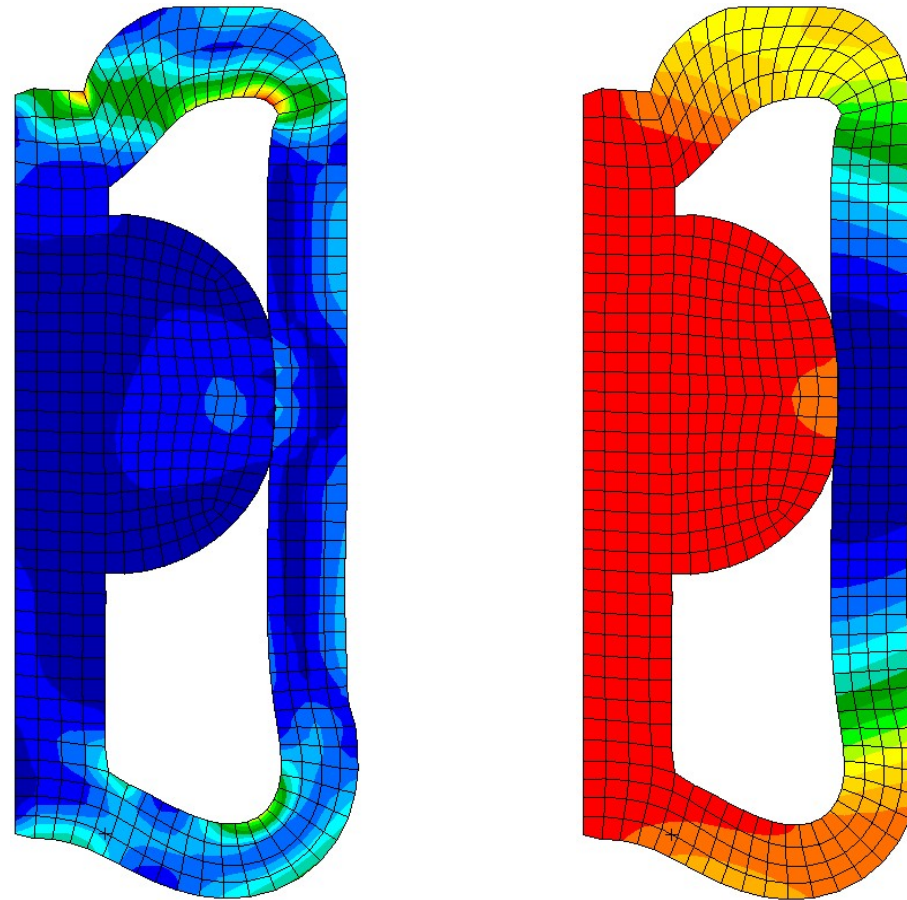
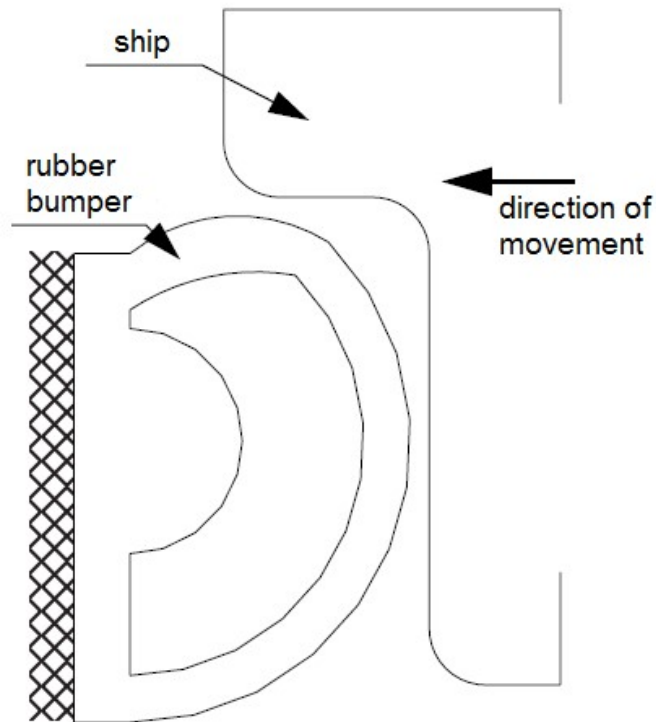


NONLINEAR MECHANICS OF STRUCTURES

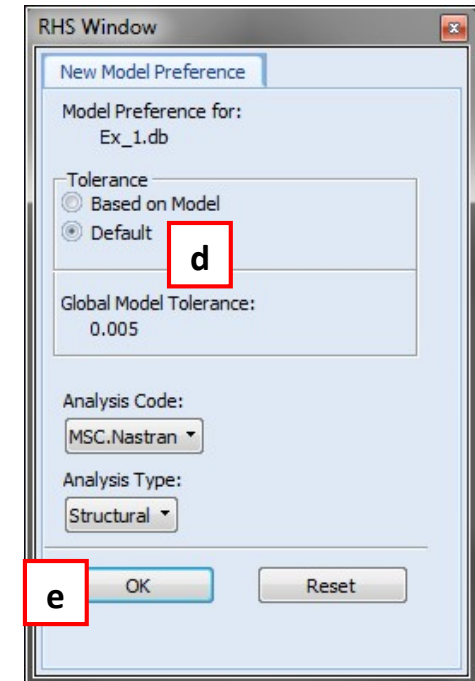
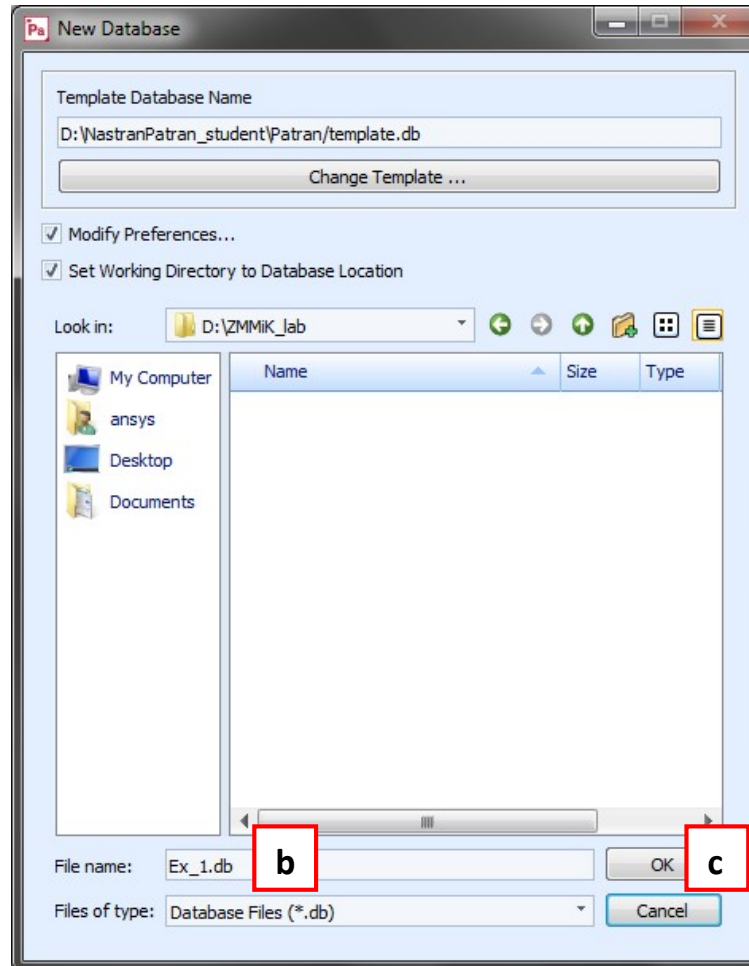
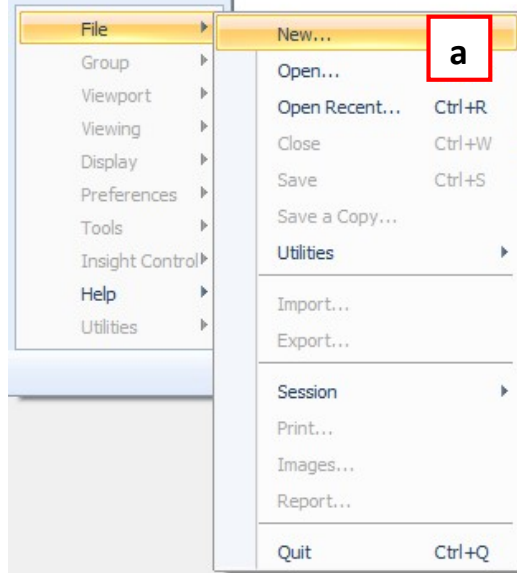
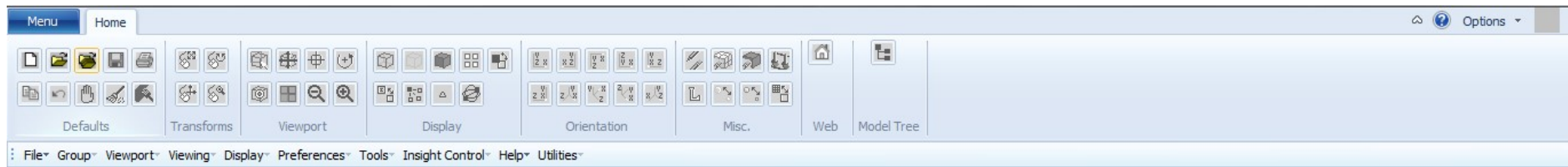
EXERCISE 1



