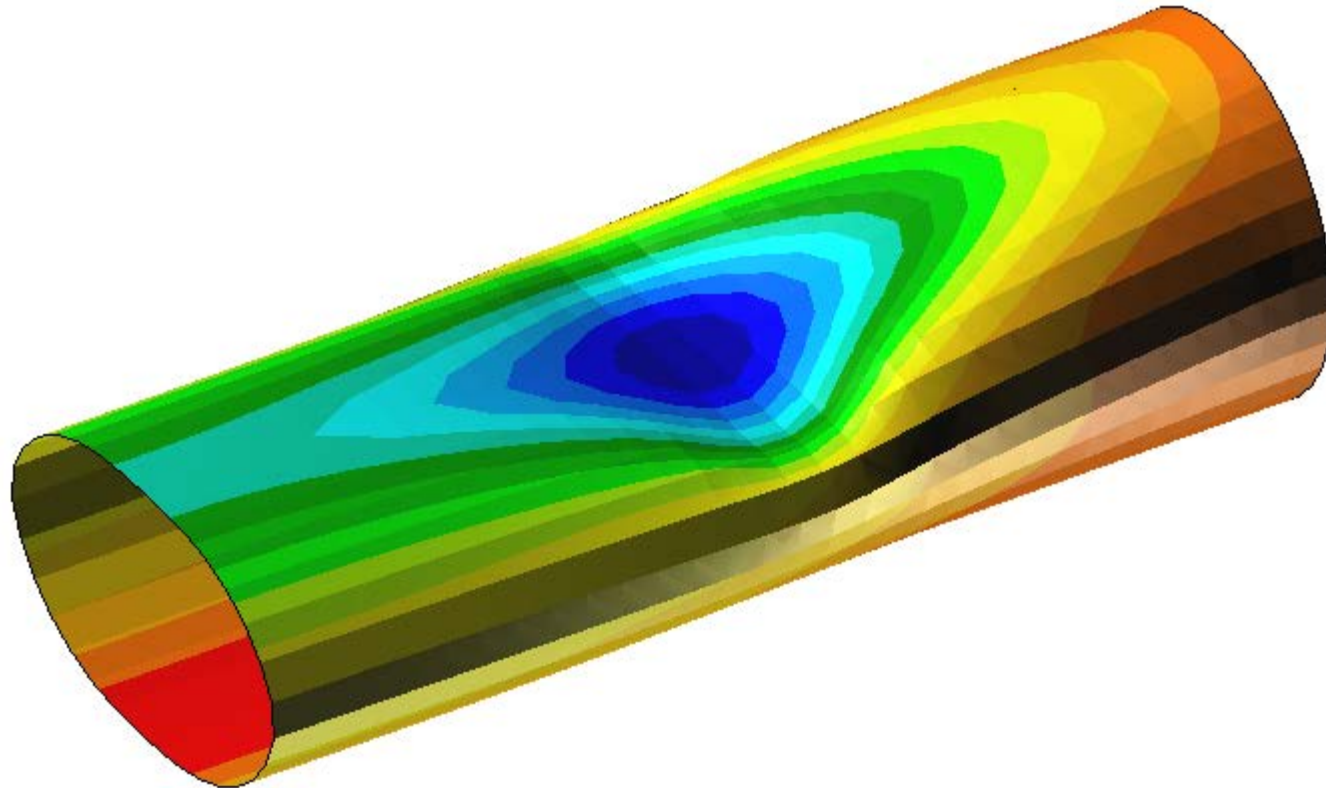
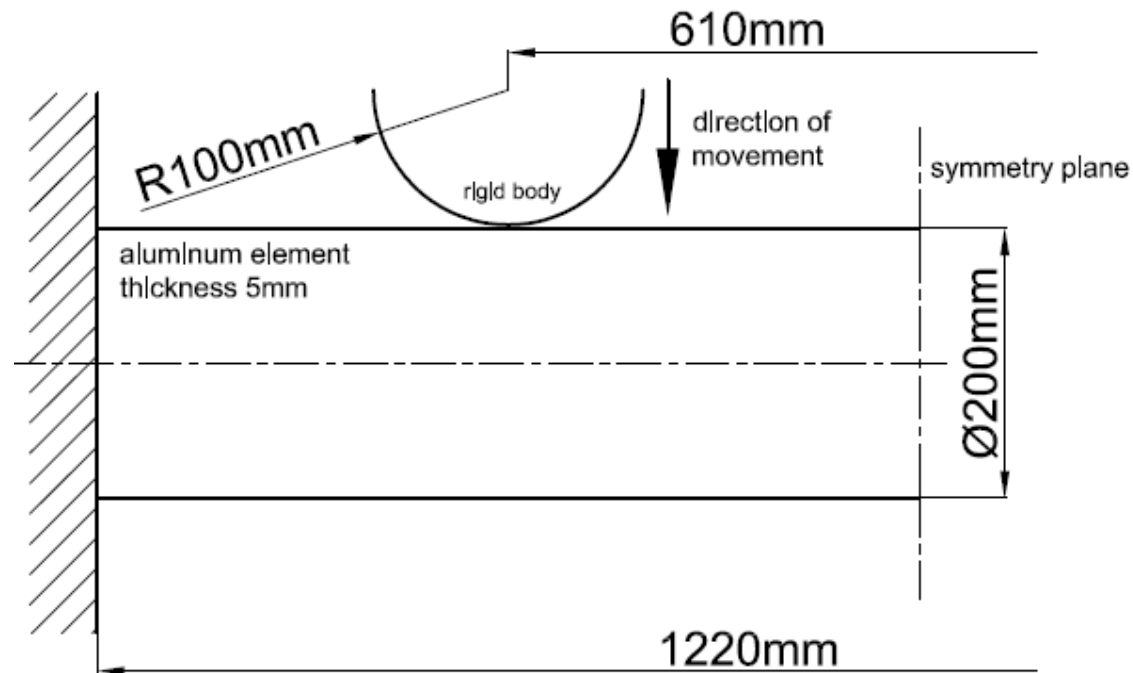


NONLINEAR MECHANICS OF STRUCTURES

EXERCISE 2

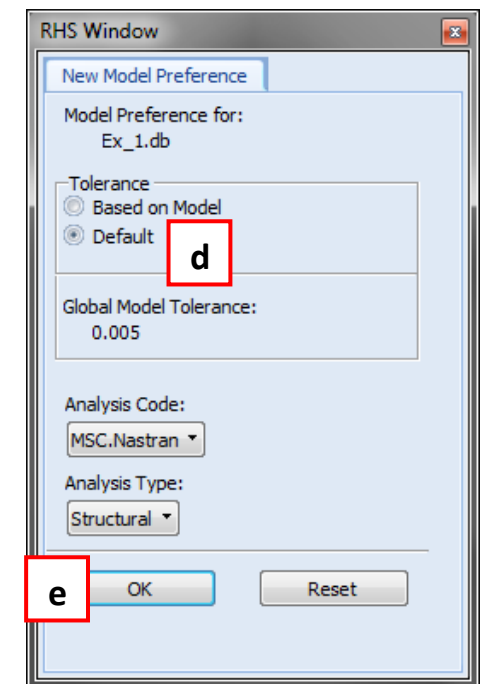
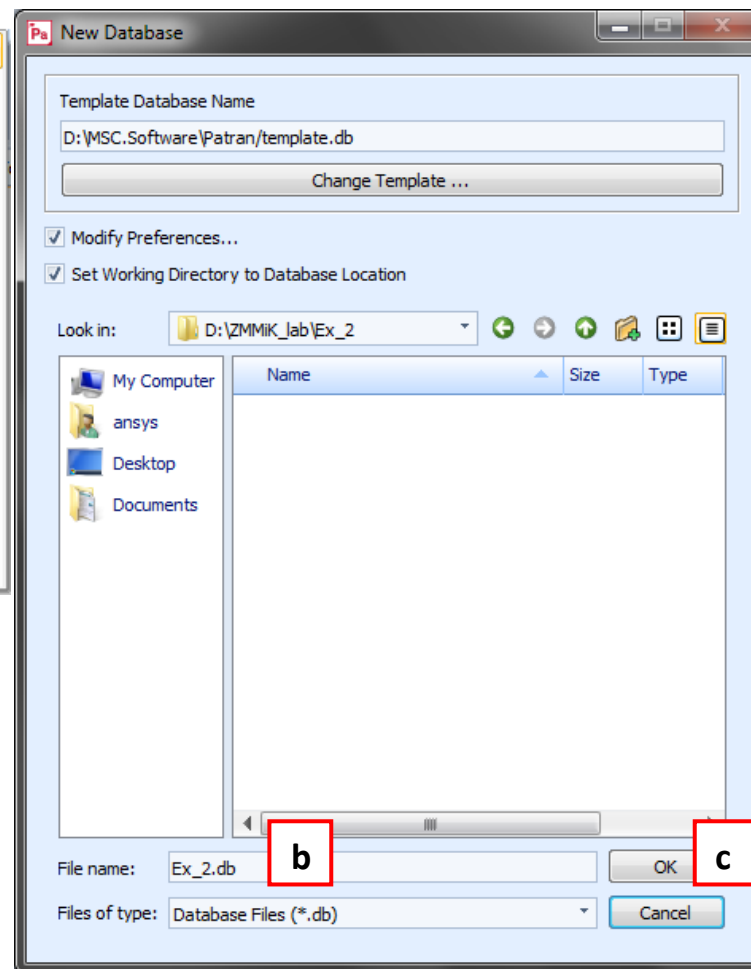
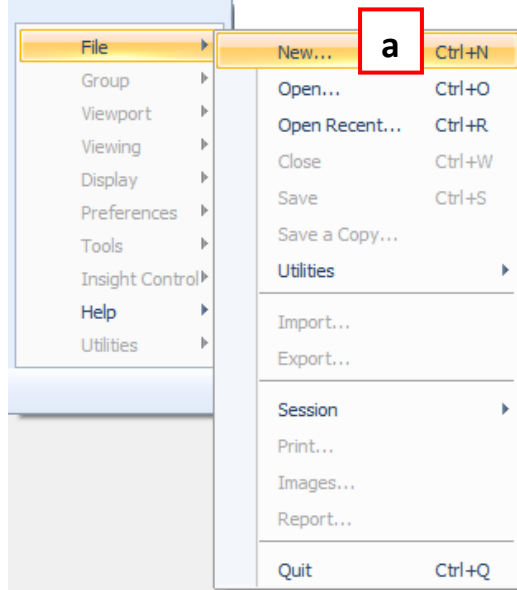


