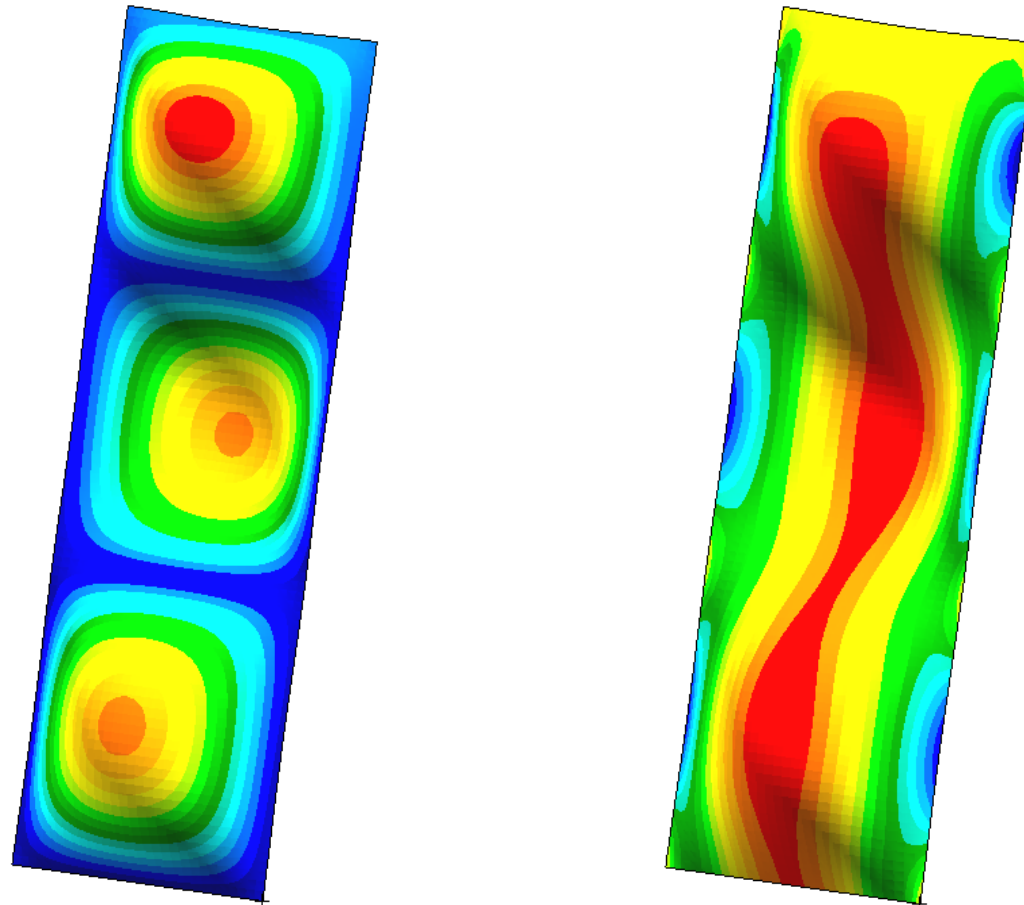


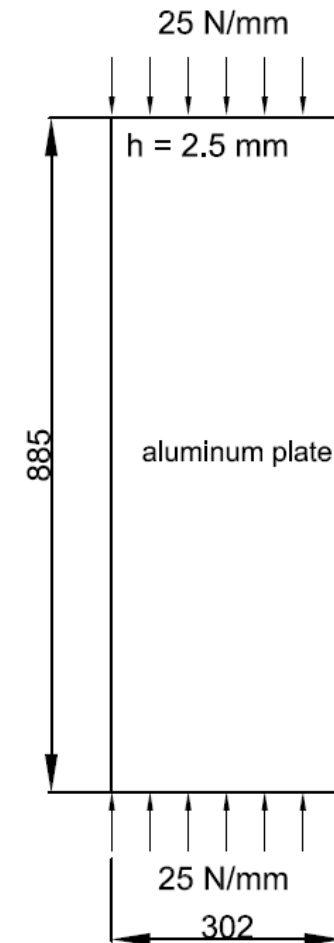
# NONLINEAR MECHANICS OF STRUCTURES

## EXERCISE 4

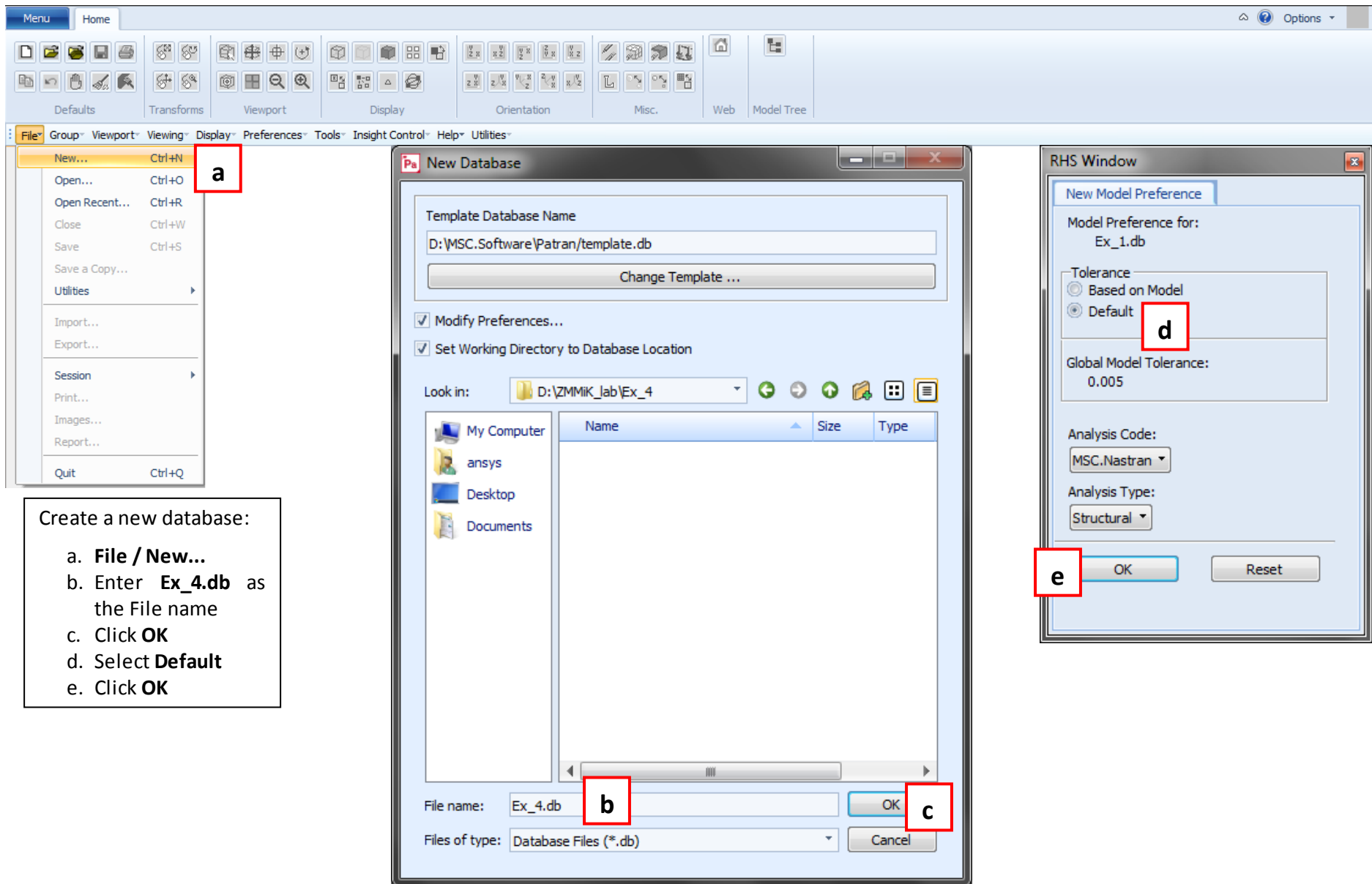


# PROBLEM DESCRIPTION

The goal of this exercise is to conduct the buckling analysis of a compressed aluminum plate. The exercise is divided into two parts. In the first part the elastic buckling behavior of the plate is analyzed. In the second part – the behavior of the same plate after the loss of stability is assessed.

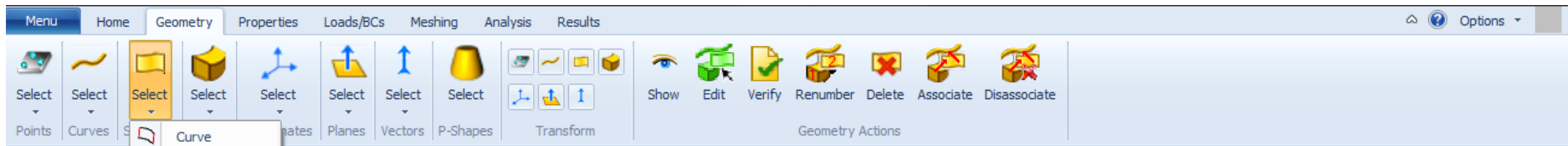


**Units: mm, N, MPa**



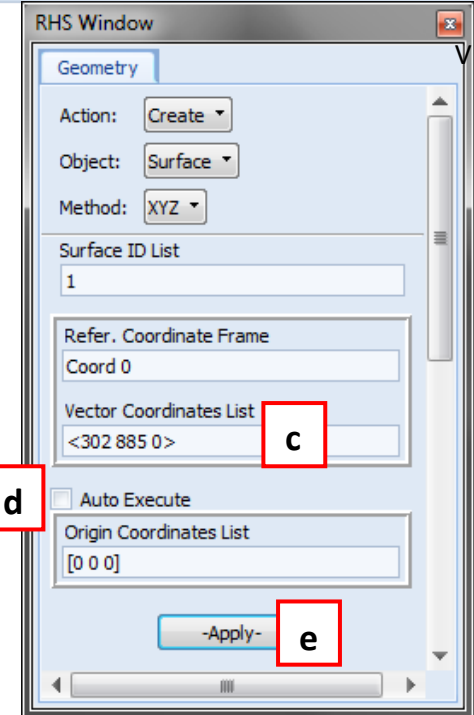
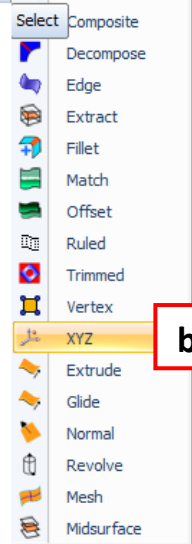
Create a new database:

- a. **File / New...**
- b. Enter **Ex\_4.db** as the File name
- c. Click **OK**
- d. Select **Default**
- e. Click **OK**



# PART 1: BUCKLING ANALYSIS

Pa Ex\_4.db - default\_viewport - default\_group - Entity



- a. Change *Viewport Color* to **Black** (click on the **Cycle Background** icon)
- Create a surface:
- b. Click on the **Geometry** icon/**Select/XYZ** (*Surfaces icon*)
  - c. Enter **<302 885 0>** as the **Vector Coordinates List**
  - d. Uncheck **Auto Execute**
  - e. Click **Apply**
  - f. Click on the **Smooth shaded** icon

The screenshot displays the software's 'Loads/BCs' menu and the 'Input Data' dialog. The 'Input Data' dialog shows the 'Translations' field set to  $\langle 0, , \rangle$ . The 'RHS Window' shows the 'New Set Name' field containing 'disp\_x' and the 'Input Data...' button highlighted.