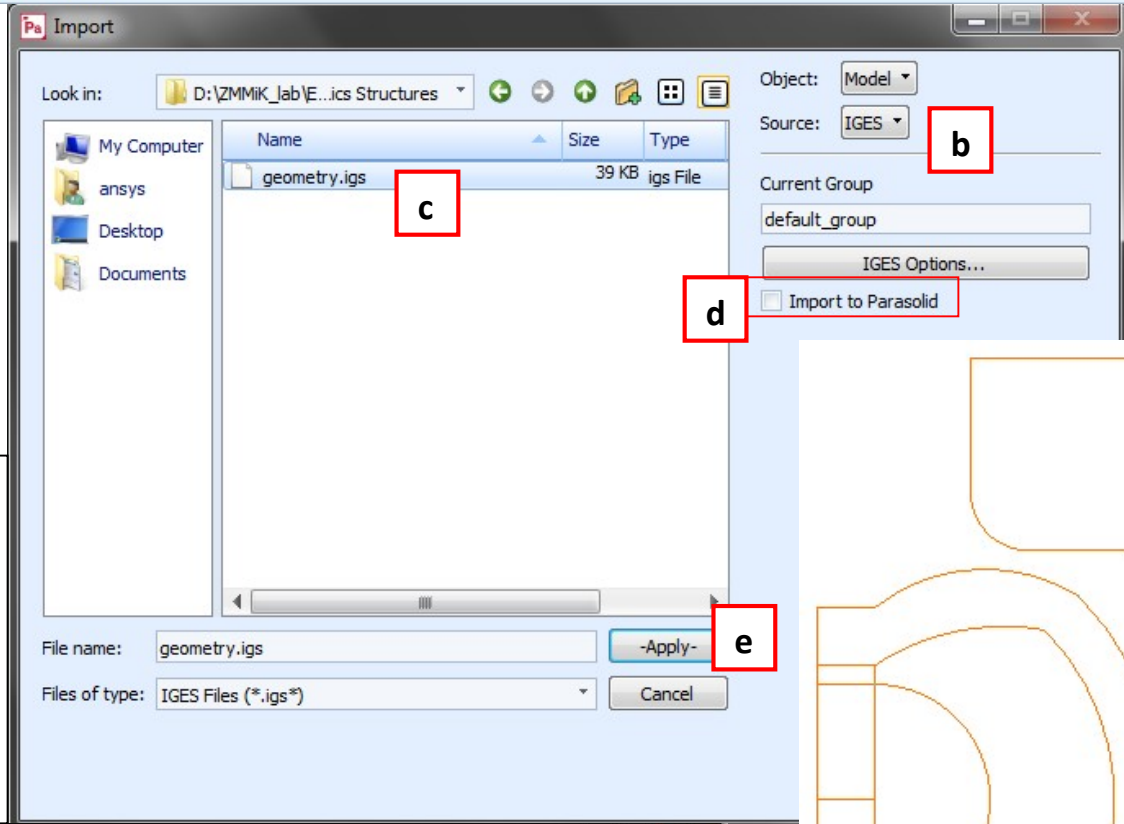
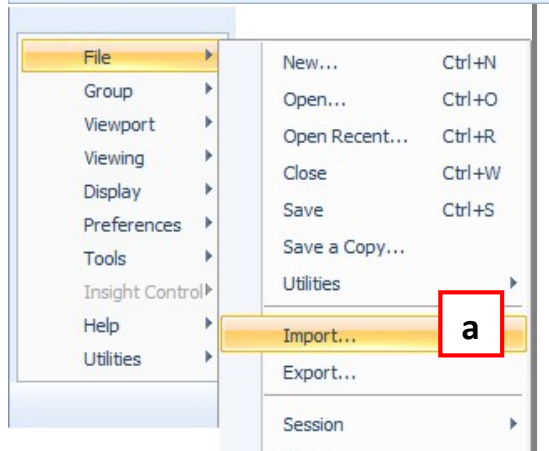
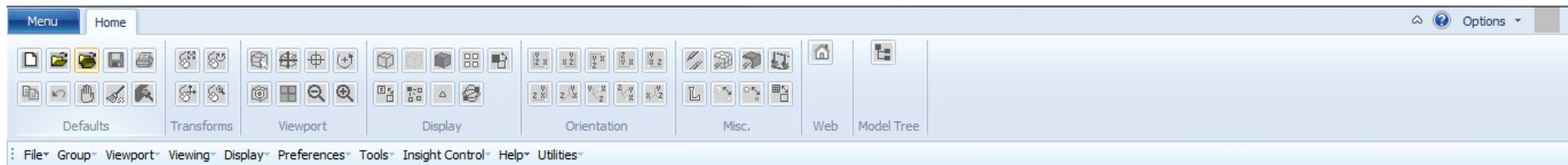
PROBLEM DESCRIPTION