PROBLEM DESCRIPTION

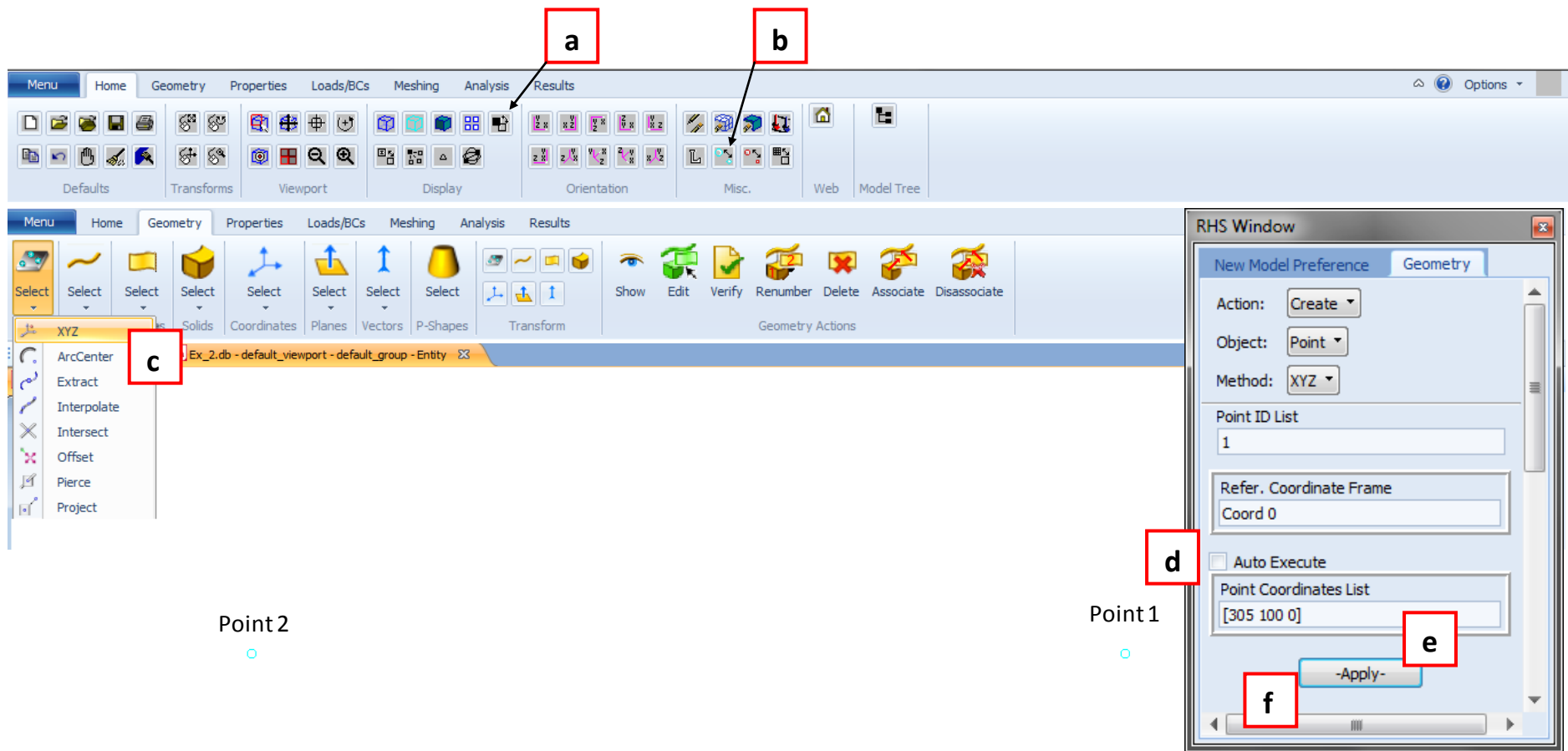


A thin-walled element (a pipe) is crushed by two cylindrical rigid bodies. Due to the symmetry of the task only half of the pipe (with appropriate boundary conditions) is modeled. The goal of this exercise is to perform a nonlinear analysis, which includes high plasticity of the used material, and assess the obtained results.

Units: mm, N, MPa

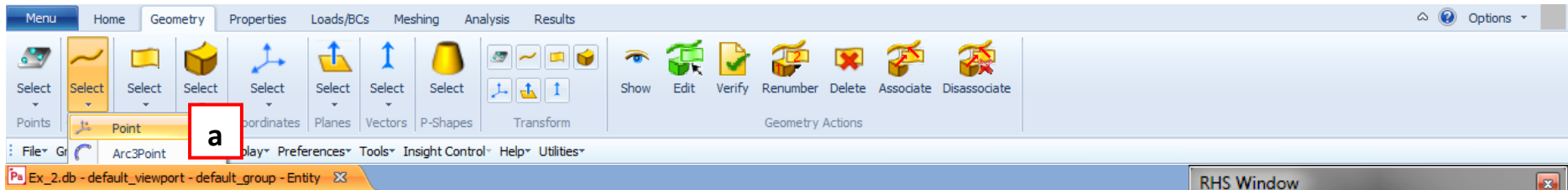


- Create a new database:
- a. **File / New...**
 - b. Enter **Ex_2.db** as the File name
 - c. Click **OK**
 - d. Select **Default**
 - e. Click **OK**



- +
- Y
X
- a. Change *Background Color* to **Black** (click on the **Cycle Background** icon)
 - b. Click on the **Point size** icon
- Create geometry points:
- c. Click on the **Geometry** icon (Points tab): **Select/XYZ**
 - d. Uncheck **Auto Execute**
 - e. Enter **[305 100 0]** as the Point Coordinates List
 - f. Click **Apply**
 - g. Create one more point using coordinates: **[-305 100 0]**

atran®
Plant Edition



Point 2



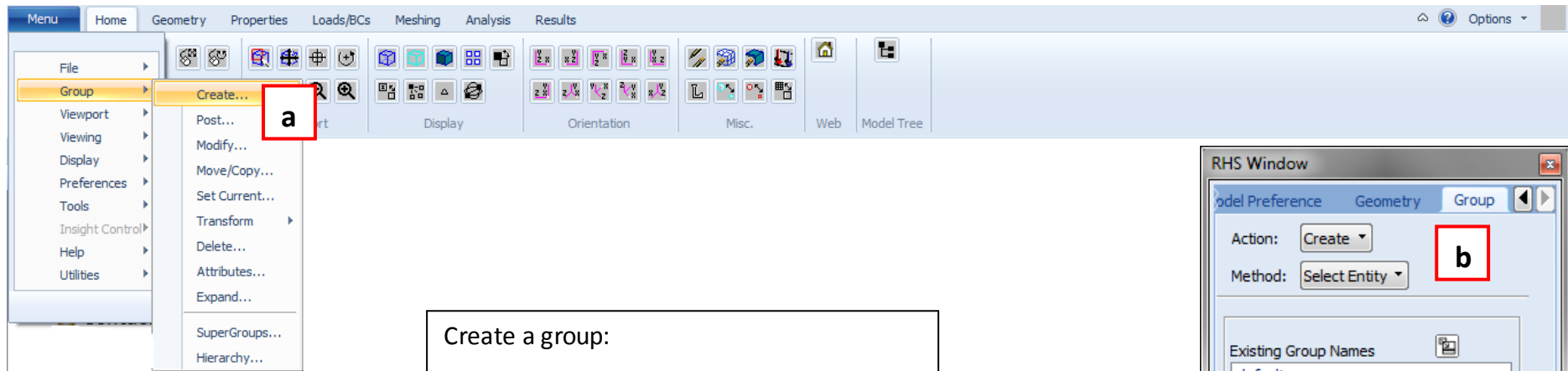
Point 1



Create a curve:

- Click on the **Geometry** icon (Curves tab): **Select/Point**
- Option: **2 Point**
- Uncheck **Auto Execute**
- Select **Point 1** as the starting point and **Point 2** as the ending point
- Click **Apply**

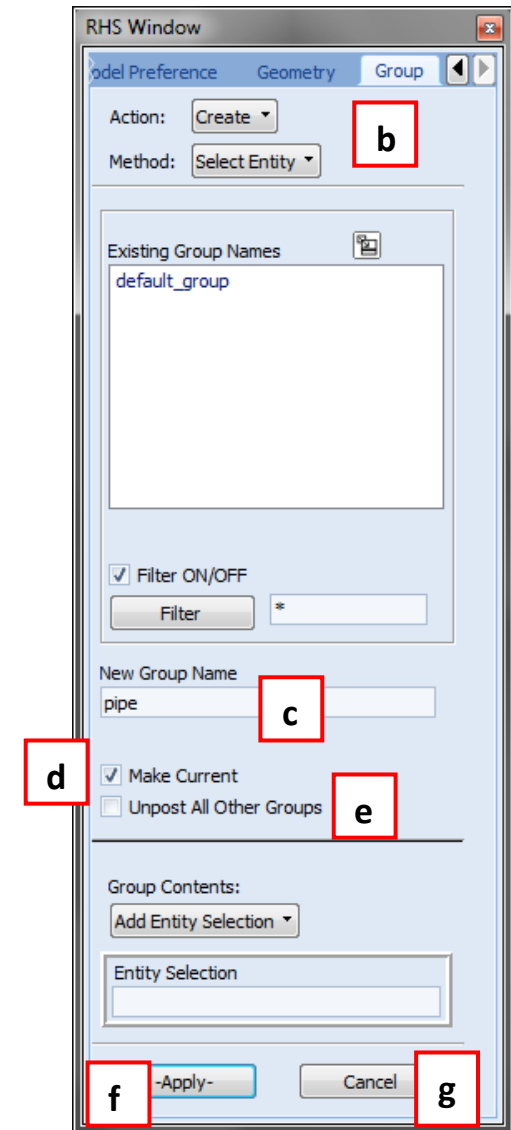
Patran®
Student Edition

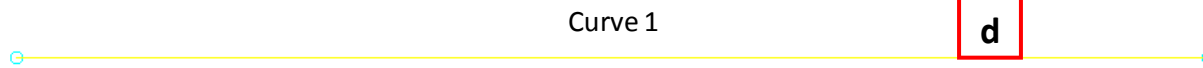
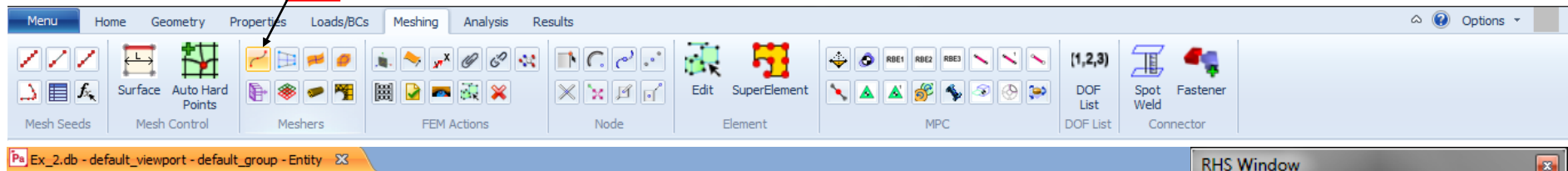


Create a group:

- a. **Group / Create...**
- b. Group: **Create/Select Entity**
- c. Enter **pipe** as the New Group Name
- d. Check **Make Current**
- e. Uncheck **Unpost All Other Groups**
- f. Click **Apply**
- g. Click **Cancel**

Remark: Mesh of the cylindrical element will be assigned to this group.

