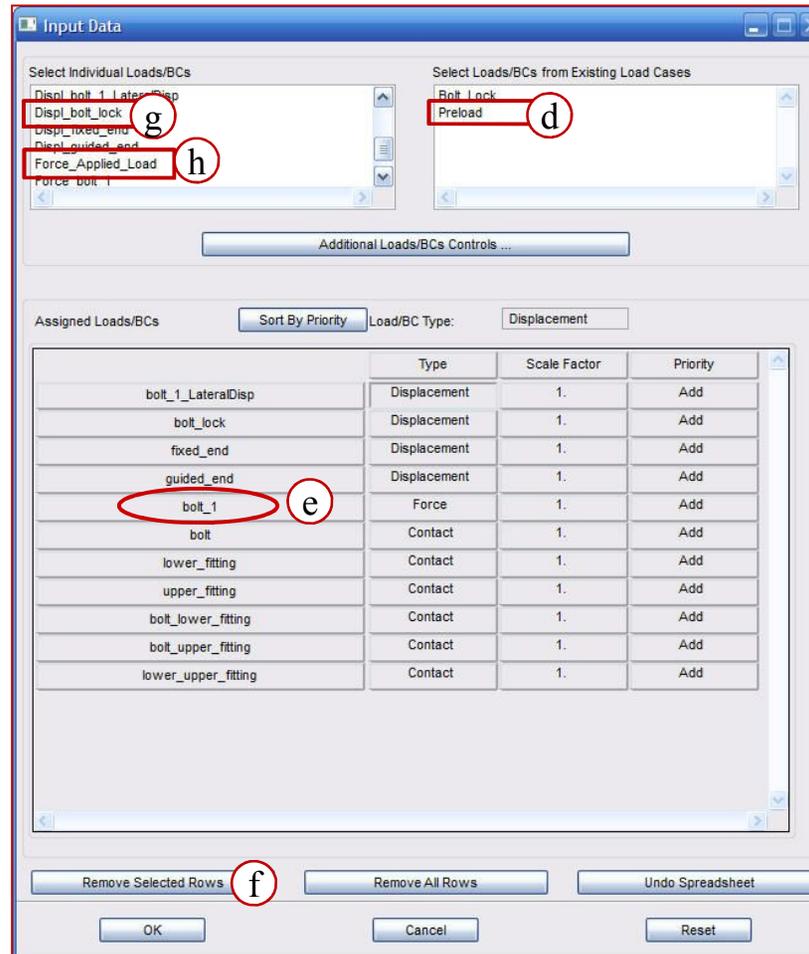
- a. The table should now look like the one shown on the right.
- b. Click **OK**.
- c. Enter **Bolt_Lock** in the *Rename Loadcase as* textbox.
- d. Click **Apply**.



Step 3. Set Up Three Load Cases (Cont.)

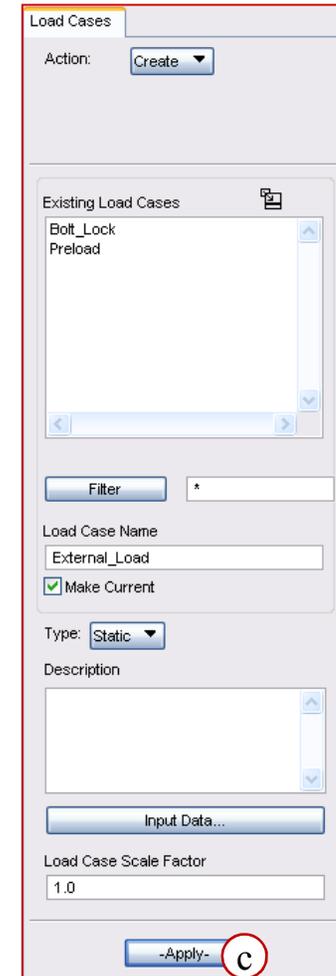
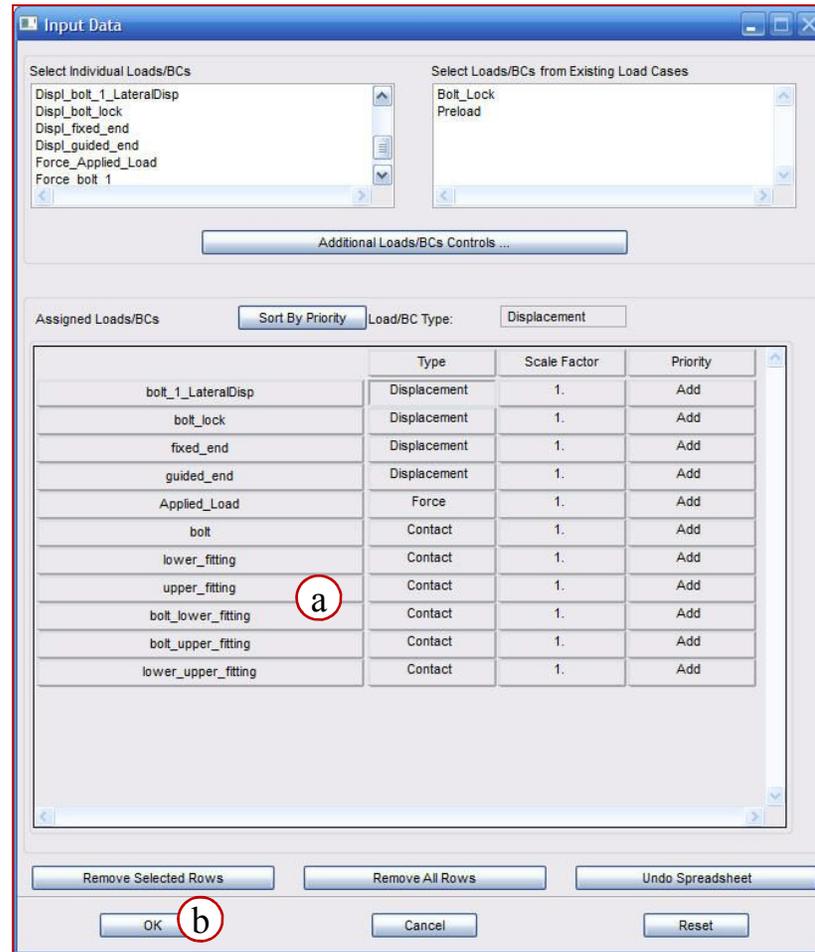
Set up a third load case to apply the external load.

- Set **Action** to **Create**.
- Enter **External_Load** as the **Load Case Name**.
- Click **Input Data**.
- Select **Preload** from the **Existing Load Cases** list.
- Click **bolt_1** to select this row.
- Click **Remove Selected Rows**.
- Click **Disp_bolt_lock** to add to table below.
- Click **Force_Applied_Load** to add to table below.

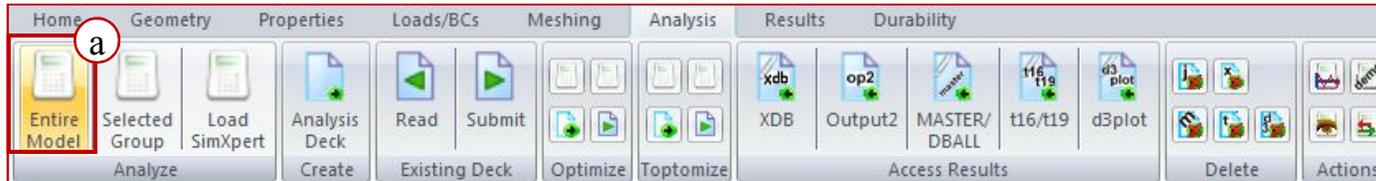


Step 3. Set Up Three Load Cases (Cont.)

- a. The table should now look like the one shown on the right.
- b. Click **OK**.
- c. Click **Apply**.

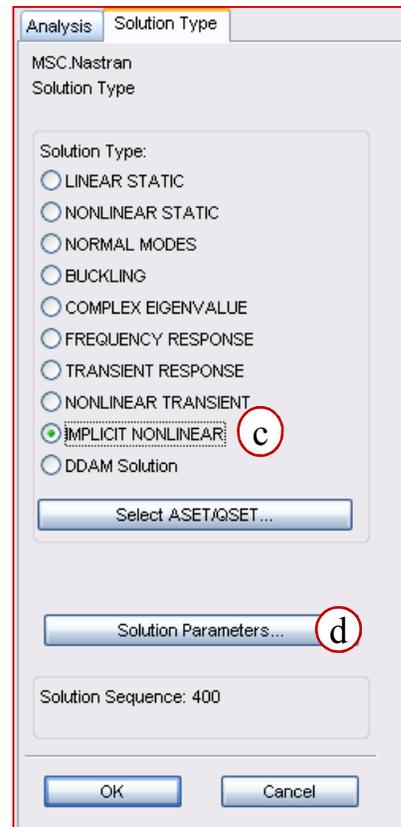


Step 4. Set Up SOL 400 and Run the Analysis



Select the Solution Type.