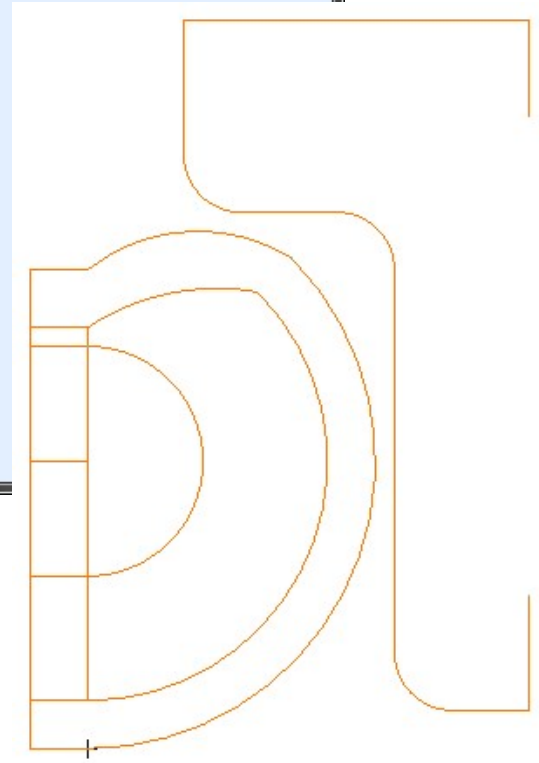
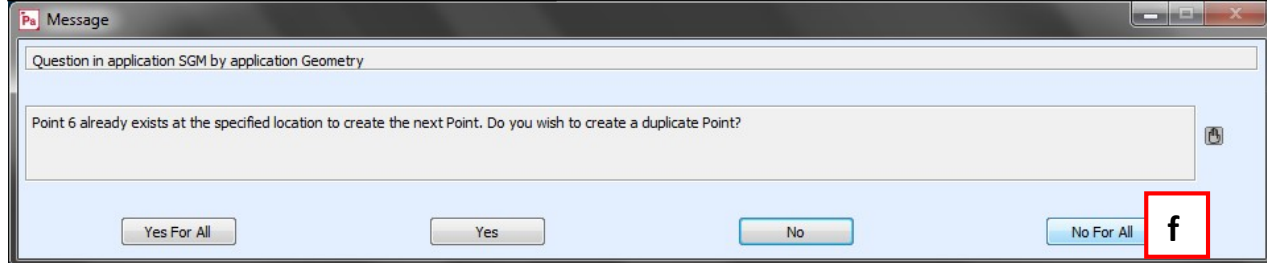
A rubber bumper comes into a contact with a ship. The geometry of the parts is described in an iges CAD file that is imported to Patran for mesh generation. The bumper is assumed to be in a state of plane strain. Contact between the bumper and the ship includes friction.



- Create a new database:
- a. File / New...
 - b. Enter **ex_1.db** as the File name
 - c. Click **OK**
 - d. Select **Default**
 - e. Click **OK**



- Import the geometry:
- a. **File / Import...**
 - b. **Source: IGES**
 - c. Find and select the **geometry.igs** file
 - d. Uncheck **Import to Parasolid**
 - e. Click **Apply**
 - f. Click **No For All**
 - g. IGES Import Summary: click **OK**



The image shows a CAD software interface with a model of a bumper-like structure. The model consists of several curves labeled C1 through C15. The 'RHS Window' panel is open, showing the 'Geometry' tab with various settings. Red boxes labeled 'a' through 'g' highlight specific UI elements and actions.

a: Change Background Color to **Black** (click on the **Cycle Background** icon)

b: Click on the **Point size** icon

Create surfaces (defining the bumper):

c: Click on the **Geometry** icon/**Select/Curve**

d: Option: **2 Curve**

e: Uncheck **Auto Execute**

f: Select the **C1** curve as the starting curve and the **C2** curve as the ending curve

g: Click **Apply**

h: Create nine more surfaces using remaining curves – start from the pair: **C2 & C3**

Remark: As the starting curve select always a curve with lower id. Ids are shown in figure on the right).

a

b

c

d

e

f

g

h

i

a. Click on the **Home/Smooth shaded** icon

Mesh the surfaces:

b. Click on the **Meshing/Surface** icon (*Meshers tab*)

c. Check out: **Create/Mesh/Surface**

d. Elem Shape: **Quad**; Mesher: **IsoMesh**; Topology: **Quad4**

e. Click on the **Surface List** panel

f. Select the surfaces by clicking and dragging the mouse

g. Uncheck **Automatic Calculation**

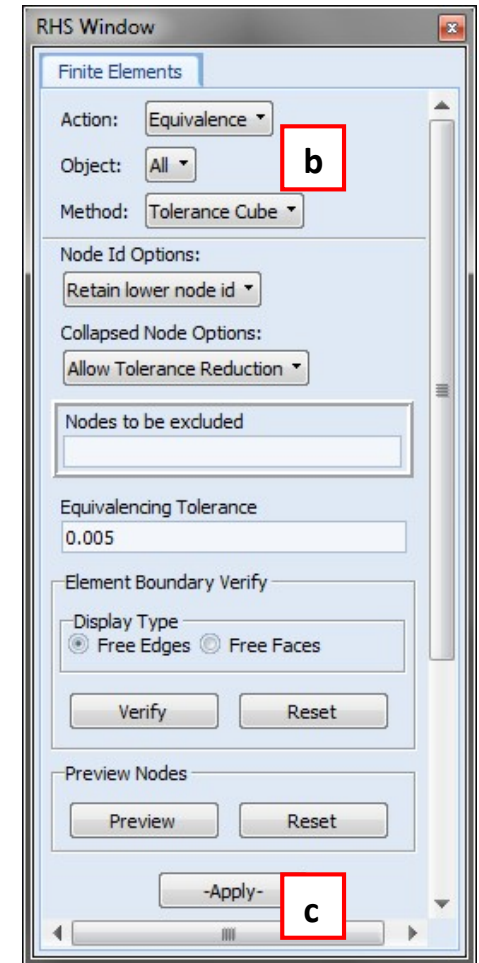
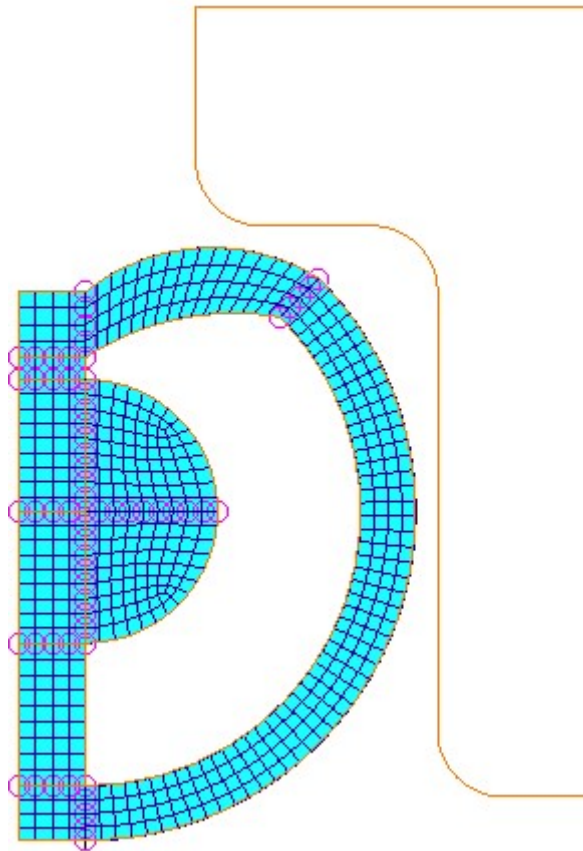
h. Enter **0.7** as the Value of the Global Edge Length

