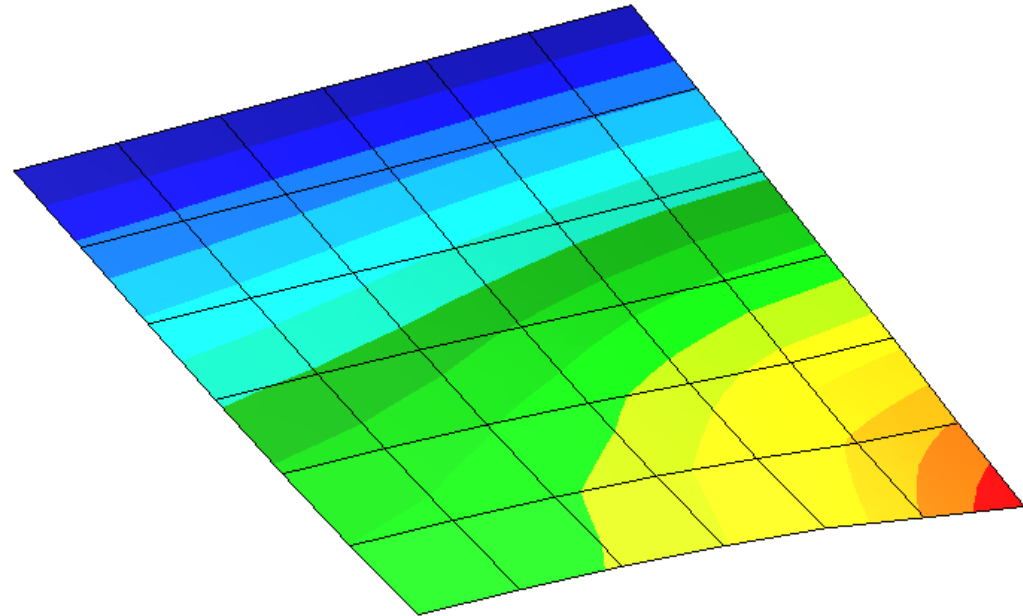
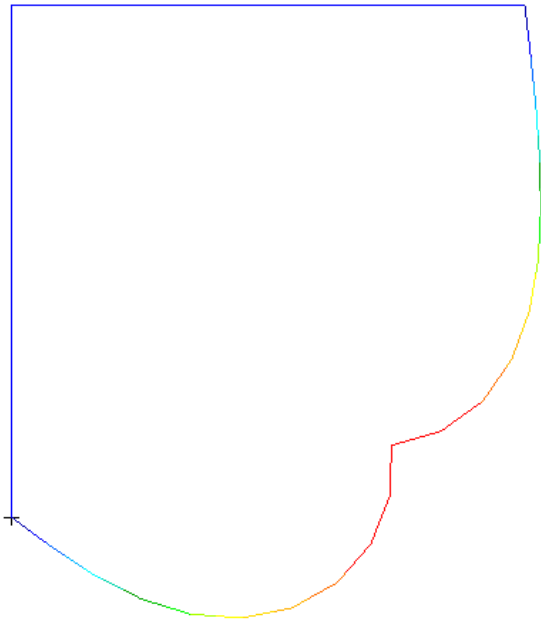


