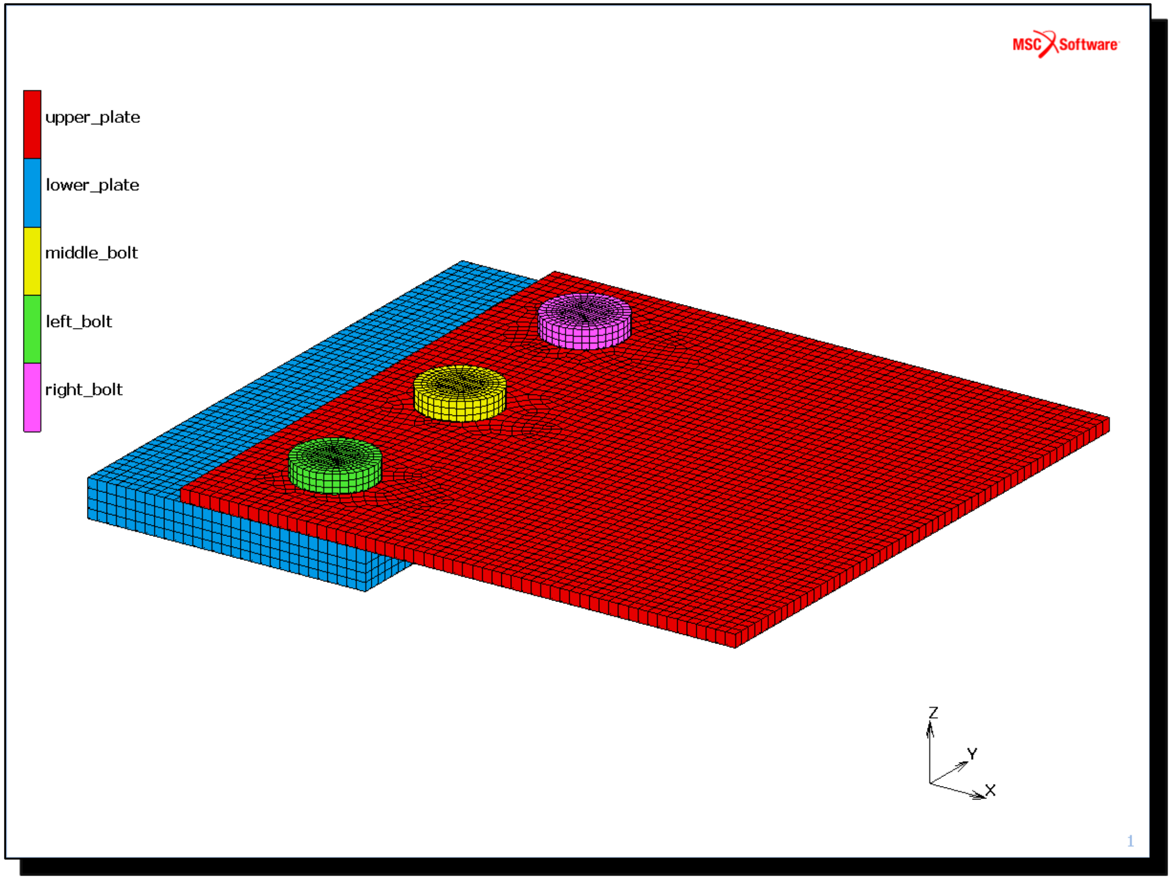
Multiple Bolt Preload Setup in Mentat



- **Objectives:**
  - This workshop illustrates Efficient and user friendly method of defining common behaviour multiple bolts using the latest Bolts feature introduced in Mentat 2018
  - This new feature is useful in assemblies with a large number of fasteners with preload or, springs. Multiple bolts can quickly be generated using a single action.
  - Using this method, after selecting the elements of bolt, Mentat will automatically create and position the control node. The axial direction of the bolt is also automatically determined. User has to define the preload applied on control node in the first degree of freedom as this new technique is based on "Cross-Section" option. The control node of the Cross-Section has only one degree of freedom.
  - During post processing, the bolt force is plotted in the bolt axial direction. Global variables for control node displacement, as well as axial and shear force are also available.
  - With this new bolt modeling technique, for Large Strain analysis, the limitation of small rotations that existed in previous releases has now been removed.
- **Software Version :**
  - Marc and Mentat 2018
- **Files Required:**
  - `bolted_plates.mud`

- **Suggested Exercise Steps:**
  1. Open the file bolted_plates.mud in Mentat
  2. Check the contact bodies
  3. Populate the contact table
  4. Create control node for bolt preload using Bolt feature
  5. Define Boundary Condition, Clamp the lower plate
  6. Define point load on the top plate
  7. Apply Bolt preload on all 3 bolts of magnitude 2000N on the control nodes
  8. Apply locking constraint on control nodes
  9. Create 3 loadcases; Bolt preload, bolt locking & External loading. Choose appropriate Loads & Boundary conditions for each of these loadcases.
  10. Create & run the job

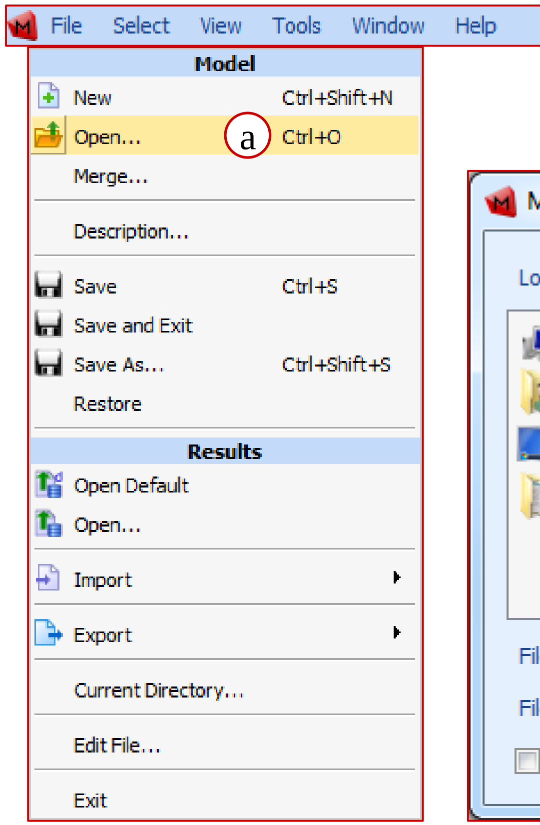
Step 1. Open An Existing Database

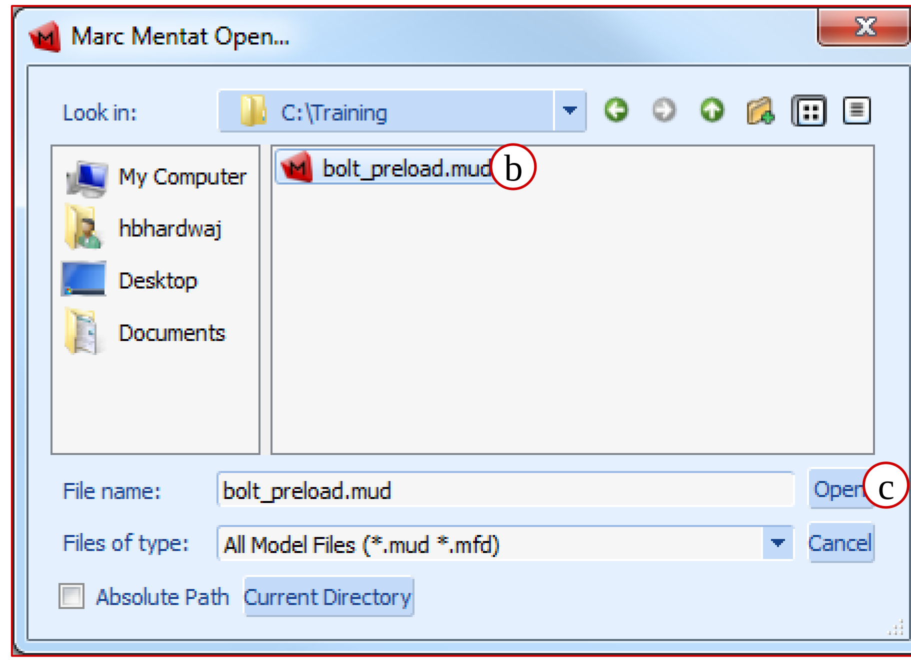
Open an existing database.

a. Select **File > Open**.

b. Select `bolt_preload.mud`.

c. Click **Open**.

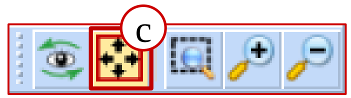


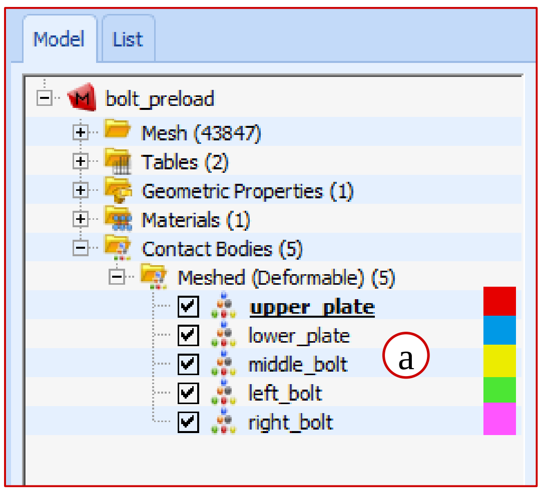


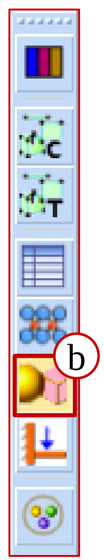
Step 2. Check Contact Bodies

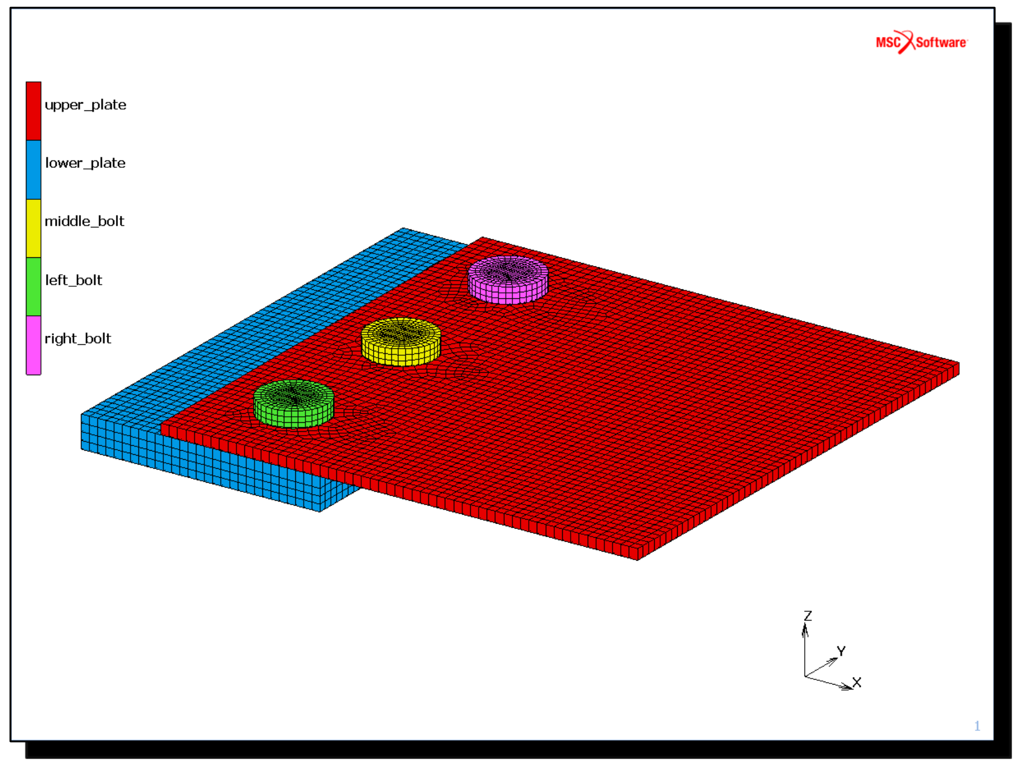
Identify contact bodies

a. Expand **Contact bodies** then **Meshed (Deformable).**
b. Click **Toggle Identification of Contact Bodies.**
c. Click **Fit.**









