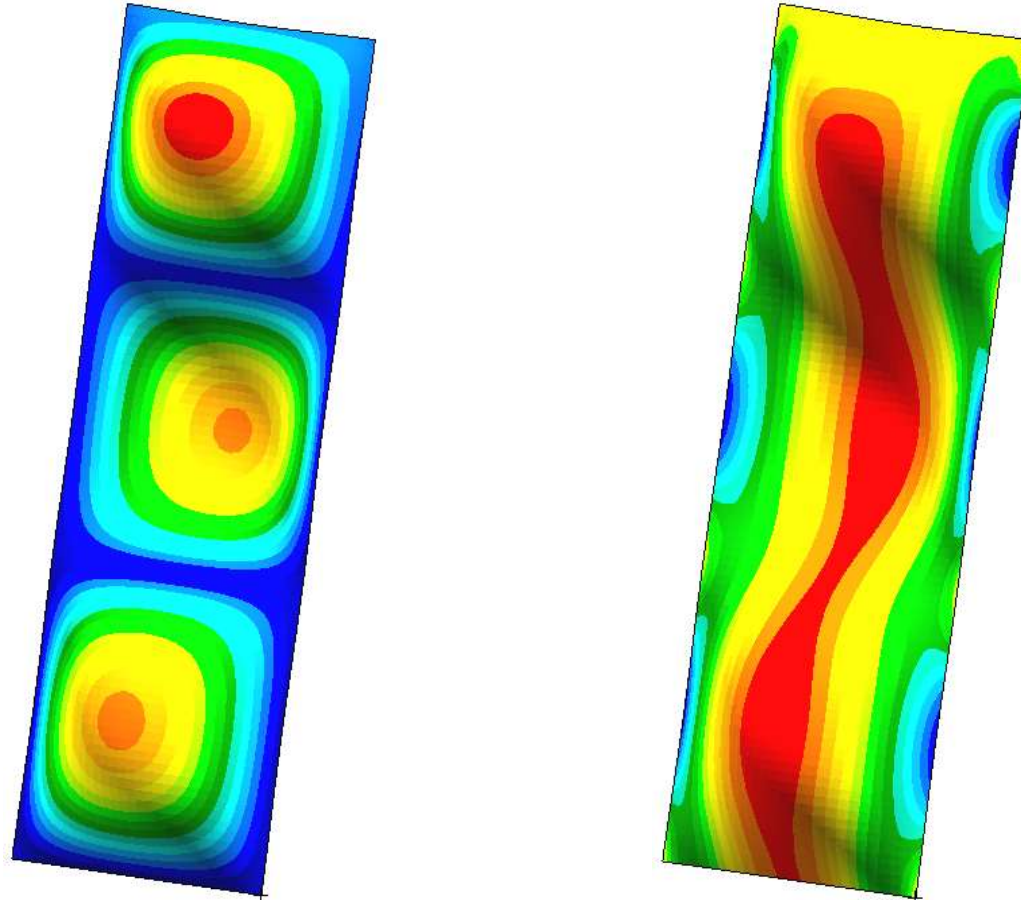
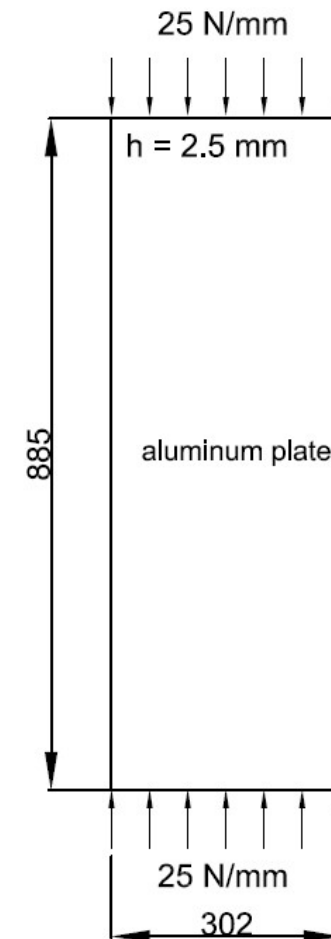