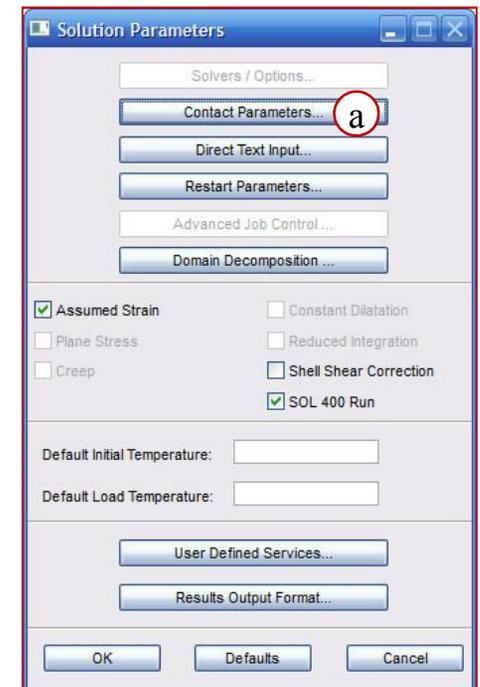
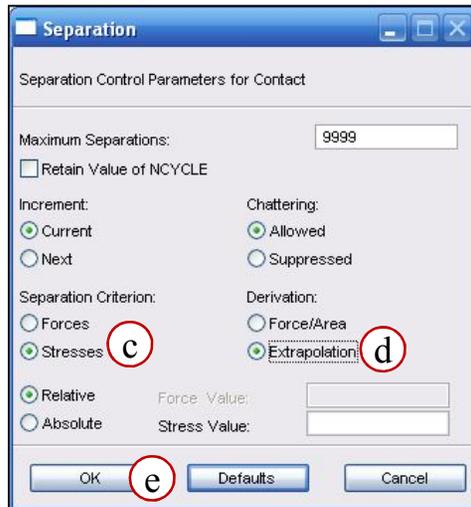
- a. Under the *Analysis* tab, click on **Entire Model** in the *Analyze* group.
- b. Click **Solution Type**.
- c. Select **IMPLICIT NONLINEAR**, SOL400
- d. Click **Solution Parameters**.



Step 4. Set Up SOL 400 and Run Analysis (Cont.)

Set up Separation Criterion

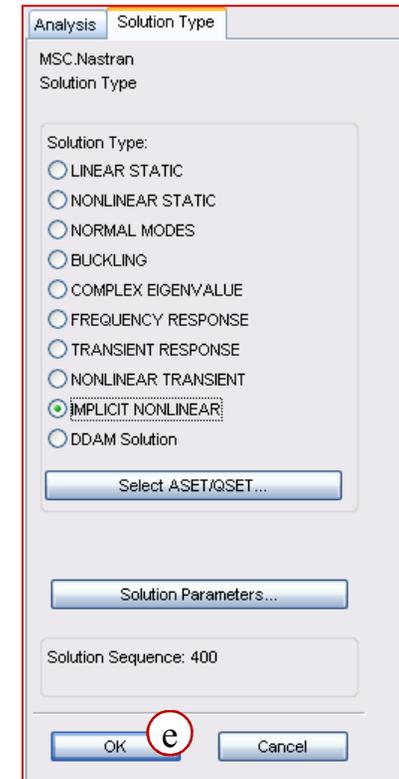
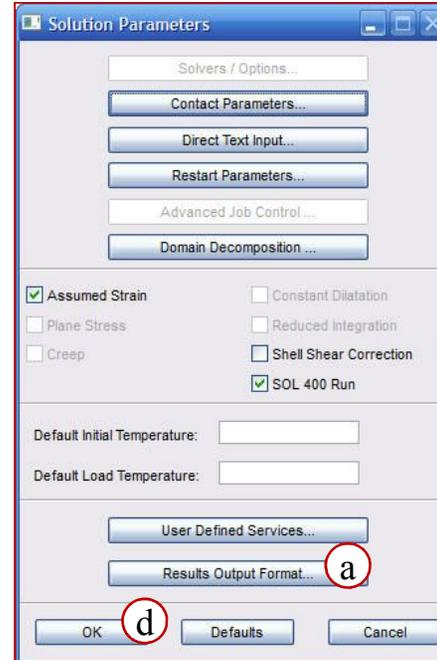
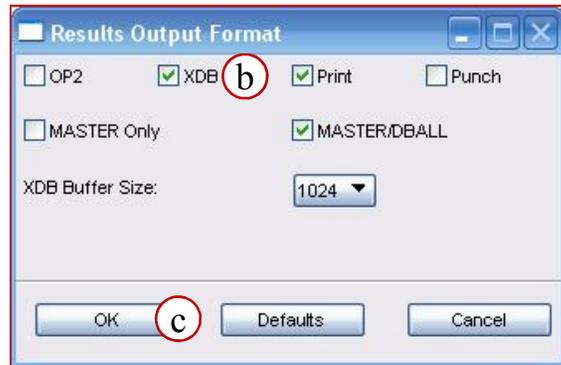
- Click **Contact Parameters**.
- Select **Separation**.
- Confirm **Stresses** for *Separation Criterion*.
- Confirm **Extrapolation** for *Derivation*.
- Click **OK**.
- Click **OK**.



Step 4. Set Up SOL 400 and Run Analysis (Cont.)

Select the Output results format.

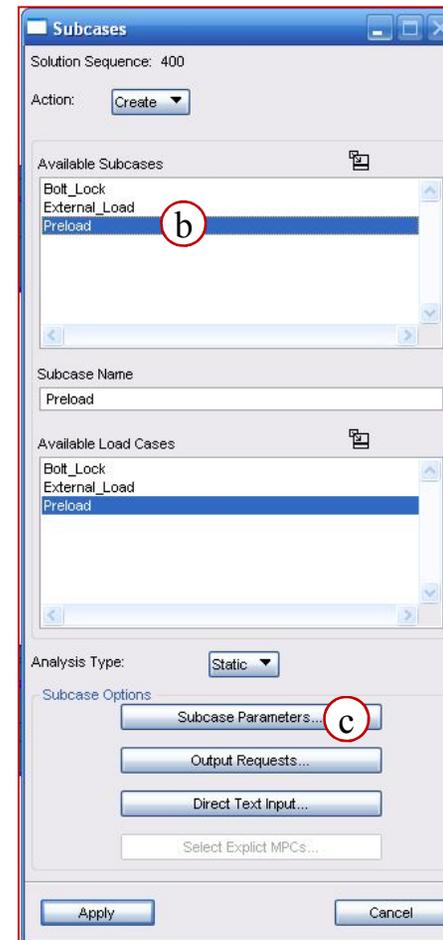
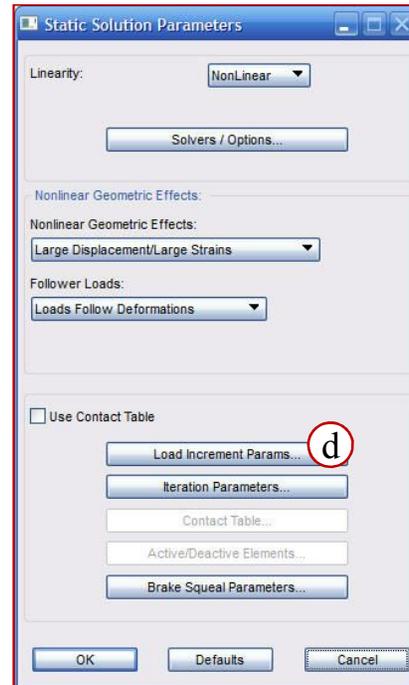
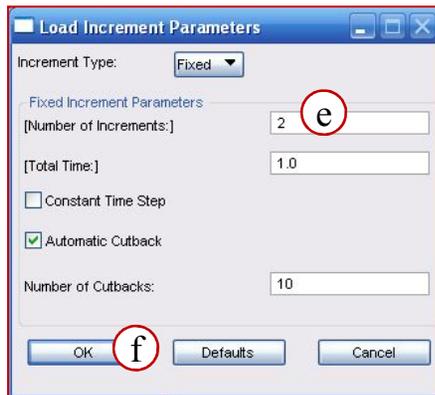
- a. Select **Results Output Format**.
- b. Check **XDB**.
- c. Click **OK**.
- d. Click **OK**.
- e. Click **OK**.