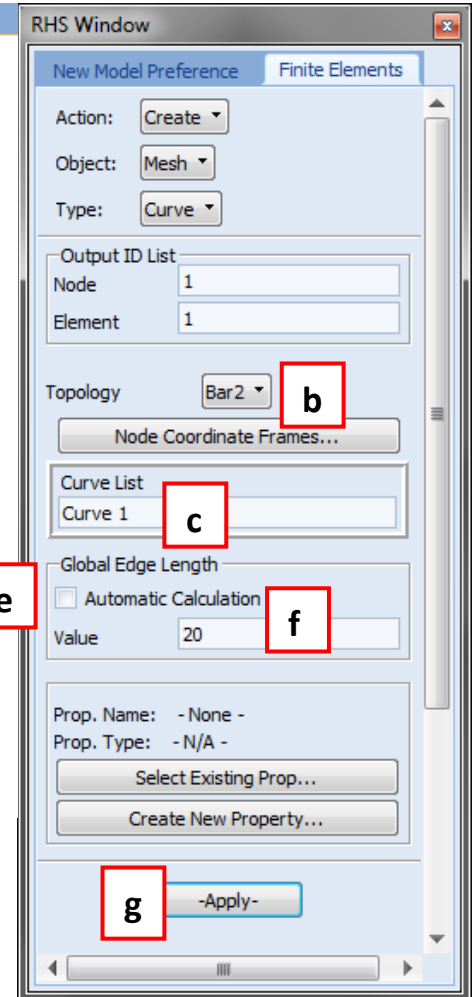




Mesh the curve:

- a. Click on the **Meshing icon/Curve** icon (Meshers tab)
- b. Topology: **Bar2**
- c. Click on the **Curve List** panel
- d. Select the horizontal curve
- e. Uncheck **Automatic Calculation**
- f. Enter **20** as the Value of the Global Edge Length
- g. Click **Apply**

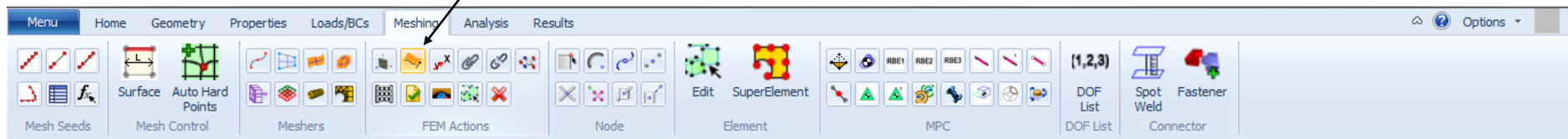
Patran®
Student Edition



The image shows the ANSYS software interface. The 'Group' menu is open, and the 'Post...' option is highlighted with a red box labeled 'a'. The 'RHS Window' dialog box is open, showing the 'Group' tab. The 'Action' is set to 'Post'. The 'Current Viewport' is 'default_viewport'. The 'Select Groups to Post' list contains 'default_group' and 'pipe', with 'pipe' selected and highlighted with a red box labeled 'b'. The 'Filter ON/OFF' checkbox is checked. The 'Apply' button is labeled 'c' and the 'Cancel' button is labeled 'd'.

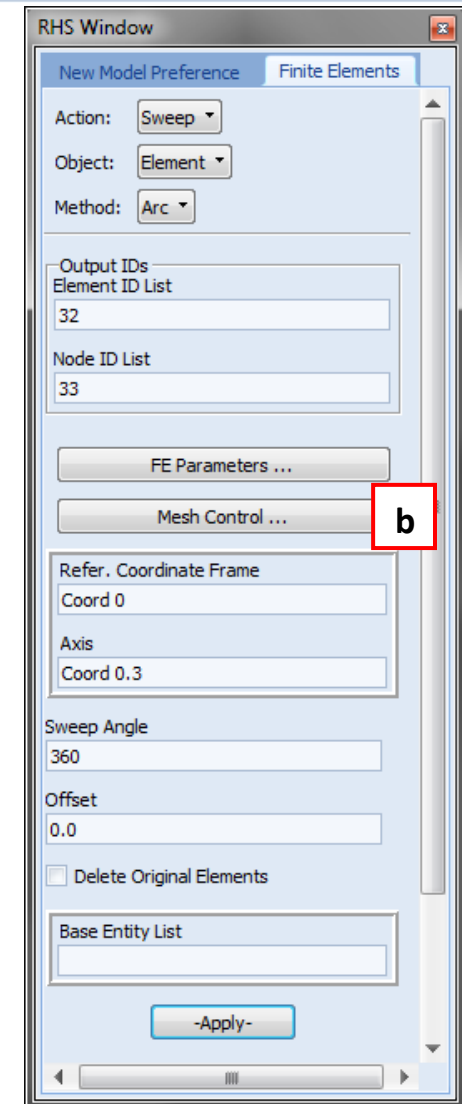
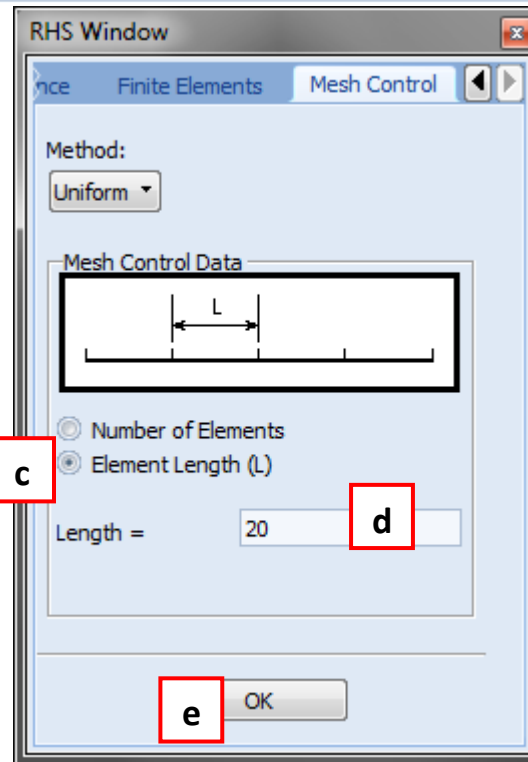
Post only the **pipe** group:

- Group / Post...**
- Select **pipe**
- Click **Apply**
- Click **Cancel**



Sweep the elements:

- a. Meshing: **Sweep** icon (FEM Actions tab)
- b. Click **Mesh Control...**
- c. Select **Element Length (L)**
- d. Enter **20** as the Length
- e. Click **OK**



Ex_2.db - default_viewport - pipe - Entity

Menu Home Geometry Properties Loads/BCs Meshing Analysis Results Options

Mesh Seeds Mesh Control Meshers FEM Actions Node Element MPC DOF List Spot Weld Fastener Connector

Finite Elements

Action: Sweep

Object: Element

Method: Arc

Output IDs

Element ID List

32

Node ID List

33

FE Parameters ...

Mesh Control ...

Refer. Coordinate Frame

Coord 0

Axis

Coord 0.1 **f**

Sweep Angle

360 **g**

Offset

0.0

Delete Original Elements **h**

Base Entity List

Elm 1:31 **i**

m -Apply-

l

j

k

Y

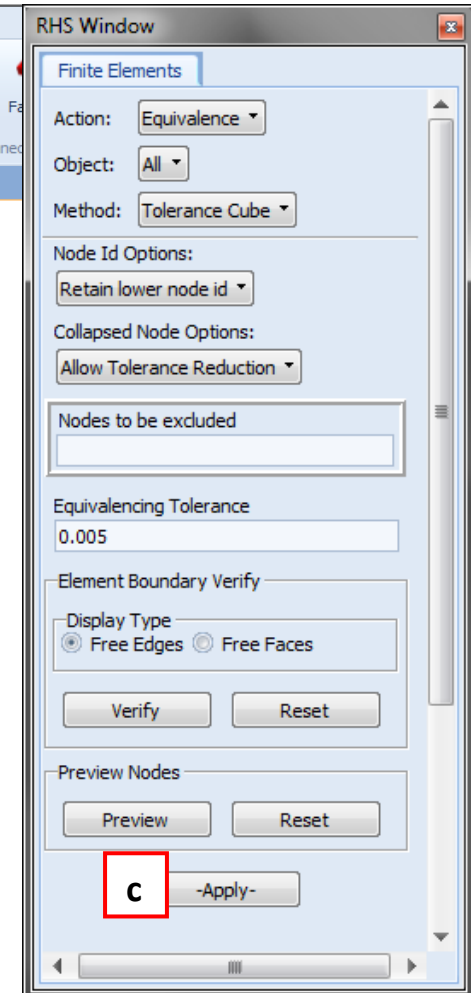
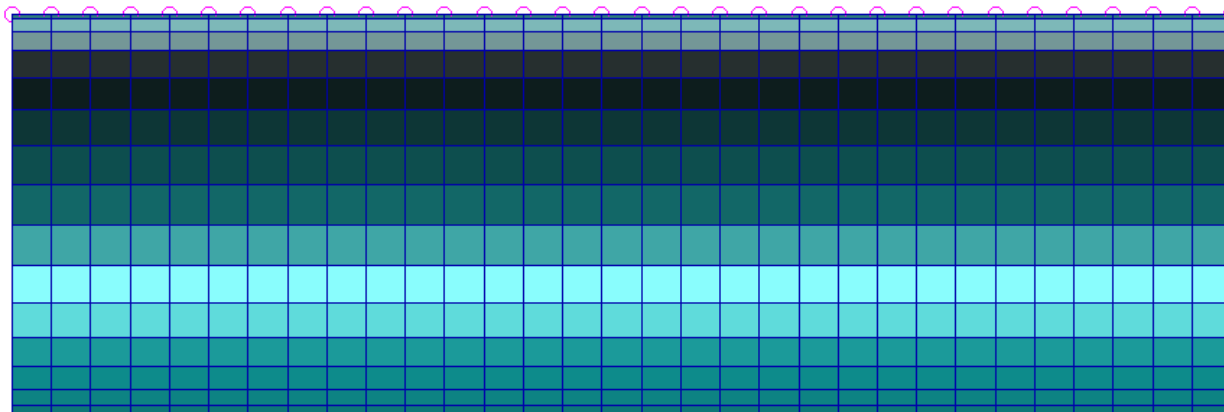
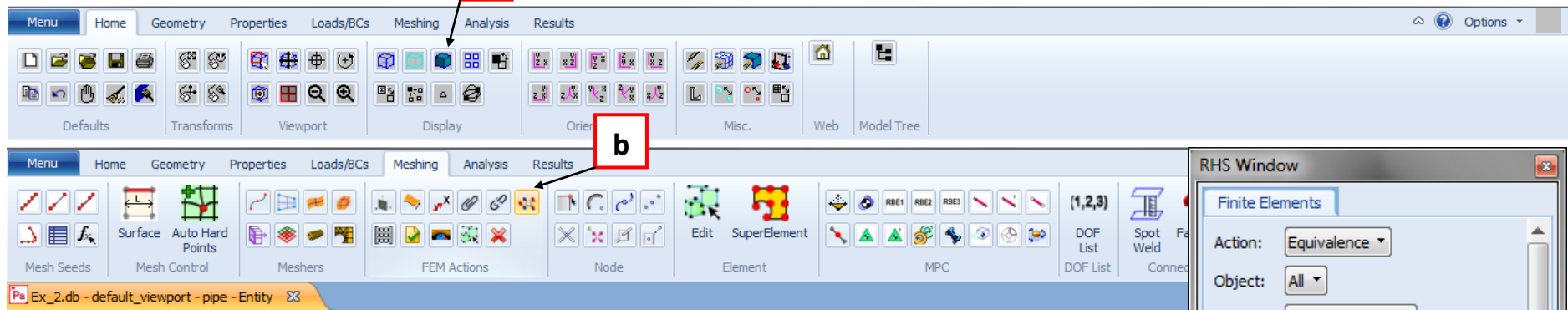
X

+

tran®
nt Edition

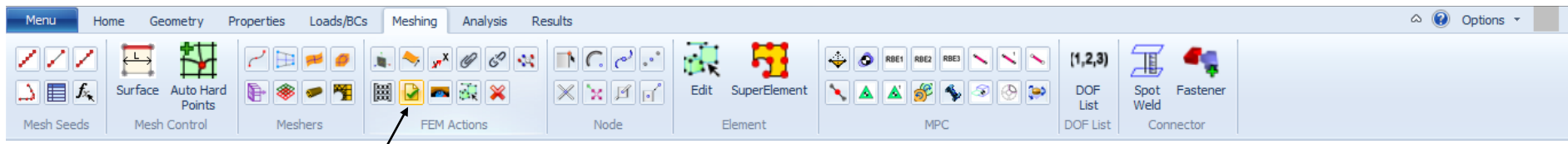
- f. Enter **Coord 0.1** as the Axis (of revolution)
- g. Enter **360** as the Sweep Angle
- h. Check **Delete Original Elements**
- i. Click on the **Base Entity List** panel
- j. Select the **Element** icon
- k. Select the **Beam element** icon
- l. Select all visible elements by clicking and dragging the mouse
- m. Click **Apply**

Remark: **Coord 0.1** means the X-axis of the global coordinate system.