Mechanics of Thin-walled Structures

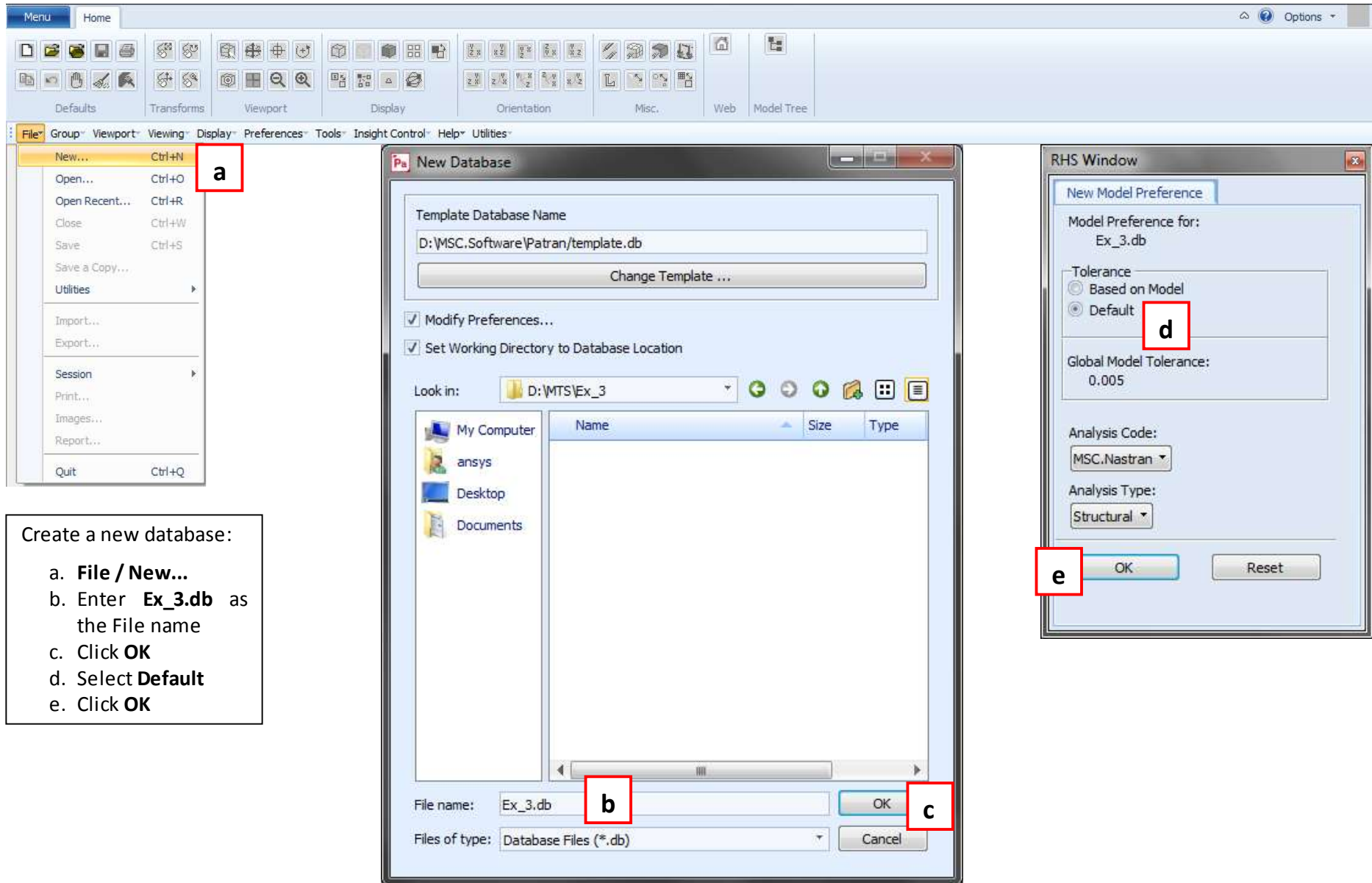


PROBLEM DESCRIPTION

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

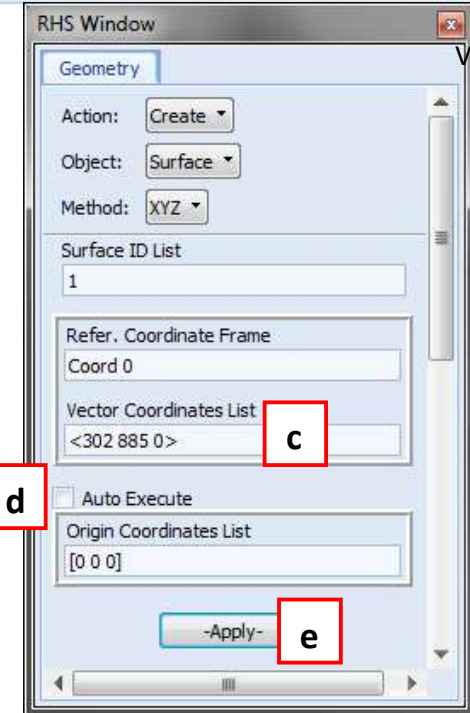


Create a new database:

- a. **File / New...**
- b. Enter **Ex_3.db** as the File name
- c. Click **OK**
- d. Select **Default**
- e. Click **OK**



PART 1: BUCKLING ANALYSIS



- 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

Apply the boundary conditions:

- Click on the **Loads/BCs** icon/**Displacement Constraint** icon
- Enter **disp_x** as the New Set Name
- Click **Input Data...**
- Enter **<0, , >** for the Translations
- Click **OK**

Menu Home Geometry Properties Loads/BCs Meshing Analysis Results