# NONLINEAR MECHANICS OF STRUCTURES

## EXERCISE 3



# PROBLEM DESCRIPTION

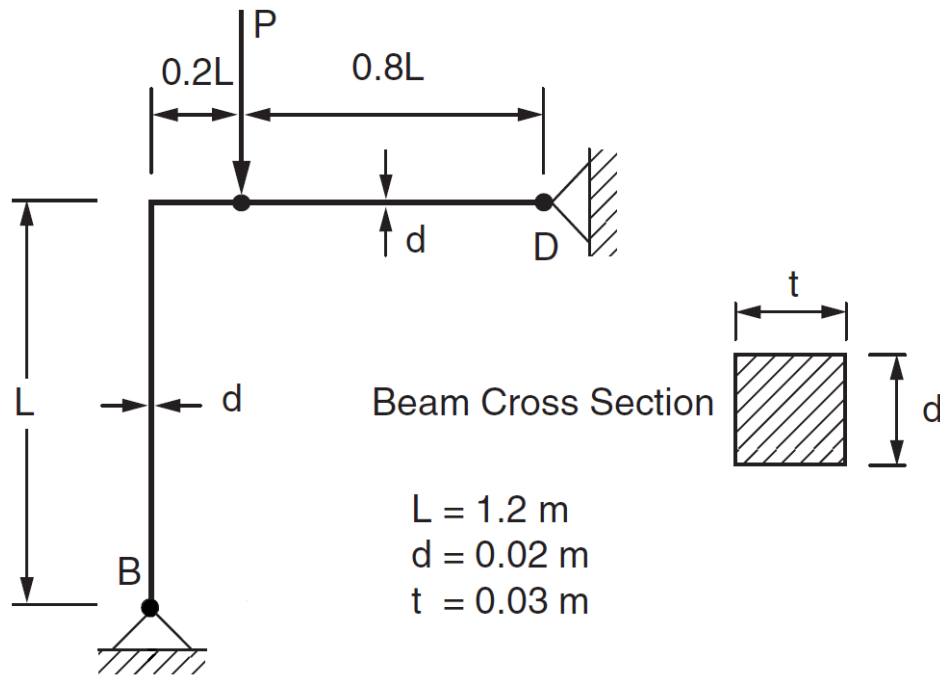


Fig. 1 Lee's Frame Buckling Problem

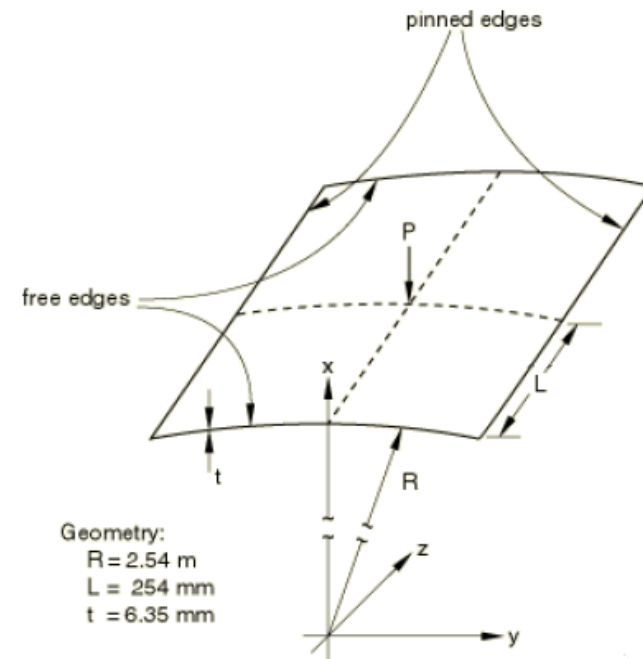
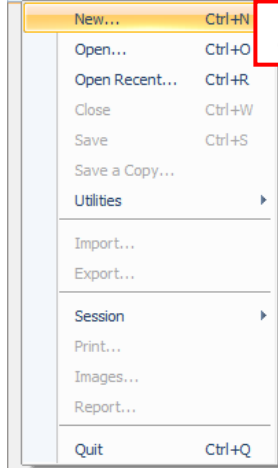