Step 4. Set Up SOL 400 and Run Analysis (Cont.)

Set up Load Increment for the Preload subcase.

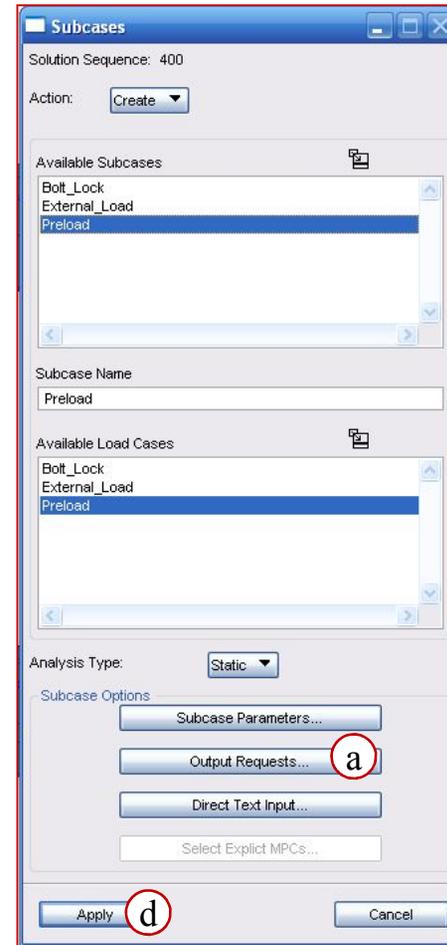
- a. Click **Subcases**.
- b. Click on **Preload** under *Available Subcases*.
- c. Click **Subcase Parameters**.
- d. Select **Load Increment Params**.
- e. Enter **2** for *Number of Increments*.
- f. Click **OK**.



Step 4. Set Up SOL 400 and Run Analysis (Cont.)

Create Output Request for the Preload subcase.

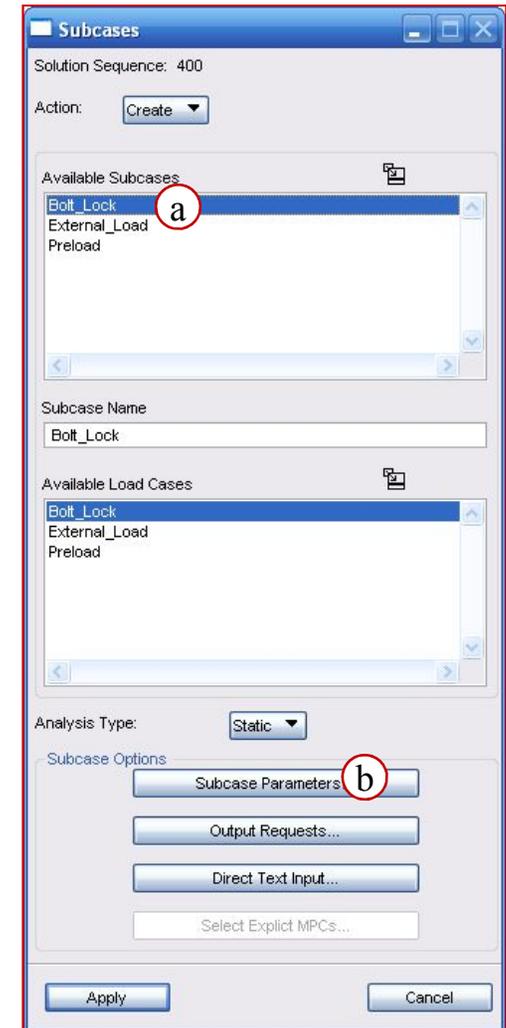
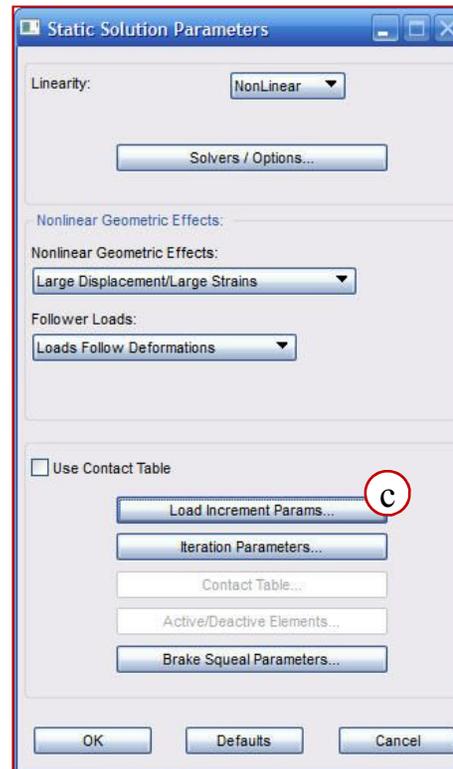
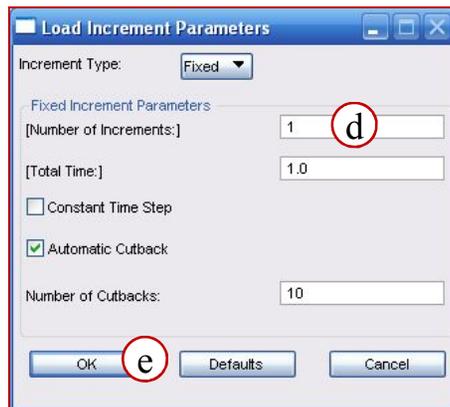
- a. Click **Output Requests**.
- b. Select **Contact Results**.
- c. Click **OK**.
- d. Click **Apply** to finish setting up the Preload subcase.



Step 4. Set Up SOL 400 and Run Analysis (Cont.)

Set up Load Increment for the Bolt_Lock subcase

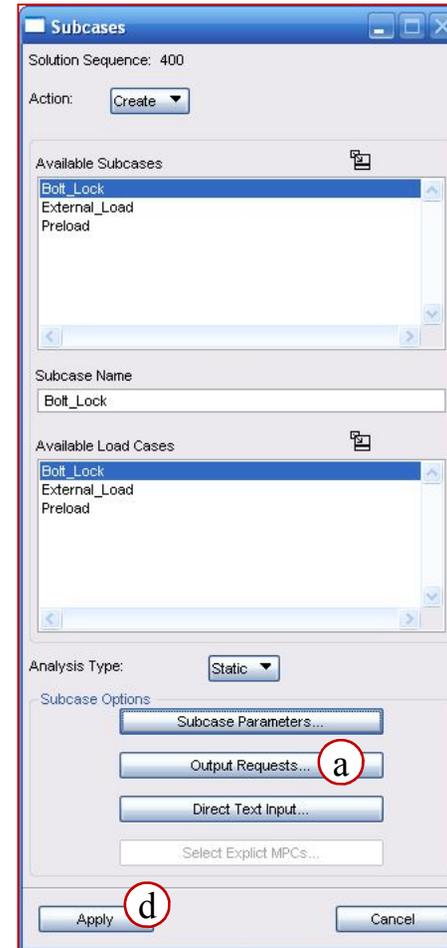
- Select **Bolt_Lock** from *Available Subcases*.
- Click **Subcase Parameters**.
- Click **Load Increment Params**.
- Enter 1 for *Number of Increments*.
- Click **OK**.



Step 4. Set Up SOL 400 and Run Analysis (Cont.)

Create Output Request for the Bolt_Lock subcase.