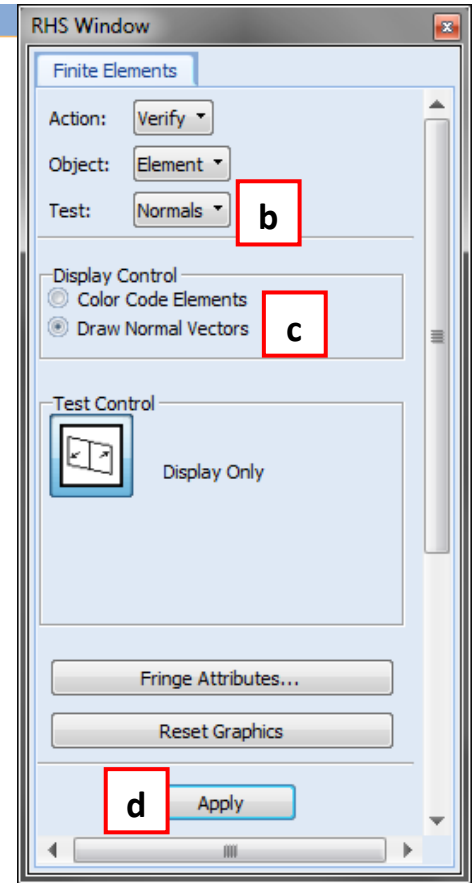
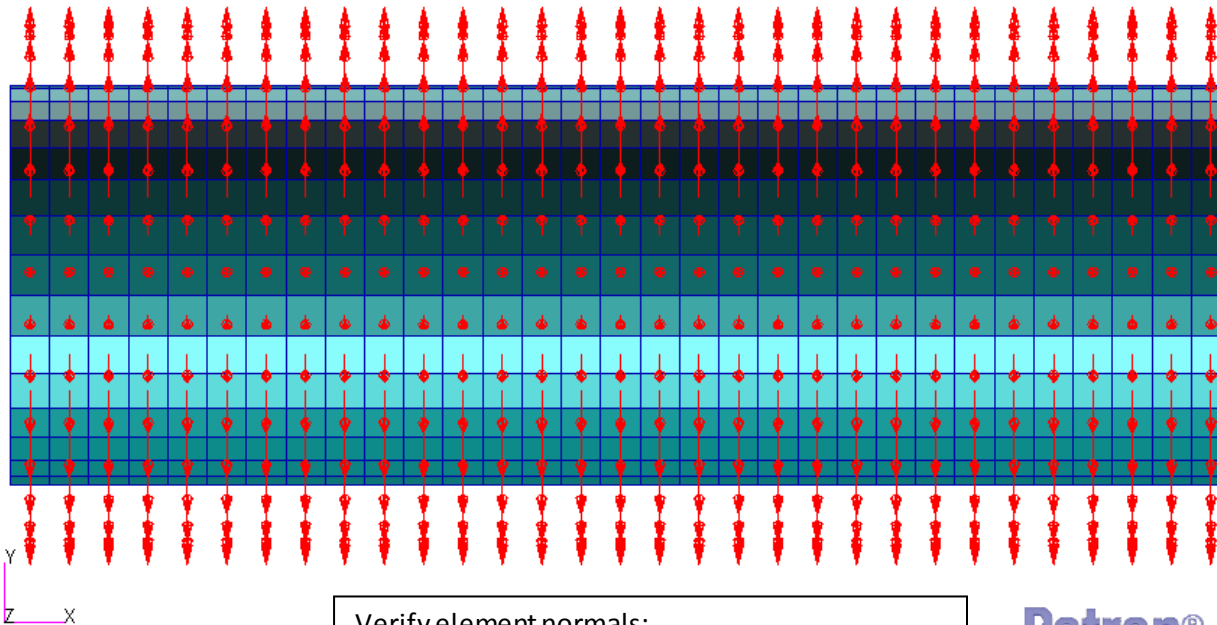
- a. Click on the **Smooth shaded** icon from **Home** icon
- Delete duplicate nodes:
- b. Meshing: **Equivalence** icon (*FEM Actions tab*)
- c. Click **Apply**

Patran®
Student Edition



Pa Ex_2.db - default_viewport - pipe - Entity

a



b

c

d

Verify element normals:

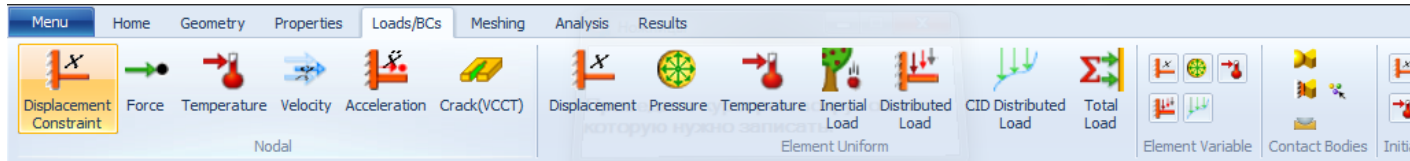
- a. Meshing: **Verify** icon
- b. Test: **Normals**
- c. Select **Draw Normal Vectors**
- d. Click **Apply**

Remark: To reverse element normals use **Elements: Modify/Element/Reverse**

Patran®
Student Edition

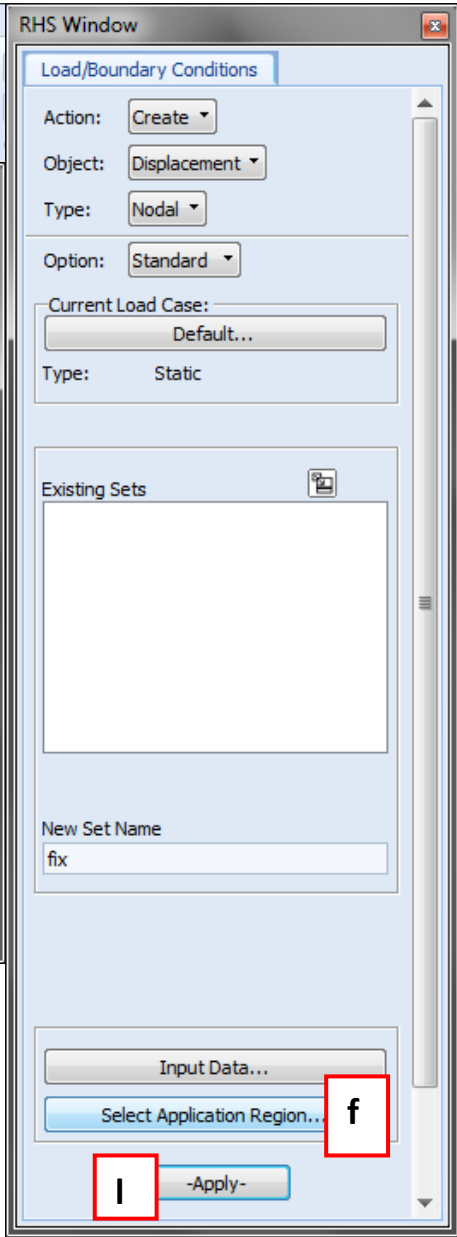
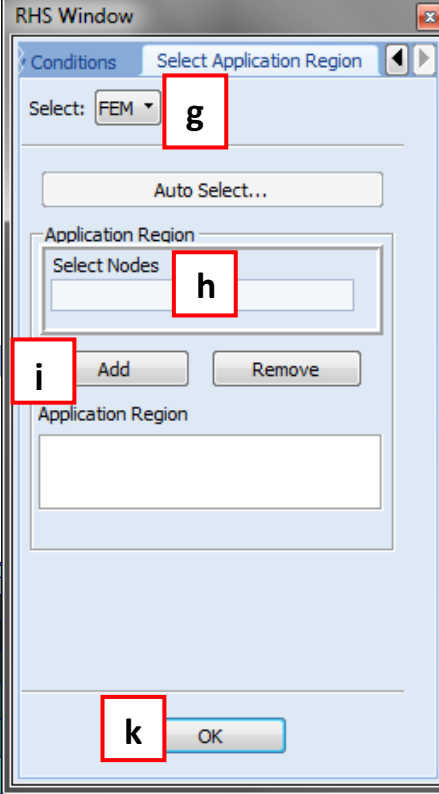
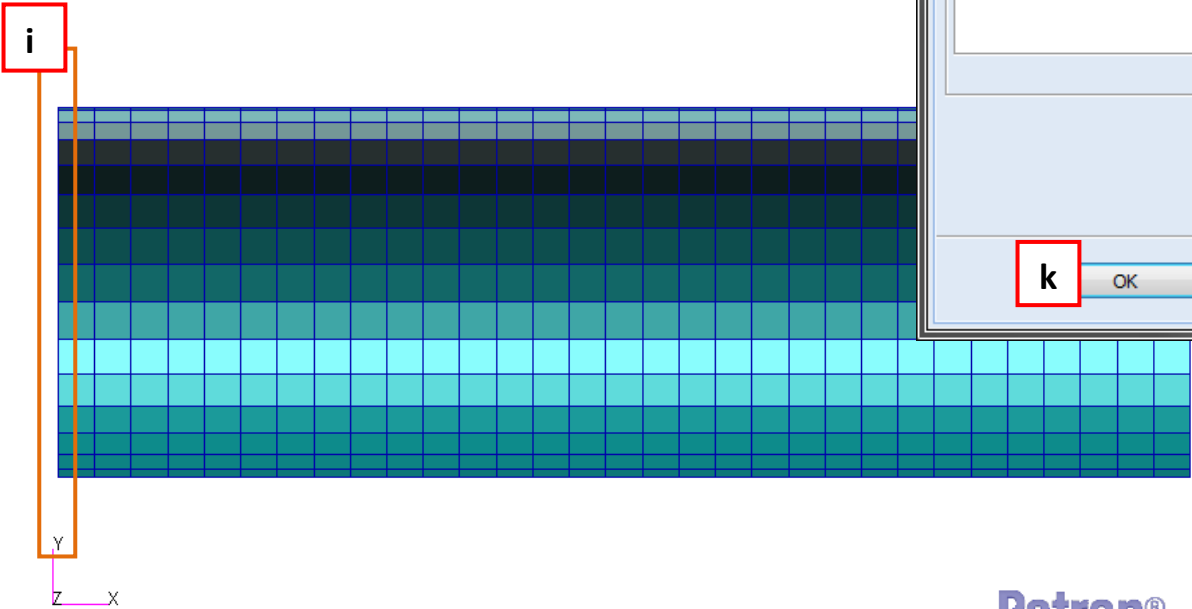
Apply the boundary conditions:

- Loads/BCs: **Displacement Constraint**
- Enter **fix** as the New Set Name
- Click **Input Data...**
- Enter **<0,0,0>** for the Translations and for the Rotations
- Click **OK**



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

Ex_2.db - default_viewport - pipe - Entity



Patran®
Student Edition

Menu Home Geometry Properties Loads/BCs Meshing Analysis Results Options

Displacement Constraint Force Temperature Velocity Acceleration Crack(VCCT) Displacement Pressure Temperature Inertial Load Distributed Load CID Distributed Load Total Load Element Variable Contact Bodies Initial

Nodal Element Uniform

n. Enter **symmetry** as the New Set Name
 o. Click **Input Data...**
 p. Enter **<0, , >** for the Translations and **<,0,0 >** for the Rotations
 q. Click **OK**

Pa Input Data

Load/BC Set Scale Factor
1.