Apply the boundary conditions:

- a. Click on the **Loads/BCs** icon/**Displacement Constraint** icon
- b. Enter **disp\_x** as the New Set Name
- c. Click **Input Data...**
- d. Enter  $\langle 0, , \rangle$  for the Translations
- e. Click **OK**

Ex\_4.db - default\_viewport - default\_group - Entity

**f.** Click **Select Application Region...**

**g.** Select **Geometry**

**h.** Click on the **Select Geometry Entities** panel

**i.** Select **Point or Vertex** icon

**j.** Select the point

**k.** Click **Add**

**l.** Click **OK**

**m.** Click **Apply**

**RHS Window**

Conditions | Select Application Region

Select: Geometry **g**

Auto Select...

Application Region

Select Geometry Entities

Surface 1.1.1 **h**

**k** Add Remove

Application Region

**l** OK

**RHS Window**

Load/Boundary Conditions

Action: Create

Object: Displacement

Type: Nodal

Option: Standard

Current Load Case: Default...

Type: Static

Existing Sets

New Set Name: disp\_x

Input Data...

Select Application Region... **f**

**m** -Apply-

n. Enter **disp\_y** as the New Set Name

o. Click **Input Data...**

p. Enter **<,0,>** for the Translations

q. Click **OK**

r. Click **Select Application Region...**

s. Select **Geometry**

t. Click on the **Select Geometry Entities** panel

u. Select **Curve or Edge** icon

v. Select the bottom edge

w. Click **Add**

x. Click **OK**

y. Click **Apply**

hh  
 ee  
 ff  
 ii  
 gg  
 jj  
 bb  
 cc  
 dd  
 kk  
 z  
 aa

Z. Enter **disp\_z** as the New Set Name  
 aa. Click **Input Data...**  
 bb. Enter **< , ,0>** for the Translations  
 cc. Click **OK**  
 dd. Click **Select Application Region...**  
 ee. Select **Geometry**  
 ff. Click on the **Select Geometry Entities** panel  
 gg. Select **Curve or Edge** icon  
 hh. Select all edges by clicking and dragging the mouse  
 ii. Click **Add**  
 jj. Click **OK**  
 kk. Click **Apply**



**Apply the load:**

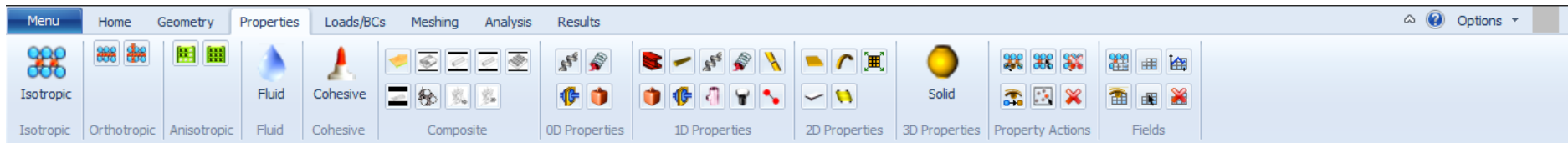
- Click on the **Loads/BCs icon/Distributed Load icon** (*Element Uniform tab*)
- Enter **load** as the New Set Name
- Target Element Type: **2D**
- Click **Input Data...**
- Enter **<0,25,0>** for the Edge Distr Load
- Click **OK**
- Click **Select Application Region...**
- Select **Geometry**
- Click on the **Select Surface Edges** panel
- Select the top edge
- Click **Add**
- Click **OK**
- Click **Apply**

Define a material:

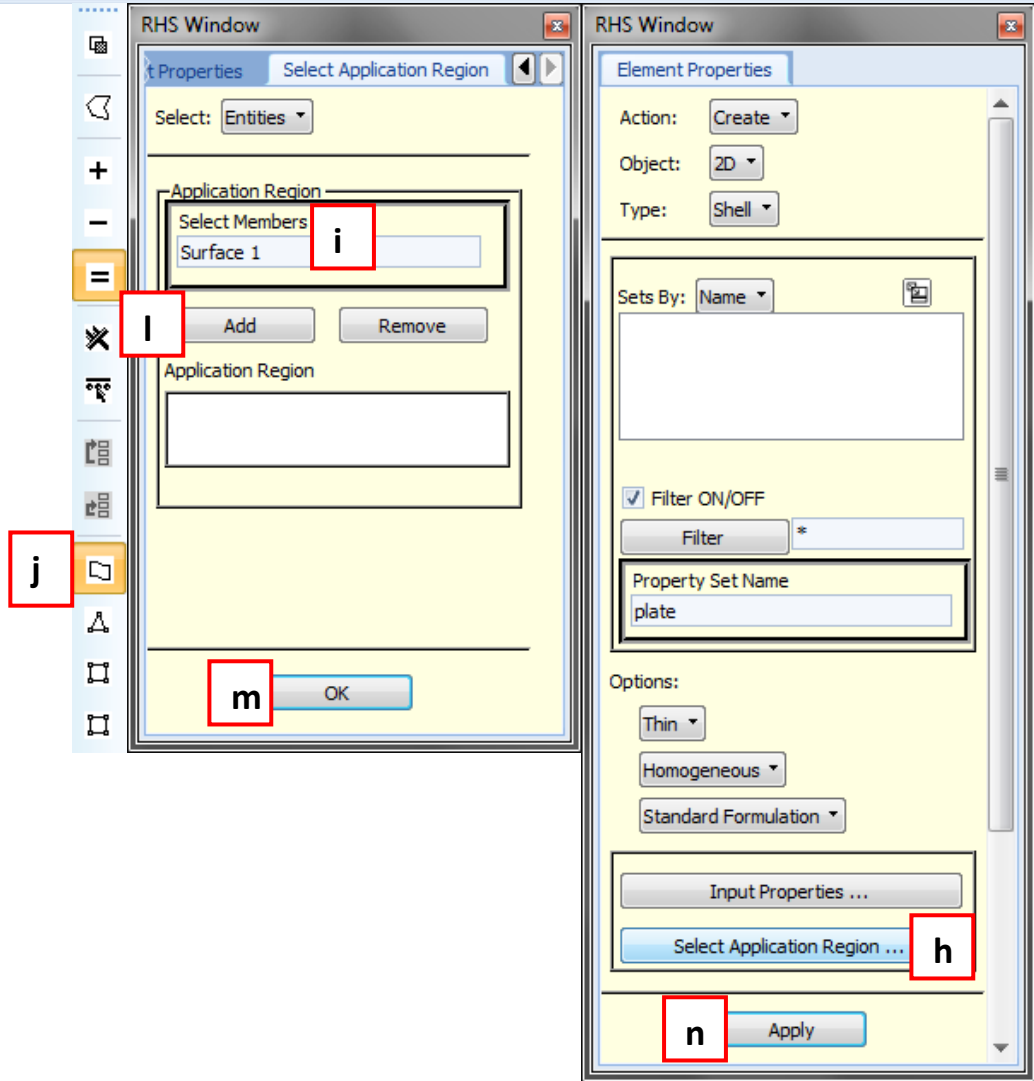
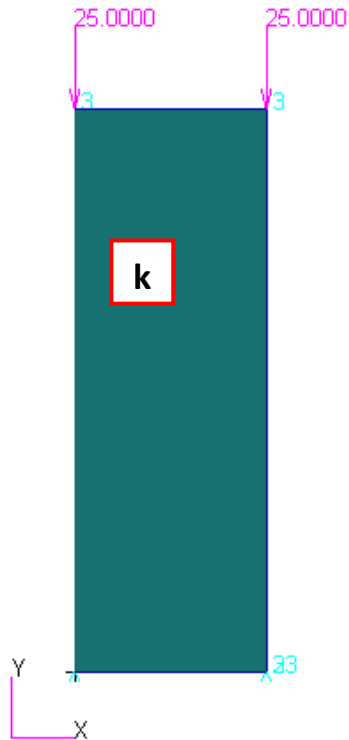
- Click on the **Properties** icon/**Isotropic** icon
- Enter **aluminum** as the Material Name
- Click **Input Properties...**
- Enter **70000** as the Elastic Modulus and **0.33** as the Poisson Ratio
- Click **OK**
- Click **Apply**

Assign the properties:

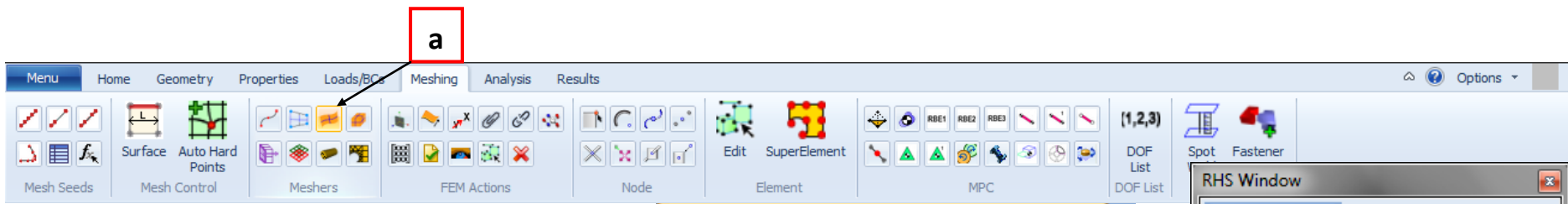
- Properties: **Shell** icon
- Enter **plate** as the Property Set Name
- Click **Input Properties...**
- Click on the **Mat Prop Name** icon
- Select **aluminum**
- Enter **2.5** as the Thickness
- Click **OK**



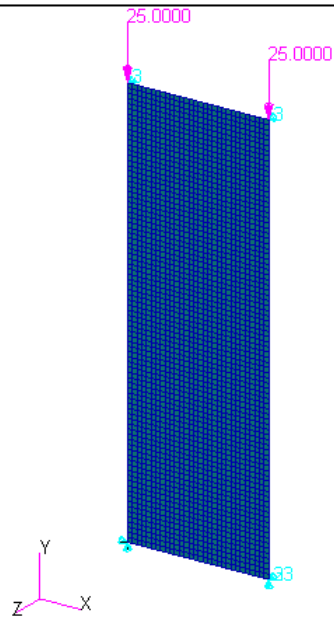
Ex\_4.db - default\_viewport - default\_group - Entity