Displacement Constraint Force Temperature Velocity Acceleration Crack(VCCT)

Nodal Element Uniform Element Variable Contact Bodies Initial Conditions LBC Actions Load Cases LBC Fields

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

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
 -Apply- m

RHS Window

Conditions Select Application Region

Select: Geometry g

Auto Select...

Application Region

Select Geometry Entities

Surface 1.1.1 h

Add k Remove

Application Region

OK l

i

j

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**

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**

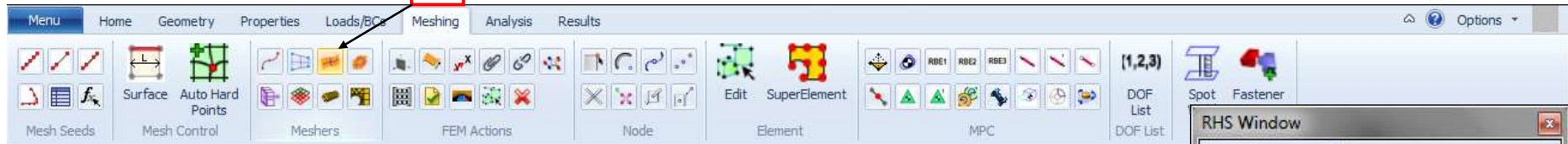
Define a material:

