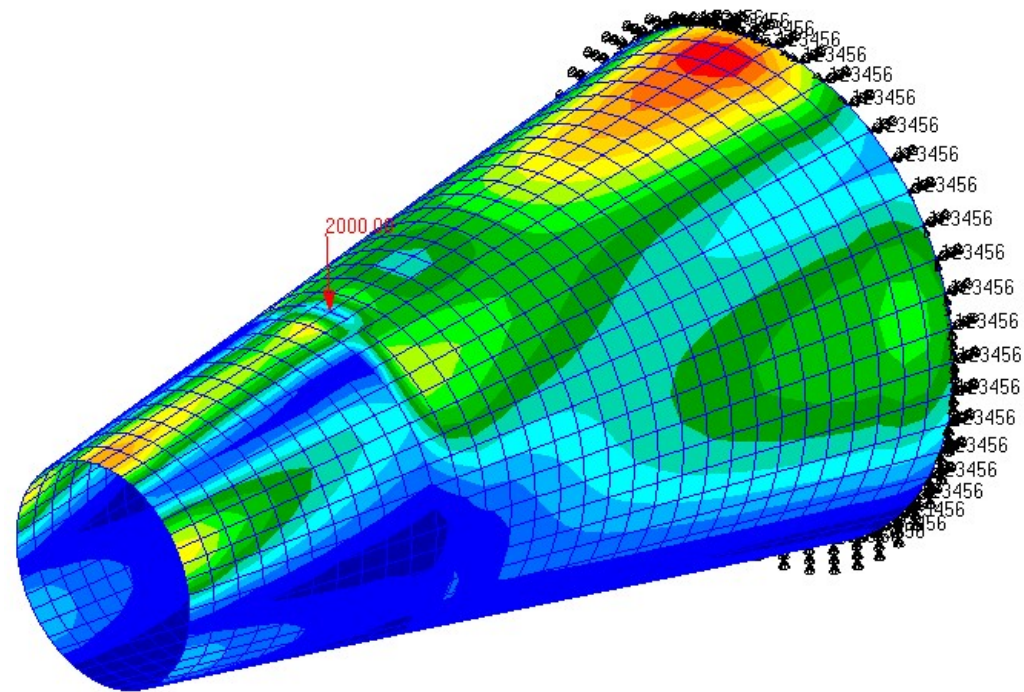
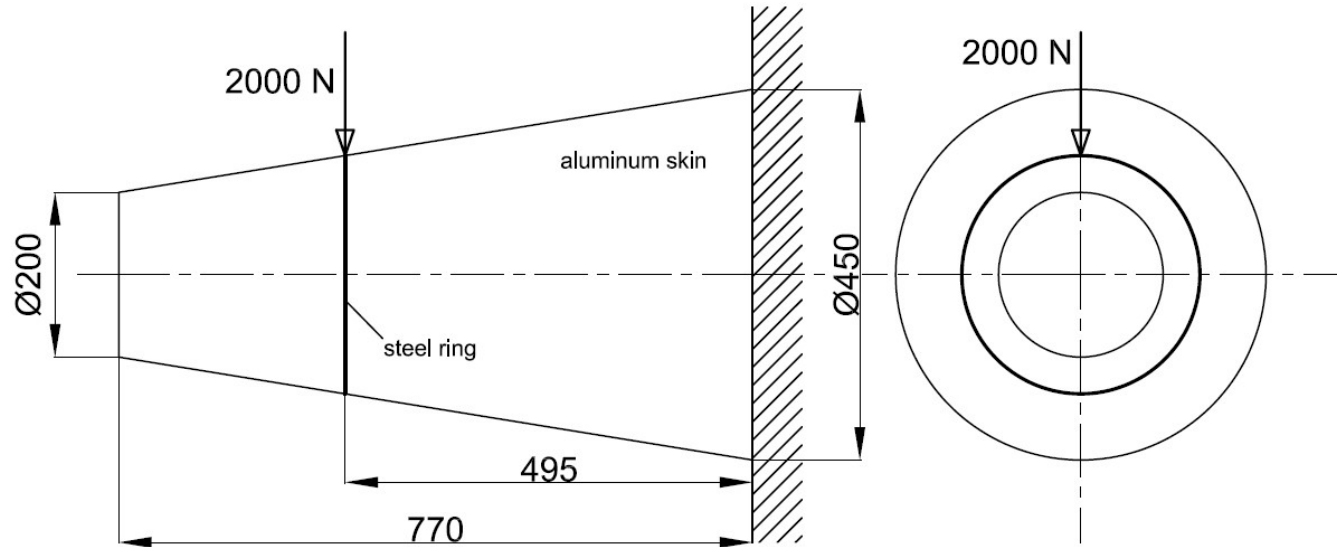


Mechanics of Thin-walled Structures



PROBLEM DESCRIPTION



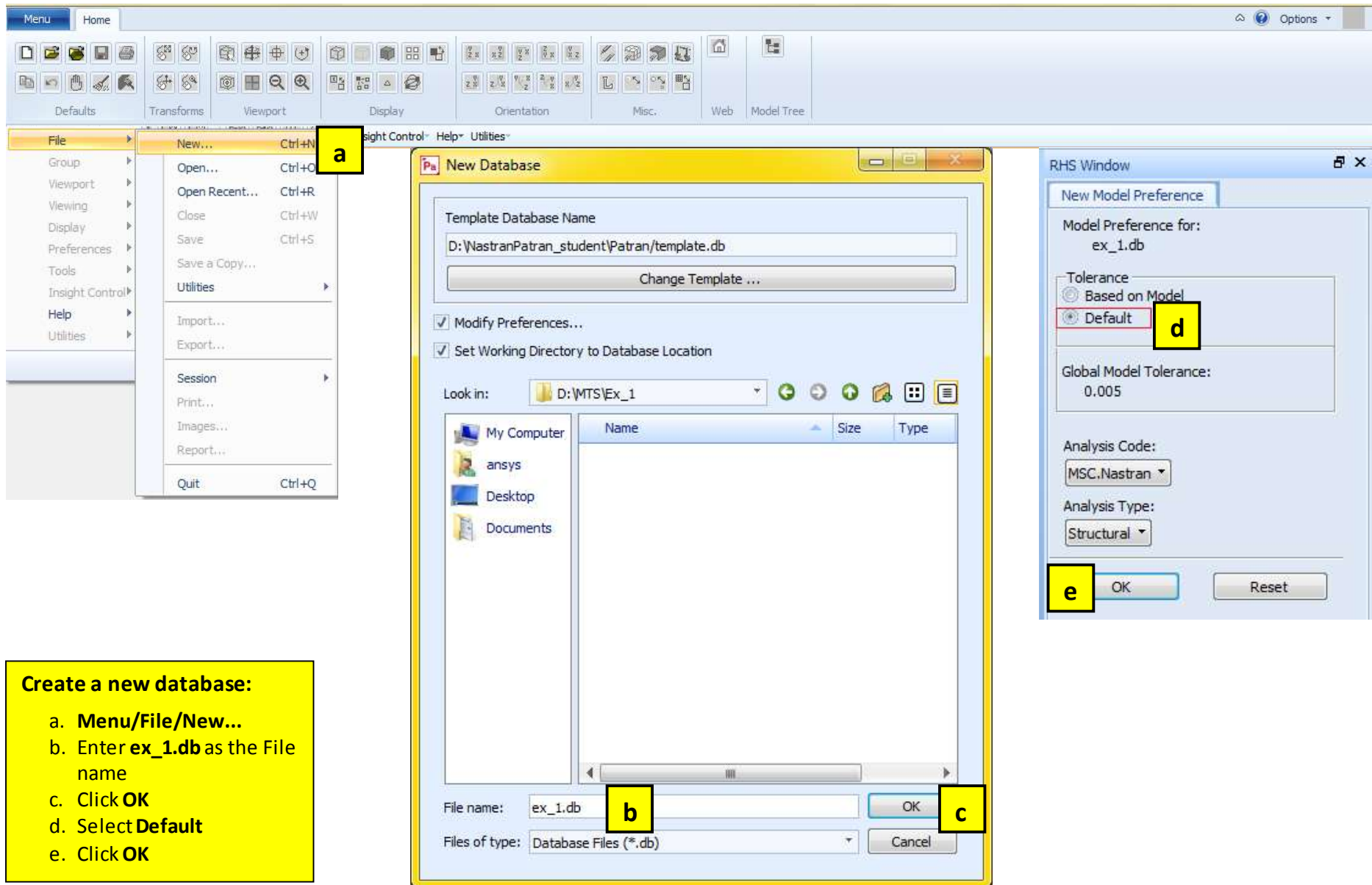
Static, linear analysis of thin-walled aluminum skin, reinforced by a stiffening ring, will be performed.