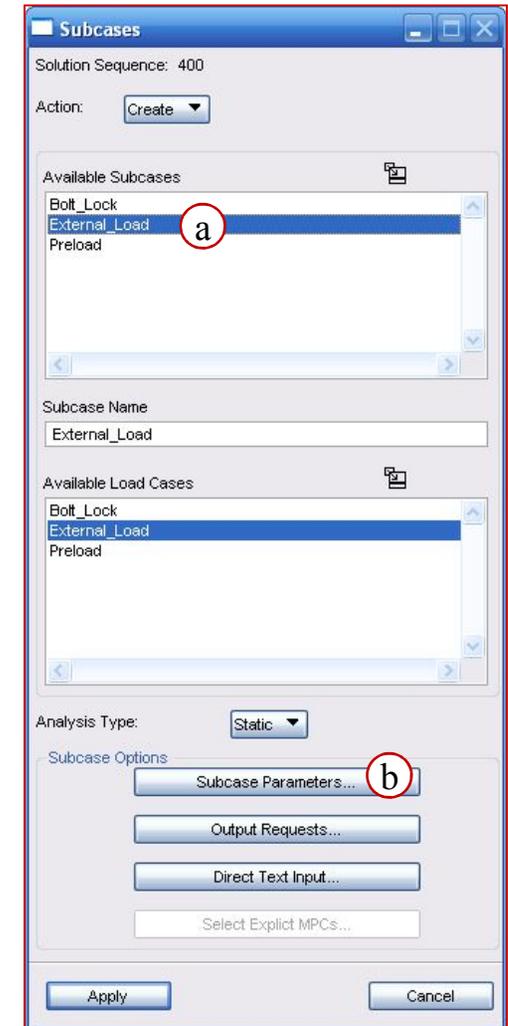
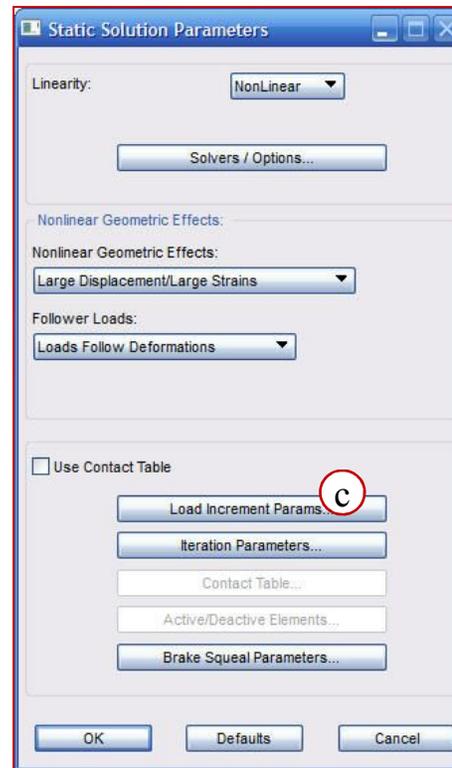
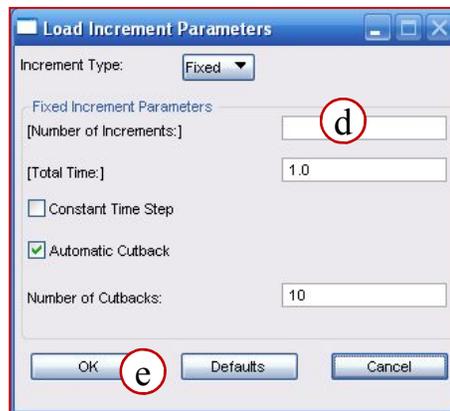
- a. Click **Output Requests**.
- b. Select **Contact Results**.
- c. Click **OK**.
- d. Click **Apply** to finish setting up the Bolt_Lock subcase.



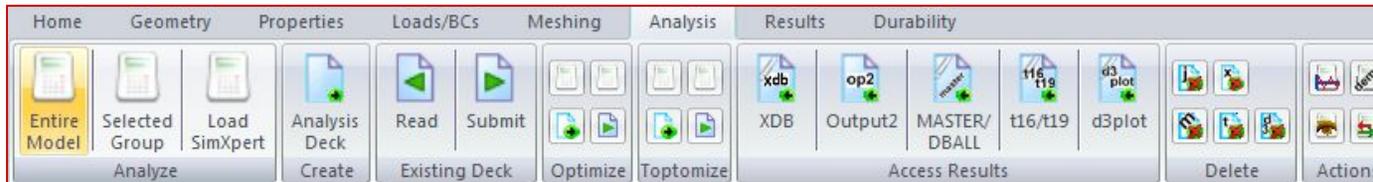
Step 4. Set Up SOL 400 and Run Analysis (Cont.)

Set up the Load Increment for the External Load subcase

- Select **External_Load** from *Available Subcases*.
- Click **Subcase Parameters**.
- Click **Load Increment Params**.
- Note that the Number of Increments is not specified, therefore the default value of 10 increments will be used. (See **NLPARM** in the *Bulk Data Entries* chapter of the *MSC Nastran Quick Reference Guide*)
- Click **OK**.

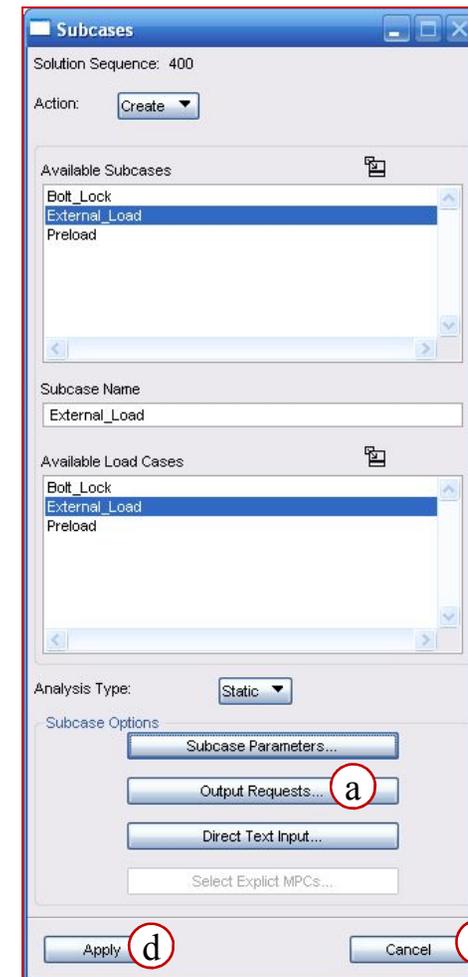


Step 4. Set Up SOL 400 and Run Analysis (Cont.)



Create Output Request for the External_Load subcase.

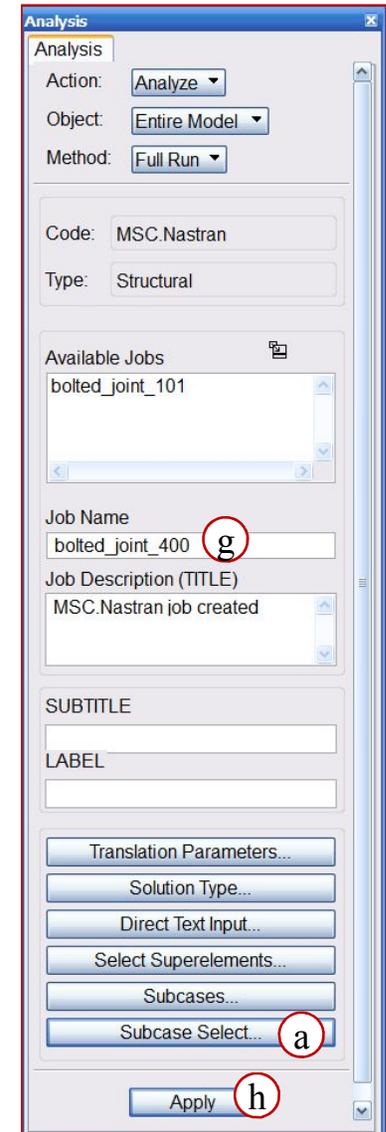
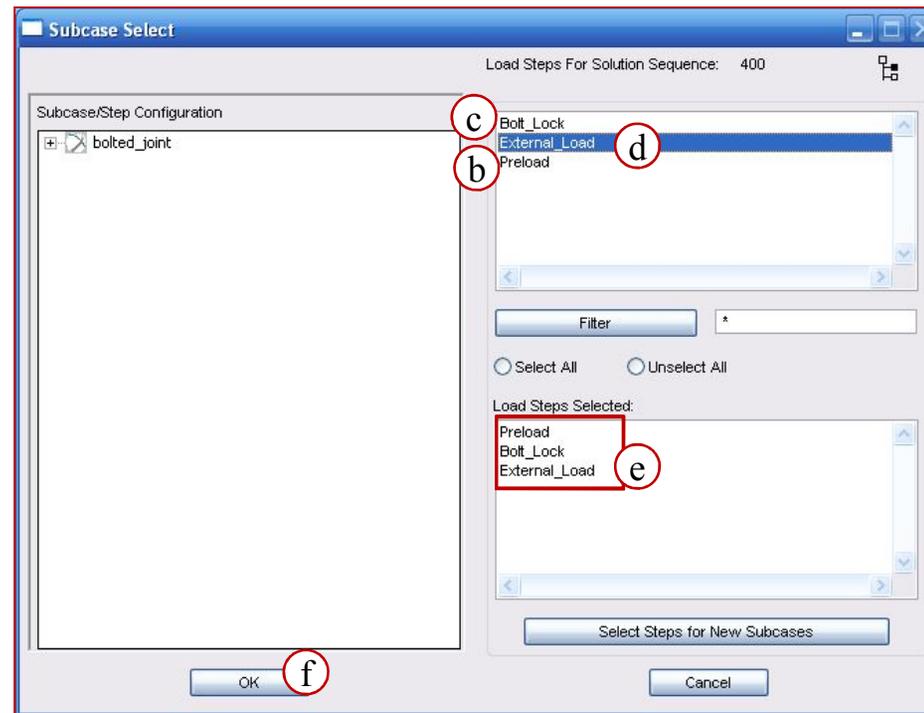
- Click **Output Requests**.
- Select **Contact Results**.
- Click **OK**.
- Click **Apply** to finish setting up the External_Load subcase.
- Click **Cancel**.