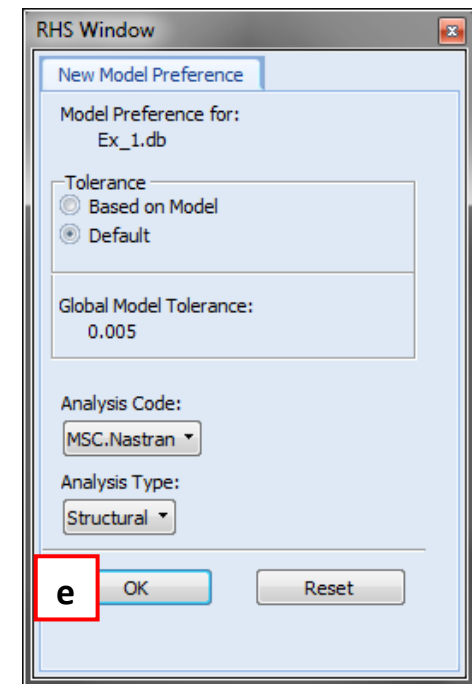
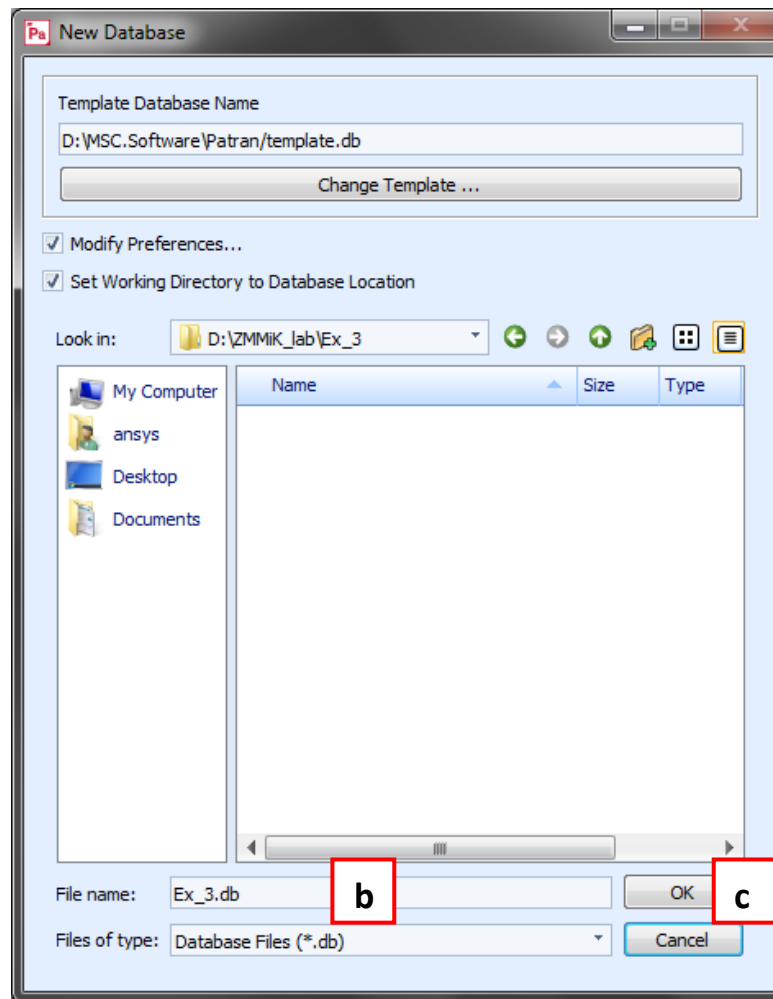
Fig. 2 Shallow, cylindrical roof under a point load