- 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**

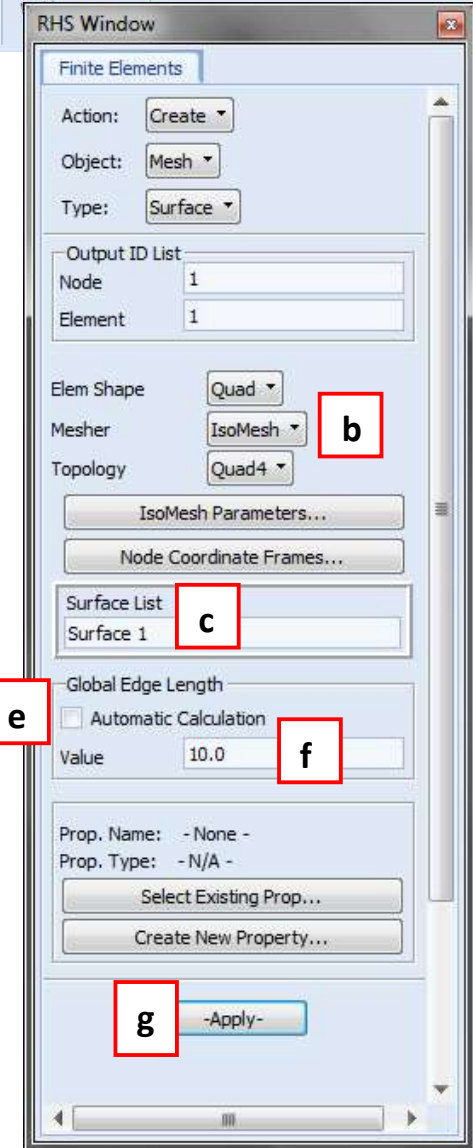
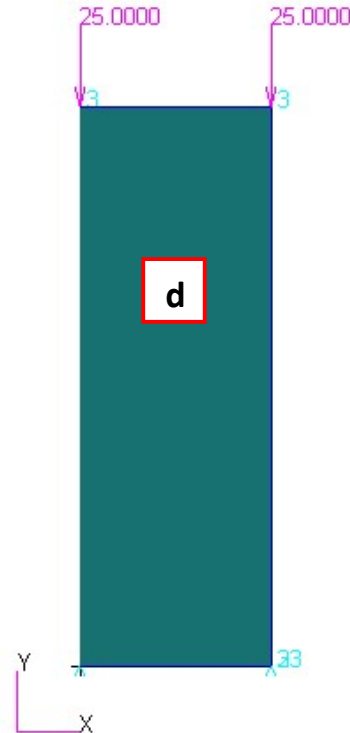
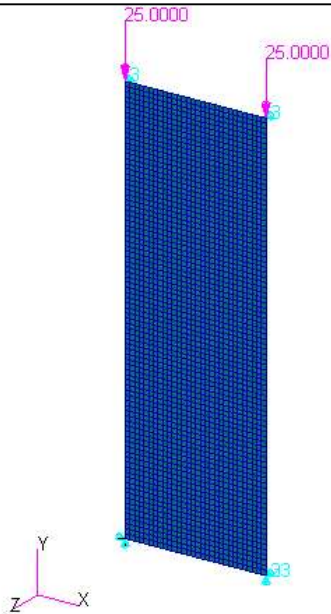
The screenshot shows the ANSYS software interface with the following components and annotations:

- Toolbar:** A row of icons for material properties. A red box labeled 'a' points to the **Shell** icon in the 2D Properties section.
- Select Material Dialog:** A list of materials with 'aluminum' selected. A red box labeled 'e' highlights the 'aluminum' entry.
- Input Properties Dialog:** A table for defining material properties. A red box labeled 'd' points to the **Mat Prop Name** icon. A red box labeled 'f' highlights the 'Thickness' field containing the value '2.5'.
- RHS Window:** A panel for defining element properties. A red box labeled 'b' highlights the 'Property Set Name' field containing the text 'plate'. A red box labeled 'c' highlights the 'Input Properties ...' button.
- Callout Box:** A white box with a red border at the bottom left containing the following instructions:
 - Properties: **Shell** icon
 - Enter **plate** as the PropertySet Name
 - Click **Input Properties...**
 - Click on the **Mat Prop Name** icon
 - Select **aluminum**
 - Enter **2.5** as the Thickness
 - Click **OK**

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**



Run a buckling analysis:

- Click on the **Analysis/Analysis Deck** icon
- Enter **ex_3_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 image shows a screenshot of the Nastran software interface with three dialog boxes open. The 'Results Output Format' dialog has 'XDB' checked and 'Print' unchecked, with a red box 'l' around the 'XDB' checkbox and 'm' around the 'OK' button. The 'Solution Parameters' dialog has 'Database Run' checked and 'Automatic Constraints' checked, with a red box 'n' around the 'OK' button and 'k' around the 'Results Output Format...' button. The 'RHS Window' dialog has 'MSC.Nastran' selected as the solution type, with a red box 'o' around the 'OK' button and 'p' around the 'Apply' button at the bottom.