Step 4. Set Up SOL 400 and Run Analysis (Cont.)

Select all three subcases and run the analysis.

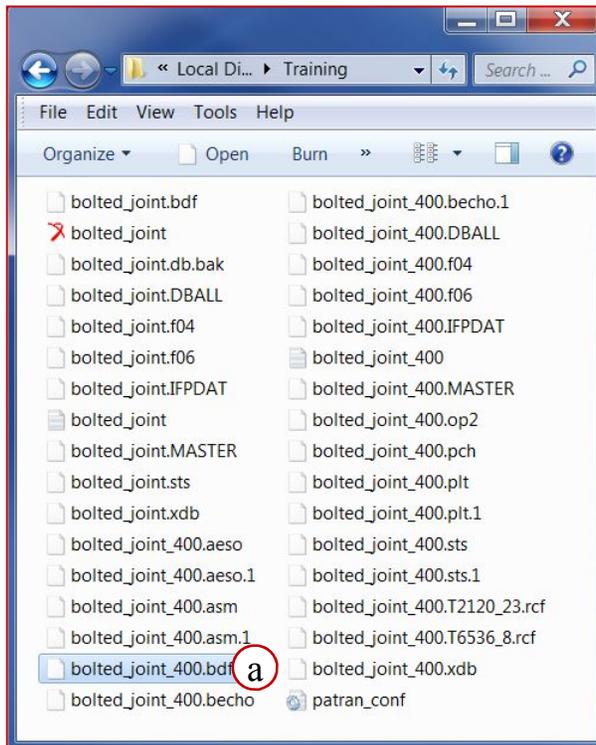
- a. Click **Subcase Select**.
- b. From the *Load Steps for Solution Sequence 400* list, select **Preload**.
- c. From the *Load Steps for Solution Sequence 400* list, select **Bolt_Lock**.
- d. From the *Load Steps for Solution Sequence 400* List, select **External_Load**.
- e. Check that all three load steps appear under *Load Steps Selected* in the order shown.
- f. Click **OK**.
- g. For the Job Name, enter **bolted_joint_400**.
- h. Click **Apply** to submit the job.



Step 5. Review Input File

While the job is running, review the MSC Nastran input file.

- a. Locate the file **bolted_joint_400.bdf**.
- b. Open **bolted_joint_400.bdf** with any text editor.



b

```
SOL 400
CEND
$ Direct Text Input for Global Case Control Data
  BCONTACT = 0
SUBCASE 1
  STEP 1
    SUBTITLE=Preload
    ANALYSIS = NLSTATIC
    NLPARM = 1
    BCONTACT = 1
    SPC = 2
    LOAD = 2
    DISPLACEMENT (SORT1, REAL)=ALL
    SPCFORCES (SORT1, REAL)=ALL
    STRESS (SORT1, REAL, VONMISES, BILIN)=ALL
    BOUTPUT (SORT1, REAL)=ALL
$ Direct Text Input for this Subcase
  STEP 2
    SUBTITLE=Bolt_lock
    ANALYSIS = NLSTATIC
    NLPARM = 2
    BCONTACT = 2
    SPC = 6
    LOAD = 3
    DISPLACEMENT (SORT1, REAL)=ALL
    SPCFORCES (SORT1, REAL)=ALL
    STRESS (SORT1, REAL, VONMISES, BILIN)=ALL
    BOUTPUT (SORT1, REAL)=ALL
$ Direct Text Input for this Subcase
  STEP 3
    SUBTITLE=External_Load
    ANALYSIS = NLSTATIC
    NLPARM = 3
    BCONTACT = 3
    SPC = 6
    LOAD = 5
    DISPLACEMENT (SORT1, REAL)=ALL
    SPCFORCES (SORT1, REAL)=ALL
    STRESS (SORT1, REAL, VONMISES, BILIN)=ALL
    BOUTPUT (SORT1, REAL)=ALL
```

Step 5. Review Input File (Cont.)

While the job is running, review the MSC Nastran input file continued.

- a. Notice the entry BOLT that defines a rigid bolt by a set of MPC constraints.
- b. Notice the three subcases that you set up.
- c. Check the BCPARA entry.

BOLT	30002	38931							
	TOP	11688	11691	11701	11708	11869	12132	12155	
		12156	12157	12158	12159	12160	12205	12206	
		12207	12208	12209	12210	12211	12216	12255	
		12716	12728	12740	12754	12764	12776	12788	
		12802	12813	12825	12839	12849	12861	12873	
		12887	12897						
	BOTTOM	38932	38933	38934	38935	38936	38937	38938	
		38939	38940	38941	38942	38943	38944	38945	
		38946	38947	38948	38949	38950	38951	38952	
		38953	38954	38955	38956	38957	38958	38959	
		38960	38961	38962	38963	38964	38965	38966	
		38967	38968						

Bolt Definition

```
BEGIN BULK
$ Direct Text Input for Bulk Data
PARAM POST 0
PARAM PRTMAXIM YES
BCPARA 0 NLGLUE 0
PARAM LGDISP 1
IBSEP 4
```

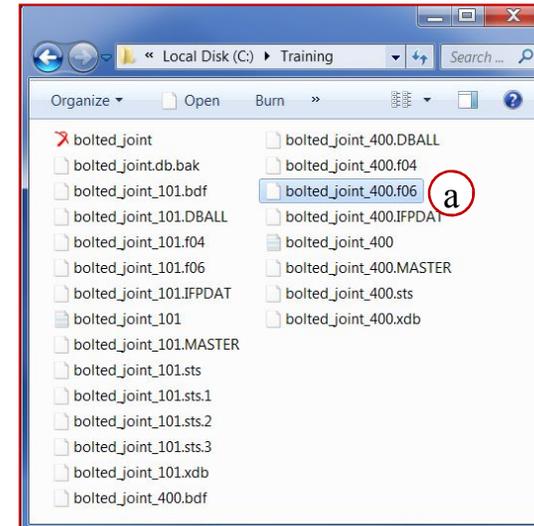
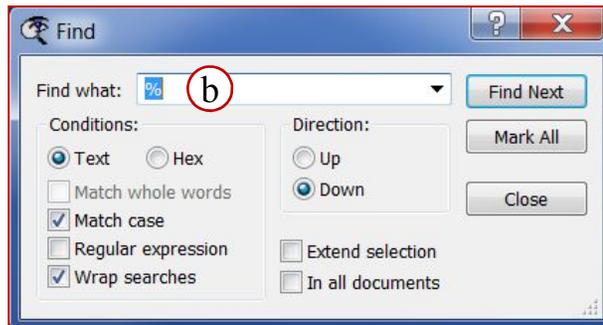
Relative stress based separation criterion

```
SUBCASE 1
STEP 1
SUBTITLE=Preload
ANALYSIS = NLSTATIC
NLPARM = 1
BCONTACT = 1
SPC = 2
LOAD = 2
DISPLACEMENT (SORT1, REAL)=ALL
SPCFORCES (SORT1, REAL)=ALL
STRESS (SORT1, REAL, VONMISES, BILIN)=ALL
BOUTPUT (SORT1, REAL)=ALL
$ Direct Text Input for this Subcase
STEP 2
SUBTITLE=Bolt_lock
ANALYSIS = NLSTATIC
NLPARM = 2
BCONTACT = 2
SPC = 6
LOAD = 3
DISPLACEMENT (SORT1, REAL)=ALL
SPCFORCES (SORT1, REAL)=ALL
STRESS (SORT1, REAL, VONMISES, BILIN)=ALL
BOUTPUT (SORT1, REAL)=ALL
$ Direct Text Input for this Subcase
STEP 3
SUBTITLE=External Load
ANALYSIS = NLSTATIC
NLPARM = 3
BCONTACT = 3
SPC = 6
LOAD = 5
DISPLACEMENT (SORT1, REAL)=ALL
SPCFORCES (SORT1, REAL)=ALL
STRESS (SORT1, REAL, VONMISES, BILIN)=ALL
BOUTPUT (SORT1, REAL)=ALL
```

Step 6. Review Results File

After completion of the run, review the .f06 results file.