In the course of this exercise, students will gain basic knowledge about: **creating geometry and mesh; applying loads and boundary conditions; setting up static, linear analysis and results post-processing.**

Two cases will be considered:

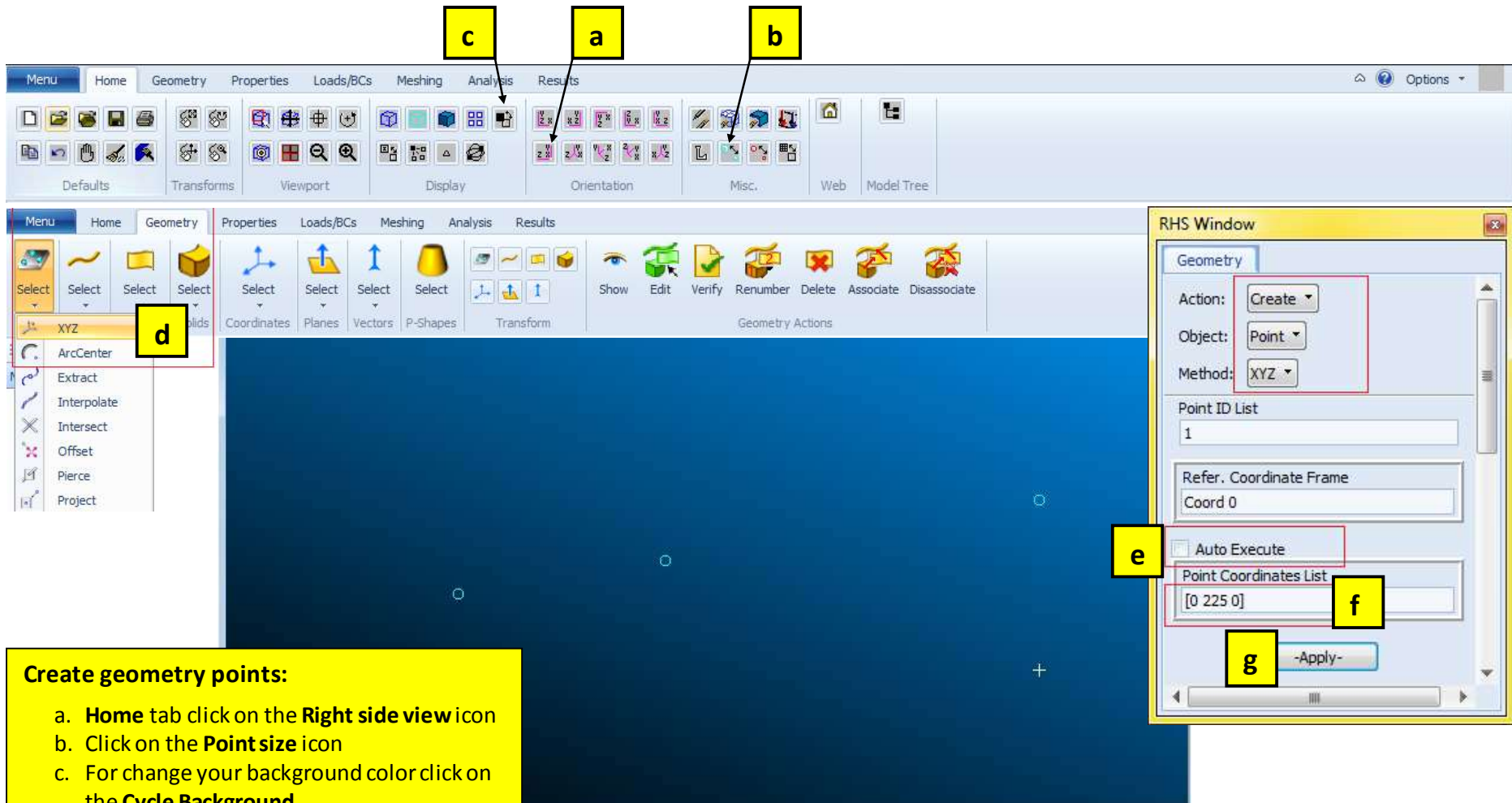
1. When stiffening ring is (relatively) **rigid**
2. When stiffening ring is made of **steel** (and its deformations influence the deformation of the aluminum skin)

Units: mm, N, MPa



Create a new database:

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

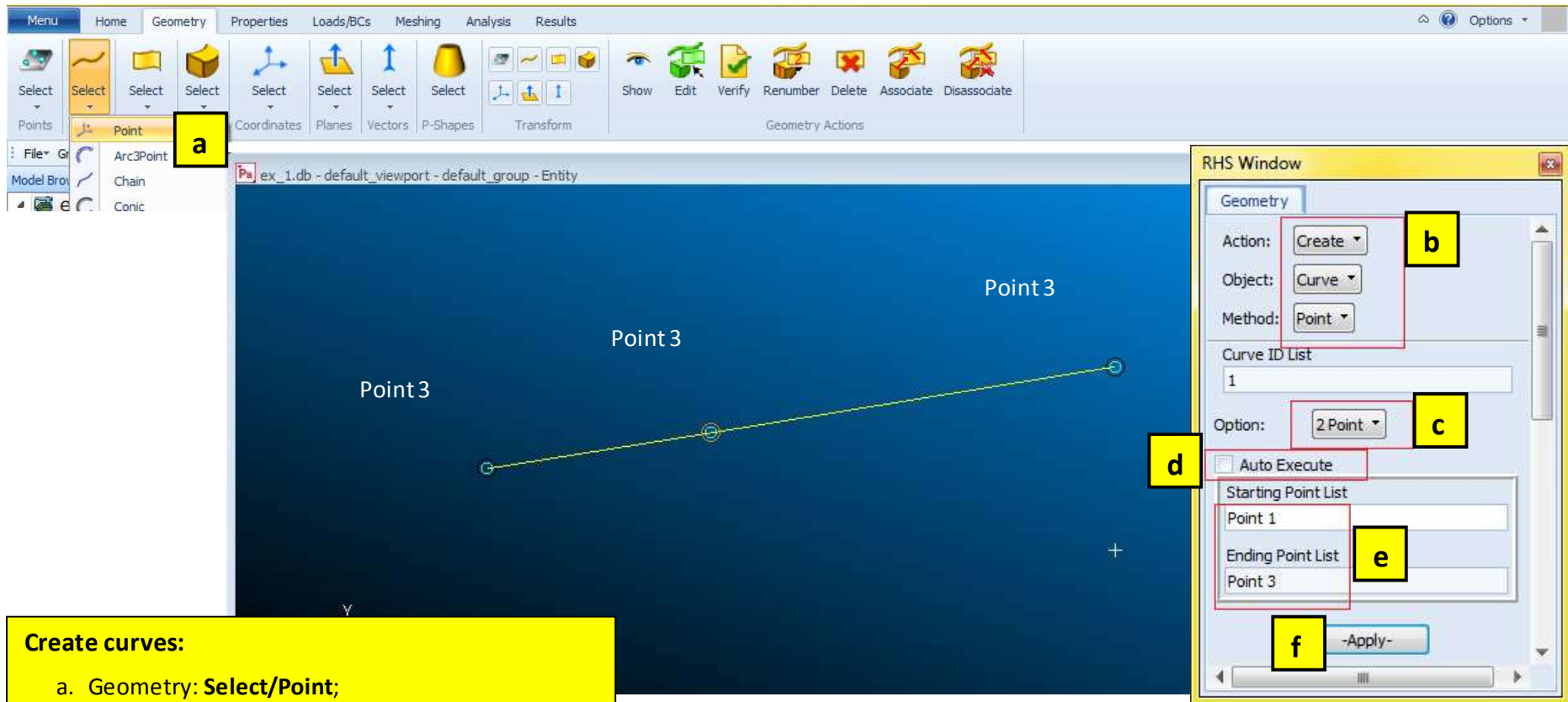


Create geometry points:

- a. Home tab click on the **Right side view** icon
- b. Click on the **Point size** icon
- c. For change your background color click on the **Cycle Background**
- d. Geometry icon: **Select/XYZ (Points icon)**
- e. Uncheck **Auto Execute**
- f. Enter **[0 225 0]** as the Point Coordinates List
- g. Click **Apply**
- h. Create two more points using coordinates: **[0 100 770]**, **[0 144 495]**

WARNING!

The **Auto Execute** option will automatically apply the operation when the last field is filled in.



Create curves:

- a. Geometry: **Select/Point**;
- b. Check out Object and Method: **Curve/Point**
- c. Option: **2 Point**;
- d. Uncheck **Auto Execute**;
- e. Select **Point 1** as the starting point and **Point 3** as the ending point;
- f. Click **Apply**;
- g. Create one more curve using **Point 3** and **Point 2**.

Remark: Points numbers may be different.