Step 3. Populate contact table

Create contact tables:

a. Right-click on the blank space in the Model Navigator and select: **Contact > New Contact Table.**

b. Click **Populate / Manipulate**

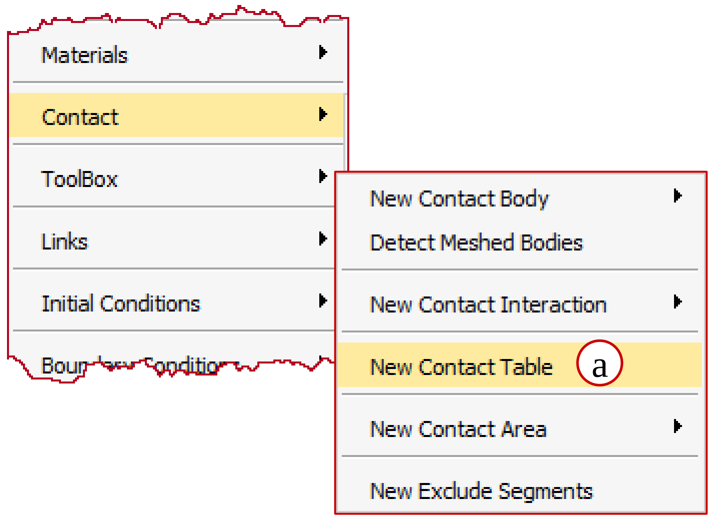
c. Enable **Contact Distance Check (Meshed Bodies Only).**

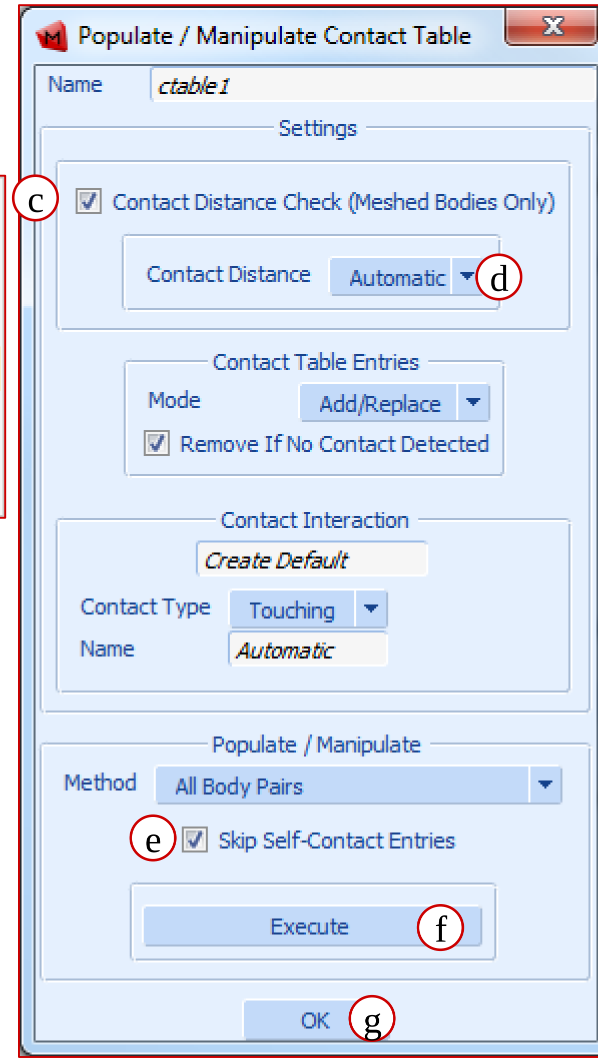
d. Select **Automatic** as **Contact Distance.**

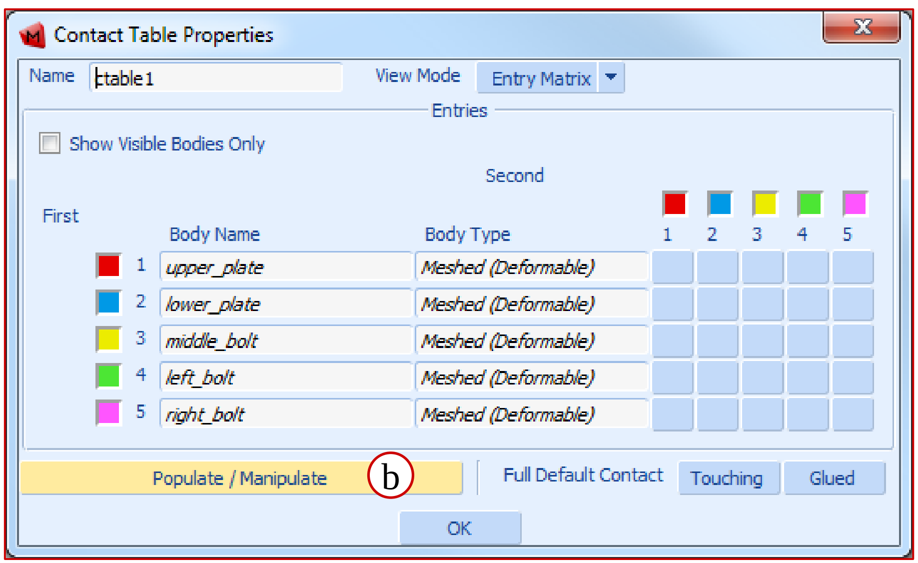
e. Enable **Skip Self-Contact Entries.**

f. Click **Execute** button

g. Click **OK**



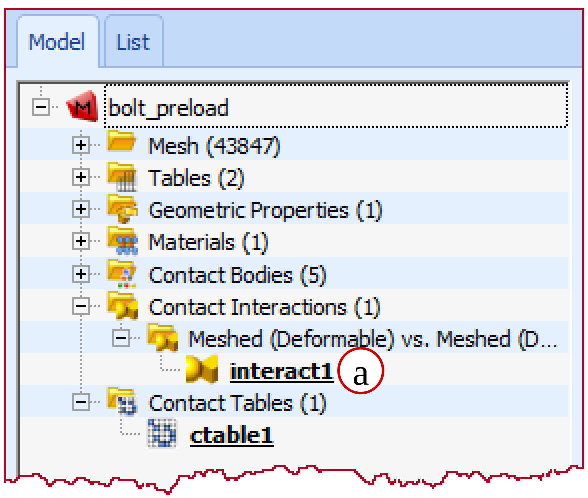


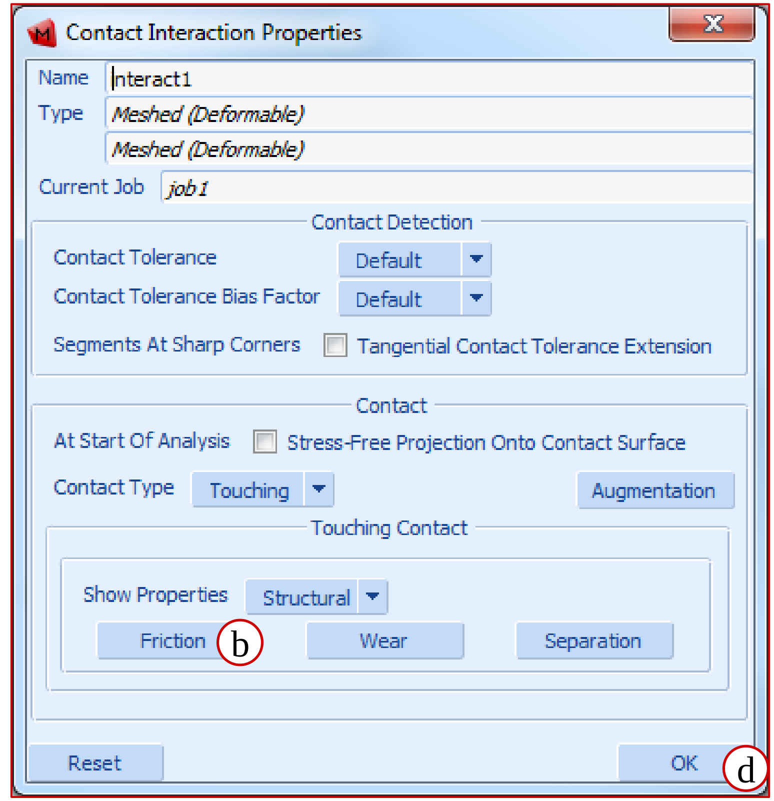


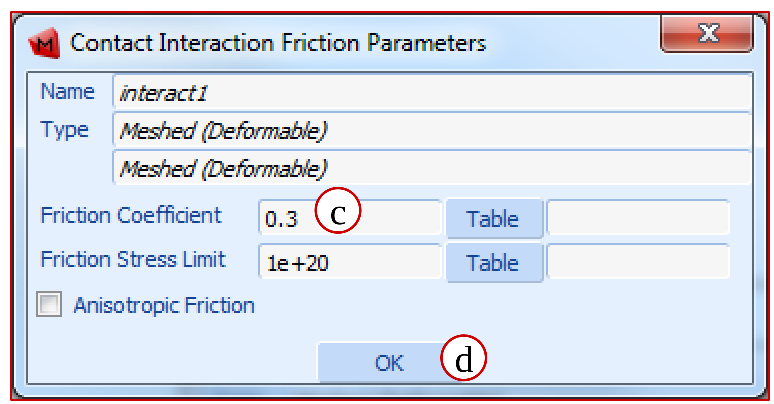
Step 4. Specify friction coefficient

Specify friction coefficient

a. Expand **Contact Interactions** and double click **interact1.**
b. Click **Friction.**
c. Enter **0.3** for **Friction Coefficient.**
d. Click **OK** twice.