Translations <T1 T2 T3>
<0, , > **p**

Rotations <R1 R2 R3>
<,0,0 > **q**

Trans Phase <Tp1 Tp2 Tp3>
< >

Rotation Phase <Rp1 Rp2 Rp3>
< >

Spatial Fields

FEM Dependent Data...

Analysis Coordinate Frame
Coord 0

q OK Reset

RHS Window

Load/Boundary Conditions

Action: Create

Object: Displacement

Type: Nodal

Option: Standard

Current Load Case:
Default...

Type: Static

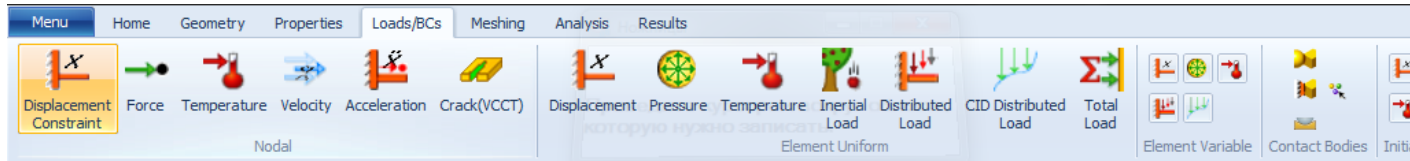
Existing Sets
fix

New Set Name
symmetry **n**

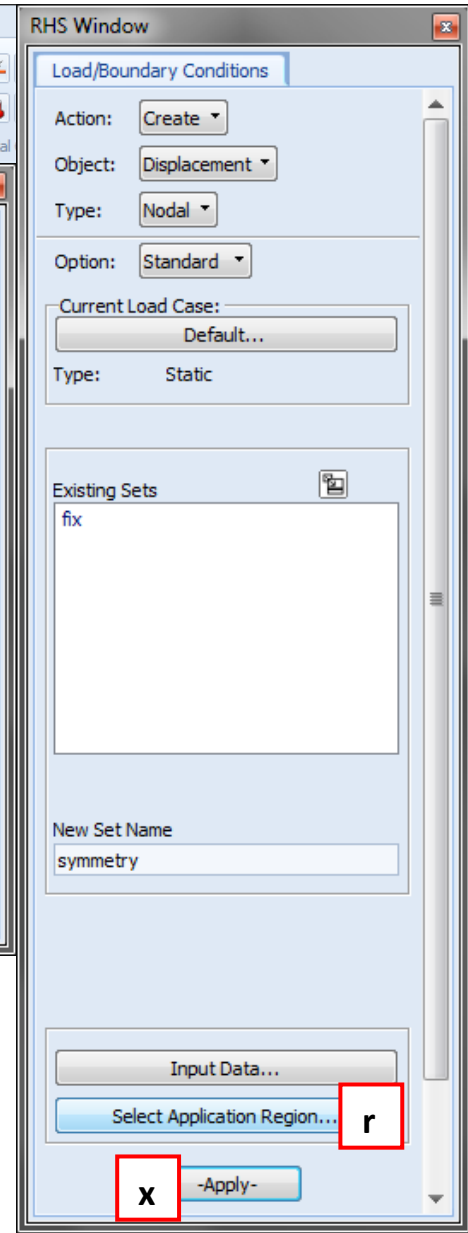
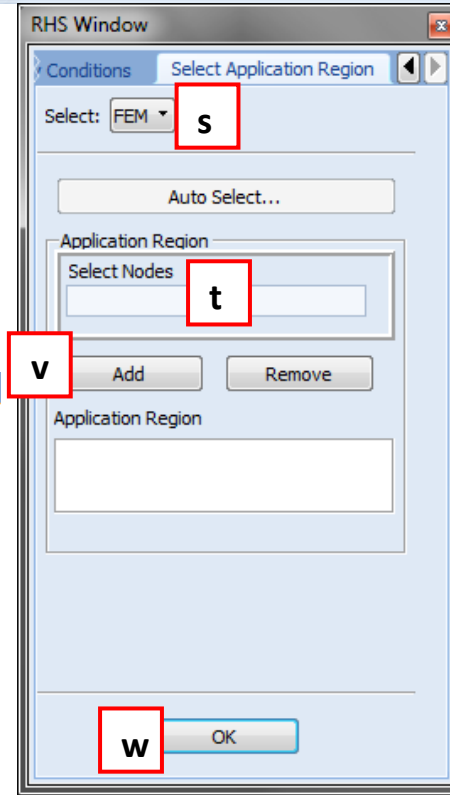
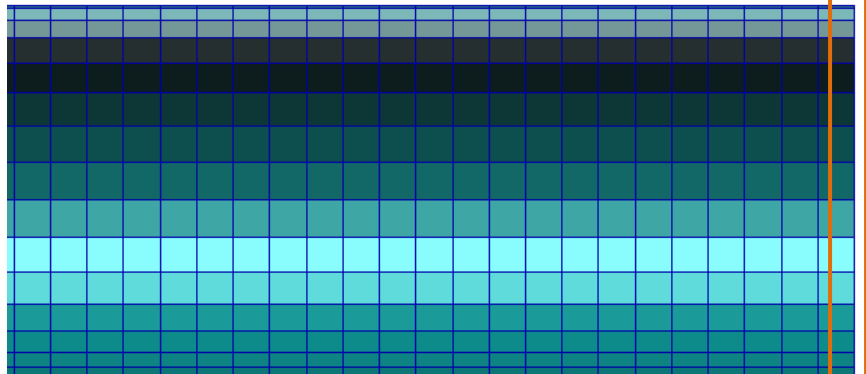
o Input Data...

Select Application Region...

-Apply-



- r. Click **Select Application Region...**
- s. Select **FEM**
- t. Click on the **Select Nodes** panel
- u. Select the nodes (belonging to the right free edge of the pipe) by clicking and dragging the mouse
- v. Click **Add**
- w. Click **OK**
- x. Click **Apply**



The screenshot shows a software interface with a ribbon menu at the top. The 'Properties' tab is active, and the 'Isotropic' icon is highlighted with a red box labeled 'a'. Below the ribbon, the 'Input Options' dialog box is open, showing a table of material properties. The 'Elastic Modulus' is set to 70000 and the 'Poisson Ratio' is set to .33, both values are highlighted with red boxes labeled 'd'. At the bottom of the dialog, the 'OK' button is highlighted with a red box labeled 'e'. To the right, the 'RHS Window' is open, showing the 'Materials' section. The 'Material Name' is set to 'aluminum', highlighted with a red box labeled 'b'. The 'Input Properties ...' button is highlighted with a red box labeled 'c', and the 'Apply' button at the bottom is highlighted with a red box labeled 'f'.

Define a material:

- Click on the **Properties** icon:
Isotropic icon
- Enter **aluminum** as the Material Name
- Click **Input Properties...**
- Enter **70000** as the Elastic Modulus and **0.33** as the Poisson Ratio
- Click **OK**
- Click **Apply**

The screenshot shows the ANSYS software interface. The top menu bar includes Menu, Home, Geometry, Properties, Loads/BCs, Meshing, Analysis, and Results. The toolbar below contains various icons for material and property definitions. The 'Input Options' dialog box is open, showing the following settings:

- Constitutive Model: Elastoplastic (labeled **i**)
- Nonlinear Data Input: Perfectly Plastic (labeled **j**)
- Yield Function: Von Mises
- Hardening Rule: None
- Strain Rate Method: Piecewise Linear
- Property Name: Yield Point = 350 (labeled **k**)
- Temperature/Strain Rate Dependent Fields: (empty)
- Current Constitutive Models: Linear Elastic - [,,,] - [Active]

The 'RHS Window' on the right shows the 'Materials' tab with the following settings:

- Action: Modify (labeled **g**)
- Object: Isotropic
- Existing Materials: aluminum (labeled **h**)
- Filter ON/OFF: Filter
- New Material Name: aluminum
- Description: (empty)
- Buttons: Input Properties ..., Change Material Status ..., Apply (labeled **m**)

On the left, a list of instructions is provided:

- Change *Action* to **Modify**
- Select **aluminum**
- Change *Constitutive Model* to **Elastoplastic**
- Nonlinear Data Input: **Perfectly Plastic**
- Enter **350** as the Yield Point
- Click **OK**
- Click **Apply**

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 'Properties' tab is active, showing various material property categories like Isotropic, Fluid, Cohesive, Composite, 0D Properties, 1D Properties, 2D Properties, and 3D Properties. The '2D Properties' section is highlighted, and the 'Shell' icon is selected, indicated by a red box labeled 'a'.

The 'Select Material' dialog is open, showing a list of materials. 'aluminum' is selected, indicated by a red box labeled 'e'.

The 'Input Properties' dialog is open for 'Stan. Homogeneous Plate(CQUAD4)'. It shows a table of properties:

Property Name	Value	Value Type
Material Name	m:aluminum	Mat Prop Name
[Material Orientation]		CID
Thickness	5	Real Scalar
[Nonstructural Mass]		Real Scalar
[Plate Offset]		Real Scalar
[Fiber Dist. 1]		Real Scalar
[Fiber Dist. 2]		Real Scalar
Nonlinear Formulation(SOL 400)		

The 'Mat Prop Name' icon is highlighted with a red box labeled 'd', and the 'Thickness' value '5' is highlighted with a red box labeled 'f'.

The 'RHS Window' is open, showing 'Element Properties'. The 'Action' is 'Create', 'Object' is '2D', and 'Type' is 'Shell'. The 'Property Set Name' is 'shell', indicated by a red box labeled 'b'. The 'Input Properties ...' button is highlighted with a red box labeled 'c'.

The 'Input Properties' dialog has 'OK', 'Clear', and 'Cancel' buttons. The 'OK' button is highlighted with a red box labeled 'g'.

A text box at the bottom of the 'Input Properties' dialog contains the text: "Enter the Thickness or select a field with the icon. Specify Element Nodal by selecting it from the dropdown."

- Assign the properties:
- Properties icon: Shell icon (2D Properties tab)**
 - Enter **shell** as the New Set Name
 - Click **Input Properties...**
 - Click on the **Mat Prop Name** icon
 - Select **aluminum**
 - Enter **5** as the Thickness
 - Click **OK**

h. Click **Select Application Region...**

i. Click on the **Select Members** panel

j. Select **Shell** element icon

k. Select all shell elements by clicking and dragging the mouse

l. Click **Add**

m. Click **OK**

n. Click **Apply**

Menu Home Geometry Properties Loads/BCs Meshing Analysis Results

File Group Viewport Viewing Display Preferences Tools Insight Control Help Utilities

Group

- Create... **a**
- Post...
- Modify...
- Move/Copy...
- Set Current...
- Transform
- Delete...
- Attributes...
- Expand...
- SuperGroups...
- Hierarchy...

Display Orientation Misc. Web Model Tree

Create one more group:

- Menu/Group/ Create...**
- Enter **rbody** as the New Group Name
- Check **Make Current** and check **Unpost All Other Groups**
- Click **Apply**
- Click **Cancel**

Remark: A moving rigid body (half of a cylinder) will be assigned to this group.