Mesh the curves:

- Meshing tab: click on the **Curve** icon
- Topology: **Bar2**
- Click on the **Curve List** panel
- Select all curves by clicking and dragging the mouse
- Uncheck **Automatic Calculation**
- Enter **25** as the Value of the Global Edge Length
- Click **Apply**
- Check out message in the **Command Window** whether elements were created

Command Window Output:

```

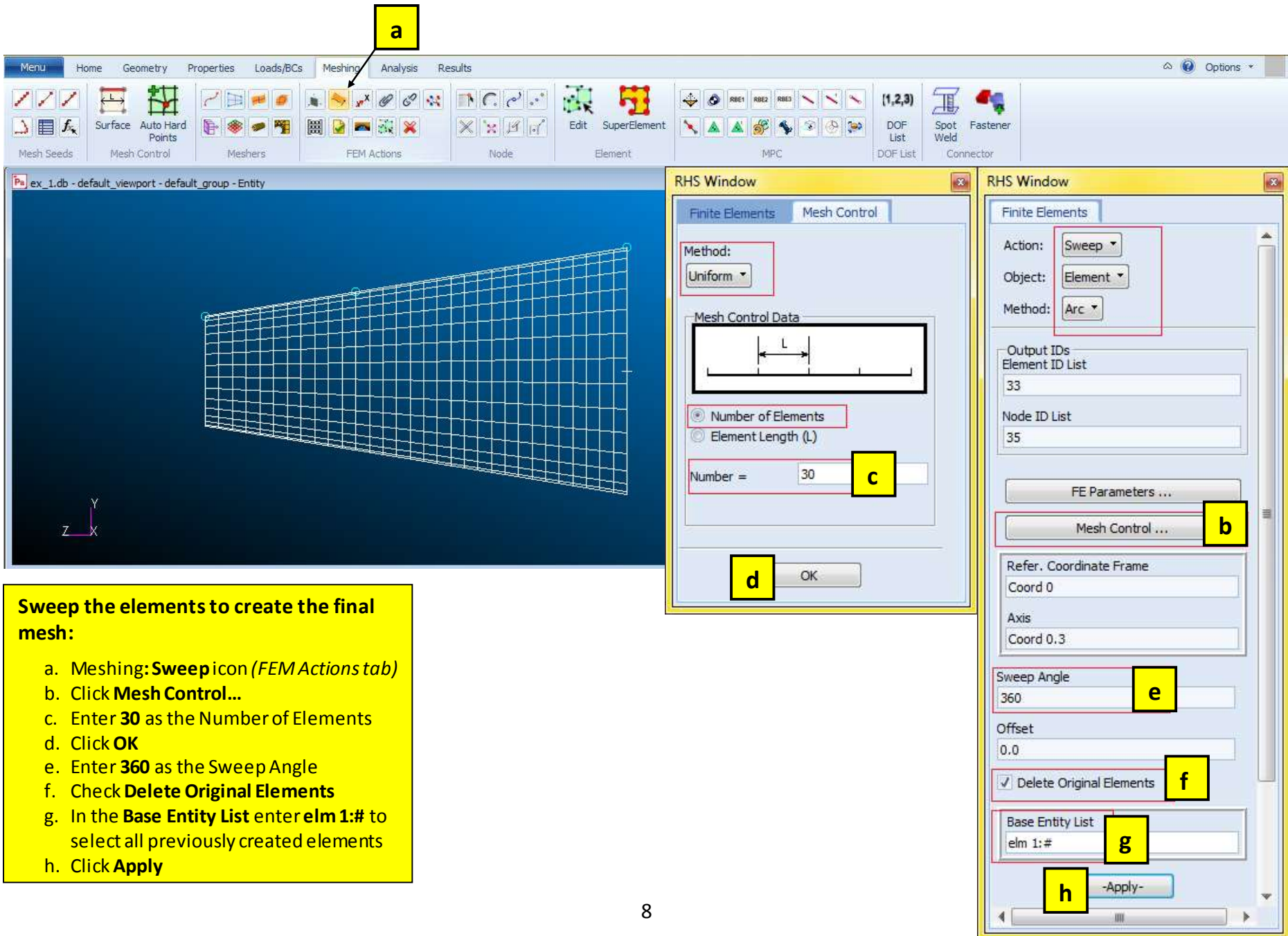
$# 12 nodes and 11 elements created for Curve 2.
$# === 33 nodes created. IDs = 1:33.
$# === 31 elements created. IDs = 1:31.

```

The image shows a software interface for creating a point element. The top toolbar has a yellow box 'a' pointing to the 'Edit' icon in the 'Meshing' section. The main viewport shows a blue background with a yellow line and a point labeled 'Point 3', with a yellow box 'f' pointing to it. The 'RHS Window' on the right has several settings: 'Action: Create', 'Object: Element', 'Method: Edit', 'Element ID List: 32', 'Shape: Point' (with a yellow box 'b'), 'Topology: Point' (with a yellow box 'b'), 'Prop. Name: - None -', 'Prop. Type: - N/A -', 'Use existing midnodes' checked, 'Auto Execute' unchecked, 'Node 1 = Point 3' (with a yellow box 'd'), and an 'Apply' button (with a yellow box 'g'). Other yellow boxes 'c' and 'e' point to icons in the 'RHS Window' toolbar.

Create a point element:

- a. Meshing: **Edit** icon
- b. Shape and Topology: **Point**
- c. Uncheck **Auto Execute**
- d. Click on the **Node 1** panel
- e. Select **Point** icon
- f. Select **Point 3**
- g. Click **Apply**



Delete the duplicate nodes:

- Click on the **Home/Smooth shaded** icon
- Click on the **Iso 1 View** icon
- Elements:
Equivalence/All/Tolerance Cube
- Click **Apply**

RHS Window

Finite Elements

Action: Equivalence

Object: All

Method: Tolerance Cube

Node Id Options:
Retain lower node id

Collapsed Node Options:
Allow Tolerance Reduction

Nodes to be excluded

Equivalencing Tolerance
0.005

Element Boundary Verify
Display Type
 Free Edges Free Faces

Verify Reset

Preview Nodes
Preview Reset

d -Apply-

e →

Verify the element normals:

- Meshing: **Verify** icon
- Test: **Normals**
- Select **Draw Normal Vectors**
- Click **Apply**

Remark:

- To reverse the element normal use **Modify/Element/Reverse**
- Click on the **Element list** panel and select **all curves** by clicking and dragging the mouse
- Click **Apply**

Apply the boundary conditions:

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

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

i. Select **FEM**

j. Click on the **Select Nodes** panel

k. Select nodes by clicking and dragging the mouse

l. Click **Add**

m. Click **OK**

n. Click **Apply**

b

Apply the force:

- Click on the **Home/Iso 1 View** icon
- Loads/BCs: **Force** icon
- Enter **load** as the New Set Name
- Click **Input Data...**
- Enter **<0,-2000,0>** for the Force
- Click **OK**

e

c

d

f

Point 3

Y
Z X

g. Click **Select Application Region...**
 h. Select **Geometry**
 i. Click on the **Select Geometry Entities** panel and enter: **Point 3**
 j. Click **Add**
 k. Click **OK**
 l. Click **Apply**