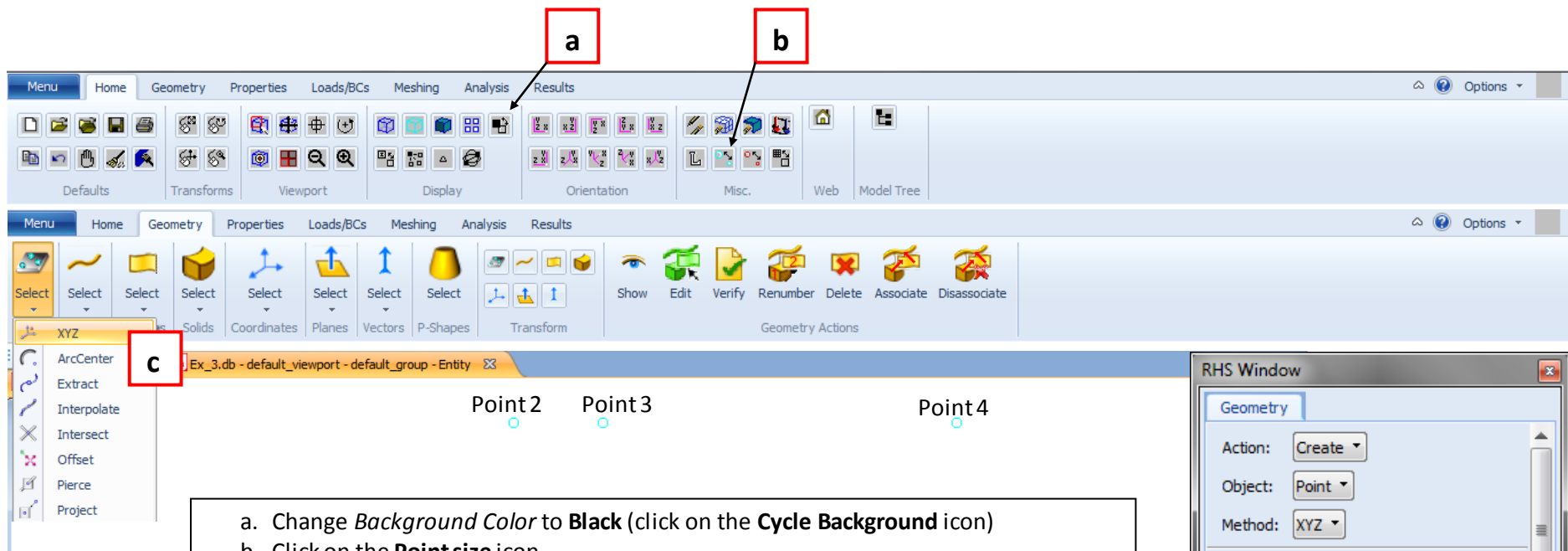
The goal of this exercise is to conduct the nonlinear analyses of two well known benchmark tests which exhibit snap-through behavior. In the first part *Lee's Frame Buckling Problem* is analyzed. In the second part – the behavior of a shallow, cylindrical roof under a point load is assessed.

Units: mm, N, MPa



- Create a new database:
- a. **File / New...**
  - b. Enter **Ex\_3a.db** as the File name
  - c. Click **OK**
  - d. Select **Default**
  - e. Click **OK**



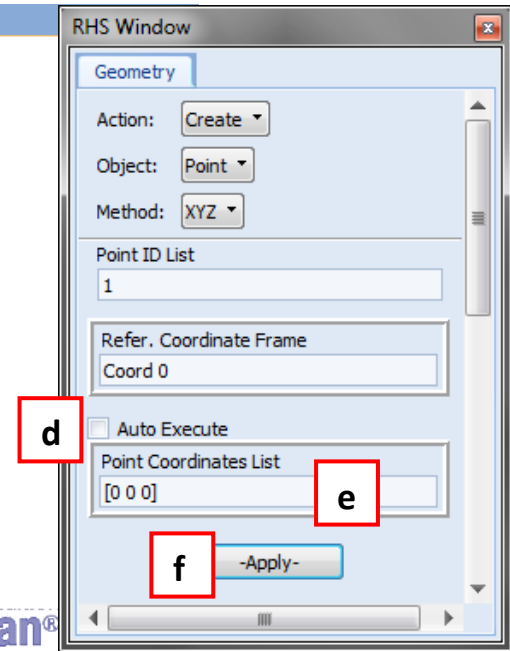


- a. Change *Background Color* to **Black** (click on the **Cycle Background** icon)
- b. Click on the **Point size** icon
- Create geometry points:
- c. Click on the **Geometry icon/Points icon/Select/XYZ**
- d. Uncheck **Auto Execute**
- e. Enter **[0 0 0]** as the Point Coordinates List
- f. Click **Apply**
- g. Create three more points using coordinates: **[0 1200 0]**, **[240 1200 0]**, **[1200 1200 0]**