- 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_3_buck.bdf** file

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

- Analysis: XDB icon
- Click **Select Results File...**
- Select **ex_3_buck.xdb** file and click **OK**
- Click **Apply**

RHS Window

Analysis

Action: Access Results

Object: Attach XDB

Method: Result Entities

Code: MSC.Nastran

Type: Structural

Available Jobs

Ex_3_buck

Job Name

Ex_3_buck

Job Description (TITLE)

SUBTITLE

LABEL

Select Results File... **b**

Translation Parameters...

d 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**

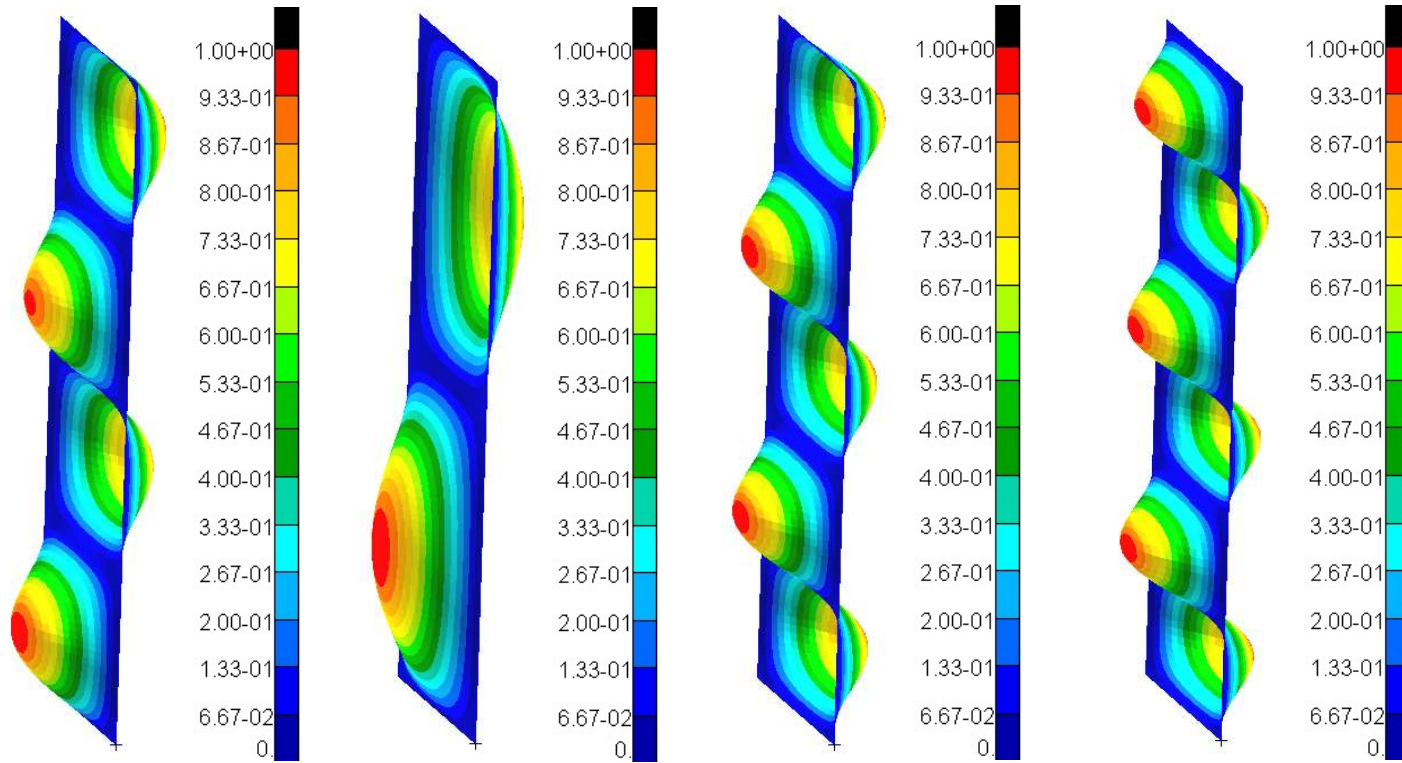
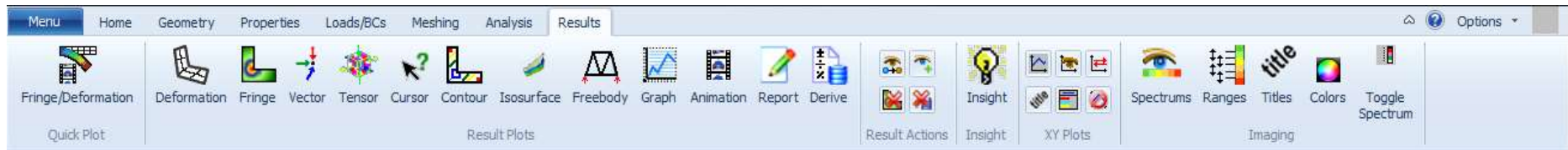
d. Click on the **Deform Attributes**