a

b

c

d

e

f

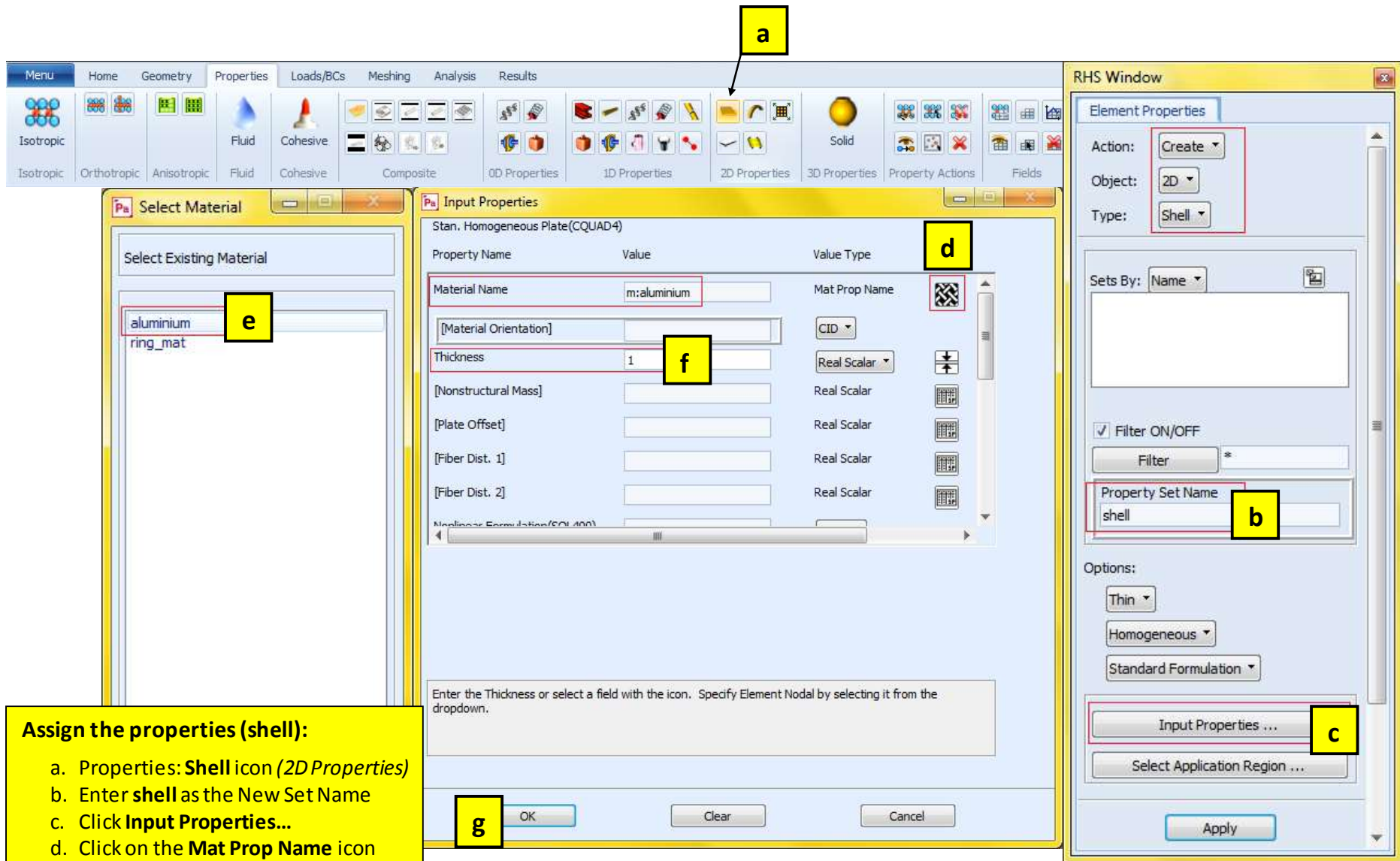
g

h

i

Define materials:

- Click on the **Home** tab/**Reset graphics** icon
- Click **Smooth shaded** icon
- Properties tab: **Isotropic** icon
- Enter **aluminum** as the Material Name
- Click **Input Properties...**
- Enter **73000** as Elastic Modulus and **0.33** as Poisson Ratio
- Click **OK**
- Click **Apply**
- Repeat steps **d-h** to define the second, relatively stiff material e.g.: **ring_mat**, $E = 2e7$, $\nu = 0.29$



Assign the properties (shell):

- Properties: **Shell** icon (*2D Properties*)
- Enter **shell** as the New Set Name
- Click **Input Properties...**
- Click on the **Mat Prop Name** icon
- Select **aluminium**
- Enter **1** 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**

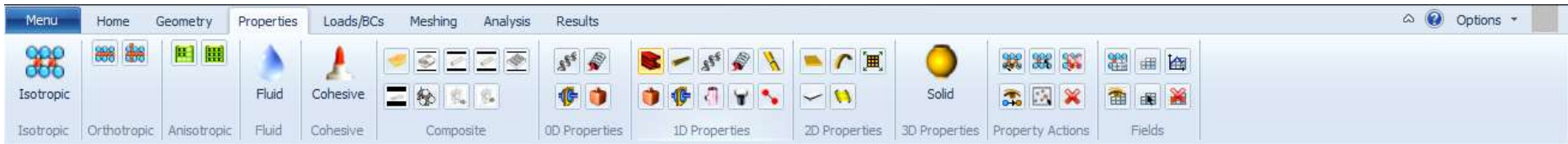
m. Click **OK**



n. Click **Apply**

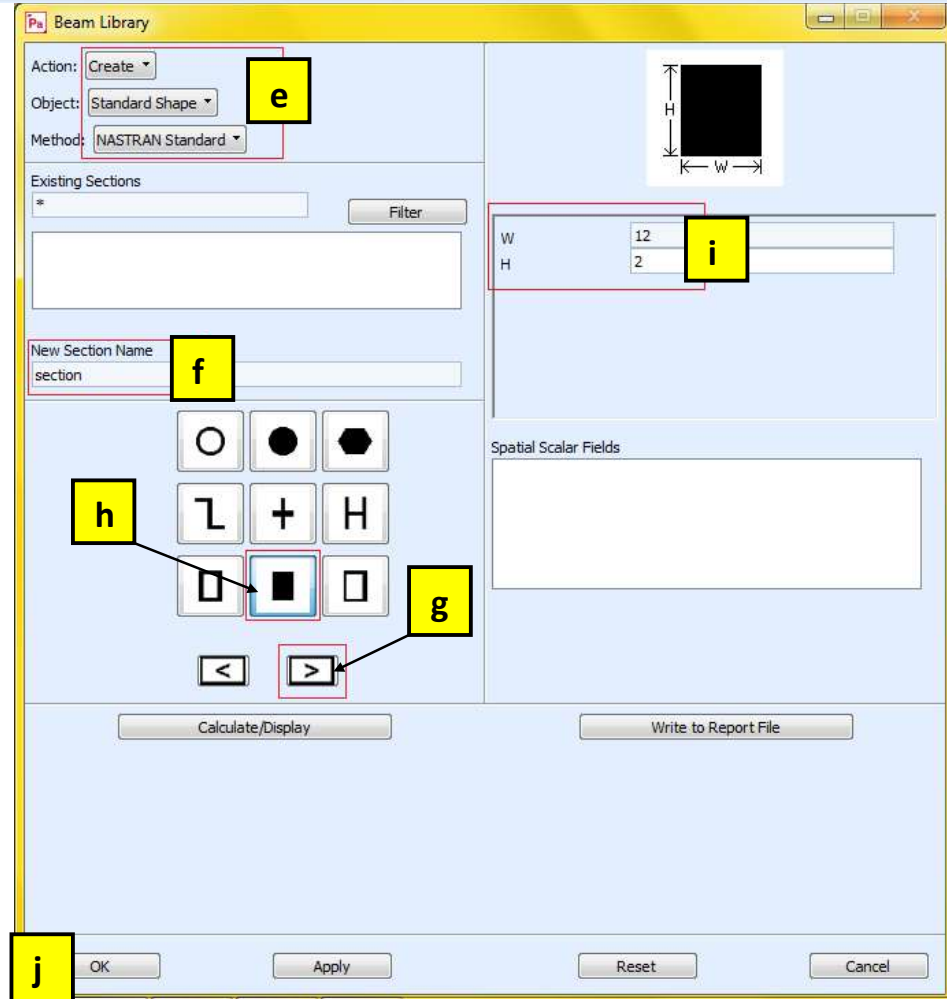
Assign the properties (ring):

- Properties: **Beam icon** (1D Properties tab)
- Enter **ring** as the New Set Name
- Click **Input Properties...**
- Click on the **Create Sections** icon

The screenshot shows the ANSYS software interface with the 'Properties' tab selected. The '1D Properties' section is active, and the 'Input Properties' dialog is open. The 'Create Sections' icon is highlighted with a yellow box labeled 'd'. The 'RHS Window' is also open, showing the 'Element Properties' section with 'Action' set to 'Create', 'Object' set to '1D', and 'Type' set to 'Beam'. The 'Property Set Name' is 'ring', highlighted with a yellow box labeled 'b'. The 'Input Properties...' button is highlighted with a yellow box labeled 'c'. The 'Beam icon' in the '1D Properties' section is highlighted with a yellow box labeled 'a'.