RHS Window

Select Application Region Group

Action: Create

Method: Select Entity

Existing Group Names

- default_group
- pipe

Filter ON/OFF

Filter *

New Group Name **b**

rbody

Make Current **c**

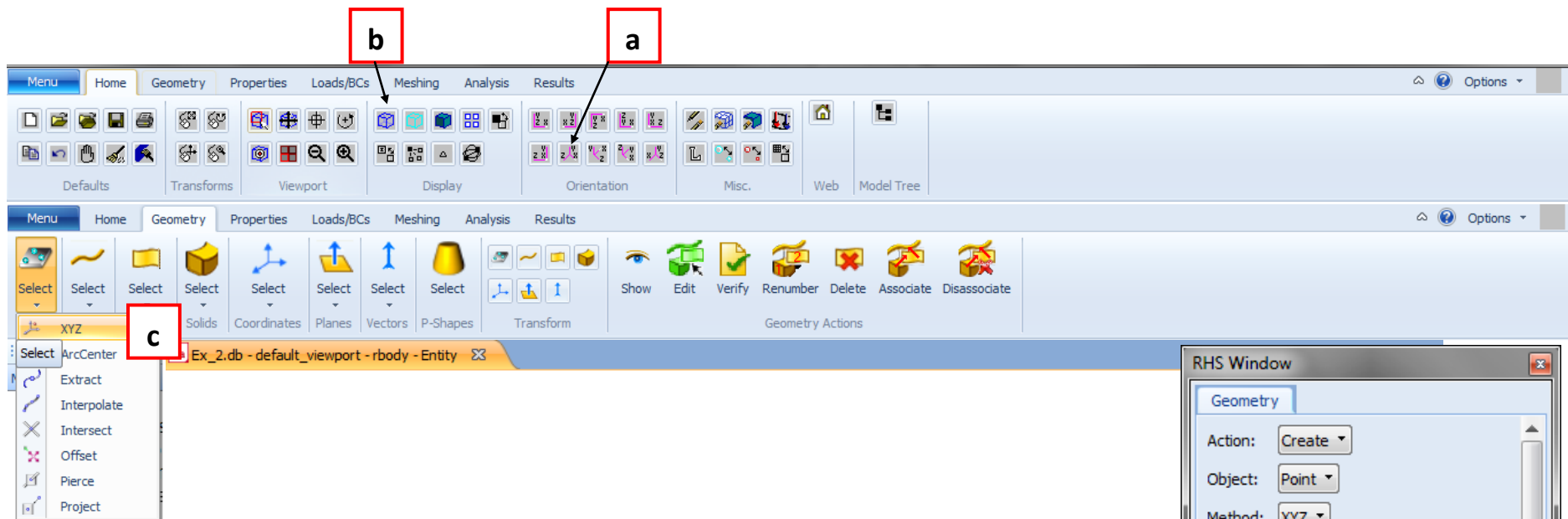
Unpost All Other Groups

Group Contents:

Add Entity Selection

Entity Selection

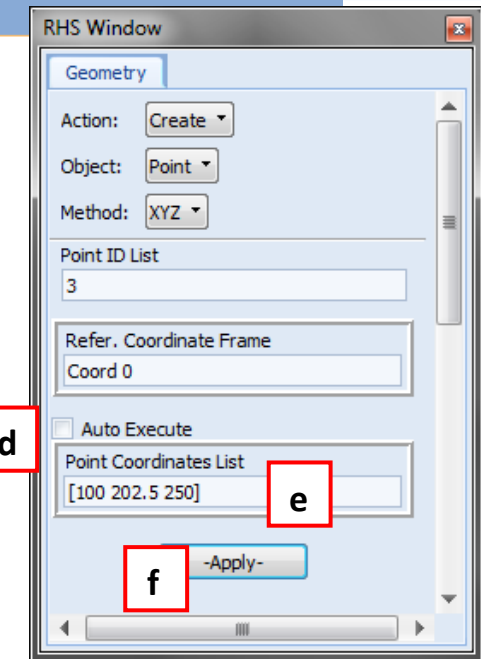
d -Apply- Cancel **e**



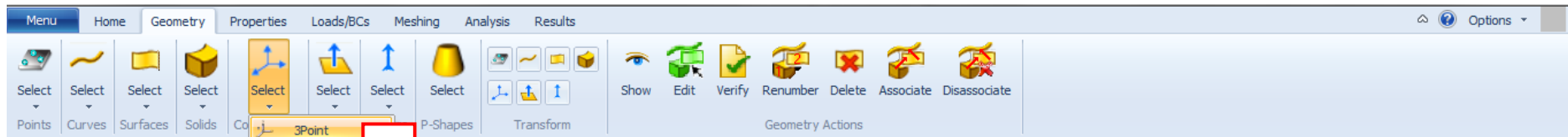
Point 3



- a. Click on the **Iso 1 View** icon
 - b. Click on the **Wireframe** icon
- Create one additional geometry point:
- c. Click on the **Geometry** icon (Points tab)/**Select/XYZ**
 - d. Uncheck **Auto Execute**
 - e. Enter **[100 202.5 250]** as the Point Coordinates List
 - f. Click **Apply**



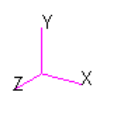
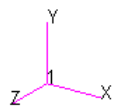
Patran®
Student Edition



Create a local coordinate system:

- Geometry** icon (Coordinates tab): **Select/3Point**
- Type: **Rectangular**
- Uncheck **Auto Execute**
- Enter **[0 202.5 250]** as the Origin (of the system)
- Enter **[0 202.5 300]** as the Point on Axis 3
- Enter **[10 202.5 10]** as the Point on Plane 1-3
- Click **Apply**
- Click on the **Fit view** icon from the **Home** panel

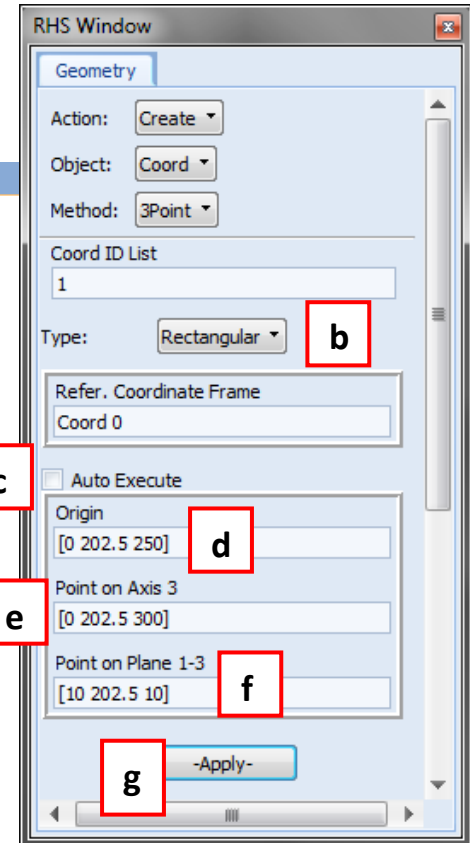
Pa Ex_2.db - default_viewport - rbody - Entity



Point 3



Patran®
Student Edition



a

b

c

d

e

f

g

Patran® Student Edition

a **c, f** **j** **b** **d** **g** **h** **i** **k** **e**

- Create an arc:
- Geometry** icon (Curves tab): **Select/Revolve**
 - Click on the **Refer. Coordinate Frame** panel
 - Select **Coord 1** (or just enter the **Coord 1**)
 - Click on the **Axis** panel
 - Select **Axis 3** icon
 - Select **Coord 1**
 - Enter **-180** as the Total Angle
 - Uncheck **Auto Execute**
 - Click on the **Point List** panel
 - Select **Point 3**
 - Click **Apply**

The image shows the Patran Student Edition software interface. The 'Geometry' menu is open, with 'Extrude' highlighted (a). The 3D view shows a half-cylinder surface (e). The 'RHS Window' is open, showing the 'Geometry' tab. The 'Action' is 'Create', 'Object' is 'Surface', and 'Method' is 'Extrude'. The 'Surface ID List' contains '1'. The 'Refer. Coordinate Frame' is 'Coord 0'. The 'Origin of Scale and Rotate' is '[0 0 0]'. The 'Translation Vector' is '<0 0 -500>' (b). The 'Sweep Parameters' section shows 'Scale Factor' as '1.0' and 'Angle' as '0.0'. The 'Auto Execute' checkbox is unchecked (c). The 'Curve List' is empty (d). The 'Apply' button is visible (f).

Create a surface (half of a cylinder):

- a. **Geometry** icon (Surfaces tab): **Select/Extrude**
- b. Enter **<0 0 -500>** as the Translation Vector
- c. Uncheck **Auto Execute**
- d. Click on the **Curve List** panel
- e. Select the arc
- f. Click **Apply**

Patran® Student Edition

Load/Boundary Conditions

Action: Create
Object: Contact
Type: Element Uniform
Option: Deformable Body

Current Load Case: Default...

Type: Static

Existing Sets

New Set Name: pipe

Target Element Type: 2D

Input Data...
Select Application Region...
-Apply-

Conditions Select Application Region

Select: FEM

Auto Select...