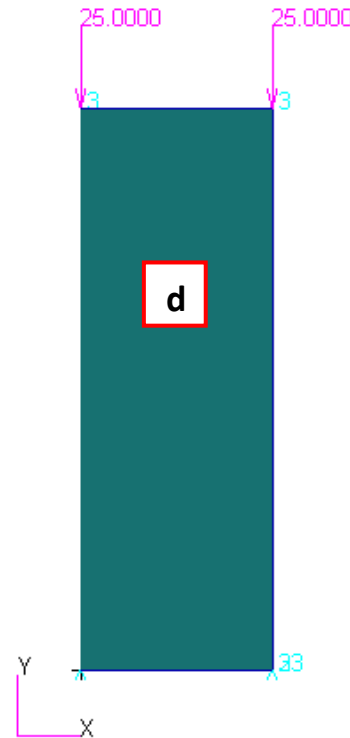
- h. Click **Select Application Region...**
- i. Click on the **Select Members** panel
- j. Select **Surface or face** icon
- k. Select the surface
- l. Click **Add**
- m. Click **OK**
- n. Click **Apply**



- Mesh the surface:
- Click on the **Meshing** icon/**Surface** icon (*Meshers tab*)
  - Elem Shape: **Quad**; Mesher: **IsoMesh**; Topology: **Quad4**
  - Click on the **Surface List** panel
  - Select the surface
  - Uncheck **Automatic Calculation**
  - Enter **10** as the Value of the Global Edge Length
  - Click **Apply**
  - Click **Iso 1 View**



Pa Ex\_4.db - default\_viewport - default\_group - Entity



**RHS Window**

Finite Elements

Action: Create

Object: Mesh

Type: Surface

Output ID List

Node	1
Element	1

Elem Shape: Quad

Mesher: IsoMesh

Topology: Quad4

IsoMesh Parameters...

Node Coordinate Frames...

Surface List

Surface 1
-----------

Global Edge Length

Automatic Calculation

Value: 10.0

Prop. Name: - None -

Prop. Type: - N/A -

Select Existing Prop...

Create New Property...

-Apply-

**a**

**b**

**c**

**d**

**e**

**f**

**g**

**h**

**i**

**j**

**Run a buckling analysis:**

- Click on the **Analysis/Analysis Deck** icon
- Enter **ex\_4\_buck** as the Job Name
- Click **Solution Type...**
- Select **BUCKLING** as the Solution Type
- Click **Solution Parameters...**
- Click **Eigenvalue Extraction...**
- Extraction Method: **Lanczos**
- Lower = **0**
- Enter **5** as the Number of Desired Roots
- Click **OK**

The screenshot displays the MSC Nastran software interface with three dialog boxes open:

- Results Output Format:** The **XDB** checkbox is checked, and the **Print** checkbox is unchecked. The **OK** button is highlighted with a red box labeled 'm'. An arrow points from the **Print** checkbox to a red box labeled 'l'.
- Solution Parameters:** The **Database Run** checkbox is checked. The **Automatic Constraints** checkbox is checked. The **Results Output Format...** button is highlighted with a red box labeled 'k'. The **OK** button is highlighted with a red box labeled 'n'.
- RHS Window:** The **Analysis** tab is selected. Under **Solution Type**, **BUCKLING** is selected. The **OK** button is highlighted with a red box labeled 'o'. The **Apply** button is highlighted with a red box labeled 'p'.

- k. Click **Results Output Format**
- l. Uncheck **Print** and check **XDB**
- m. Click **OK**
- n. Click **OK**
- o. Click **OK**
- p. Click **Apply**
- q. Run **Nastran** analysis using **ex\_4\_buck.bdf** file

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, titled 'RHS Window', has an 'Analysis' tab. 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). A section titled 'Available Jobs' contains a list with 'Ex\_4\_buck'. Below this are fields for 'Job Name' (Ex\_4\_buck) and 'Job Description (TITLE)'. Further down are fields for 'SUBTITLE' and 'LABEL'. At the bottom of the window are two buttons: 'Select Results File...' (highlighted with a red box labeled 'b') and 'Apply' (highlighted with a red box labeled 'd').

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

- Analysis: **XDB** icon
- Click **Select Results File...**
- Select **ex\_4\_buck.xdb** file and click **OK**
- Click **Apply**



a. Hide (erase) all geometry entities

b. Results: **Fringe/Deformation**

c. Select Result: **Cases: A1:Mode 1**  
 Select Fringe Result: **Eigenvectors, Translational**  
 Select Deformation Result: **Eigenvectors Translational**