Step 5. Make the plates invisible for bolt creation

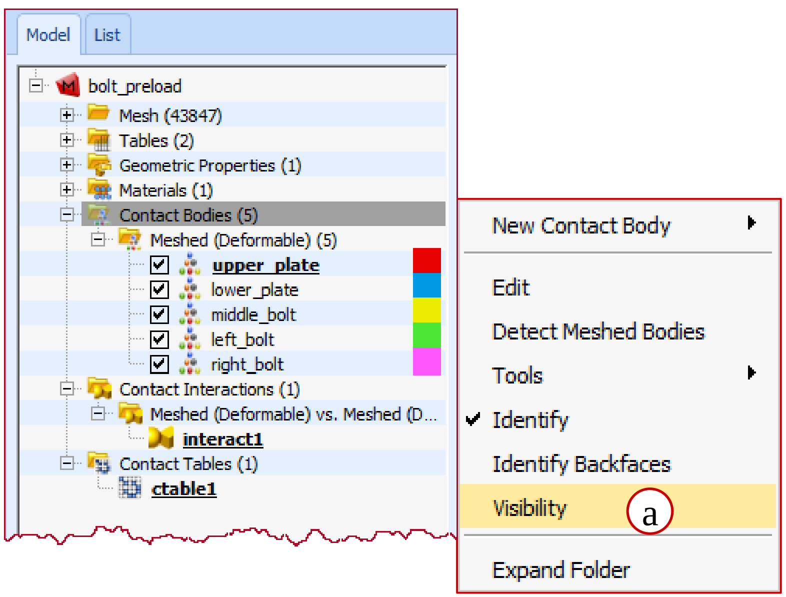
Display only bolts and hide the plates

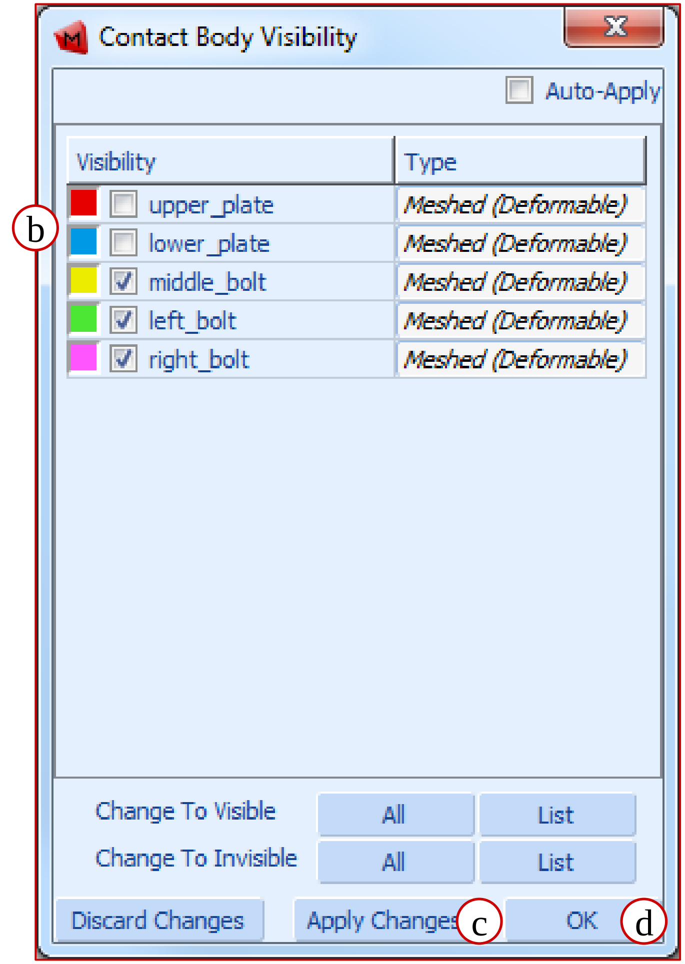
a. Right click on **Contact Bodies** and click **Visibility.**

b. Disable **upper_plate** and **lower_plate** contact bodies.

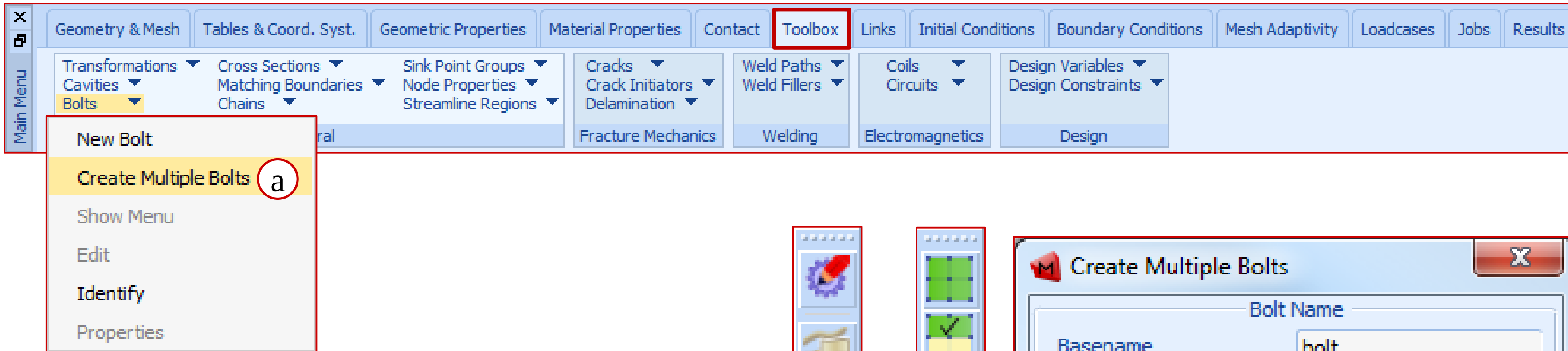
c. Click **Apply Changes.**

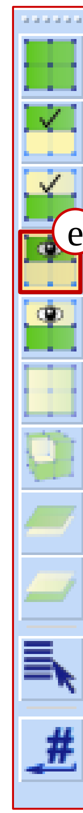
d. Click **OK.**

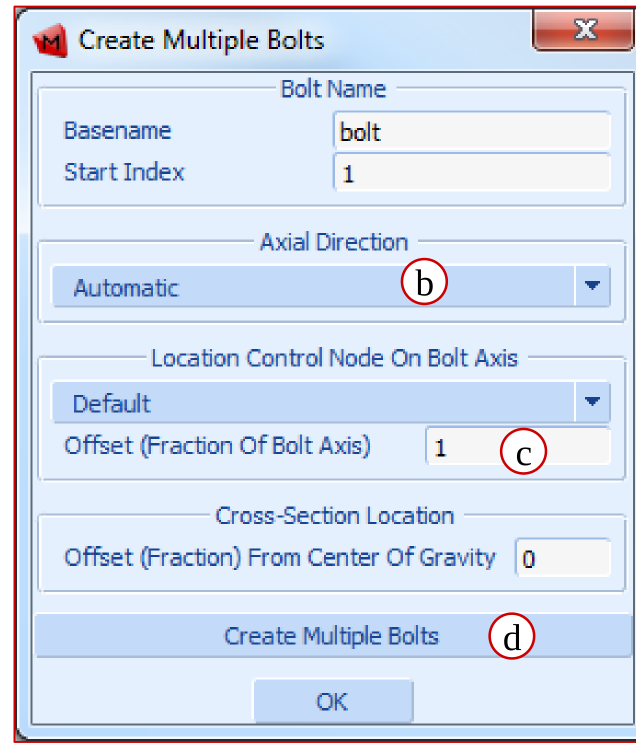




Step 6. Bolt creation





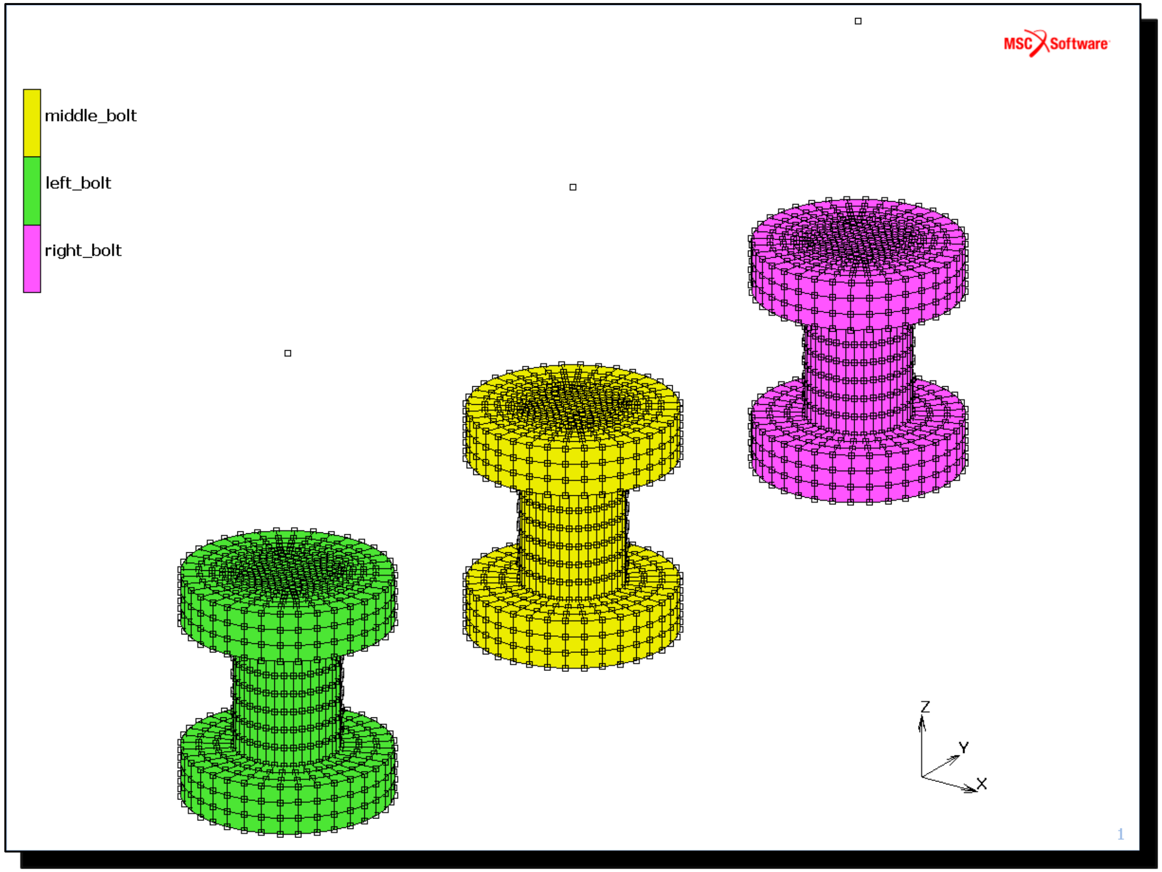


Create bolts

a. Select **Toolbox > Bolts > Create Multiple Bolts.**
b. Verify **Automatic** is selected as **Axial Direction.**
c. To specify the location of control node on bolt axis, set **Offset** as **“1”.**
d. Click **Create Multiple Bolts.**
e. Click **All Visible Elements.**
f. Select **Toggle Nodes Display.**

Step 6. Bolt creation (Contd.)

Note that control nodes are created for each bolt in the normal direction, at distance of 1, as specified in previous step.