- e. Create Sections: **Create/Standard Shape/NASTRAN Standard**
- f. Enter **section** as the New Section Name
- g. Click 
- h. Click 
- i. Enter **12** as the value of **W** and **2** as the value of **H**
- j. Click **OK**



k. Click on the **Mat Prop Name** icon
 l. Select **ring_mat**
 m. Enter **<0 0 1>** for the **Bar Orientation**
 n. Click **OK**

o. Click **Select Application Region...**

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

q. Select **Beam element** icon

r. Select all beam elements by clicking and dragging the mouse

s. Click **Add**

t. Click **OK**

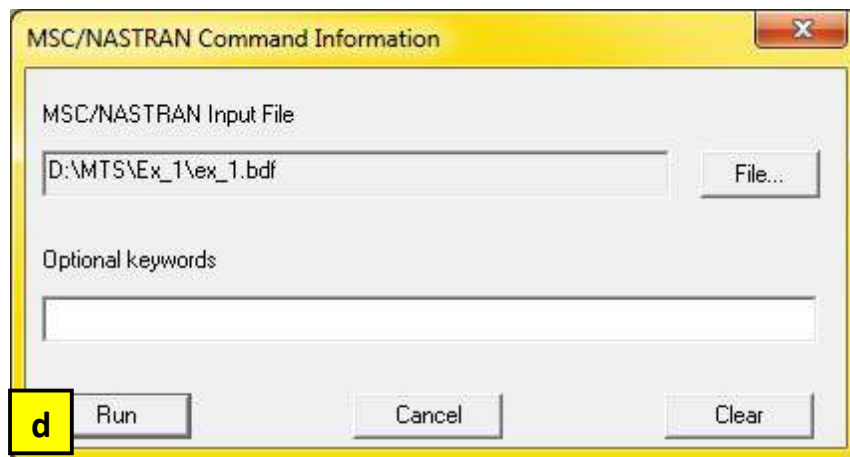
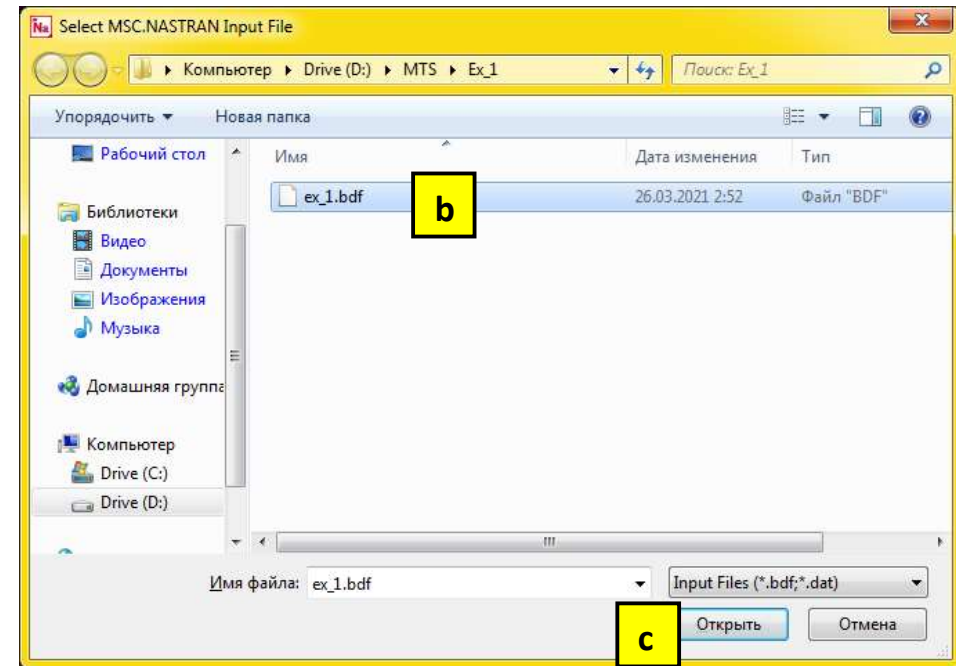
u. Click **Apply**

Run a linear analysis:

- Click on the Analysis icon: **Analysis Deck** icon
- Click **Solution Type...**
- Select **LINEAR STATIC** as the Solution Type
- Click **Solution Parameters...**
- Click **Results Output Format...**
- Uncheck **Print** and check **XDB**
- Click **OK**
- Click **OK**
- Click **OK**
- Click **Apply**

Run **Nastran** analysis:

- a. Open the **NASTRAN**
- b. Select **ex_1.bdf** file
- c. Click **Open**
- d. Click **Run**



The image shows the MSC Nastran software interface with three main components: a ribbon menu, a file selection dialog, and a right-hand side (RHS) window.

- Ribbon Menu:** The 'Analysis' tab is active, and the 'XDB' icon is highlighted with a yellow box labeled 'a'.
- Select File Dialog:** A 'Select File' dialog box is open, showing the file 'ex_1.xdb' selected in the file list. The file name 'ex_1.xdb' is also entered in the 'File name' field. A yellow box labeled 'c' highlights the selected file.
- RHS Window:** The 'Analysis' section is active. The 'Action' is set to 'Access Results', 'Object' to 'Attach XDB', and 'Method' to 'Result Entities'. The 'Available Jobs' list contains 'ex_1'. The 'Job Name' is 'ex_1' and the 'Job Description (TITLE)' is 'MSC.Nastran job created on 24-Mar-21 at 02:53:31'. At the bottom, the 'Select Results File...' button is highlighted with a yellow box labeled 'b', and the 'Apply' button is highlighted with a yellow box labeled 'd'.

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

- Click **Analysis** tab/**XDB** icon
- Click **Select Results File...**
- Select **ex_1.xdb** file and click **OK**
- Click **Apply**

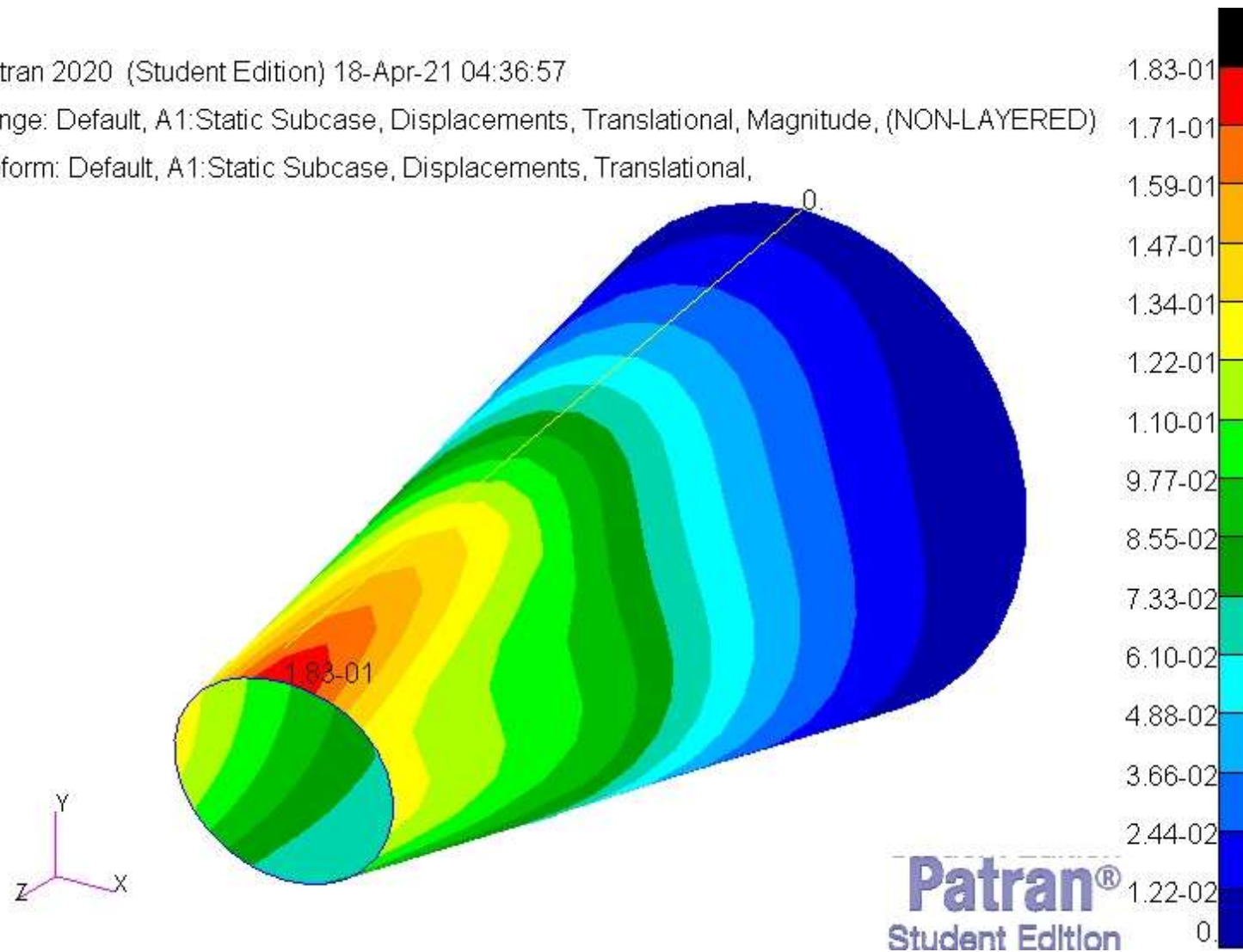
Post-process the results:

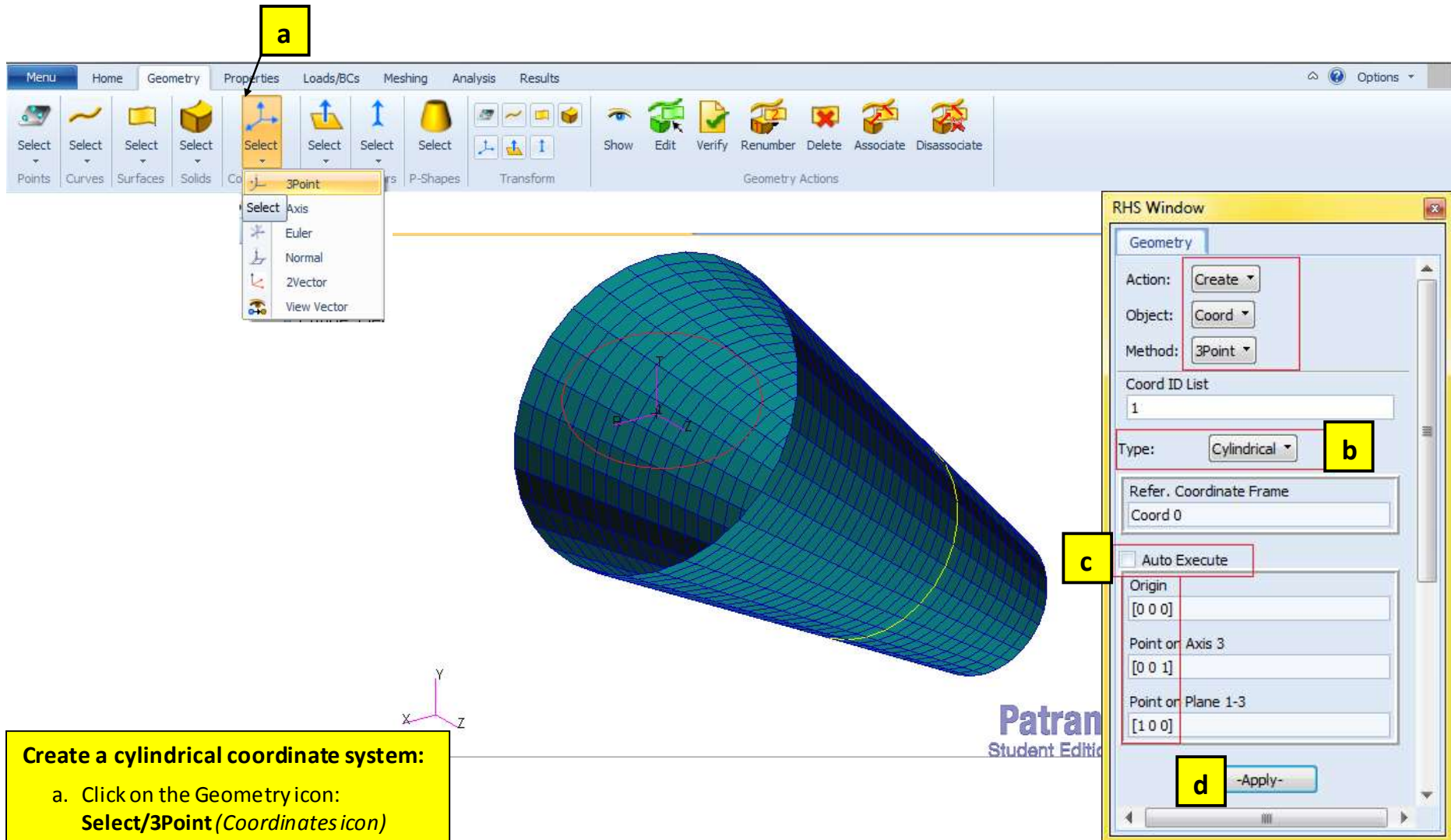
- Click **Home/Plot/Erase Geometry** icon
- Click on the Results icon: **Fringe/Deformation** icon
- Select Result Cases: **Default, A1:Static Subcase**
- Select Fringe Result: **Displacements, Translational**
- Quantity: **Y Component**
- Select Deformation Result: **Displacements, Translational**
- Click on the **Deform Attributes** icon
- Enter **0.04** as the Scale Factor
- Uncheck **Show Undeformed**
- Click **Apply**

Patran 2020 (Student Edition) 18-Apr-21 04:36:57

Fringe: Default, A1:Static Subcase, Displacements, Translational, Magnitude, (NON-LAYERED)

Deform: Default, A1:Static Subcase, Displacements, Translational,





Create a cylindrical coordinate system:

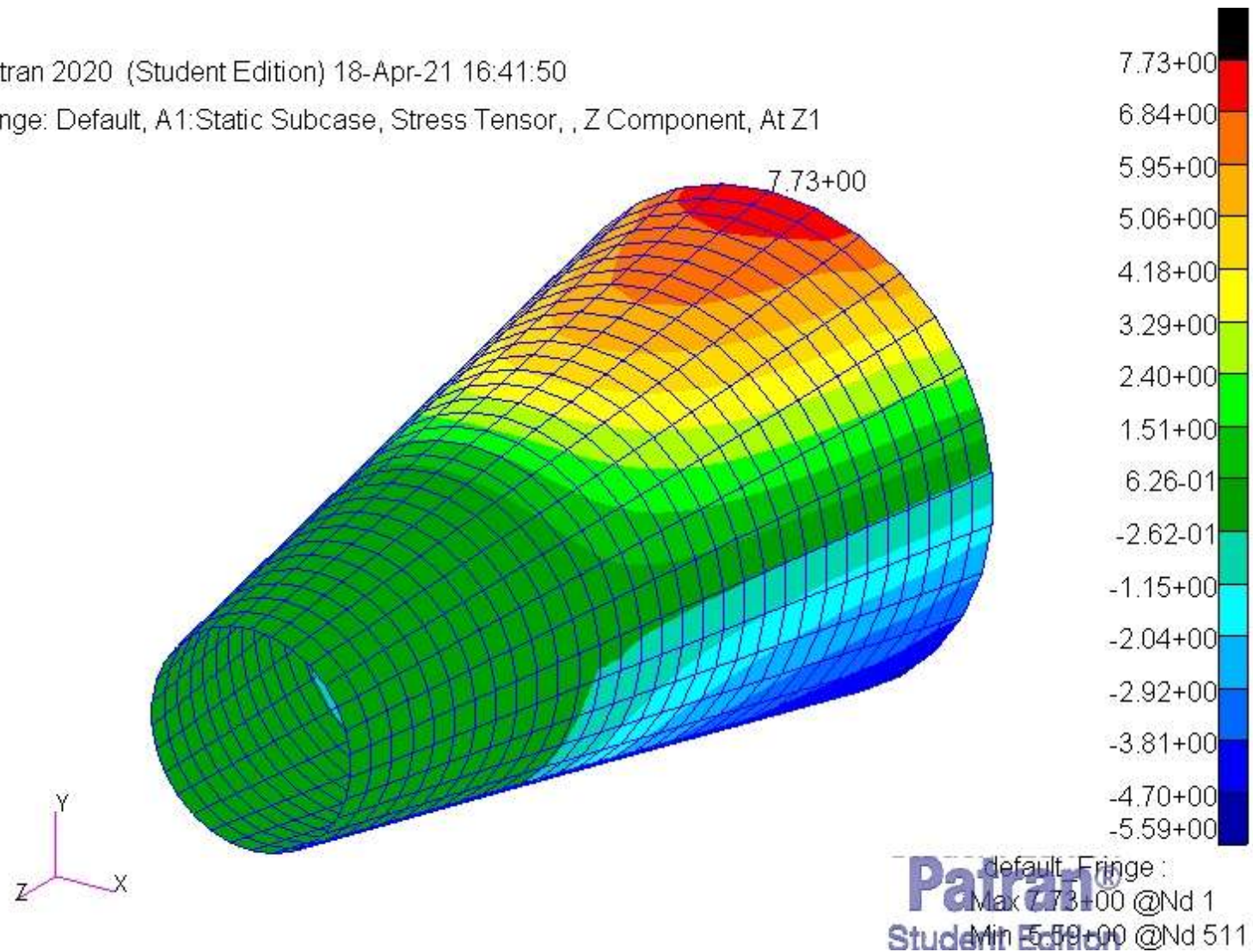
- Click on the Geometry icon: **Select/3Point (Coordinates icon)**
- Type: **Cylindrical**
- Uncheck **Auto Execute**
- Click **Apply**
- Click **Home/Smooth shaded** icon
- Click on the **Iso 4 View** icon

Post-process the results:

- Click on the Results: **Fringe** icon
- Select Result Cases: **Default, A1:StaticSubcase**
- Select Fringe Result: **Stress Tensor**
- Quantity: **Z Component** (axial stress)
- Click on the **Plot Options** icon
- Select **CID**
- Click on the **Select Coordinate Frame** panel
- Select the cylindrical coordinate system
- Click **Apply**