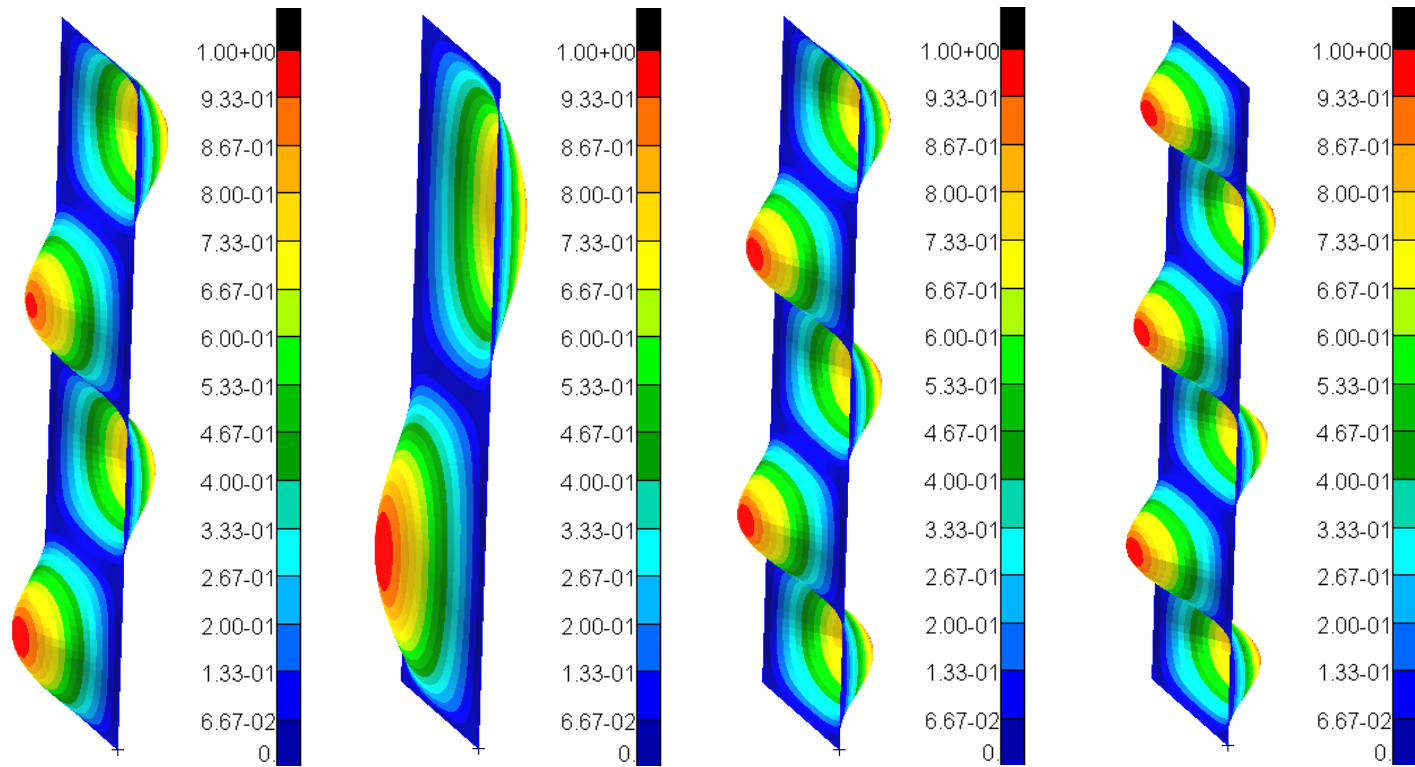
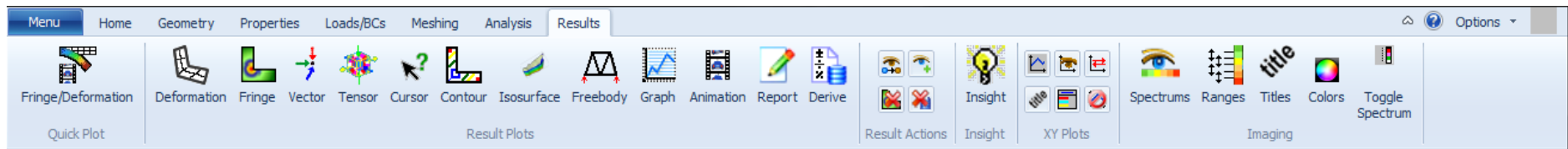
d. Click on the **Deform Attributes**

e. Change **Scale Factor** to **0.1**

f. Uncheck **Show Undeformed**

g. Click **Apply**

Mode 1: Factor = 1.728



Mode 2: Factor = 1.899

Mode 3: Factor = 1.994

Mode 4: Factor = 2.27

Mode 5: Factor = 2.78

**PART 2: PLATE BEHAVIOR  
AFTER LOSS OF STABILITY  
(NONLINEAR ANALYSIS)**

a. Click on the **Reset graphics** icon  
b. Click on the **Plot/Erase Geometry** icon  
c. Click on the **Smooth shaded** icon  
d. Click on the **Loads/BCs** icon  
e. Modify the Distributed Load (change its value to **88.55**)

Create a pressure load:

f. Click on the **Loads/BCs icon/Pressure icon** (*Element Uniform tab*)  
g. Enter **pressure** as the New Set Name  
h. Target Element Type: **2D**  
i. Click **Input Data...**  
j. Enter **1e-4** as the Top Surf Pressure  
k. Click **OK**

Ex\_4.db - default\_viewport - default\_group - Entity

8.86+01  
8.86+01  
8.86+01  
8.86+01

Y  
X  
Z

**o** **q** **r** **m** **n** **p**

**l** **s**

- l. Click **Select Application Region...**
- m. Select **Geometry**
- n. Click on the **Select Surfaces or Edges** panel
- o. Select **Surface or Face** icon
- p. Select the surface
- q. Click **Add**
- r. Click **OK**
- s. Click **Apply**

The image shows a screenshot of the MSC Nastran software interface with several dialog boxes open. Red boxes labeled 'a' through 'j' highlight specific steps in the workflow:

- a**: Analysis Deck icon in the top toolbar.
- b**: Job Name field in the RHS Window dialog.
- c**: Solution Type... button in the RHS Window dialog.
- d**: NONLINEAR STATIC radio button in the Solution Parameters dialog.
- e**: Solution Parameters... button in the Solution Parameters dialog.
- f**: Results Output Format... button in the Solution Parameters dialog.
- g**: XDB checkbox in the Results Output Format dialog.
- h**: OK button in the Results Output Format dialog.
- i**: OK button in the Solution Parameters dialog.
- j**: OK button in the RHS Window dialog.