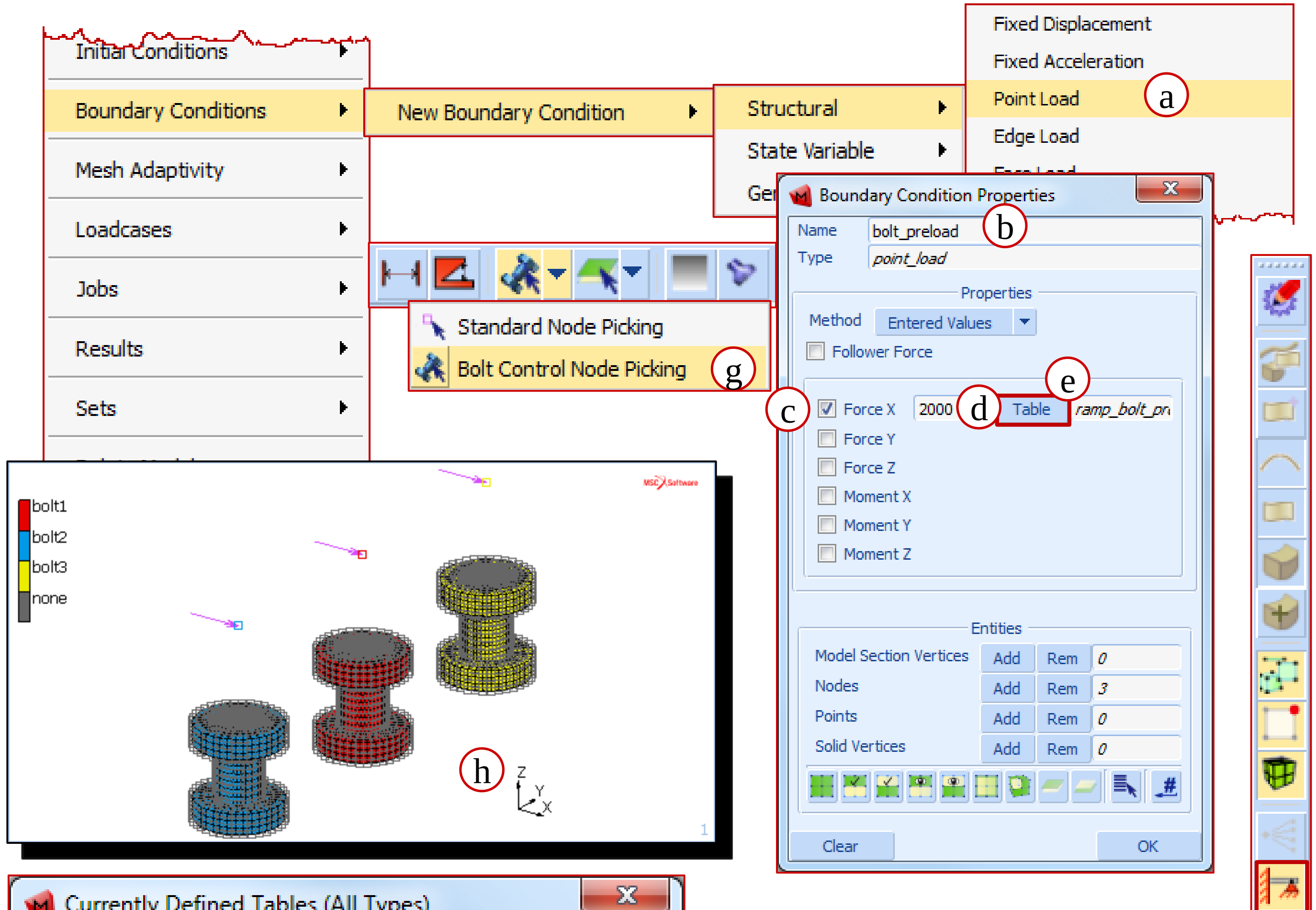


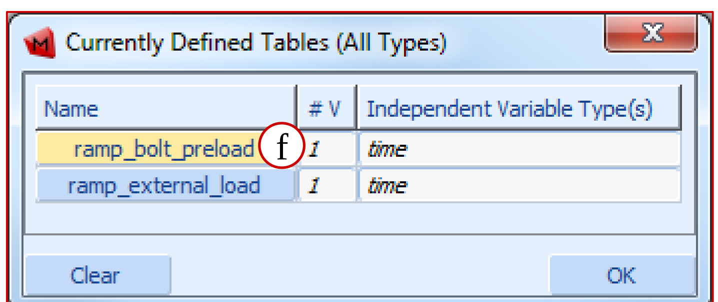
Note -The control node has only 1 dof in the direction of the cross section normal vector

Step 7. Apply boundary conditions

Specify bolt preload

a. Right-click on the Model Navigator and select: **Boundary Conditions > New Boundary Condition > Structural > Point Load.**
b. Enter **boundary condition Name:** `bolt_preload`.
c. Enable **Force X**
d. Enter **2000** for **Force X.**
e. Click **table**
f. Select `ramp_bolt_preload`.
g. In the miscellaneous menu, change the node picking option from Standard Node Picking to **Bolt Control Node Picking**
h. Click on any single element on each bolt to select the control node of it's corresponding bolt.
i. Click **Toggle Boundary Conditions Display** to turn display on.



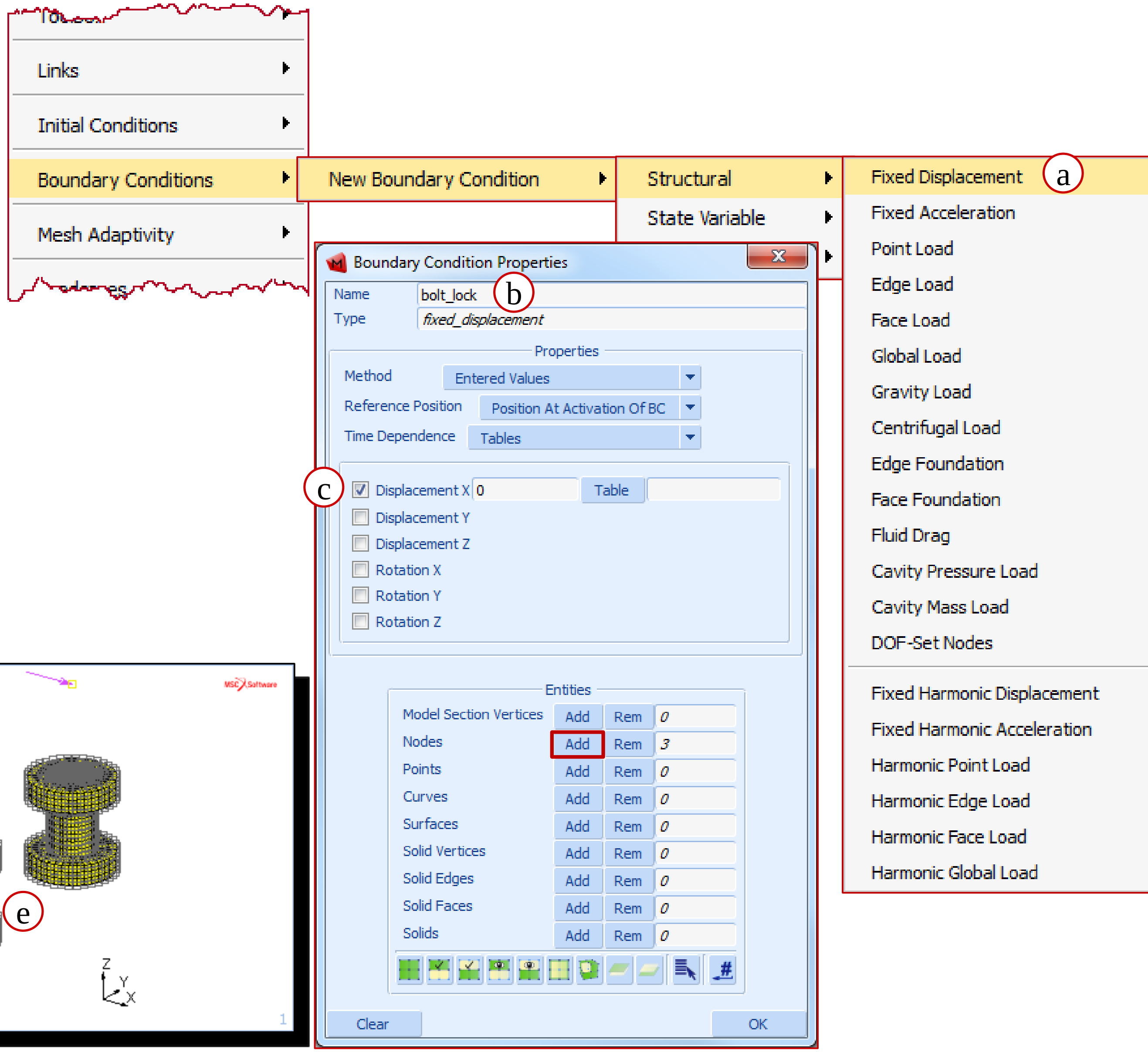


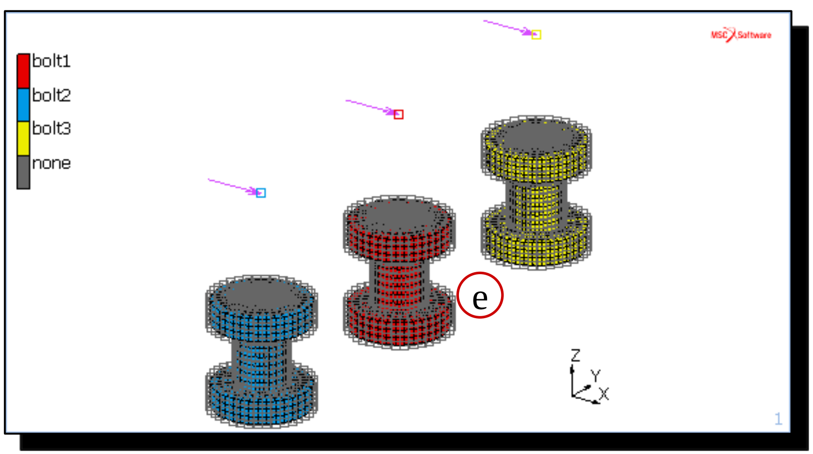
Note – Preload is defined by specifying a force on the first degree of freedom of control node. Notice that this force works in the direction of the cross section normal vector.

Step 7. Apply boundary conditions (Contd.)