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**

1. 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**

Run a nonlinear analysis:

- Click on the **Analysis/Analysis Deck** icon
- Enter **ex_3_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 screenshot displays the MSC Nastran software interface with three main panels open:

- Subcase Parameters:** The "Number of Load Increments" field is set to 4, highlighted with a red box labeled 'n'. Other fields include "Total Time", "Matrix Update Method" (Automatic), "Number of Iterations per Update" (5), and "Allowable Iterations per Increment" (25). There are checkboxes for "Convergence Criteria" (Displacement Error, Load Error, Work Error) and "Arc-Length Method".
- Subcases:** The "Available Subcases" list shows "Default" selected, highlighted with a red box labeled 'l'. The "Subcase Options" section includes a button for "Subcase Parameters..." highlighted with a red box labeled 'm'.
- RHS Window:** The "Analysis" tab is active. The "Action" is "Analyze", "Object" is "Entire Model", and "Method" is "Analysis Deck". The "Code" is "MSC.Nastran" and "Type" is "Structural". The "Job Name" is "Ex_3_nl". At the bottom, the "Subcases..." button is highlighted with a red box labeled 'k'.

In the bottom-left corner, a text box contains the following instructions:

- 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 main toolbar contains icons for Analyze, Create, Existing Deck, Optimize, Toptimize, Access Results, Delete, Monitor, and Actions.

The **Output Requests** dialog box is open, showing the following settings:

- Form Type: **Advanced** (labeled **q**)
- Select Result Type: Displacements, Element Stresses, Constraint Forces, Multi-Point Constraint Forces, Element Forces, Applied Loads, Element Strain Energies, Element Strains.
- Output Requests list:
 - DISP(SORT1,REAL)=ALL FEM
 - STRESS(SORT1,REAL,VONMISES,BILIN)=ALL FEM;PARAM,NOCO
 - SPCFORCES(SORT1,REAL)=ALL FEM
- Options:
 - Sorting: By Node/Element
 - Format: Rectangular
 - Tensor: Von Mises
 - Element Points: Bilinear
 - Plate Strain Curv: Plane Curv.
 - Composite Plate Opt: Ply Stresses
 - Suppress Print for Result Type:
 - Velocity option:
 - Power option: (labeled **r**)
- Intermediate Output Option: **Yes** (labeled **r**)
- Buttons: Delete, OK (labeled **s**), Defaults, Cancel.

The **Subcases** dialog box is open, showing the following settings:

- Solution Sequence: 106
- Action: Create
- Available Subcases: Default
- Subcase Name: Default
- Available Load Cases: Default
- Subcase Options:
 - Subcase Parameters... (labeled **p**)
 - Output Requests... (labeled **p**)
 - Direct Text Input...
 - Select Explicit MPCs...
- Buttons: Apply (labeled **t**), Cancel (labeled **u**).

The **RHS Window** is open, showing the following settings:

- Analysis:
 - Action: Analyze
 - Object: Entire Model
 - Method: Analysis Deck
- Code: MSC.Nastran
- Type: Structural
- Available Jobs: Ex_3_buck
- Job Name: Ex_3_nl
- Job Description (TITLE):
- SUBTITLE:
- LABEL:
- Buttons: Translation Parameters..., Solution Type..., Direct Text Input..., Select Superelements..., Subcases..., Subcase Select..., Apply (labeled **v**).