Patran 2020 (Student Edition) 18-Apr-21 16:41:50

Fringe: Default, A1:Static Subcase, Stress Tensor, , Z Component, At Z1



Create a graph:

- Click **Reset Graphics** icon
- Click **Smooth shaded** icon
- Click **Top view** icon
- Results: **Graph** icon
- Select Result Cases: **Default, A1:Static Subcase**
- Select Fringe Result: **Stress Tensor**
- Position: **At Z2** (top layer)
Quantity: **Z Component** (axial stress)
- X: **Path Length**
- Click on the **Target Entities** icon
- Click on the **Select Path Points** panel
- Select **Node** icon
- Select nodes by clicking and dragging the mouse
- Click on the **Plot Options** icon

The image shows a software interface with a ribbon menu at the top and a graph window in the center. The ribbon menu includes sections for 'Quick Plot' (Fringe/Deformation, Deformation, Fringe, Vector, Tensor, Cursor, Contour, Isosurface, Freebody, Graph, Animation, Report, Derive), 'Result Plots' (Result Actions, Insight, XY Plots), and 'Spectrums'. The central graph window, titled 'ex_1.db - 3', displays a red parabolic curve representing 'Result1: Stress Tensor, ZZ'. The y-axis is labeled 'Stress Tensor' and ranges from $-7,500 \times 10^0$ to $7,500 \times 10^0$. The x-axis is labeled 'Path Length' and ranges from $0,000 \times 10^0$ to $1,500 \times 10^3$. To the right is the 'RHS Window' configuration panel. It has tabs for 'Plot/Erase' and 'Results'. The 'Results' tab is active, showing settings for 'Action: Create', 'Object: Graph', and 'Method: Y vs X'. A red box labeled 'q' highlights a set of icons. Below this, 'Coordinate Transformation:' is set to 'CID' (labeled 'n'), and 'Select Coordinate Frame' is set to 'Coord 1' (labeled 'o'). Other settings include 'Scale Factor: 1.0', 'Filter Values: None', 'Averaging Definition: Domain: All Entities, Method: Derive/Average', and 'Extrapolation: Shape Fn.'. At the bottom, an 'Apply' button is labeled 'p' and a 'Reset' button is also visible.

n. Select **CID**
o. Select the cylindrical coordinate system
p. Click **Apply**
q. Click **Select Results** icon

The image displays the ANSYS Workbench interface. The main window shows a plot of Stress Tensor (Pa) versus Path Length. The plot contains two curves: a red line for 'Result1: Stress Tensor, ZZ' and a green line for 'Result1: Stress Tensor, ZZ'. The Y-axis ranges from $-7,500 \times 10^0$ to $7,500 \times 10^0$ Pa, and the X-axis ranges from $0,000 \times 10^0$ to $1,500 \times 10^3$ Path Length.

Two configuration windows are open:

- Left RHS Window:** Shows settings for the plot. The 'Method' is 'Y vs X'. The 'Curve Fit' is 'Linear'. The 'X Axis Label' is 'Path Length'. The 'Y Axis Label' is 'Stress Tensor,'. The 'Y Axis Scale' is 'Linear'. The 'XY Window Name' is '3'. The 'Append Curves in XY Window' checkbox is checked.
- Right RHS Window:** Shows settings for the plot. The 'Method' is 'Y vs X'. The 'Y' is 'Result'. The 'Select Y Result' list includes 'Stress Tensor,'. The 'Position...(At Z1)' button is highlighted. The 'Quantity' is 'Z Component'. The 'X' is 'Path Length'.

Yellow boxes with letters 'r', 's', 't', and 'u' highlight specific UI elements:

- r:** Position...(At Z1) button in the right RHS window.
- s:** Display Attributes icon in the right RHS window.
- t:** Append Curves in XY Window checkbox in the left RHS window.
- u:** Apply button in the left RHS window.

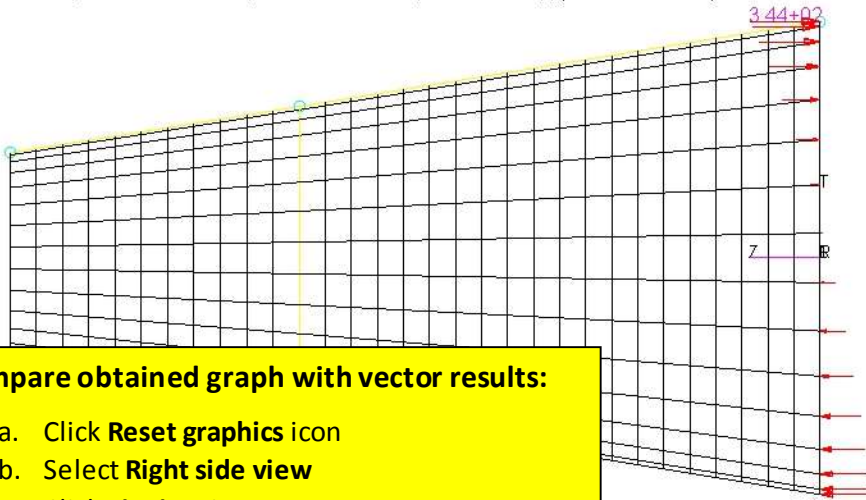
- r. Change **Position...** to **At Z1** (bottom layer)
- s. Click on the **Display Attributes** icon
- t. Check **Append Curves in XY Window**
- u. Click **Apply**

Why do results for both layers differ? Compare foregoing graphs with the constraint forces distribution from the next slide.



Patran 2020 (Student Edition) 18-Apr-21 05:11

Vector: Default, A1:Static Subcase, Constraint Forces, Translational, (NON-LAYERED)



Patran Student Edition
 Default Vector:
 Max: 1.11+02 @Nd 1
 Min: 0.00 @Nd 2

- Compare obtained graph with vector results:**
- Click **Reset graphics** icon
 - Select **Right side view**
 - Click **Fit view** icon
 - Click on Results: **Vector** icon
 - Select Vector Result: **Constraint Forces, Translational**
 - Change option to **Component** and check only **ZZ** option
 - Click on **Display Attributes** icon
 - Check **Constant** vector color option and choose red color for **ZZ**: component
 - Change **Length** to **Model Scaled**
 - Uncheck **Show Vector Label**
 - Click **Apply**

RHS Window

Plot/Erase Results

Action: Create
 Object: Marker
 Method: Vector

Spectrum
 Constant
 XX: ■ YY: ■ ZZ: ■
 Y+Z: ■ Z+X: ■ X+Y: ■
 X+Y+Z: ■

Show Viewport Legend

Vector definition

Length: Model Scaled
 Scale Factor: 0.1
 Anchor Style:
 Head Style:
 Line Style:

Title Editor...

Show Title Lock Title

Show Max/Min Labels
 Show Vector Label
 Label Style...

Show on Deformed

Apply Reset

RHS Window

Plot/Erase Results

Action: Create
 Object: Marker
 Method: Vector

Select Result Cases

Default, A1:Static Subcase;-MSC.NASTRAN

Select Vector Result

Constraint Forces, Rotational
 Constraint Forces, Translational
 Displacements, Rotational
 Displacements, Translational
 Stress Tensor

Show As: Component
 XX YY ZZ
 Y+Z Z+X X+Y
 X+Y+Z

Animate

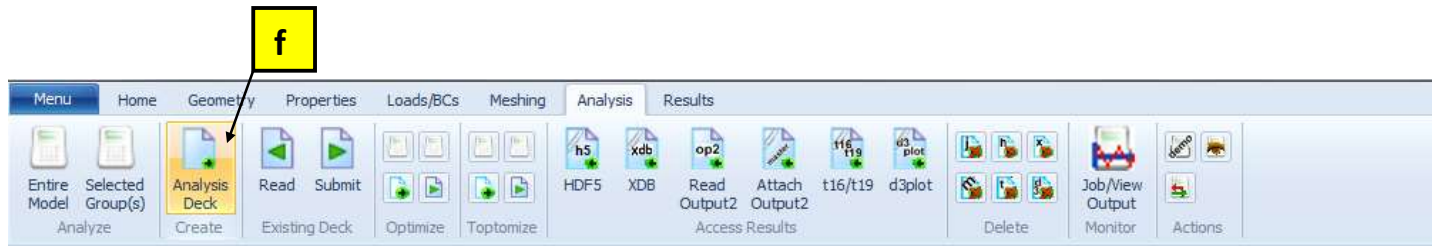
Apply Reset

Change ring material properties, run second analysis and compare results with recently obtained:

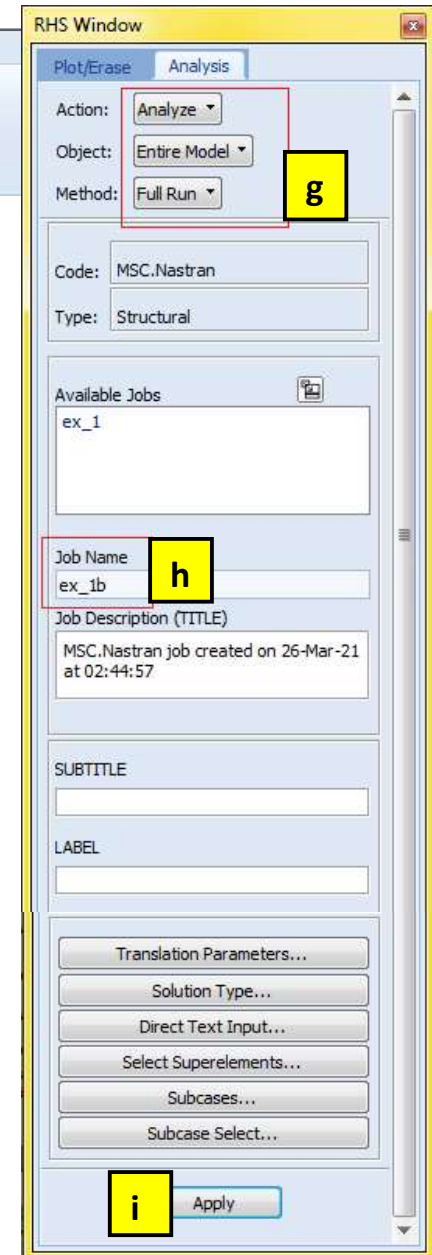
- Go to **Properties** tab and select **Modify/Isotropic**
- Choose **ring_mat**
- Change Young's Modulus to **2e5** (Note that previous material was 100 times stiffer)
- Click **OK**
- Click **Apply**

The screenshot shows the ANSYS software interface with the following components:

- Properties Toolbar:** The **Isotropic** button is highlighted with a yellow box labeled 'a'.
- Input Options Dialog:**
 - The **Constitutive Model** is set to **Linear Elastic**.
 - The **Elastic Modulus** is set to **200000.**, highlighted with a yellow box labeled 'c'.
 - The **Poisson Ratio** is set to **0.28999999**.
 - The **OK** button is highlighted with a yellow box labeled 'd'.
- RHS Window:**
 - The **Materials** tab is active.
 - The **Action** is set to **Modify**, highlighted with a yellow box labeled 'a'.
 - The **Object** is set to **Isotropic**, highlighted with a yellow box labeled 'a'.
 - The **Existing Materials** list contains **aluminum** and **ring_mat**, with **ring_mat** selected and highlighted with a yellow box labeled 'b'.
 - The **Filter** checkbox is checked.
 - The **New Material Name** is **ring_mat**.
 - The **Description** shows **Date: 26-Mar-21** and **Time: 02:24:14**.
 - The **Apply** button is highlighted with a yellow box labeled 'e'.



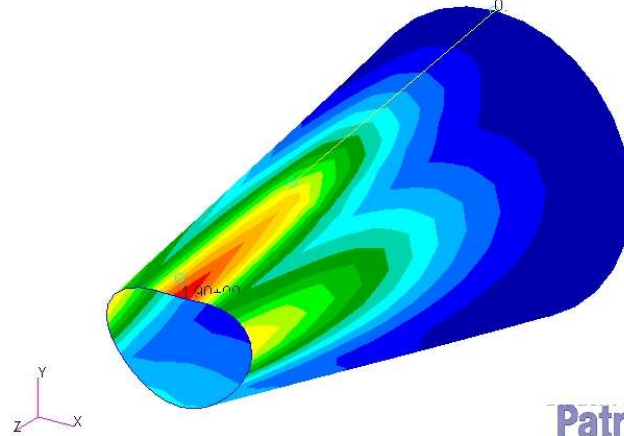
- f. Click Analysis tab: **Analysis Deck** icon
- g. Method: **Full Run**
- h. Change **Job Name** to **ex_1b**
- i. Select **Apply**
- j. After analysis is done, **attach ex_1b.xdb** file and **Plot results** in the same manner as previously. Results referring to the second analysis will be tagged as **A2: Static subcase**



Use following reference to verify the results of the second analysis

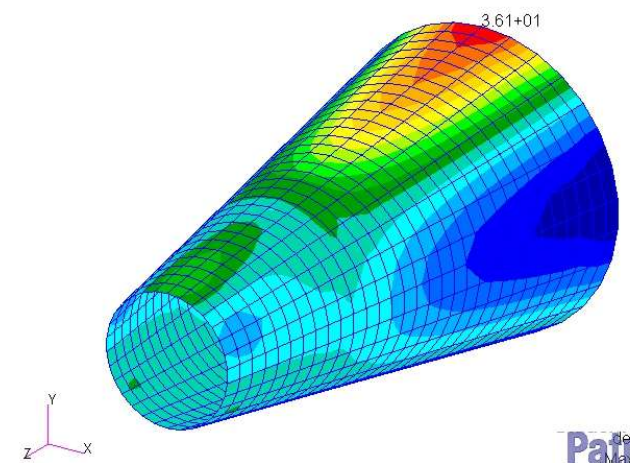
- What qualitative changes can be noticed?
- Which calculation yield results more similar to the theoretical solution? Why?

Patran 2020 (Student Edition) 18-Apr-21 16:20:23
 Fringe: Default, A2:Static Subcase, Displacements, Translational, Magnitude, (NON-LAYERED)
 Deform: Default, A2:Static Subcase, Displacements, Translational,

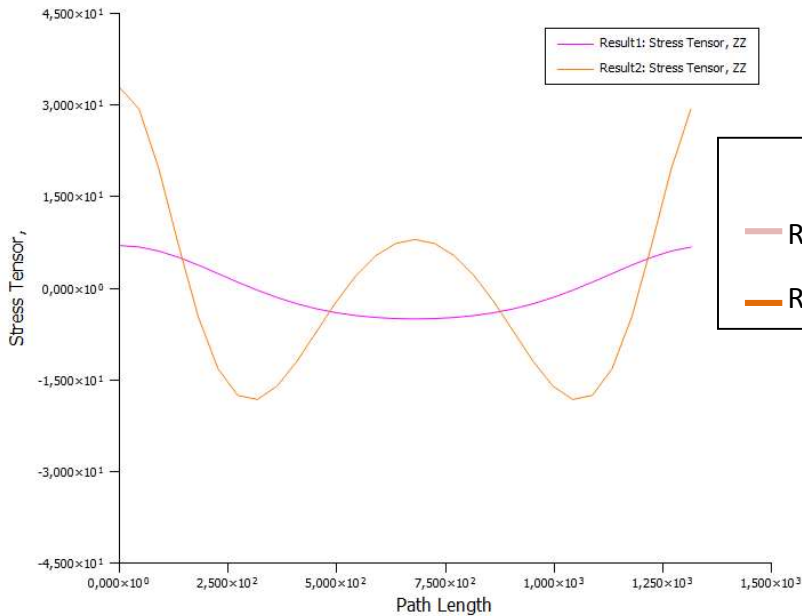


Patran®
Student Edition

Patran 2020 (Student Edition) 18-Apr-21 16:45:06
 Fringe: Default, A2:Static Subcase, Stress Tensor, Z Component, At Z1

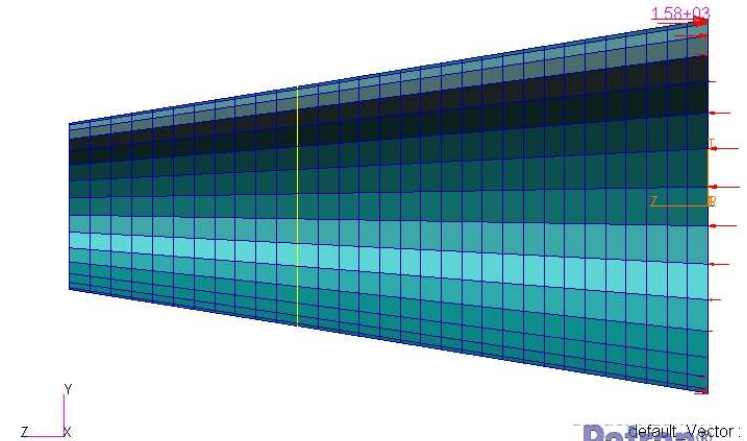


Patran®
Student Edition



Legend:
 — Ring stiffness: $E=2e5$, at Z2
 — Ring stiffness: $E=2e7$, at Z2

Patran 2020 (Student Edition) 18-Apr-21 16:34:10
 Vector: Default, A2:Static Subcase, Constraint Forces, Translational, (NON-LAYERED)



Patran®
Student Edition