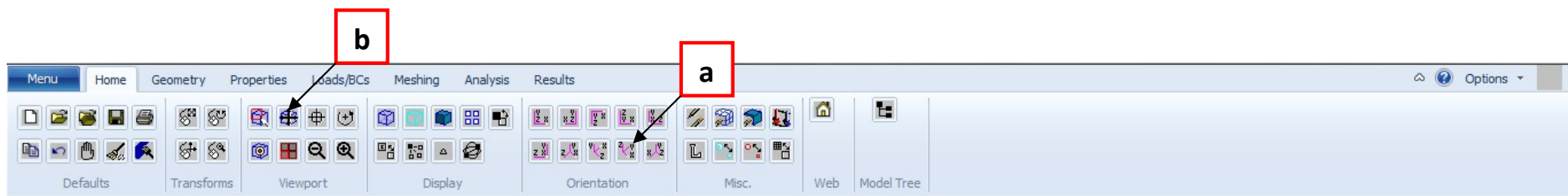
i. Click **Apply**



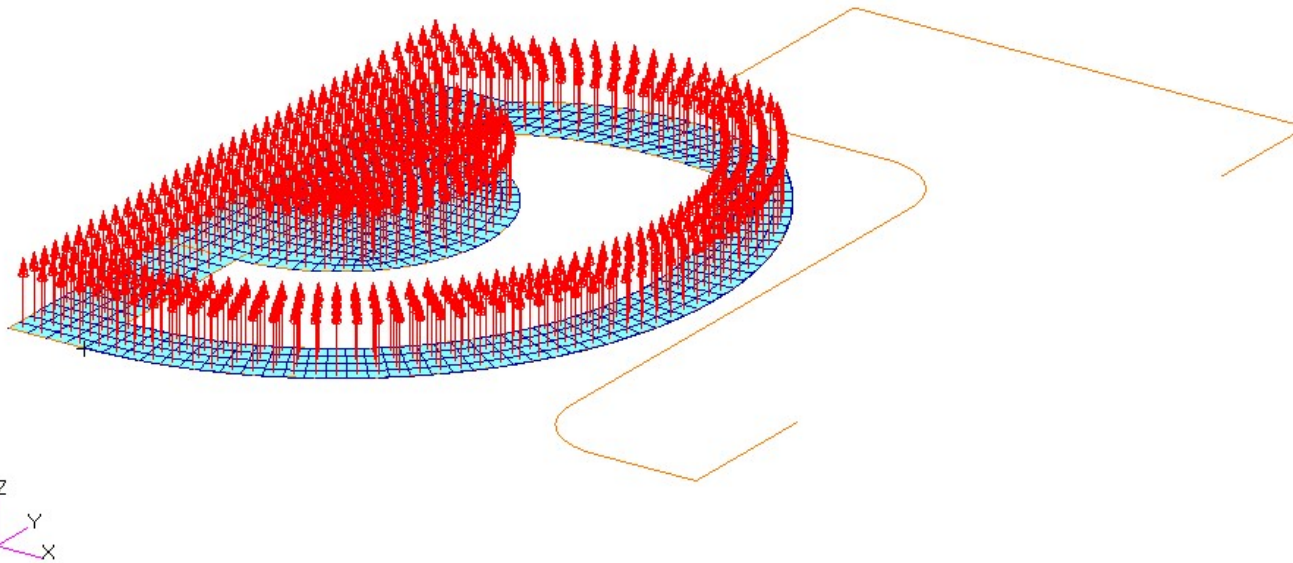
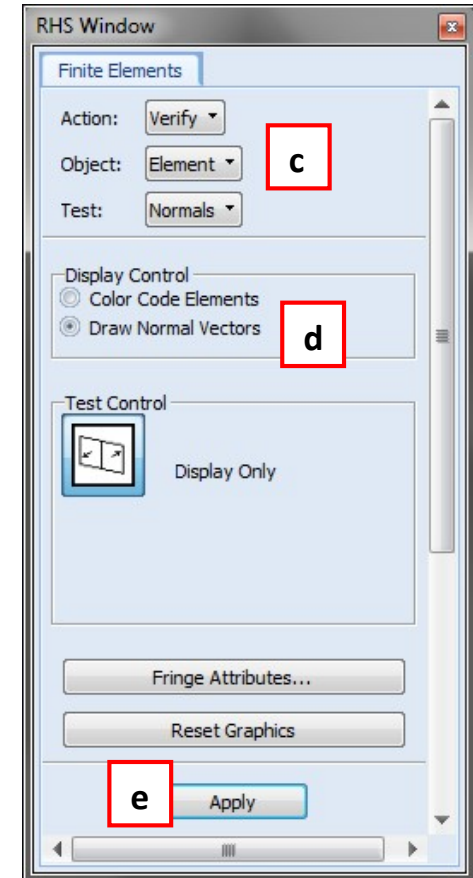
Delete duplicate nodes:

- a. Meshing: **Equivalence** icon
- b. Check out: **Equivalence/All/Tolerance Cube**
- c. Click **Apply**





- a. Click on the **Iso 3 View** icon
 b. Click on the **Fit view** icon
- Verify element normals:
- c. Elements: **Verify/Element/Normals**
 - d. Select **Draw Normal Vectors**
 - e. Click **Apply**
- Remark: Make sure that all normal vectors of the elements are pointed towards the positive direction of Z-axis. If not, use **Elements: Modify/Element/Reverse** to reverse element normals.



The screenshot shows the ANSYS Workbench software interface. The ribbon menu at the top includes 'Menu', 'Home', 'Geometry', 'Properties', 'Loads/BCs', 'Meshing', 'Analysis', and 'Results'. The 'Loads/BCs' tab is active, showing various icons for applying boundary conditions. The 'Displacement Constraint' icon is highlighted with a red box labeled 'c'. The 'Input Data' dialog box is open, showing the 'Load/BC Set Scale Factor' set to 1. The 'Translations <T1 T2 T3>' and 'Rotations <R1 R2 R3>' fields are both set to '<0 0 0>', with the 'Rotations' field highlighted by a red box labeled 'g'. The 'Spatial Fields' section is empty. The 'Analysis Coordinate Frame' is set to 'Coord 0'. The 'OK' button is highlighted with a red box labeled 'h'. The 'RHS Window' on the right shows the 'Load/Boundary Conditions' tab. The 'Action' is set to 'Create', the 'Object' is 'Displacement', and the 'Type' is 'Nodal', with the 'Displacement' dropdown highlighted by a red box labeled 'd'. The 'Option' is 'Standard'. The 'Current Load Case' is 'Default...'. The 'Type' is 'Static'. The 'Existing Sets' list is empty. The 'New Set Name' field contains 'fix', highlighted by a red box labeled 'e'. The 'Input Data...' button is highlighted by a red box labeled 'f'. The '-Apply-' button is at the bottom.

a. Click on the **Front view** icon
b. Click on the **Fit view** icon

Apply the boundary conditions:

c. Click on the **Loads/BCs** icon/**Displacement Constraint** icon
d. Check out: **Create/Displacement/Nodal**
e. Enter **fix** as the New Set Name
f. Click **Input Data...**
g. Enter **<0,0,0>** for the Translations and for the Rotations
h. Click **OK**