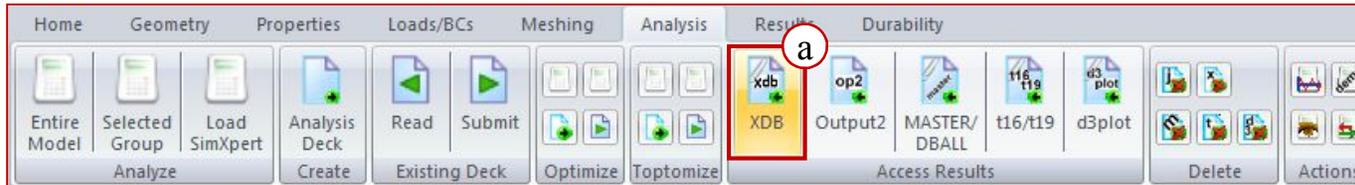
- Locate the file **bolted_joint_400.f06** and open with any text editor.
- Search for “%” and scan the .f06 file
- Close the .f06 file.



```

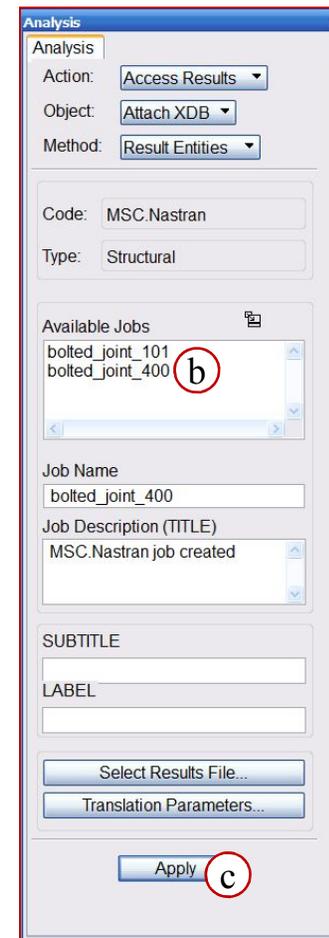
0
0
NON - LINEAR ITERATION MODULE OUTPUT
SUBCASE 1 STEP 1
STIFFNESS UPDATE TIME 0.01 SECONDS
ITERATION TIME 3.84 SECONDS
SUBCASE 1 STEP 1
LOAD NO. -- ERROR FACTORS -- CONV ITR MAT NO. AVG TOTL -- -- -- -- DISP -- -- -- -- LINE_S NO. TOT TOT
STEP INC ITR DISP LOAD WORK RATE DIV DIV BIS R_FORCE WORK AVG MAX AT GRID C FACT NO QNV KUD ITR
%5.00000E-01 1 1 1.00E+00 2.79E-02 1.00E+00 1.000 0 1 0 4.07E-04 4.927E-01 2.47E-05 9.854E-04 38931 2 1.00 0 0 0 1
*** USER INFORMATION MESSAGE 4550 (nl3con)
*** THE STIFFNESS MATRIX IS UPDATING TO THE INITIAL STATE FOR NEW LMFAC=9.725004E+02, PENFN=9.725004E+02 ***
%5.00000E-01 1 1 1.00E+00 2.79E-02 1.00E+00 1.000 0 1 0 4.07E-04 4.927E-01 2.47E-05 9.854E-04 38931 2 1.00 0 0 1 2
%5.00000E-01 1 2 4.80E-03 3.10E-07 3.53E-04 0.000 0 1 0 7.37E-08 4.924E-01 2.47E-05 9.850E-04 38931 2 1.00 0 0 2 3
%1.00000E+00 2 1 1.00E+00 1.39E-02 1.00E+00 1.000 0 1 0 4.08E-04 1.970E+00 4.95E-05 1.970E-03 38931 2 1.00 0 0 3 4
%1.00000E+00 2 2 1.06E-02 6.57E-07 5.72E-04 0.000 0 1 0 1.18E-07 1.969E+00 4.98E-05 1.969E-03 38931 2 1.00 0 0 4 5
    
```

Step 7. Attach Results File

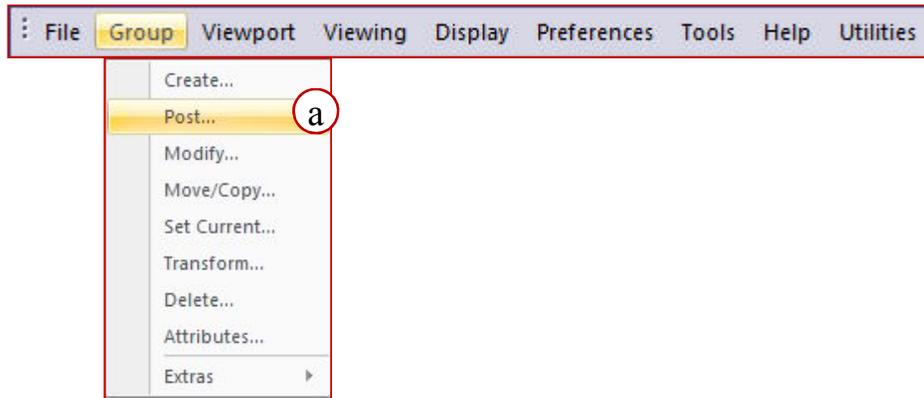


Post process analysis results in Patran.

- a. Under the *Analysis* tab, click **XDB** from the *Access Results* Group.
- b. Under the Available Jobs, select **bolted_joint_400**.
- c. Click **Apply**.

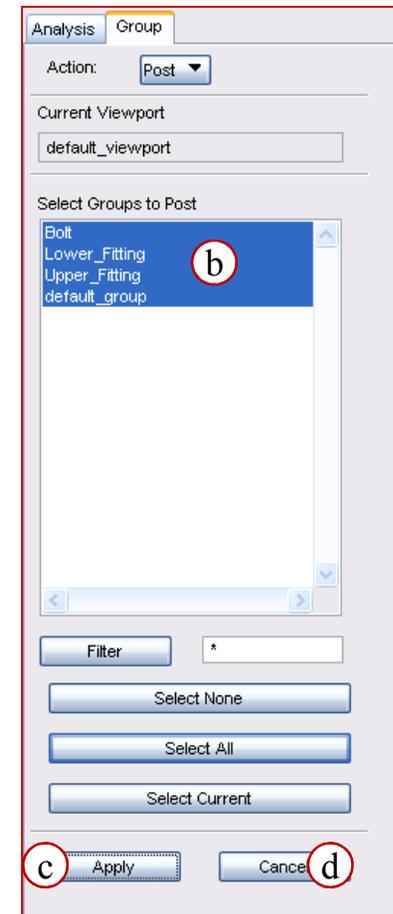


Step 8. Review Analysis Results

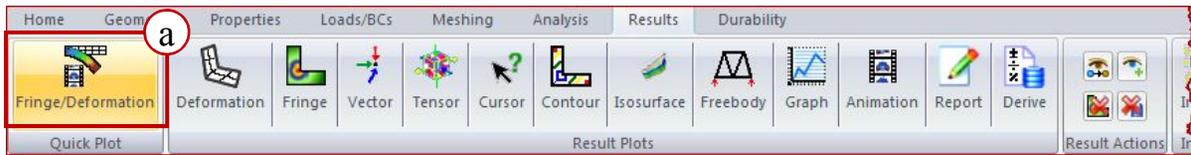


Post the entire bolted_joint model.