Instructions:

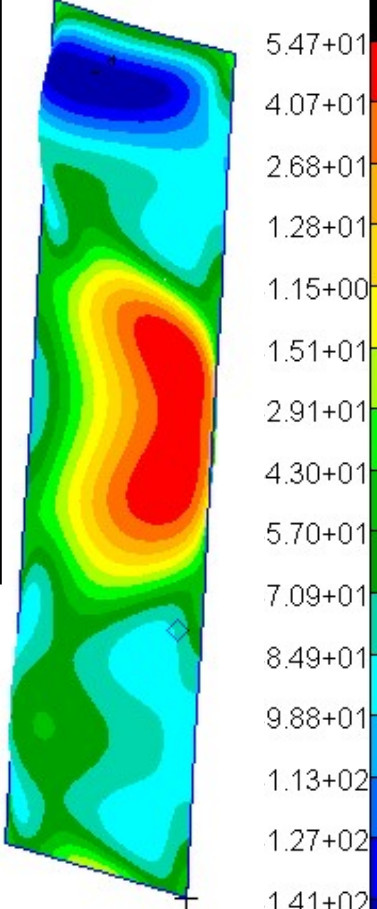
- p. Click **Output Requests...**
- q. Form type: **Advanced**
- r. Intermediate Output Option: **Yes**
- s. Click **OK**
- t. Click **Apply**
- u. Click **Cancel**
- v. Click **Apply**
- w. Run **Nastran** analysis using **ex_3_nl.bdf** file

Menu Home Geometry Properties Loads/BCs Meshing Analysis Results

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

Quick Plot **c** Result Plots

a. Attach the **ex_3_nl.xdb** results file
 b. Click **Reset Graphics** icon
 c. Results: **Fringe/Deformation**
 d. Select Result: **A2:Non-linear: 100% of Load**
 Select Fringe Result: **Nonlinear Stresses, Stress Tensor**
 Select Deformation Result: **Displacements, Translational**
 e. Select: **Position...(At Z2)**
 Quantity: **Y Component**
 f. Click on the **Deform Attributes**
 g. Change **Scale Factor** to **0.1**
 h. Uncheck **Show Undeformed**
 i. Click **Apply**



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

100% of Load; Y stress;
top surface

RHS Window

Results

Action: Create
Object: Quick Plot **f**

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 Ai

Position...(At Z2) **e**

Quantity: Y Component **e**

Select Deformation Result

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

Animate

RHS Window

Results

Action: Create
Object: Quick Plot

Show Viewport Legend

Deformed:

Render Style: Wireframe **d**

Line Style:

Line Width:

Scale Interpretation

Model Scale True Scale **d**

Scale Factor: 0.1 **g**

Show Undeformed **h**

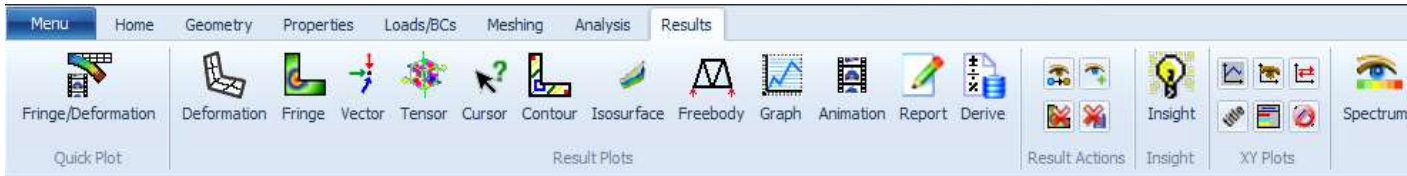
Title Editor...