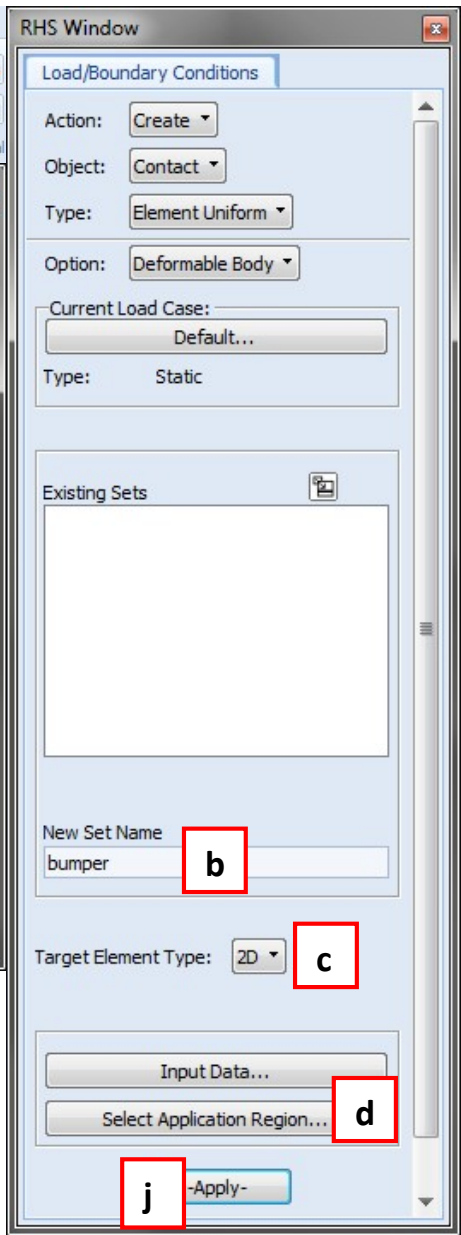
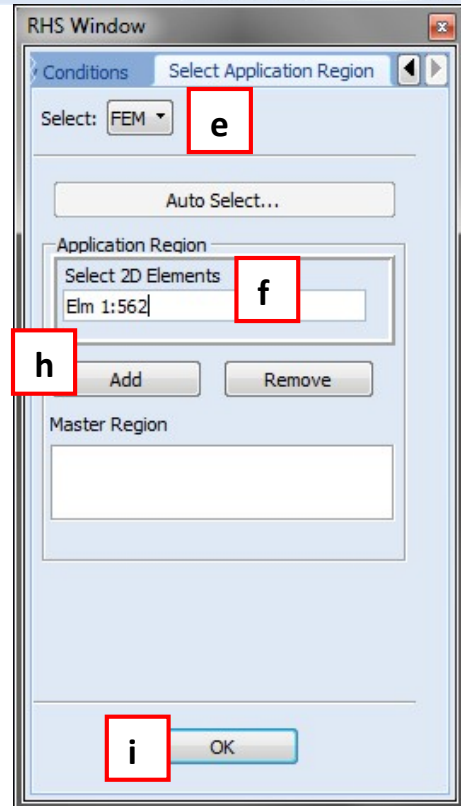
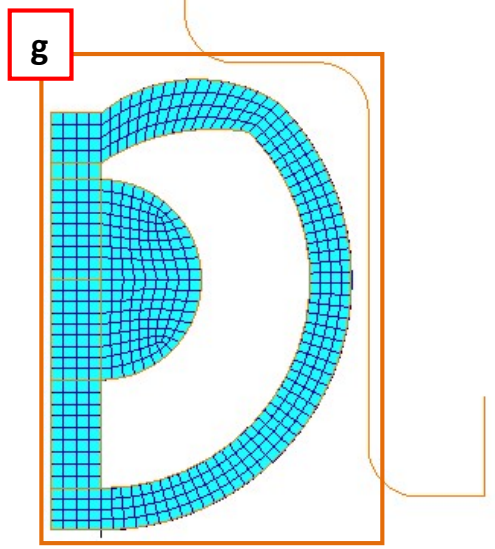
The image shows the ANSYS software interface with a meshed bumper model. A red box labeled 'l' highlights the left edge of the bumper. Two dialog boxes are open: 'Select Application Region' and 'Load/Boundary Conditions'. In the 'Select Application Region' dialog, 'FEM' is selected (j), 'Select Nodes' is active (k), 'Add' is highlighted (m), and 'OK' is highlighted (n). In the 'Load/Boundary Conditions' dialog, 'Create' is highlighted (i) and '-Apply-' is highlighted (o).

- i. Click **Select Application Region...**
- j. Select **FEM**
- k. Click on the **Select Nodes** panel
- l. Select the nodes (belonging to the left free edge of the bumper) by clicking and dragging the mouse
- m. Click **Add**
- n. Click **OK**
- o. Click **Apply**



Define the contact between the rigid body (ship) and the rubber element (bumper):

- Loads/BCs: **Deformable** icon
- Enter **bumper** as the New Set Name
- Target Element Type: **2D**
- Click **Select Application Region...**
- Select **FEM**
- Click on the **Select 2D Elements** panel
- Select the elements by clicking and dragging the mouse
- Click **Add**
- Click **OK**
- Click **Apply**



The image shows the ANSYS Workbench software interface. At the top is a ribbon menu with tabs for Menu, Home, Geometry, Properties, Loads/BCs, Meshing, Analysis, and Results. Below the ribbon are various tool icons for applying loads and boundary conditions. On the left, a blue meshed model of a ship is shown. Three 'RHS Window' panels are open:

- Left Panel:** Shows 'Conditions' and 'Select Application Region'. The 'Geometry Filter' is set to 'Geometry'. The 'Application Region' section has 'Select Curves' highlighted. An 'Add' button is visible.
- Middle Panel:** Shows 'Load/Boundary Conditions' and 'Input Data'. The 'Motion Control' is set to 'Velocity'. The 'Velocity (vector)' field contains '<-2, 0., 0.,>'. The 'Friction Coefficient (MU)' field contains '0.2'. An 'OK' button is at the bottom.
- Right Panel:** Shows 'Load/Boundary Conditions' and 'Input Data'. The 'Action' is 'Create', 'Object' is 'Contact', and 'Type' is 'Element Uniform'. The 'Option' is 'Rigid Body'. The 'Current Load Case' is 'Default...'. The 'Type' is 'Static'. The 'New Set Name' field contains 'ship'. The 'Target Element Type' is '1D'. An 'Input Data...' button is visible.

A text box on the left contains the following instructions:

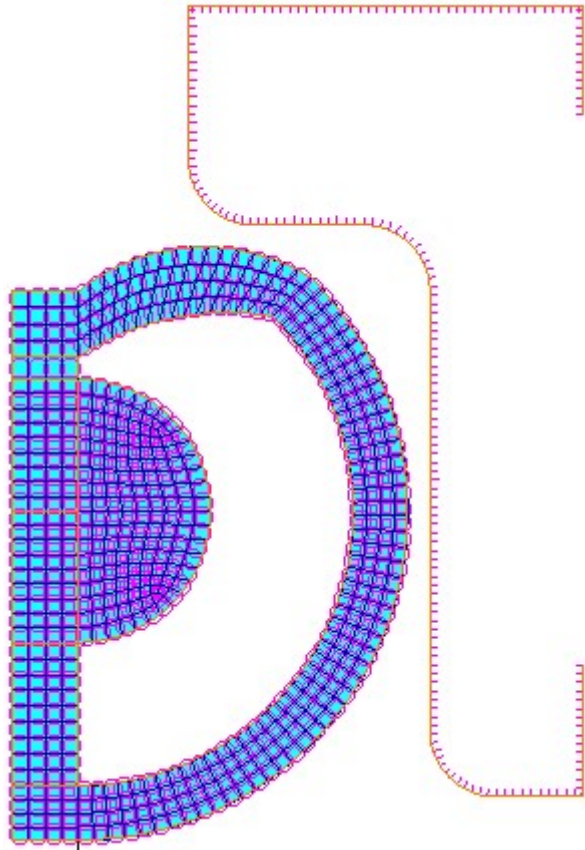
- Loads/BCs: **Rigid** icon
- Enter **ship** as the New Set Name
- Target Element Type: **1D**
- Click **Input Data...**
- Motion Control: **Velocity**
- Enter **<-2 0 0>** as the Velocity (vector)
- Enter **0.2** as the Friction Coefficient (see p. 24)
- Click **OK**
- Click **Select Application Region...**
- Geometry Filter: **Geometry**
- Click on the **Select Curves** panel
- Select the curves defining the ship (click on one of them, then press the **Shift Key** and holding it down select all other curves)
- Click **Add**
- Click **OK**
- Click **Apply**

Verify your model:

a. Check if your model looks like the one shown on the right

Remark: If you find out that the arrows point the wrong way (i.e. pointing away from the ship) after creating the rigid contact curves, modify this LBC set (**Modify/Contact/Element Uniform**):

- Select **ship**
- Click **Modify Data...**
- Toggle **Flip Contact Side** to **ON**
- Accept the modifications