- a. Select **Post** from the *Group* menu.
- b. Click **Select All**.
- c. Click **Apply**.
- d. Click **Cancel**.

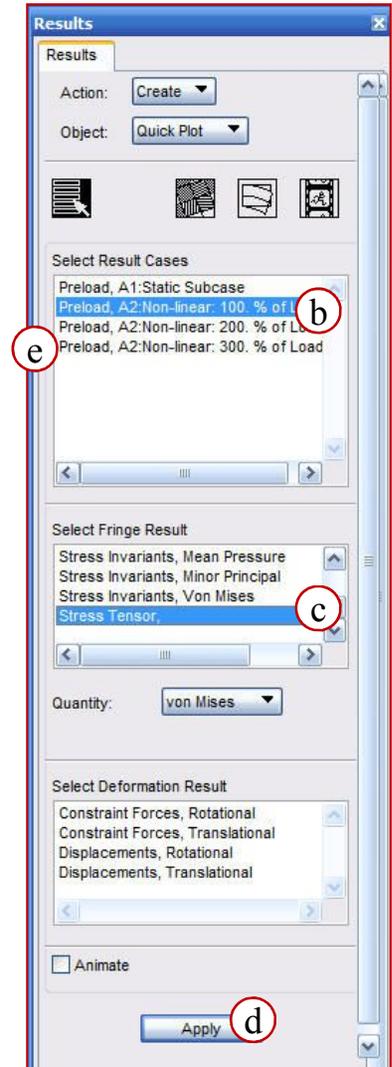
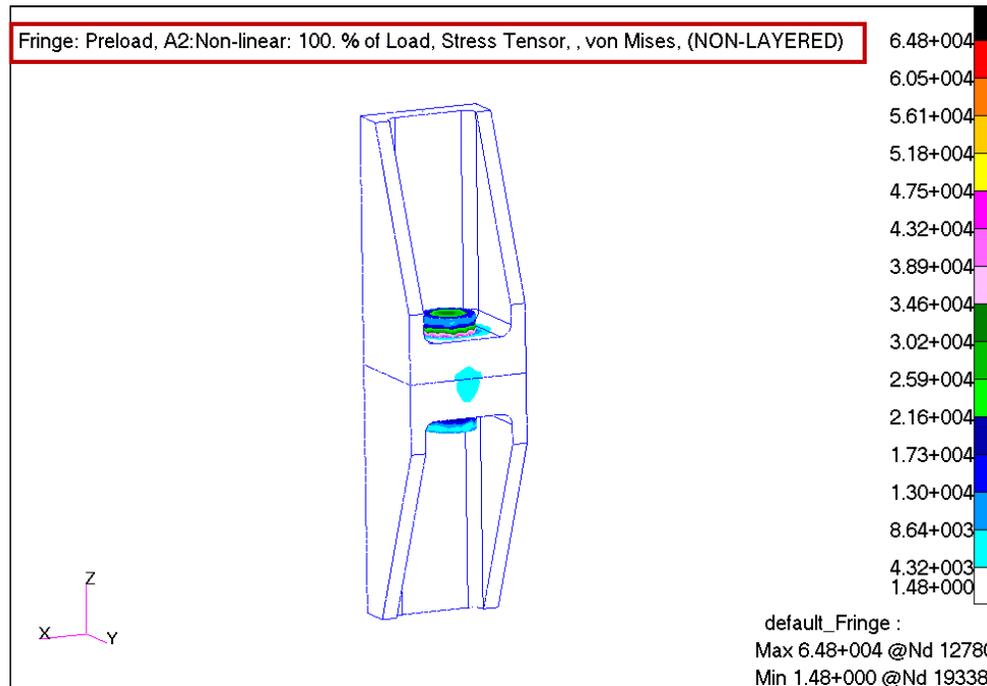


Step 8. Review Analysis Results (Cont.)

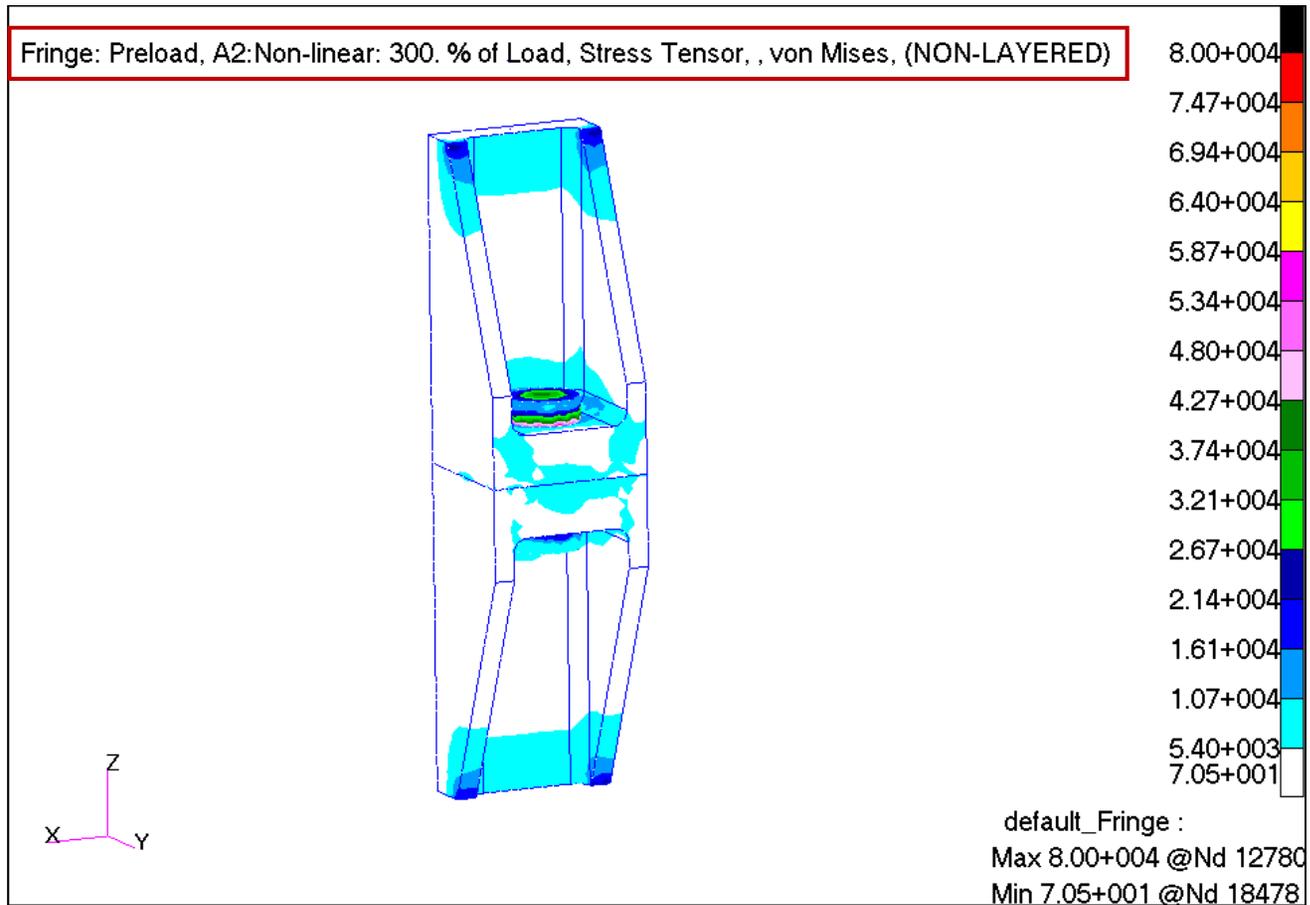


Examine von Mises stresses.

- Under the *Results* tab, click **Fringe/Deformation** from the *Quick Plot* group.
- Select **100% of Load**.
- Select **Stress Tensor** for Fringe Result.
- Click **Apply**.
- Repeat for **300% of Load**. This plot shown on the next page.