Specify Bolt lock boundary condition

a. Right-click on the blank space in the Model Navigator and select: **Boundary Conditions > New Boundary Condition > Structural > Fixed Displacement.**
b. Enter **boundary condition Name:** `bolt_lock`**.**
c. Enable **Displacement X**
d. Click **Add.**
e. Select all the control nodes of the bolt.
f. Repeat step a**.**

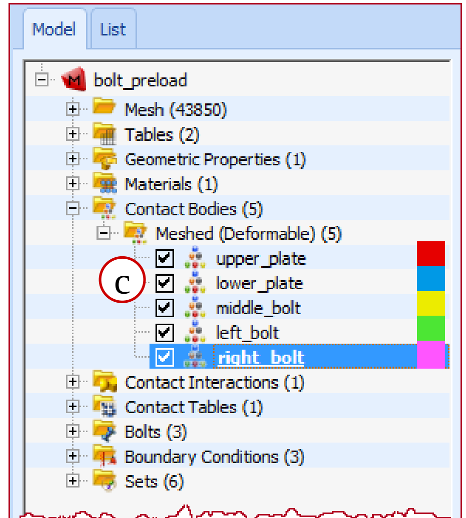


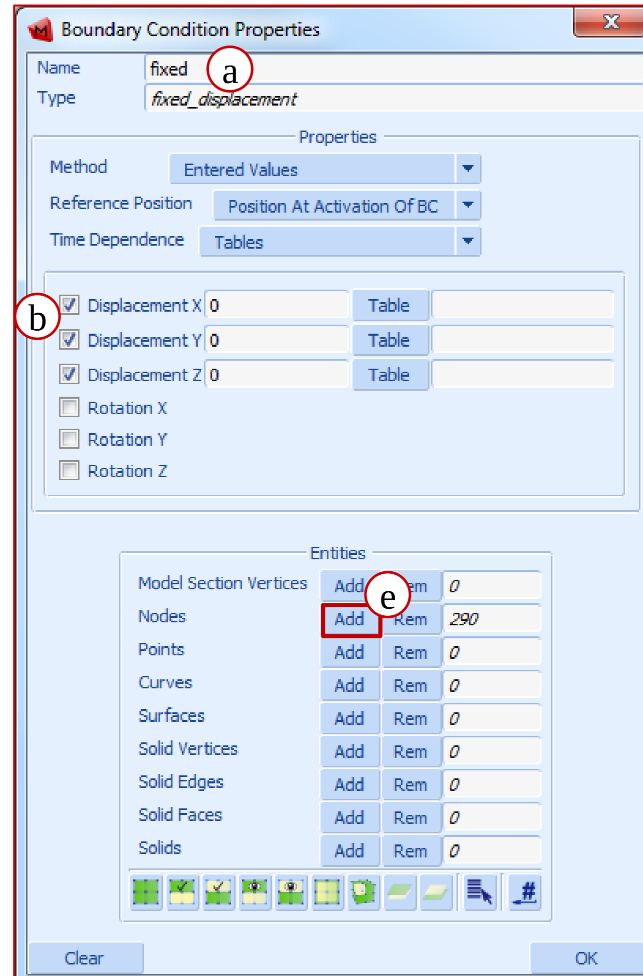


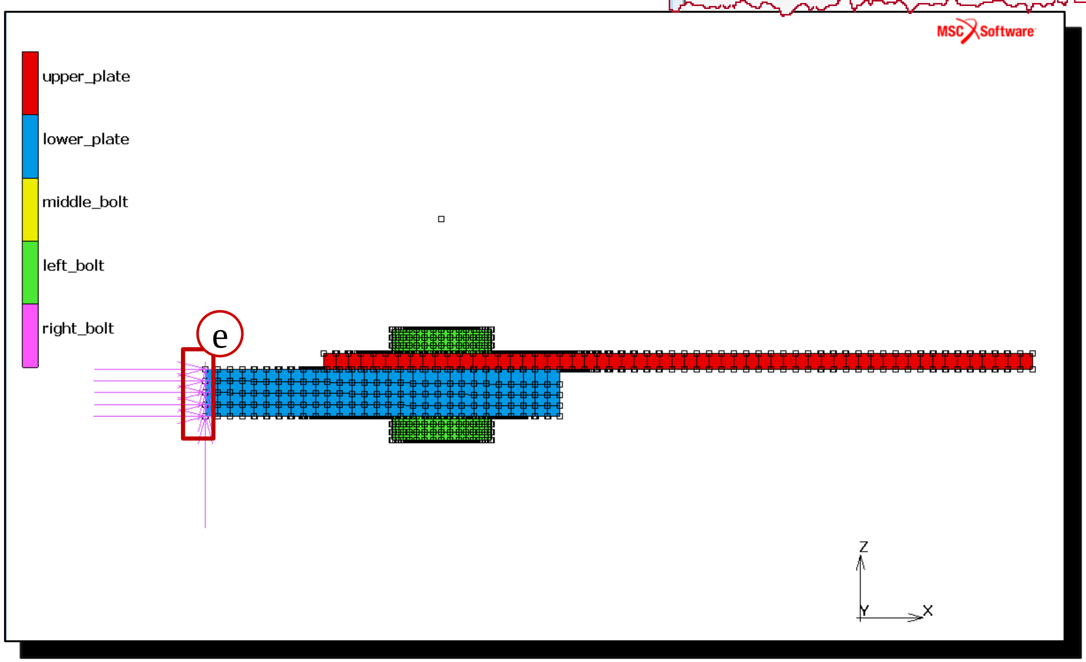
Step 7. Apply boundary conditions (Cont.)

Specify fixed boundary conditions

a. Enter `fixed` for **Name.**
b. Enable **Displacement X, Displacement Y, Displacement Z.**
c. Enable visibility of every contact bodies.
d. In the **miscellaneous** menu, change the node **picking option** from Bolt Control Node Picking to **Standard Node Picking.**
e. Click **Nodes: Add.**
f. Select nodes on left face of bottom plate.
g. Click **OK.**







Step 7. Apply boundary conditions (Cont.)

Specify point load on top plate

a. Right-click on the Model Navigator and select: **Boundary Conditions > New Boundary Condition > Structural > Point Load.**

b. Enter `external_load` for **Name.**

c. Enable **Force Z.**

d. Enter **50** for **Force Z.**

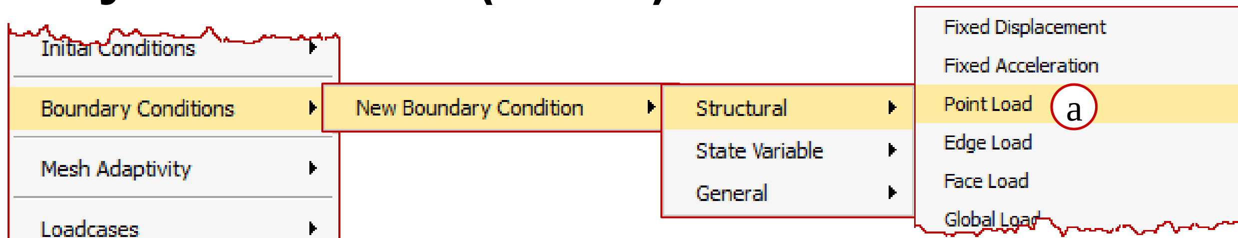
e. Click **Table.**

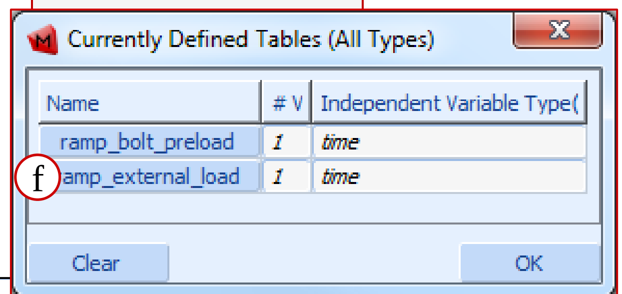
f. Select `ramp_external_load.`

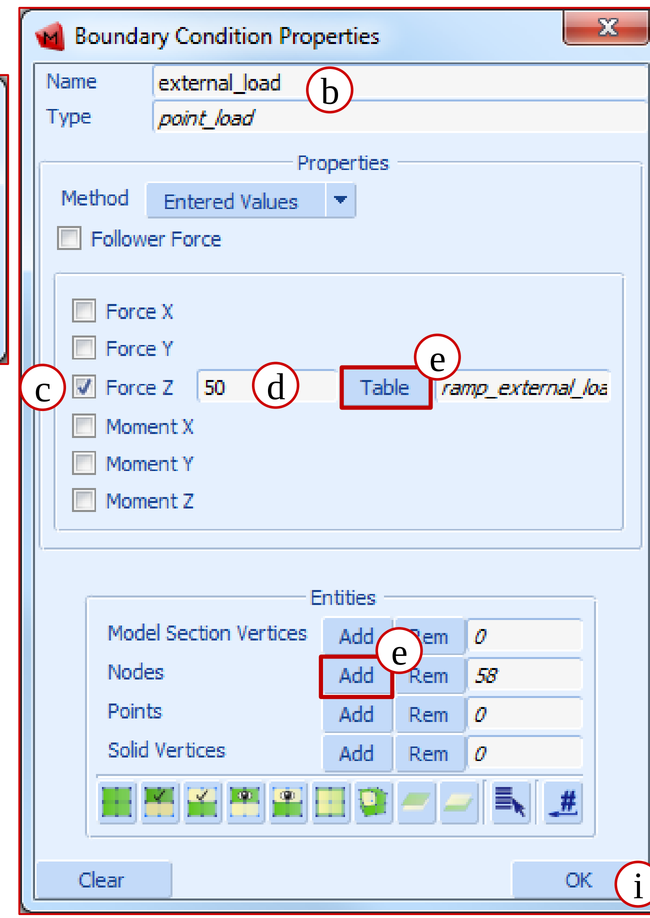
g. Click **Nodes: Add**

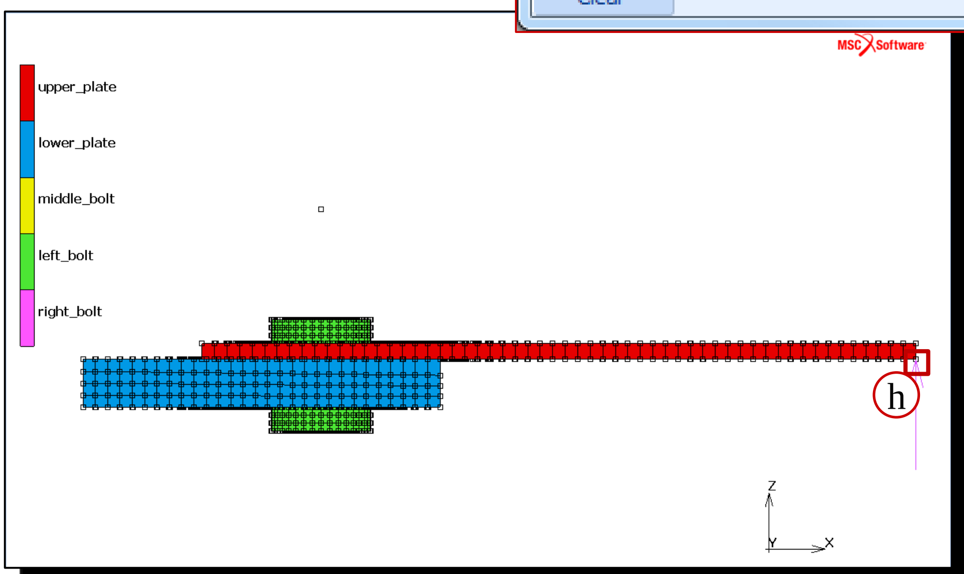
h. Select right lower edge nodes of top plate.

i. Click **OK**









Bolt Modeling – 3 Steps

Step 1
Bolt Preload

- Fixed Constraints on bottom plate
- Touching contact between all three Contact Bodies
- Bolt Pretension Load Point Load Fx