Application Region

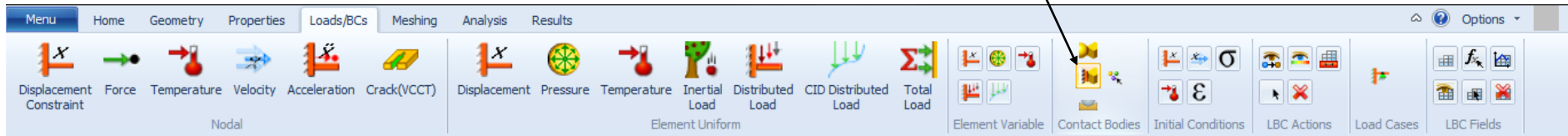
Select 2D Element

Add Remove

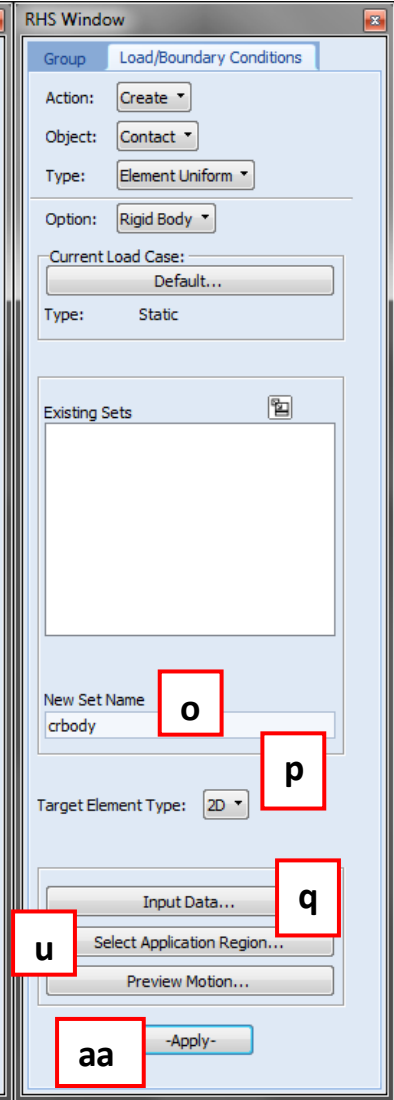
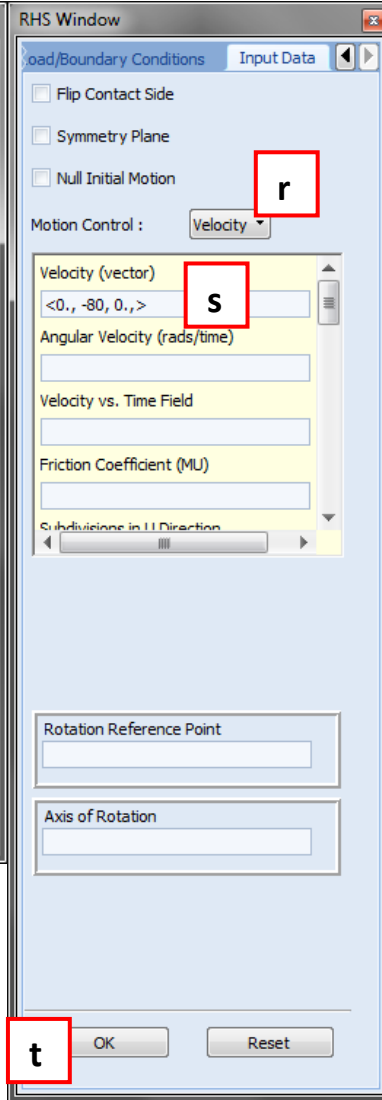
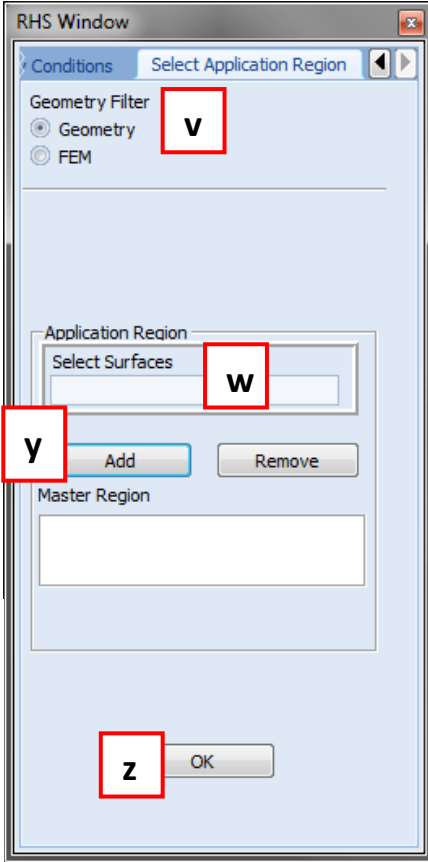
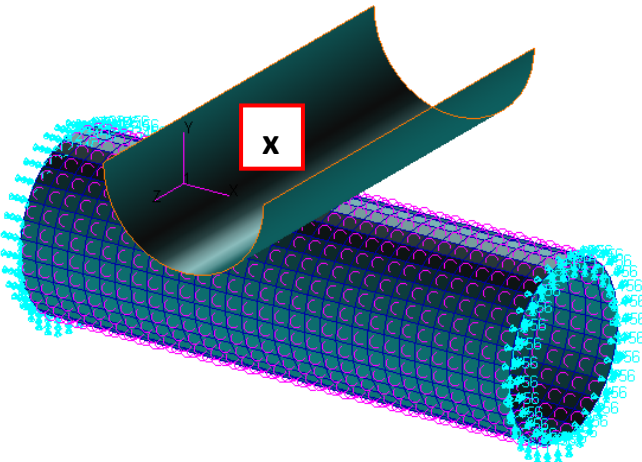
Register Region

OK

- a. Post two groups: **rbody** and **pipe**
 - b. Click on the **Smooth shaded** icon
- Define the contact between the rigid body and the cylindrical element:
- c. Click on the **Loads/BCs** icon
 - d. Loads/BCs: **Contact Bodies** tab/**Deformable** icon
 - e. Enter **pipe** as the New Set Name
 - f. Target Element Type: **2D**
 - g. Click **Select Application Region...**
 - h. Select **FEM**
 - i. Click on the **Select 2D Elements** panel
 - j. Select the elements by clicking and dragging the mouse
 - k. Click **Add**
 - l. Click **OK**
 - m. Click **Apply**



- n. Loads/BCs: **Contact Bodies** tab/**Rigid** icon
- o. Enter **crbody** as the New Set Name
- p. Target Element Type: **2D**
- q. Click **Input Data...**
- r. Motion Control: **Velocity**
- s. Enter **<0 -80 0>** as the Velocity (vector)
- t. Click **OK**
- u. Click **Select Application Region...**
- v. Geometry Filter: **Geometry**
- w. Click on the **Select Surfaces** panel
- x. Select the surface (half of the cylinder)
- y. Click **Add**
- z. Click **OK**
- aa. Click **Apply**



The screenshot displays the ANSYS software interface with several dialog boxes open. The 'Analysis Deck' dialog is at the top, with the 'Analysis Deck' icon highlighted by a red box labeled 'a'. Below it, the 'Results Output Format' dialog is open, with the 'XDB' checkbox checked and 'Print' unchecked, highlighted by a red box labeled 'g'. The 'Solution Parameters' dialog is in the center, with the 'SOL 400 Run' checkbox checked, highlighted by a red box labeled 'e'. The 'RHS Window' dialog is on the right, with the 'IMPLICIT NONLINEAR' radio button selected, highlighted by a red box labeled 'c'. The 'Solution Parameters' dialog also has a 'Results Output Format...' button highlighted by a red box labeled 'f'. The 'RHS Window' dialog has a 'Solution Parameters...' button highlighted by a red box labeled 'd'. The 'RHS Window' dialog also has a 'Solution Type...' button highlighted by a red box labeled 'b'. The 'Results Output Format' dialog has an 'OK' button highlighted by a red box labeled 'h'. The 'Solution Parameters' dialog has an 'OK' button highlighted by a red box labeled 'i'. The 'RHS Window' dialog has an 'OK' button highlighted by a red box labeled 'j'. A red box labeled 'a' is also present in the top left corner of the main window.

- Run a nonlinear analysis:
- Click on the **Analysis icon/ Analysis Deck**
 - Click **Solution Type...**
 - Select **IMPLICIT NONLINEAR** as the Solution Type
 - Click **Solution Parameters...**
 - Check **SOL 400 Run**
 - Click **Results Output Format...**
 - Uncheck **Print** and check **XDB**
 - Click **OK**
 - Click **OK**
 - Click **OK**

The screenshot displays the ANSYS Workbench software interface. The top ribbon includes tabs for Menu, Home, Geometry, Properties, Loads/BCs, Meshing, Analysis, and Results. The Analysis tab is active, showing various analysis-related icons.

Three dialog boxes are open over the main interface:

- Load Increment Parameters:** The **Increment Type** is set to **Adaptive** (labeled 'o'). The **OK** button is highlighted (labeled 'p').
- Static Solution Parameters:** The **Linearity** is set to **NonLinear**. The **Load Increment Params...** button is highlighted (labeled 'n'). The **OK** button is highlighted (labeled 'q').
- Subcases:** The **Solution Sequence** is 400. The **Default** subcase is selected in the **Available Subcases** list (labeled 'l'). The **Subcase Parameters...** button is highlighted (labeled 'm'). The **Apply** button is highlighted (labeled 'k').

The **RHS Window** on the right shows the **Analysis** group. The **Action** is **Analyze**, **Object** is **Entire Model**, and **Method** is **Analysis Deck**. The **Code** is **MSC.Nastran** and the **Type** is **Structural**. The **Job Name** is **Ex_2**. The **Subcase Select...** button at the bottom is highlighted (labeled 'k').

- k. Click **Subcases...**
- l. Select **Default**
- m. Click **Subcase Parameters...**
- n. Click **Load Increment Params...**
- o. Change *Increment Type* to **Adaptive**
- p. Click **OK**
- a. Click **OK**

Menu Home Geometry Properties Loads/BCs Meshing Analysis Results

Analyze Create Existing Deck Optimize Topoptimize Access Results Delete Monitor Actions

Entire Model Selected Group(s) Analysis Deck Read Submit HDF5 XDB Read Output2 Attach Output2 t16/t19 d3plot Job/View Output

r. Click **Output Requests...**

s. Select **Element Strains**

t. Click **OK**

u. Click **Apply**

v. Click **Cancel**

w. Click **Apply**

x. Run **Nastran** analysis using **Ex_2.bdf** file

Standard Results

SUBCASE NAME: Default
SOLUTION SEQUENCE: 400

Form Type: Basic

Select Result Type

- Element Stresses
- Constraint Forces
- Multi-Point Constraint Forces
- Element Forces
- Applied Loads
- Element Strain Energies
- Element Strains** s
- Grid Point Stresses

Output Requests

DISPLACEMENT(SORT1,REAL)=All FEM
STRESS(SORT1,REAL,VONMISES,BILIN)=All FEM;PARAM,NOCO
SPCFORCES(SORT1,REAL)=All FEM
STRAIN(SORT1,REAL,VONMISES,STRCUR,BILIN)=All FEM

Delete

TITLE
This is a default subcase.

SUBTITLE
Default

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

t 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... r
Direct Text Input...
Select Explicit MPCs...

u Apply Cancel v

RHS Window

Group Analysis

Action: Analyze

Object: Entire Model

Method: Analysis Deck

Code: MSC.Nastran

Type: Structural

Available Jobs

Job Name
Ex_2

Job Description (TITLE)

SUBTITLE

LABEL

Translation Parameters...
Solution Type...
Direct Text Input...
Select Superelements...
Subcases...
Subcase Select...

w Apply

The image shows a software interface with a ribbon menu and a right-hand side (RHS) window. The ribbon menu includes tabs for Menu, Home, Geometry, Properties, Loads/BCs, Meshing, Analysis, and Results. The Analysis tab is active, and the XDB icon is highlighted with a red box labeled 'a'. The RHS window is titled 'RHS Window' and has an 'Analysis' tab selected. It contains several dropdown menus: 'Action' set to 'Access Results', 'Object' set to 'Attach XDB', and 'Method' set to 'Result Entities'. Below these are input fields for 'Code' (MSC.Nastran) and 'Type' (Structural). An 'Available Jobs' list contains 'Ex_2'. There are also input fields for 'Job Name' (Ex_2) and 'Job Description (TITLE)'. Below these are fields for 'SUBTITLE' and 'LABEL'. At the bottom of the dialog, there are three buttons: 'Select Results File...' (labeled 'b'), 'Translation Parameters...', and 'Apply' (labeled 'd').