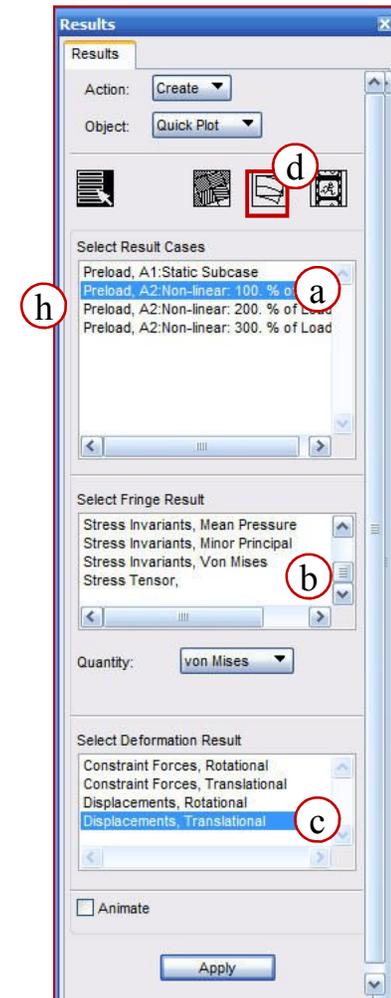
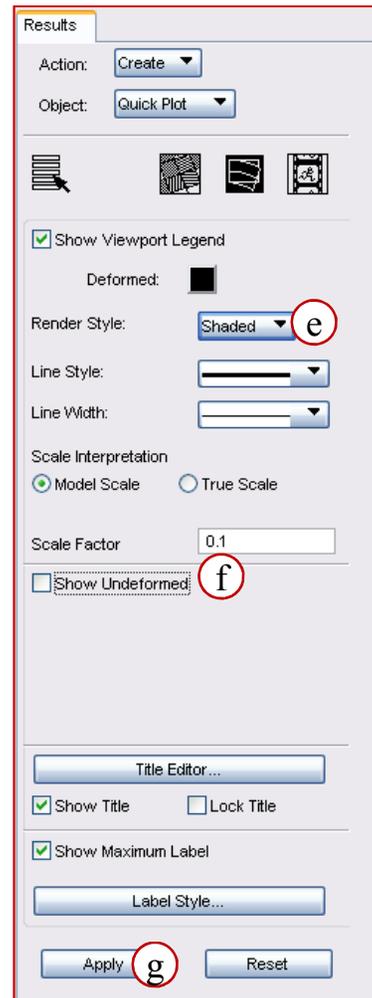
Step 8. Review Analysis Results (Cont.)



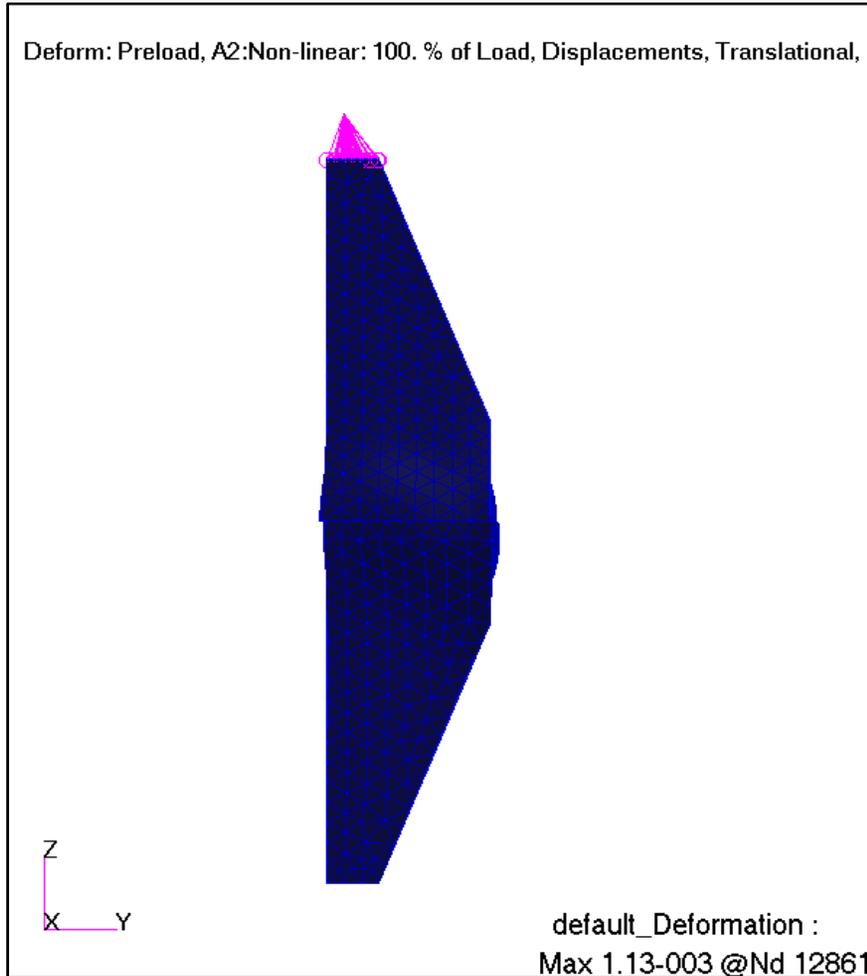
Step 8. Review Analysis Results (Cont.)

Examine deformation.

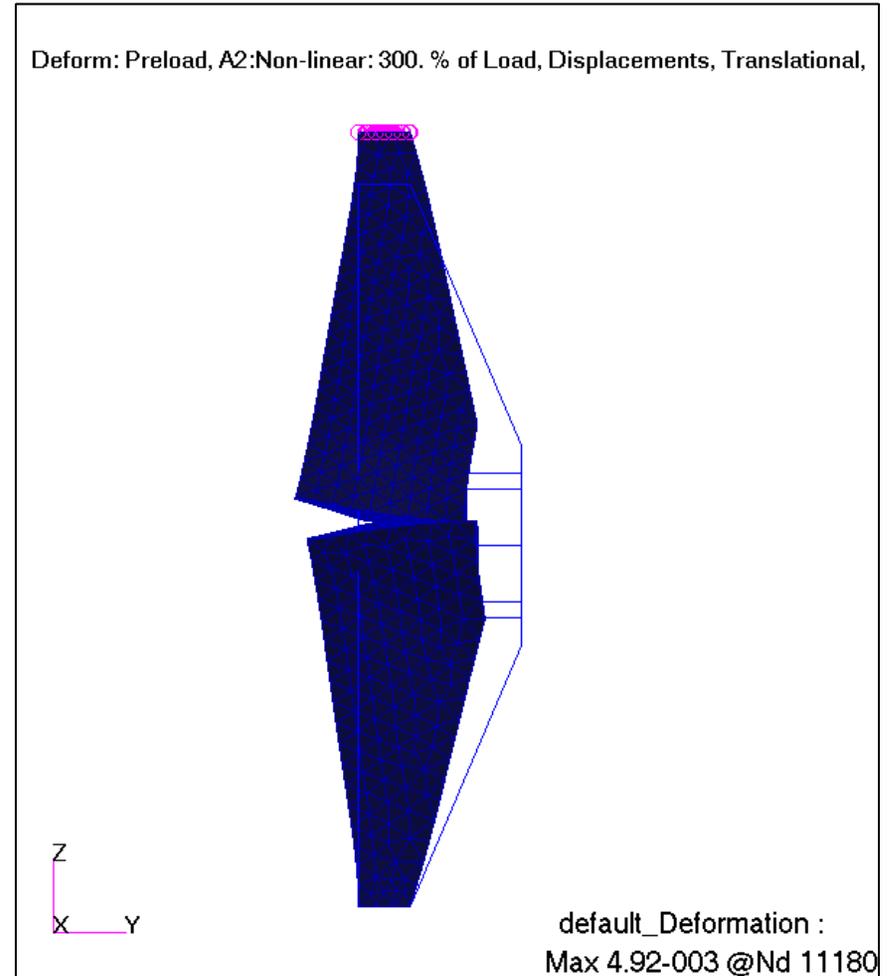
- a. Select **100% of Load**.
- b. Unselect **Stress Tensor** by clicking on it.
- c. Select **Displacement, Translational** for deformation result.
- d. Click the **Deform Attributes** icon.
- e. Select **Shaded** for the *Render Style*.
- f. Uncheck *Show Undeformed*.
- g. Click **Apply**.
- h. Repeat for **300% of Load**.
- i. Both plots are shown on the following page.



Step 8. Review Analysis Results (Cont.)



100% of Load



300% of Load