RHS Window

Load/Boundary Conditions Input Data

Flip Contact Side

Symmetry Plane

Null Initial Motion

Motion Control : Velocity ▾

Velocity (vector)

<-2., 0., 0.>

Angular Velocity (rads/time)

Velocity vs. Time Field

Friction Coefficient (MU)

0.2

Subdivisions for Curves

Temperature-Dependent Fields

Rotation Reference Point

Axis of Rotation

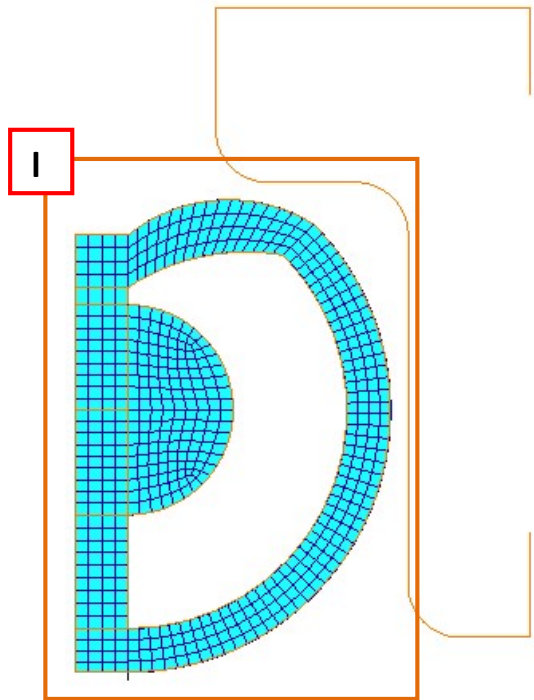
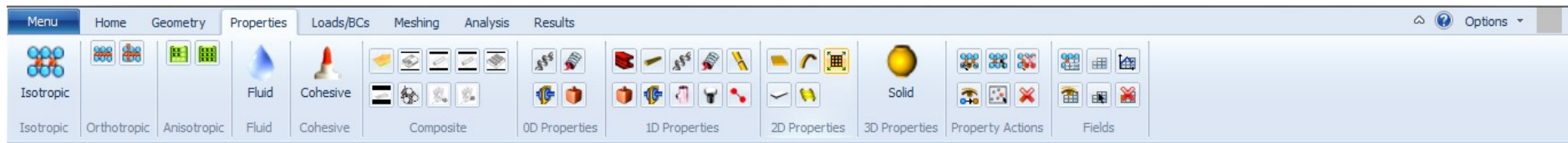
OK Reset

Define a material:

- Click on the Properties: **Isotropic** icon
- Enter **rubber** as the Material Name
- Click **Input Properties...**
- Change *Constitutive Model* to **Hyperelastic**
Change *Data Type* to **Coefficients**
Select **Mooney Rivlin MATHE**
- Enter **0.84** as the C10 coefficient and **0.21** as the C01 coefficient
- Click **OK**

Assign the properties:

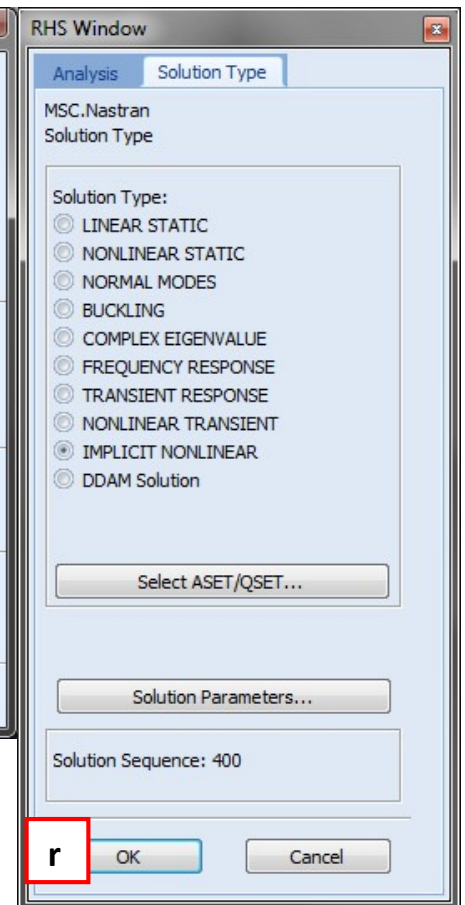
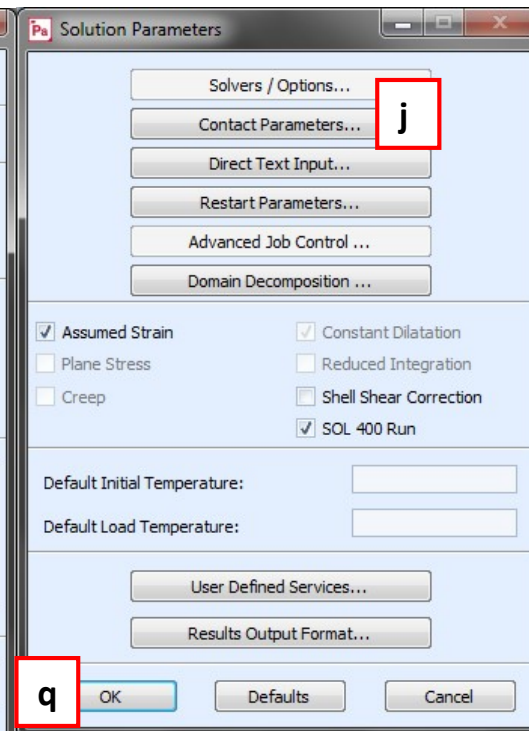
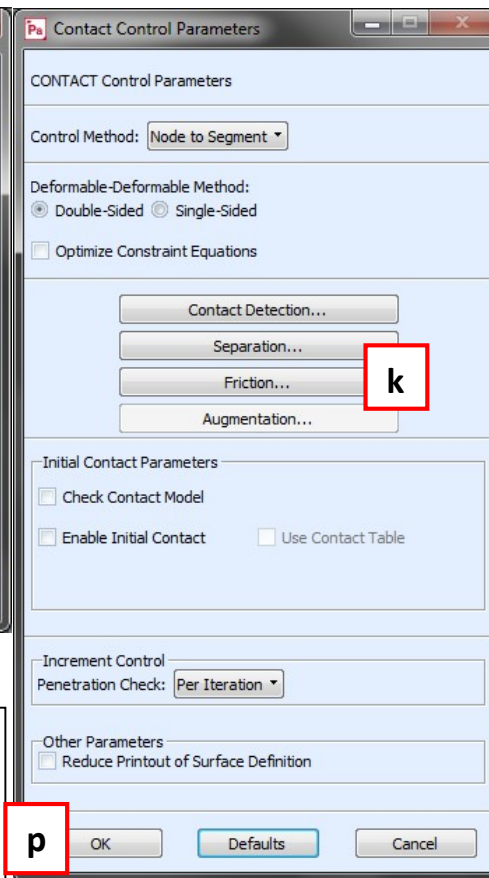
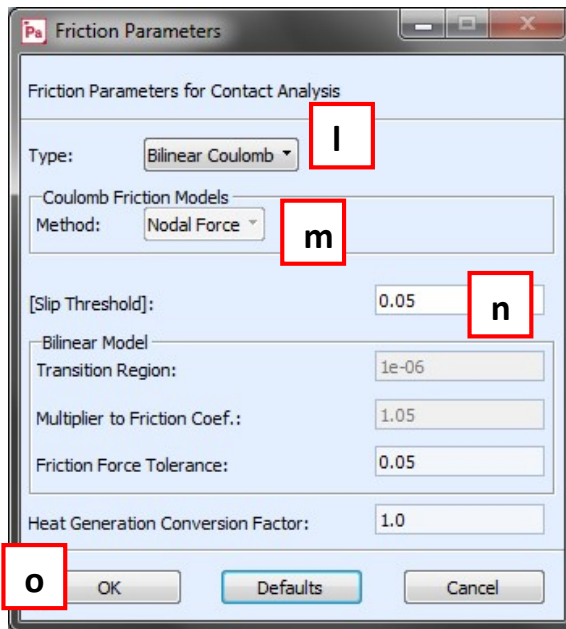
- Properties: **2D Solid** icon
- Enter **bumper** as the New Set Name
- Options: **Plane Strain**
- Click **Input Properties...**
- Click on the **Mat Prop Name** icon
- Select **rubber**
- Click **OK**