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

- a. Click on the **Analysis** tab/**XDB** icon
- b. Click **Select Results File...**
- c. Select **Ex_2.xdb** file and click **OK**
- d. Click **Apply**

Patran 2020 (Student Edition) 31-Mar-21 19:57:22

Fringe: Default, A1:Non-linear:100. % of Load, Displacements, Translational, Y Component, (NON-LAYERED)

Deform: Default, A1:Non-linear: 100. % of Load, Displacements, Translational,

6.72+00
9.45+01
-4.83+00
-1.06+01
-1.64+01
-2.22+01
-2.79+01
-3.37+01
-3.95+01
-4.53+01
-5.10+01
-5.68+01
-6.26+01
-6.84+01
-7.41+01
-7.99+01

a **b** **d** **e** **f** **g** **h** **i** **j** **k** **l**

a. Click on the **Reset graphics** icon and change **Cycle Background** color to white

b. Click on the **Smooth shaded** icon

c. Post only the **pipe** group: **Menu/Group/Post**

Post-process the results:

d. Click on the **Results** icon/**Fringe/Deformation** icon (Quick Plot tab)

e. Select Result Cases: **A1:Non-linear: 100% of Load**

f. Select Fringe Result: **Displacements, Translational**

g. Quantity: **Y Component**

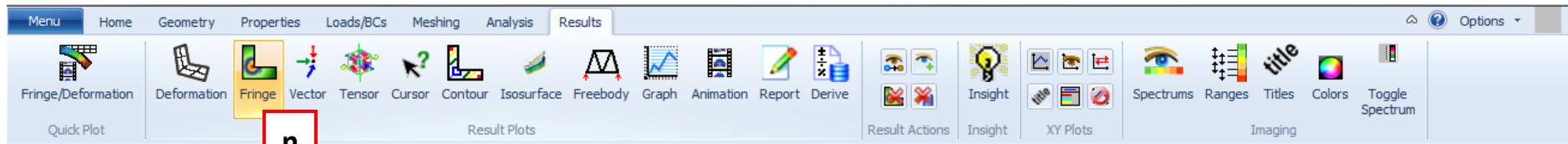
h. Select Deformation Result: **Displacements, Translational**

i. Click on the **Deform Attributes** icon

j. Select **True Scale** with the ScaleFactor equals to **1.0**

k. Uncheck **Show Undeformed**

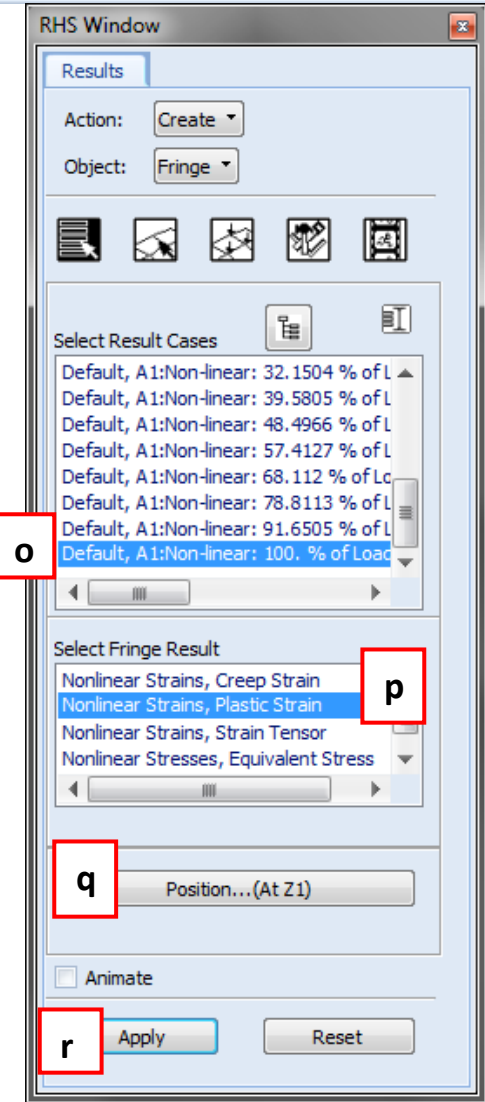
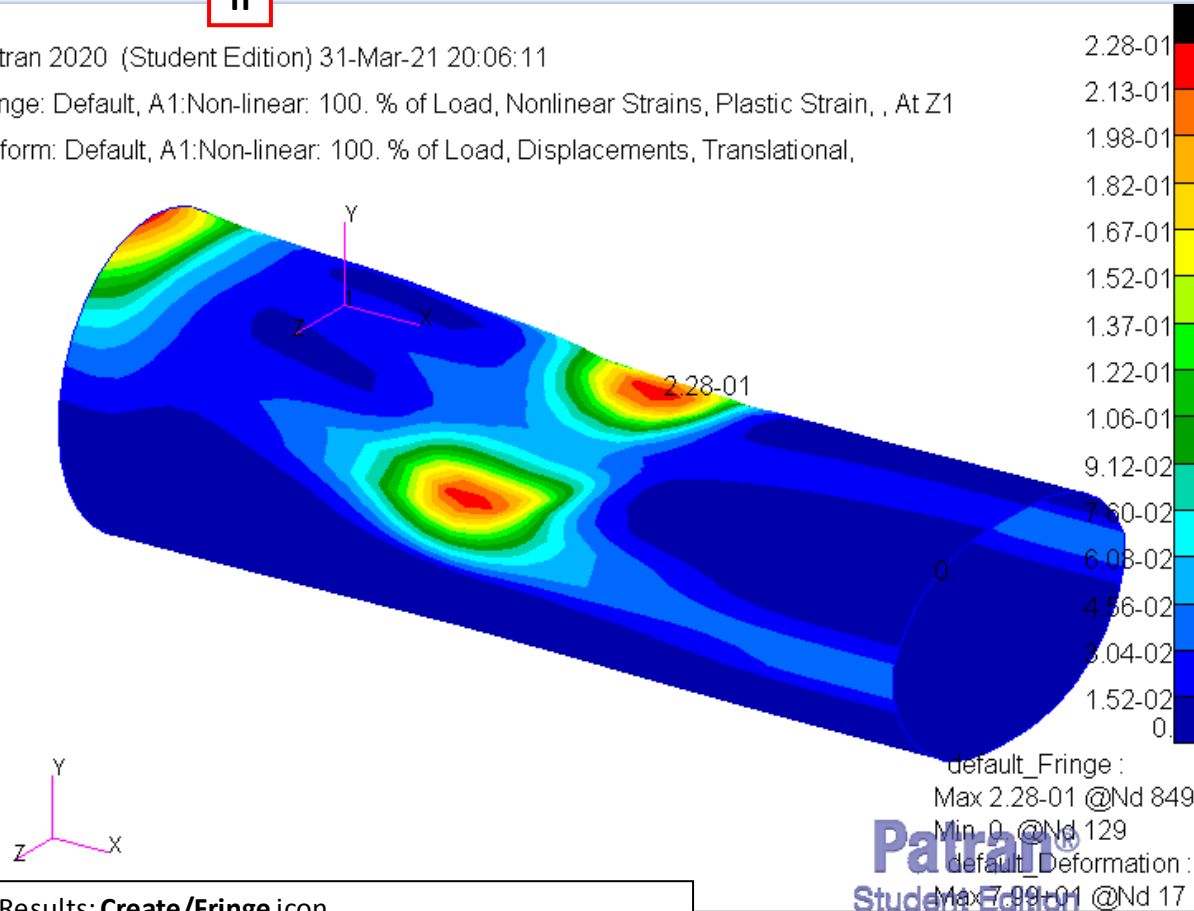
l. Click **Apply**



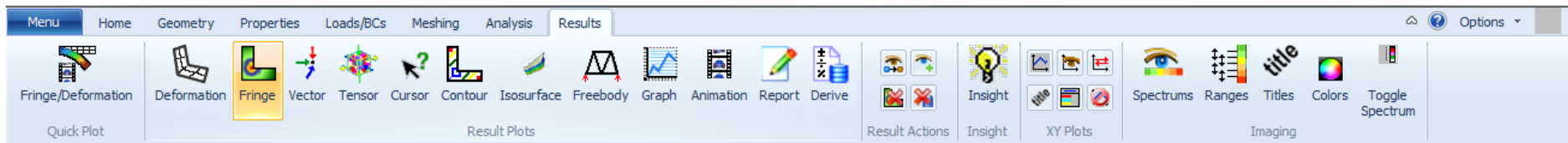
Patran 2020 (Student Edition) 31-Mar-21 20:06:11

Fringe: Default, A1:Non-linear: 100. % of Load, Nonlinear Strains, Plastic Strain, , At Z1

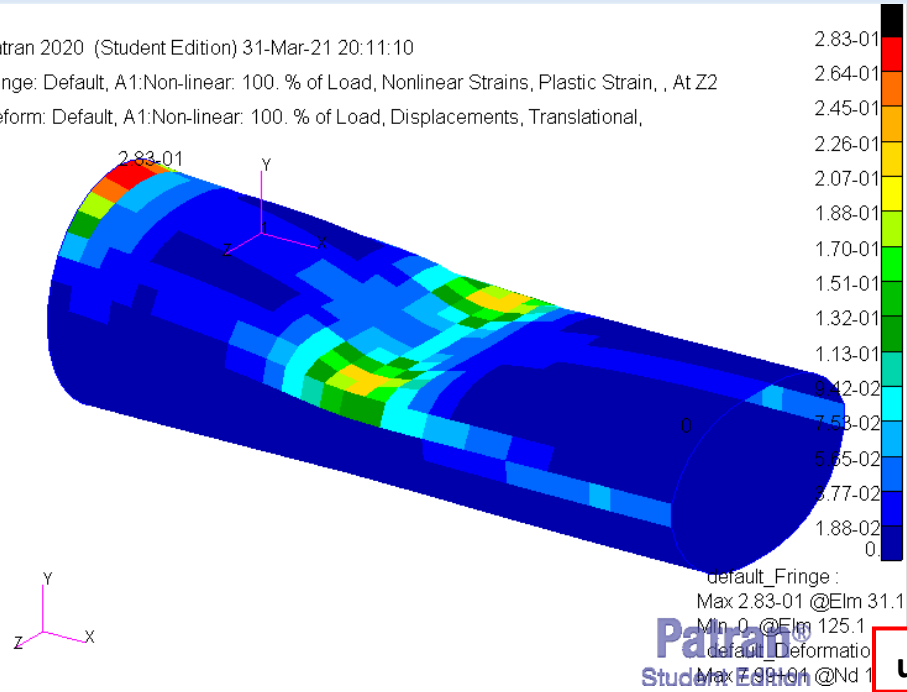
Deform: Default, A1:Non-linear: 100. % of Load, Displacements, Translational,



- n. Results: **Create/Fringe** icon
- o. Select Result Cases: **A1:Non-linear: 100% of Load**
- p. Select **Nonlinear Strains, Plastic Strain**
- q. Position: **At Z1** (bottom layer– in our case: inner layer)
- r. Click **Apply**



Patran 2020 (Student Edition) 31-Mar-21 20:11:10
 Fringe: Default, A1:Non-linear: 100. % of Load, Nonlinear Strains, Plastic Strain, , At Z2
 Deform: Default, A1:Non-linear: 100. % of Load, Displacements, Translational,



RHS Window

Results

Action: Create

Object: Fringe

Scale Factor: 1.0

Filter Values: None

Averaging Definition:

Domain: None

Method: Derive/Average

Extrapolation: Shape Fn.

Use PCL Expression

Define PCL Expression...

Existing Fringe Plots...

Save Fringe Plot As:

v Apply Reset

RHS Window

Results

Action: Create

Object: Fringe

t

Select Result Cases

- Default, A1:Non-linear: 32.1504 % of L
- Default, A1:Non-linear: 39.5805 % of L
- Default, A1:Non-linear: 48.4966 % of L
- Default, A1:Non-linear: 57.4127 % of L
- Default, A1:Non-linear: 68.112 % of L
- Default, A1:Non-linear: 78.8113 % of L
- Default, A1:Non-linear: 91.6505 % of L
- Default, A1:Non-linear: 100. % of Load

Select Fringe Result

- Nonlinear Strains, Creep Strain
- Nonlinear Strains, Plastic Strain
- Nonlinear Strains, Strain Tensor
- Nonlinear Stresses, Equivalent Stress

S Position...(At Z2)

Animate

Apply Reset

- s. Change Position to **At Z2** (top layer – in our case: outer layer)
- t. Click on the **Plot Options** icon
- u. Domain: **None**
- v. Click **Apply**
- w. Save date base and close Patran