Step 2
Bolt Lock

- Fixed Constraints on bottom plate
- Touching contact between all three Contact Bodies
- Lock Bolt Control Node Displacement Constraint (x)

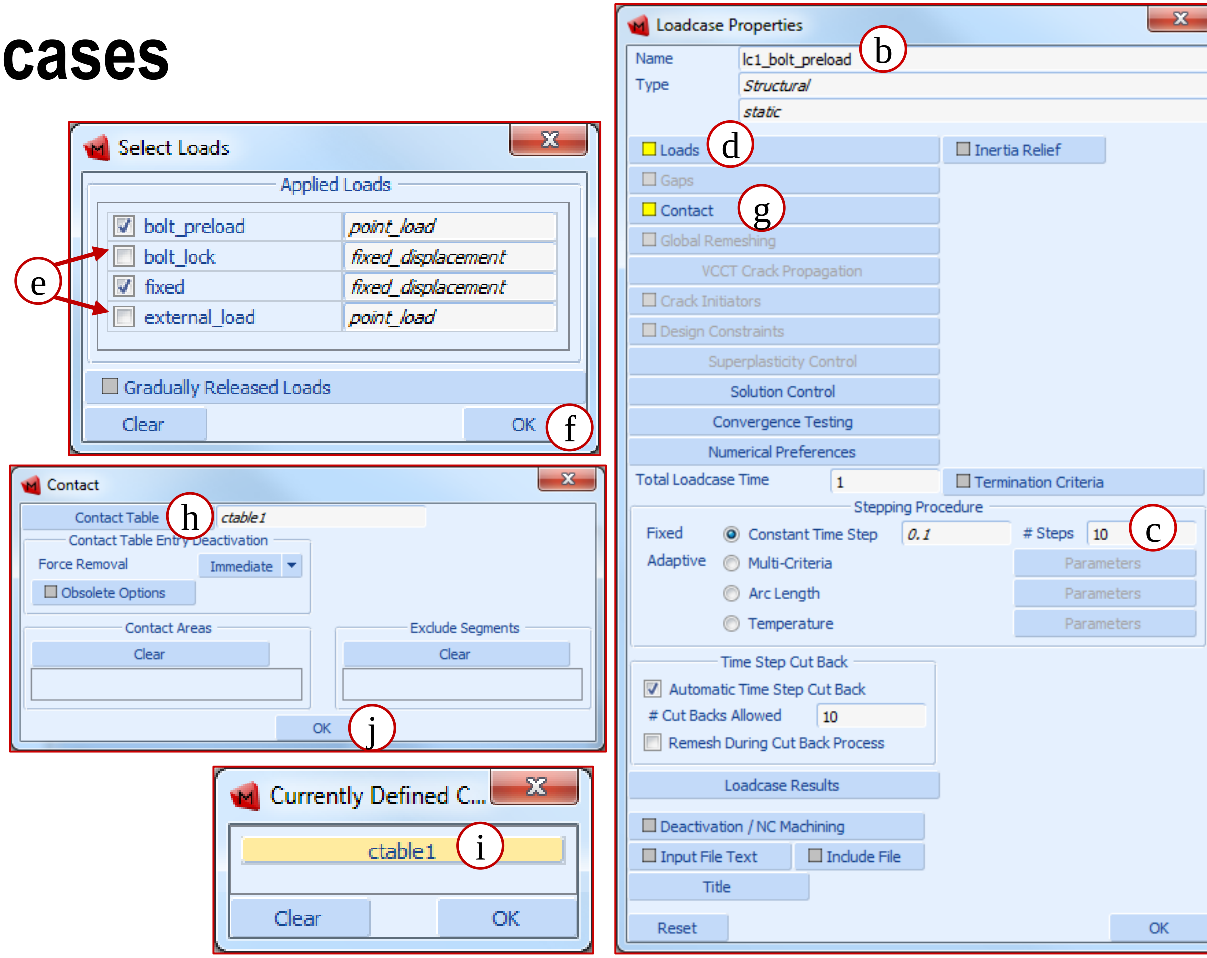
Step 3
External Load

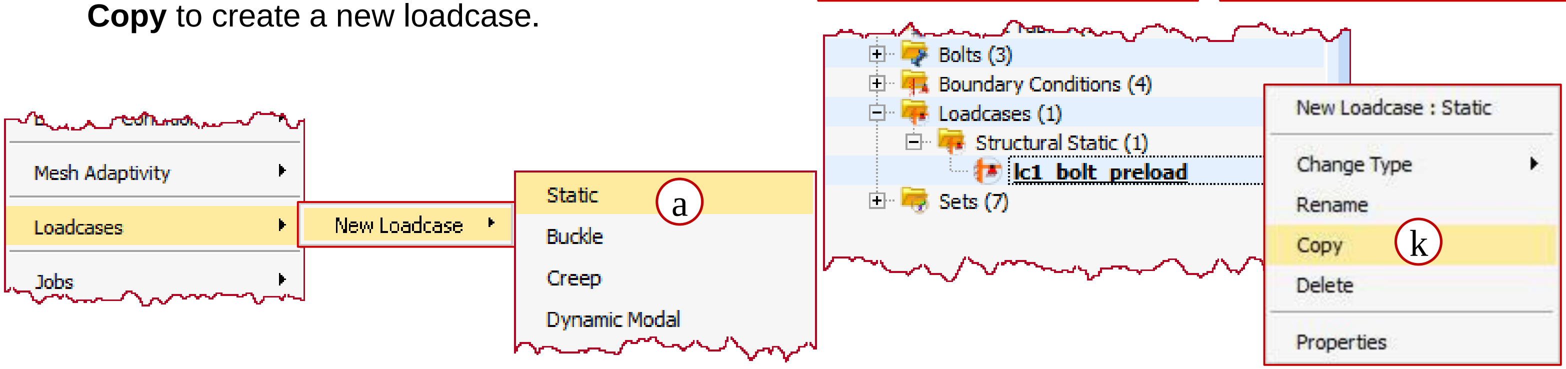
- Fixed Constraints on bottom plate
- Touching contact between all three Contact Bodies
- Lock Bolt Control Node Displacement Constraint (x)
- Apply External Load point load Fz

Step 7. Define Loadcases

Create First Loadcase to preload the bolt:

a. Right-click a blank place in the Model Navigator. Select **Loadcases > New Loadcase > Static.**
b. Enter `lc1_bolt_preload` for **Name**.
c. Specify **Number of Steps** as **10.**
d. Click **Loads**
e. Under **Applied Loads**, disable `bolt_lock` and `external_load`.
f. Click **OK.**
g. Click **Contact.**
h. Click **Contact Table.**
i. Select `ctable1`.
j. Click **OK.**
k. Right click on **lc1_bolt_preload** in the Model Navigator and select **Copy** to create a new loadcase.

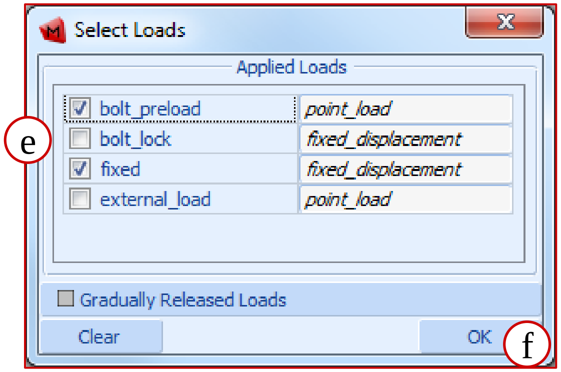


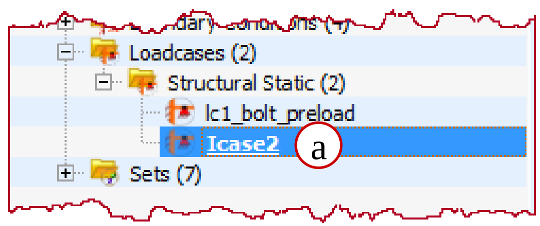


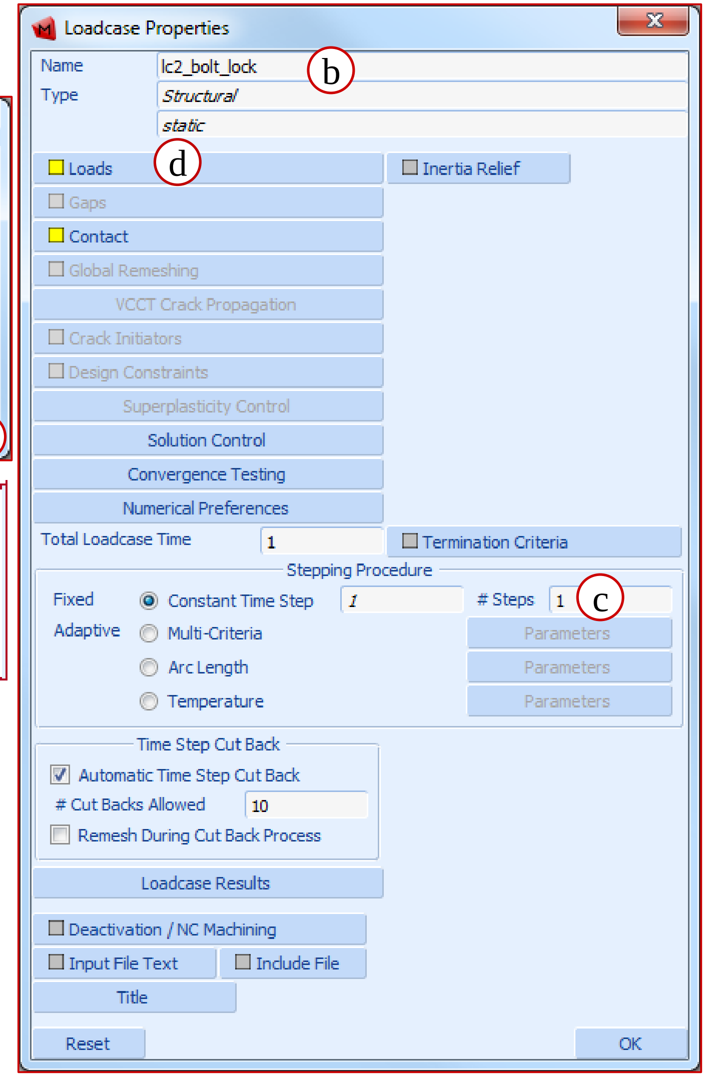
Step 7. Define Loadcase (Cont.)

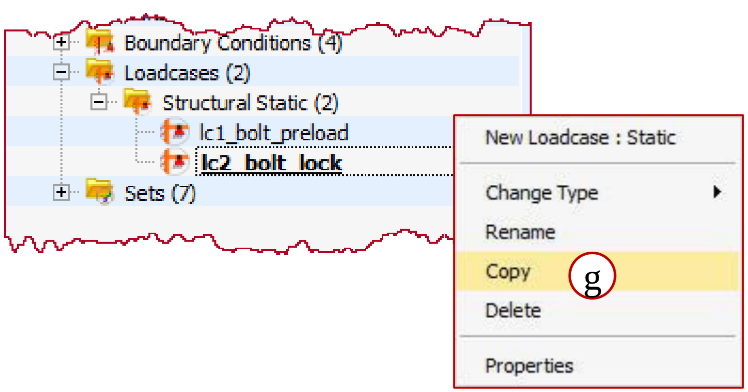
Define Second Loadcase to lock the bolt:

a. Double click on `lcase2` loadcase.
b. Enter `lc2_bolt_lock` for **Name.**
c. Specify **Number of Steps** as **1.**
d. Click **Loads.**
e. Verify `fixed` and `bolt_lock` is selected.
f. Click **OK.**
g. Right click on **lc1_bolt_preload** in the Model Navigator and select **Copy** to create a new loadcase.