- i. Click **Select Application Region...**
- j. Click on the **Select Members** panel
- k. Select **Shell element** icon
- l. Select all shell elements by clicking and dragging the mouse
- m. Click **Add**
- n. Click **OK**
- o. Click **Apply**

Run a nonlinear analysis:

- Click on the **Analysis** icon
- Click on the **Analysis Deck** icon
- Click **Solution Type...**
- Select **IMPLICIT NONLINEAR** as the Solution Type
- Click **Solution Parameters...**
- i. **Removed**



- j. Click **Contact Parameters...**
- k. Click **Friction...**
- l. Type: **Bilinear Coulomb**
- m. Method: **Nodal Force**
- n. Enter **0.05** as the relative Sliding Velocity
- o. Click **OK**
- p. Click **OK**
- q. Click **OK**
- r. Click **OK**

s. Click **Subcases...**

t. Select **Default**

u. Click **Subcase Parameters...**

v. Click **Load Increment Params...**

w. Increment Type: **Fixed**

x. Enter **45** as the Number of Increments and **4.5** as the Total Time

y. Click **OK**

z. Click **OK**

hh on next page



Standard Results

SUBCASE NAME: Default
SOLUTION SEQUENCE: 400

Form Type: **Advanced** **bb**

Select Result Type

- Applied Loads
- Element Strain Energies
- Element Strains
- Grid Point Stresses
- Grid Point Force Balance
- Non-Linear Stress
- Contact Results** **bb1**

Output Requests

- DISPLACEMENT(SORT1,REAL)=All FEM
- STRESS(SORT1,REAL,VONMISES,BILIN)=All FEM;PARAM,NOCOI
- SPCFORCES(SORT1,REAL)=All FEM
- BOUTPUT(SORT1,REAL)=All FEM**

Intermediate Output Option: Yes **cc**

TITLE
This is a default subcase.

SUBTITLE
Default **bb**

LABEL
This load case is the default load case that always appears

dd OK Defaults Cancel

Subcases

Solution Sequence: 400

Action: Create

Available Subcases

- Default

Subcase Name
Default

Available Load Cases

- Default

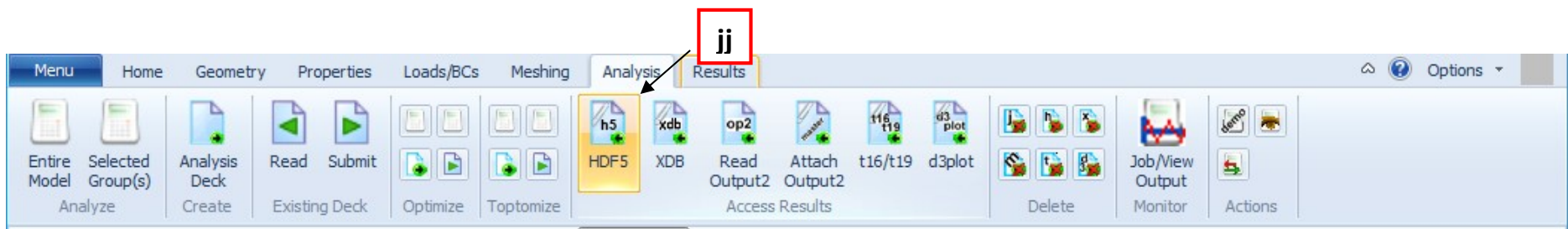
Analysis Type: Static

Subcase Options

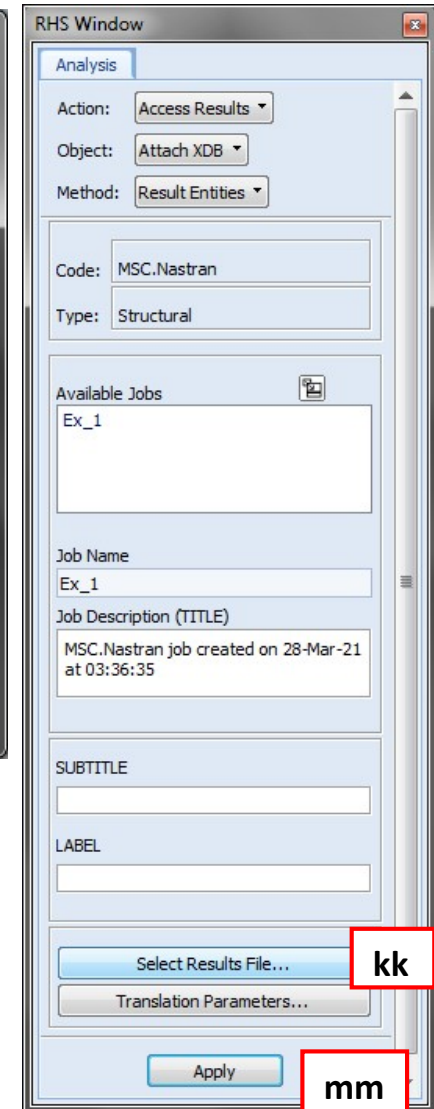
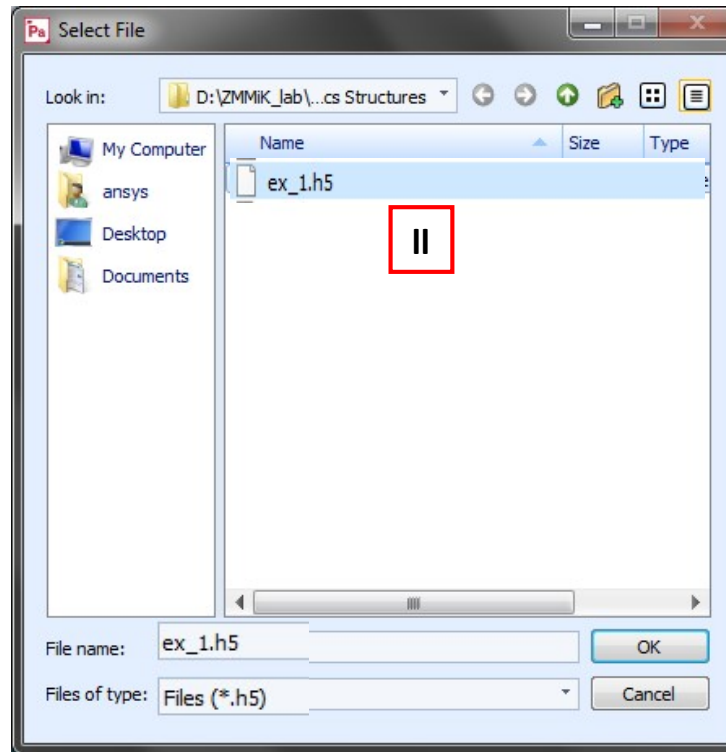
- Subcase Parameters...
- Output Requests...** **aa**
- Direct Text Input...
- Select Explicit MPCs...