Show Title Lock Title **d**

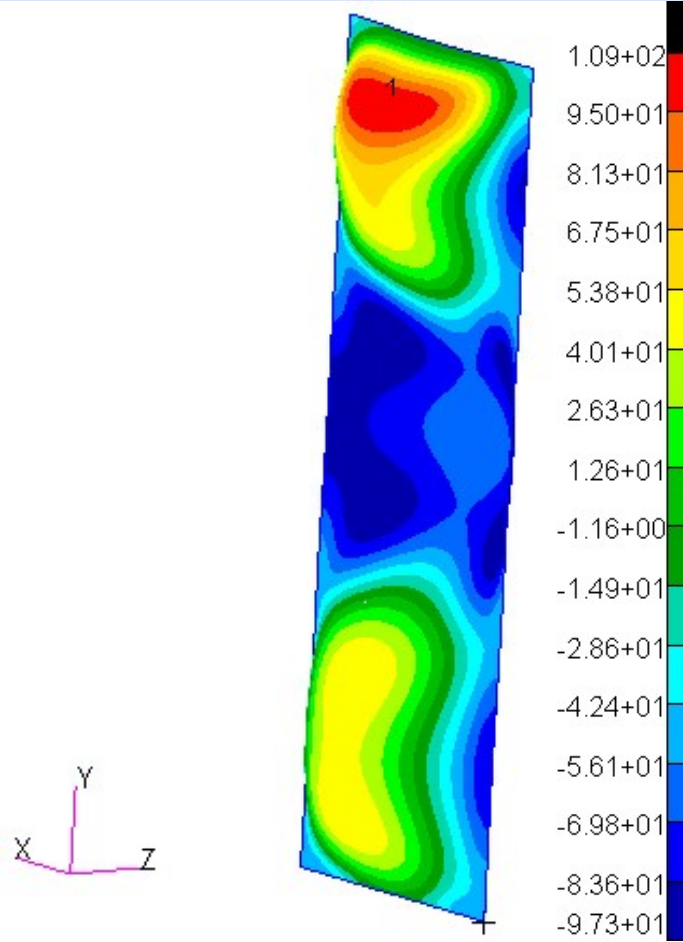
Show Maximum Label

Label Style...

i Apply Reset



j. Select: **Position...(At Z1)**
 k. 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

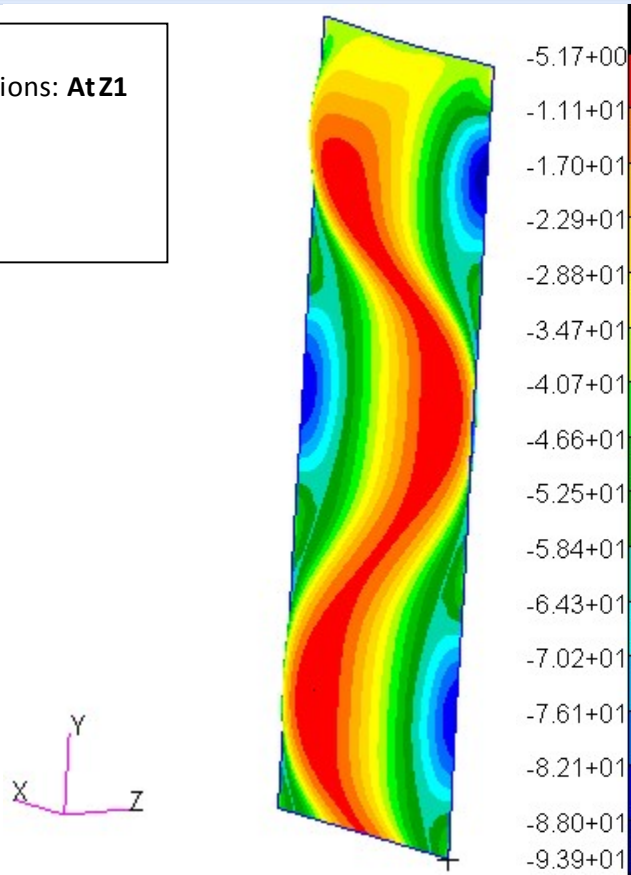
k Apply

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

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



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

RHS Window

Results Select...

Positions

At Z1 **m**

At Z2

Filter *

Option: Average **n**

o 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) **l**

Quantity: Y Component

Select Deformation Result

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

Animate

p Apply