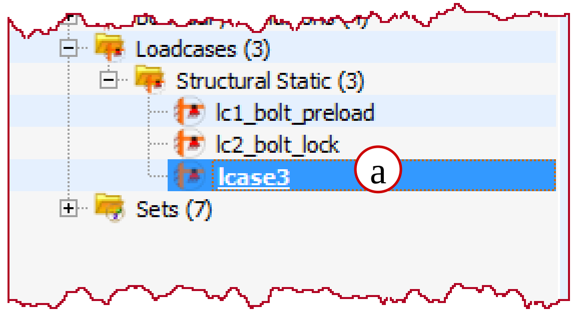


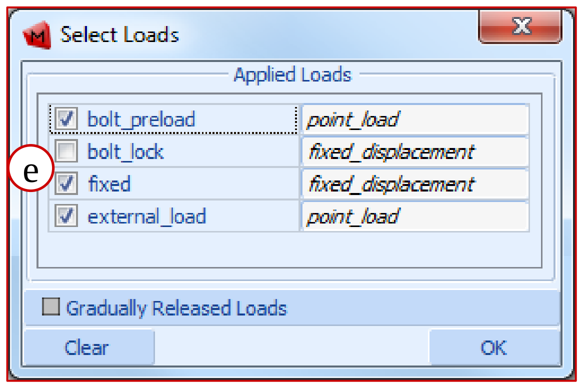


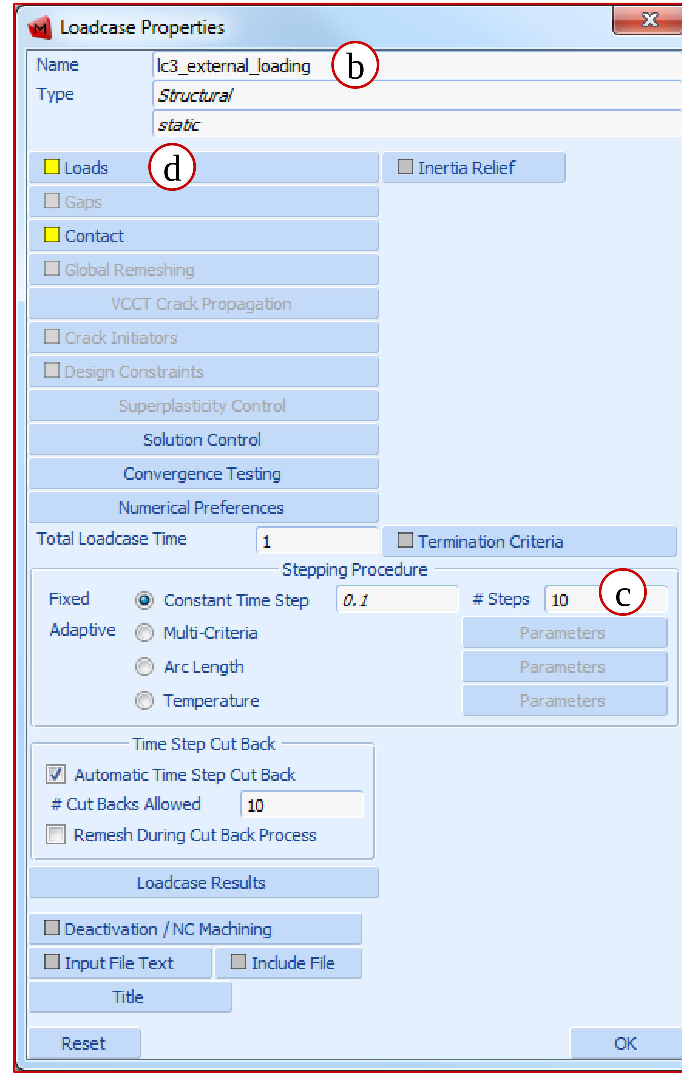
Step 7. Define Loadcase (Cont.)

Define Third Loadcase to keep the bolt locked and apply point load :

a. Double click on `Icase3` loadcase.
b. Enter `lc3_external_loading` for **Name.**
c. Specify **Number of Steps** as **10.**
d. Click **Loads.**
e. Select `fixed`, `bolt_lock and external_load.`
f. Click **OK.**
g. Click **OK.**







Step 8. Create a Job

Create a Job

a. Right-click a blank place in the Model Navigator. Select **Jobs > New Job > Structural.**

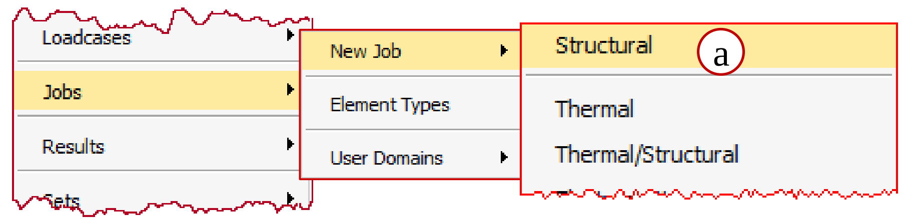
b. Select `lc1_bolt_preload, lc2_bolt_lock` and `lc3_external_loading.`

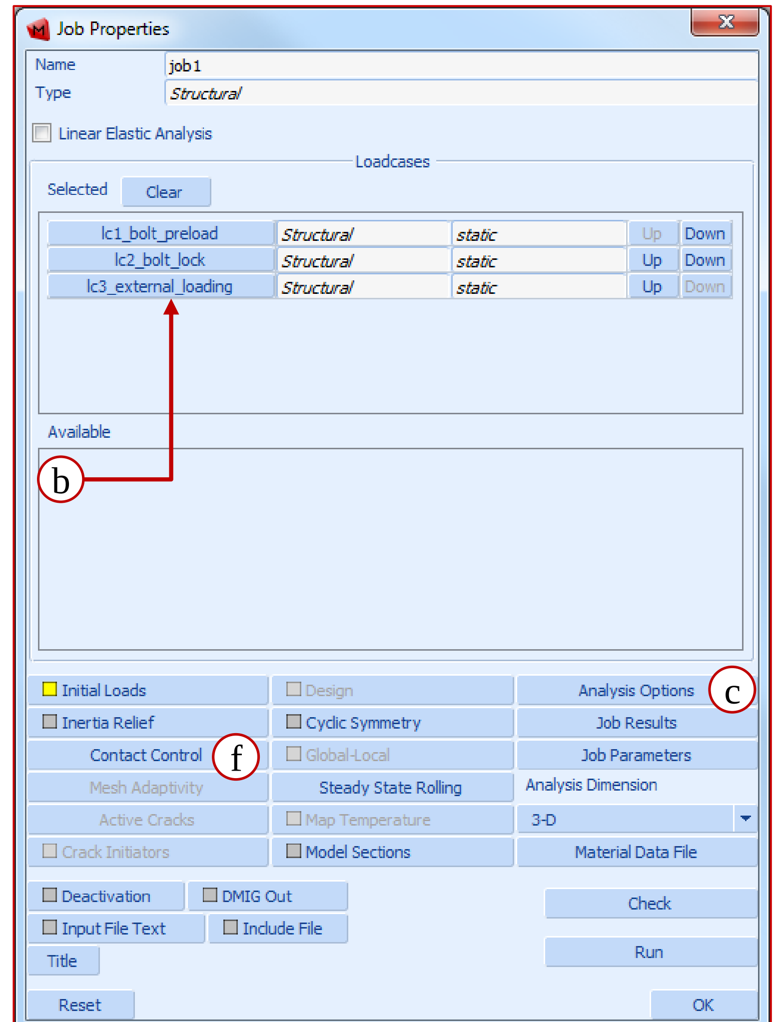
c. Click **Analysis Options.**

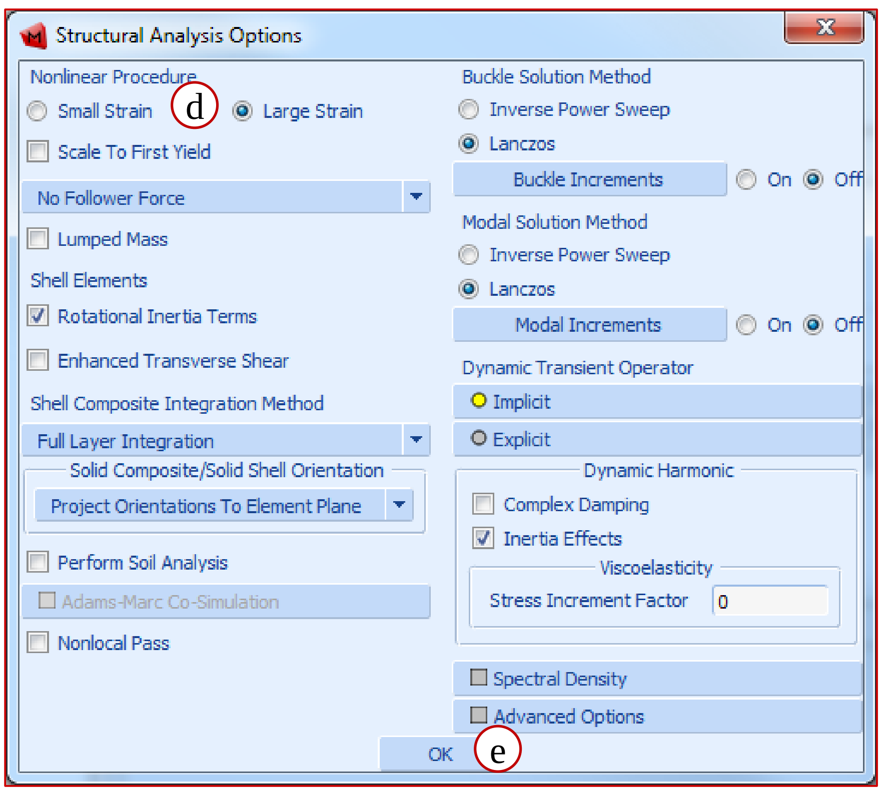
d. Click **Large Strain.**

e. Click **OK.**

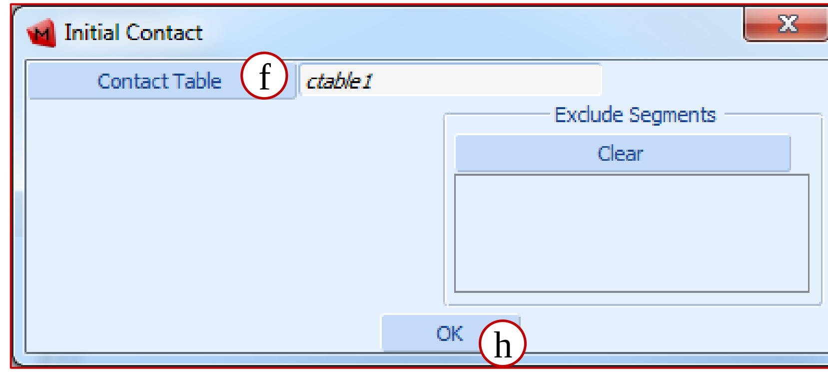
f. Click **Contact Control.**







Step 8. Create a Job (Cont.)