- Run a nonlinear analysis:
- Click on the **Analysis/Analysis Deck** icon
  - Enter **ex\_4\_nl** as the Job Name
  - Click **Solution Type...**
  - Select **NONLINEARSTATIC** as the Solution Type
  - Click **Solution Parameters...**
  - Click **Results Output Format...**
  - Uncheck **Print** and check **XDB**
  - Click **OK**
  - Click **OK**
  - Click **OK**

The image shows the ANSYS Workbench software interface with three panels open: Subcase Parameters, Subcases, and RHS Window. Red boxes highlight specific elements corresponding to the instructions in the text box on the left.

- Subcase Parameters Panel:**
  - n:** The text '4' in the 'Number of Load Increments' field.
  - o:** The 'OK' button at the bottom left.
- Subcases Panel:**
  - l:** The 'Default' subcase selected in the 'Available Subcases' list.
  - m:** The 'Subcase Parameters...' button in the 'Subcase Options' section.
- RHS Window Panel:**
  - k:** The 'Subcases...' button at the bottom of the 'Analysis' section.

- k. Click **Subcases...**  
 l. Select **Default**  
 m. Click **Subcase Parameters...**  
 n. Enter **4** as the Number of Load Increments  
 o. Click **OK**

The screenshot displays the MSC Nastran software interface. The top menu bar includes 'Menu', 'Home', 'Geometry', 'Properties', 'Loads/BCs', 'Meshing', 'Analysis', and 'Results'. The 'Analysis' menu is active, showing options like 'HDF5', 'XDB', 'Read Output2', 'Attach Output2', 't16/t19', 'd3plot', 'Delete', 'Monitor', and 'Actions'. Three dialog boxes are open: 'Output Requests', 'Subcases', and 'RHS Window'. The 'Output Requests' dialog has 'Form Type' set to 'Advanced' (labeled 'q') and 'Intermediate Output Option' set to 'Yes' (labeled 'r'). The 'Subcases' dialog has 'Output Requests...' selected (labeled 'p'). The 'RHS Window' shows 'Action' set to 'Analyze' and 'Type' set to 'Structural'. A text box at the bottom left contains the following instructions:

- p. Click **Output Requests...**
- q. Form type: **Advanced**
- r. Intermediate Output Option: **Yes**
- s. Click **OK**
- t. Click **Apply**
- u. Click **Cancel**
- v. Click **Apply**
- w. Run **Nastran** analysis using **ex\_4\_nl.bdf** file



Menu Home Geometry Properties Loads/BCs Meshing Analysis Results

Quick Plot **b** Result Plots

100% of Load; Y stress; top surface

5.47+01  
4.07+01  
2.68+01  
1.28+01  
1.15+00  
1.51+01  
2.91+01  
4.30+01  
5.70+01  
7.09+01  
8.49+01  
9.88+01  
1.13+02  
1.27+02  
1.41+02  
1.55+02

X Y Z

**a.** Attach the results file  
**b.** Results: **Fringe/Deformation**  
**c.** Select Result: **A2:Non-linear: 100% of Load**  
 Select Fringe Result: **Nonlinear Stresses, Stress Tensor**  
 Select Deformation Result: **Displacements, Translational**  
**d.** Select: **Position...(At Z2)**, Option: **Average**  
 Quantity: **Y Component**  
**e.** Click on the **Deform Attributes**  
**f.** Change **Scale Factor** to **0.1**  
**g.** Uncheck **Show Undeformed**  
**h.** Click **Apply**

**RHS Window**

Results  
 Action: Create  
 Object: Quick Plot

Select Result Cases

Default, A1:Mode 2 : Factor = 1.8998;  
 Default, A1:Mode 3 : Factor = 1.9942;  
 Default, A1:Mode 4 : Factor = 2.2704;  
 Default, A1:Mode 5 : Factor = 2.7812;  
 Default, A2:Non-linear: 25. % of Load;  
 Default, A2:Non-linear: 50. % of Load;  
 Default, A2:Non-linear: 75. % of Load;  
 Default, A2:Non-linear: 100. % of Load

Select Fringe Result

Nonlinear Strains, Strain Tensor  
 Nonlinear Stresses, Equivalent Stress  
 Nonlinear Stresses, Stress Tensor  
 Principal Stress Direction, Zero Shear A

Position...(At Z2)

Quantity: Y Component

Select Deformation Result

Constraint Forces, Rotational  
 Constraint Forces, Translational  
 Displacements, Rotational  
 Displacements, Translational

Apply

**RHS Window**

Results  
 Action: Create  
 Object: Quick Plot

Show Viewport Legend

Deformed:

Render Style: Wireframe

Line Style:

Line Width:

Scale Interpretation  
 Model Scale  True Scale

Scale Factor: 0.1

Show Undeformed:

Title Editor...