ee Apply Cancel **ff**

- aa. Click **Output Requests...**
- bb. Form Type: **Advanced**
- bb1.** Select **"Contact Results"**
- cc. Intermediate Output Option: **Yes**
- dd. Click **OK**
- ee. Click **Apply**
- ff. Click **Cancel**
- gg. Click **Apply**
- hh. Click **Apply** (see page 19)



- ii. Run **Nastran analysis** using **Ex_1.bdf** file from your working directory folder
- Attach the results file, when the analysis job is completed:
- jj. Analysis: **HDF5** icon
 - kk. Click **Select Results File...**
 - ll. Select **ex_1.h5** file and click **OK**
 - mm. Click **Apply**



Patran 2020 (Student Edition) 01-May-21 19:09:32
 Fringe: SC1:Step 1:DEFAULT, A1:Time=4.5, Displacements, Translational, Magnitude, (NON-LAYERED)
 Deform: SC1:Step 1:DEFAULT, A1:Time=4.5, Displacements, Translational,

8.09+00
7.55+00
7.01+00
6.47+00
5.93+00
5.39+00
4.85+00
4.31+00
3.77+00
3.24+00
2.70+00
2.16+00
1.62+00
1.08+00
5.39-01
0

Patran®
Student Edition

Y
X

8.09+00

a **d** **b** **c**

e

j

f

g

h

i

k

l

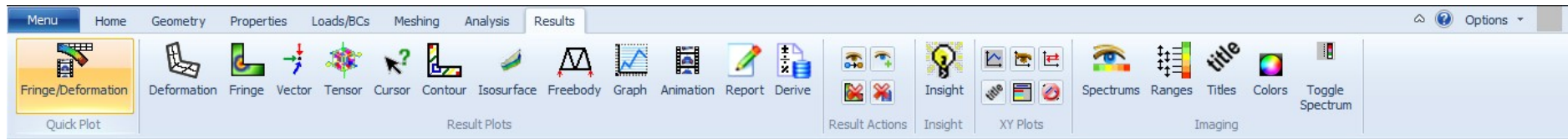
m

Instructions:

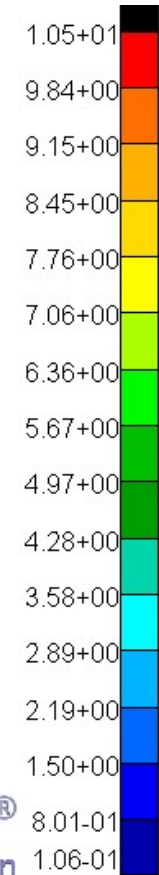
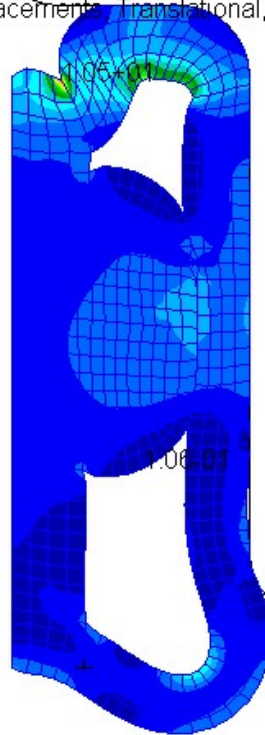
- Click on the **Reset graphics** icon
- Click on the **Smooth shaded** icon
- Click on the **Plot/Erase Geometry** icon
- Click on the **Fit view** icon

Post-process the results:

- Click on the Results: **Fringe/Deformation** icon
- Select **last** step
- Select Fringe Result: **Displacements, Translational**
- Quantity: **Magnitude**
- Select Deformation Result: **Displacements, Translational**
- Click on the **Deform Attributes** icon
- Select **True Scale** with the Scale Factor equals to **1.0**
- Uncheck **Show Undeformed**
- Click **Apply**



Patran 2020 (Student Edition) 01-May-21 19:16:04
 Fringe: SC1:Step 1:DEFAULT, A1:Time=4.5, Cauchy stresses, , von Mises, (NON-LAYERED)
 Deform: SC1:Step 1:DEFAULT, A1:Time=4.5, Displacements, Translational,



RHS Window

Results

Action: Create

Object: Quick Plot

Select Result Cases

- Default, A1:Non-linear: 344. % of Load
- Default, A1:Non-linear: 352. % of Load
- Default, A1:Non-linear: 360. % of Load
- Default, A1:Non-linear: 368. % of Load
- Default, A1:Non-linear: 376. % of Load
- Default, A1:Non-linear: 384. % of Load
- Default, A1:Non-linear: 392. % of Load
- Default, A1:Non-linear: 400. % of Load

Select Fringe Result

- Cauchy Stresses, **p**
- Constraint Forces, Rotational
- Constraint Forces, Translational
- Displacements, Rotational
- Displacements, Translational

Quantity: von Mises **q**

Select Deformation Result

- Constraint Forces, Rotational
- Constraint Forces, Translational
- Displacements, Rotational
- Displacements, Translational **r**

Animate

s Apply

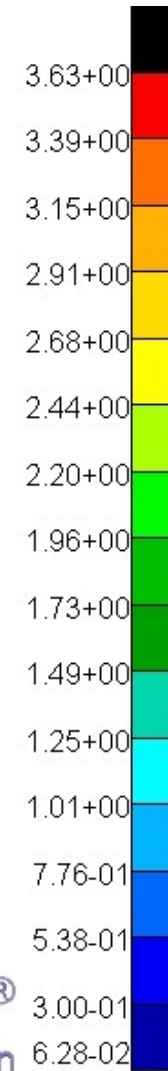
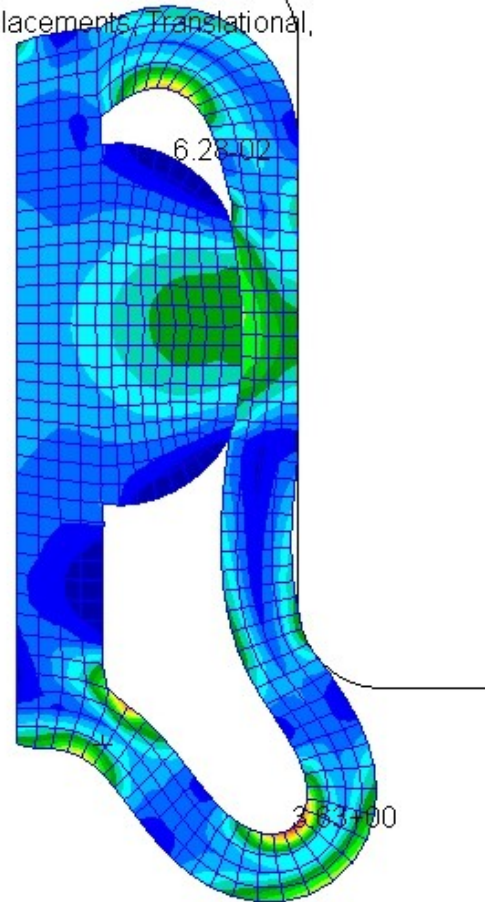
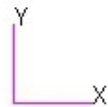
- n. Click on the **Select Results** icon
- o. Make sure that the **last** result step is selected
- p. Select Fringe Result: **Cauchy Stresses**
- q. Quantity: **von Mises**
- r. Select Deformation Result: **Displacements, Translational**
- s. Click **Apply**

And below – the same run, but **WITHOUT FRICTION** (slips down :-)

Patran 2020 (Student Edition) 01-May-21 19:22:48

Fringe: SC1:Step 1:DEFAULT, A2:Time=4.5, Cauchy stresses, , von Mises, (NON-LAYERED)

Deform: SC1:Step 1:DEFAULT, A2:Time=4.5, Displacements, Translational,



Patran®
Student Edition