Define Contact Settings

a. Select **Segment To Segment** as **Method.**

b. Select **Version 2** as **Default Settings.**

c. Verify **Finite Sliding** is selected as **Model.**

d. Select **Coulomb Bilinear (Displacement)** as **Type.**

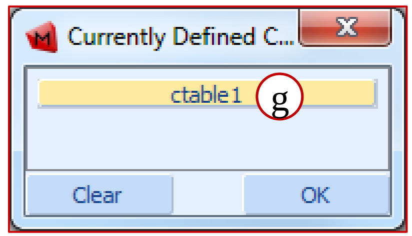
e. Click **Initial Contact.**

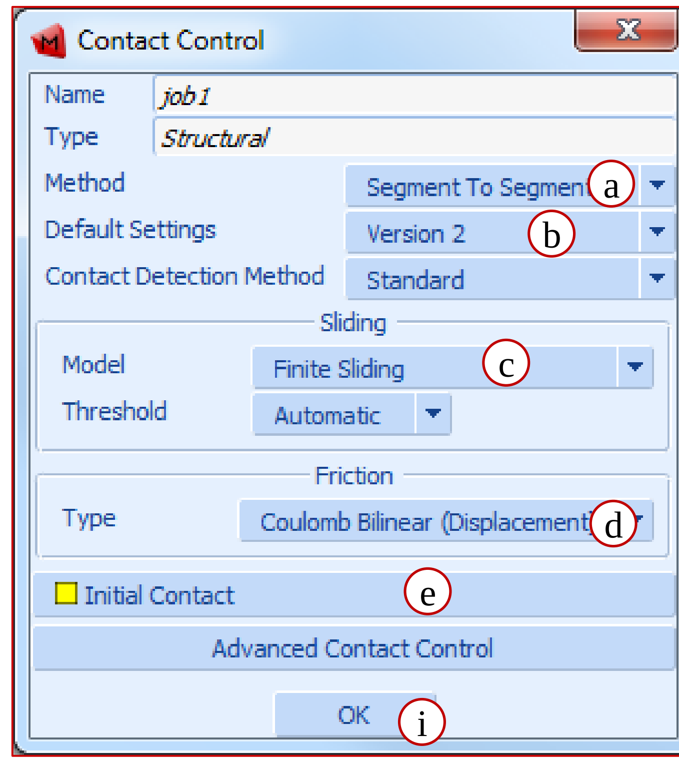
f. Click **Contact Table.**

g. Select `ctable1`**.**

h. Click **OK.**

i. Click **OK.**

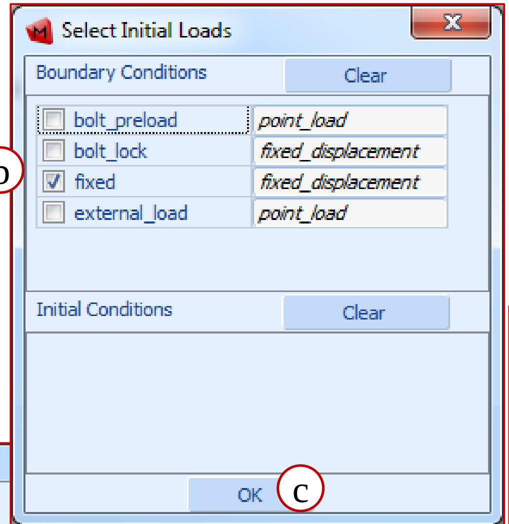


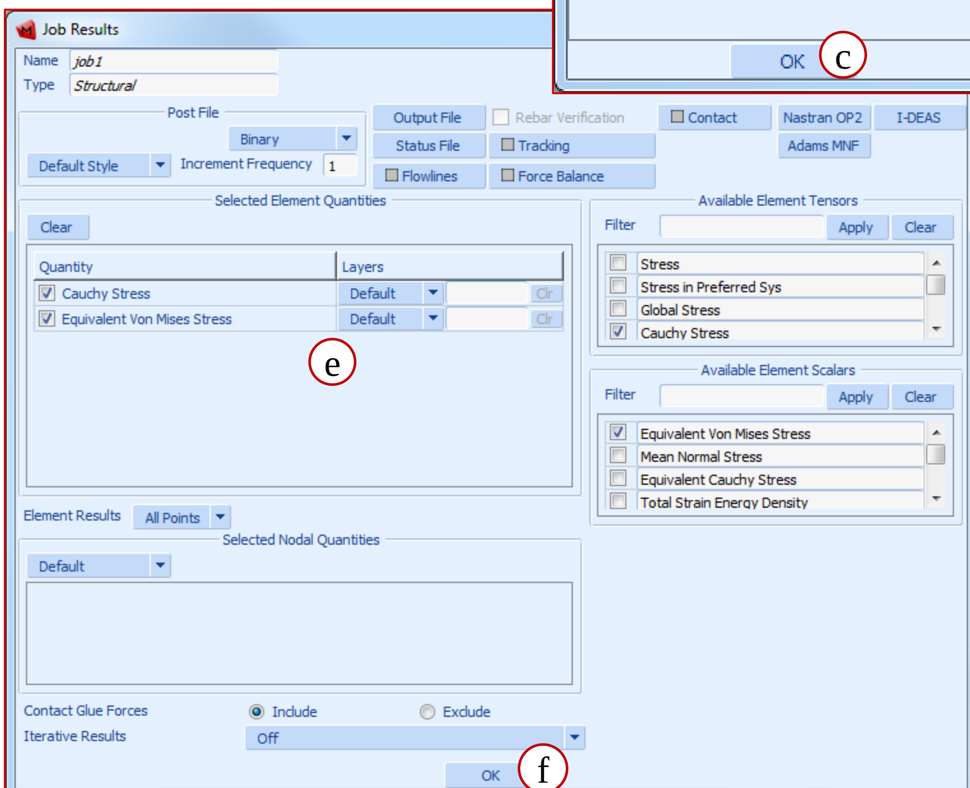


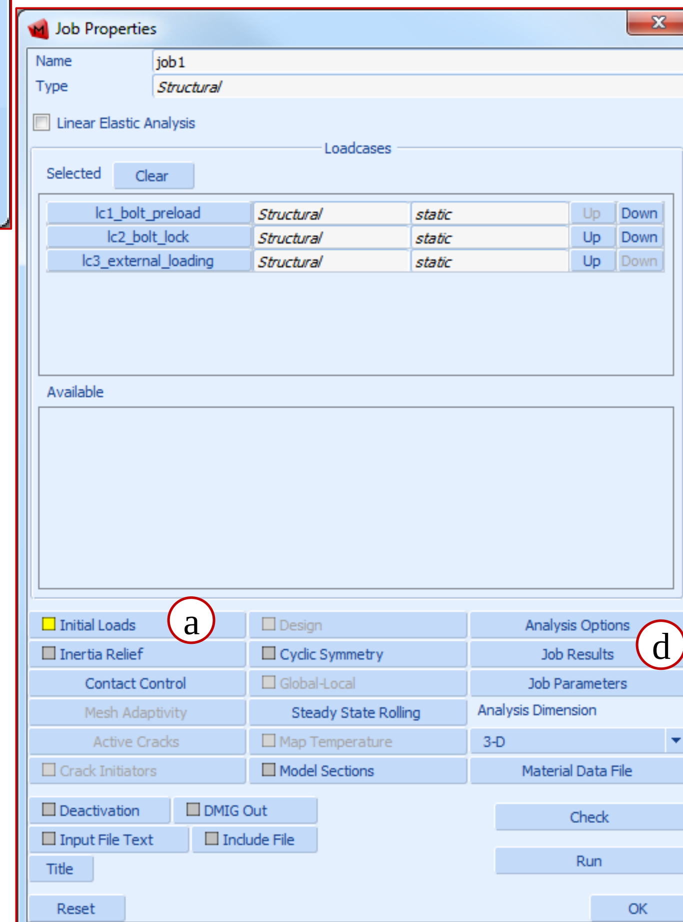
Step 8. Create a Job (Cont.)

Select initial loads

a. Click **Initial Loads.**
b. Select `Fixed`**.**
c. Click **OK.**
d. Click **Job Results.**
e. Select **Cauchy Stress** and **Equivalent Von Mises Stress** to request these outputs.
f. Click **Run.**



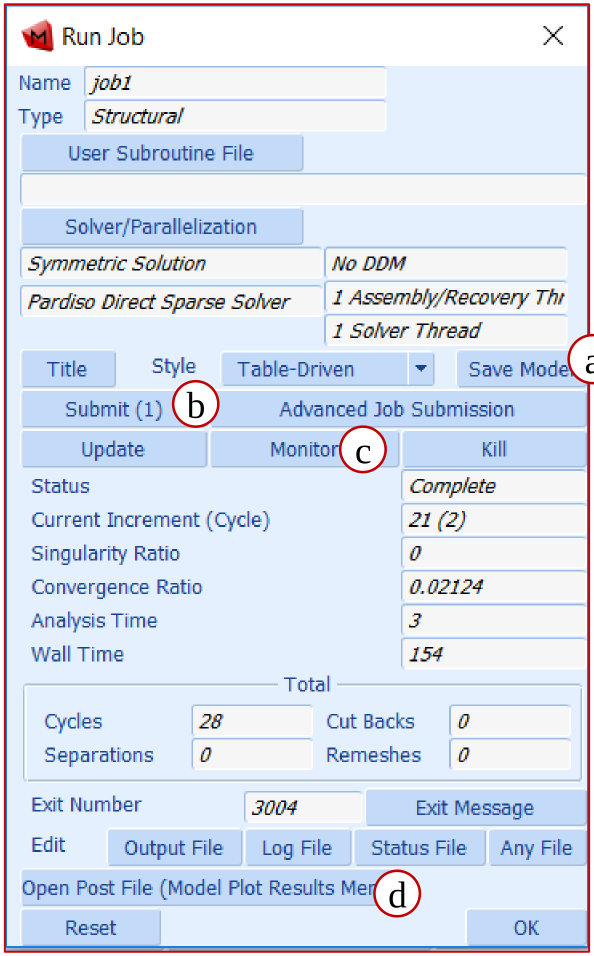




Step 9. Run the Job

Run the Analysis

a. Click **Save Model.**
b. Click **Submit (1).**
c. Click **Monitor.**
d. When status shows **Complete,** Click **Open Post File (Model Plot Results Menu).**



Step 10. Create Equivalent Von Mises Stress Contour Plot

Create a Equivalent Von Mises Stress fringe plot.

a. Under **Scalar Plot,** select **Contour Bands** as **Style.**

b. Click **Scalar**.

c. Select **Equivalent Von Mises Stress.**

d. Click the **Monitor Results** File icon.