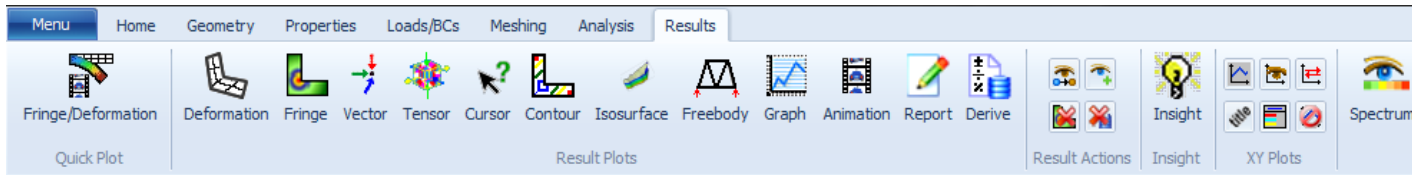
Show Title:  Lock Title:

Show Maximum Label:

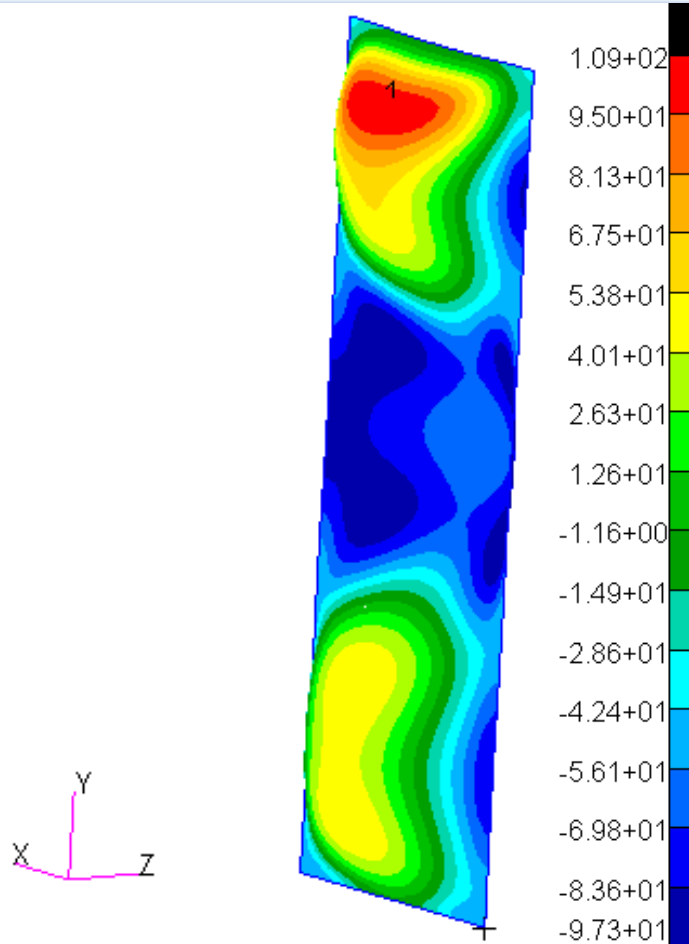
Label Style...

Apply Reset





- i. Select: **Position...(At Z1)**
- j. Click **Apply**



100% of Load; Y stress;  
bottom surface

RHS Window

Results

Action: Create

Object: Quick Plot

Select Result Cases

- Default, A1:Mode 2 : Factor = 1.8998;
- Default, A1:Mode 3 : Factor = 1.9942;
- Default, A1:Mode 4 : Factor = 2.2704;
- Default, A1:Mode 5 : Factor = 2.7812;
- Default, A2:Non-linear: 25. % of Load;
- Default, A2:Non-linear: 50. % of Load;
- Default, A2:Non-linear: 75. % of Load;
- Default, A2:Non-linear: 100. % of Load

Select Fringe Result

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

Position...(At Z1) **i**

Quantity: Y Component

Select Deformation Result

- Constraint Forces, Rotational
- Constraint Forces, Translational
- Displacements, Rotational
- Displacements, Translational

Animate

**i** Apply

Menu Home Geometry Properties Loads/BCs Meshing Analysis Results

Fringe/Deformation Deformation Fringe Vector Tensor Cursor Contour Isosurface Freebody Graph Animation Report Derive

Quick Plot Result Plots Result Actions Insight XY Plots Spectrums

k. Click **Position...**  
 l. Select both positions: **At Z1** and **At Z2**  
 m. Option: **Average**  
 n. Click **Close**  
 o. Click **Apply**

100% of Load;  
 membrane stress  
 (Y stress; middle surface)

**RHS Window**

Results Select...

Positions

- At Z1
- At Z2

Filter \*

Option: Average

Close

**RHS Window**

Results

Action: Create

Object: Quick Plot

Select Result Cases

- Default, A1:Mode 2 : Factor = 1.8998;
- Default, A1:Mode 3 : Factor = 1.9942;
- Default, A1:Mode 4 : Factor = 2.2704;
- Default, A1:Mode 5 : Factor = 2.7812;
- Default, A2:Non-linear: 25. % of Load;
- Default, A2:Non-linear: 50. % of Load;
- Default, A2:Non-linear: 75. % of Load;
- Default, A2:Non-linear: 100. % of Load;

Select Fringe Result

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

Position...(At Z1>

Quantity: Y Component

Select Deformation Result

- Constraint Forces, Rotational
- Constraint Forces, Translational
- Displacements, Rotational
- Displacements, Translational

Animate

Apply