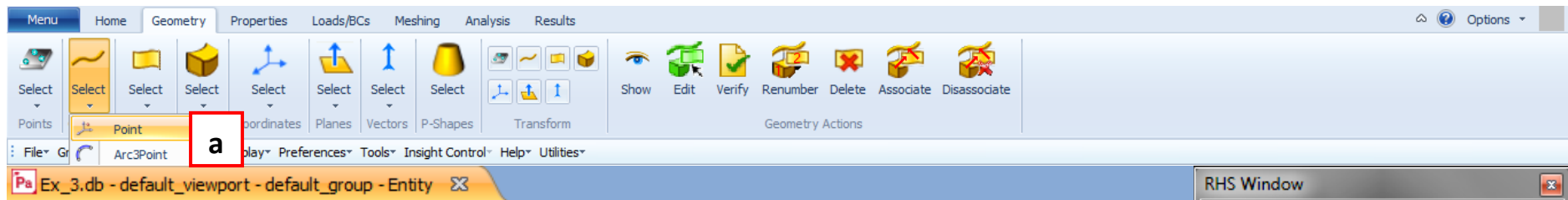


Point 1  
+

Point 2    Point 3    Point 4



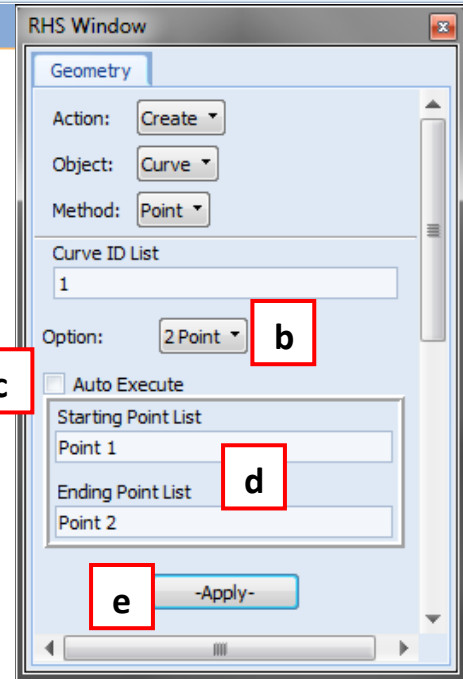
Patran®



Point 2      Point 3                      Point 4



- Create curves:
- a. Click on the **Geometry** icon: **Curves** icon/**Select/Point**
  - b. Option: **2 Point**
  - c. Uncheck **Auto Execute**
  - d. Select **Point 1** as the starting point and **Point 2** as the ending point
  - e. Click **Apply**
  - f. Create two more curves using:
    - **Point 2** as the starting point and **Point 3** as the ending point
    - **Point 3** as the starting point and **Point 4** as the ending point



**Patran**<sup>®</sup>  
Student Edition

Ex\_3.db - default\_viewport - default\_group - Entity

Mesh the curves:

- Click on the **Meshing** icon/**Curve** icon (*Meshers tab*)
- Topology: **Bar2**
- Click on the **Curve List** panel
- Select all visible curves by clicking and dragging the mouse
- Click **Apply**

Patran®  
Student Edition

RHS Window

Finite Elements

Action: Create

Object: Mesh

Type: Curve

Output ID List

Node: 1

Element: 1

Topology: Bar2

Node Coordinate Frames...

Curve List

Curve 1:3

Global Edge Length

Automatic Calculation

Value: 120.0

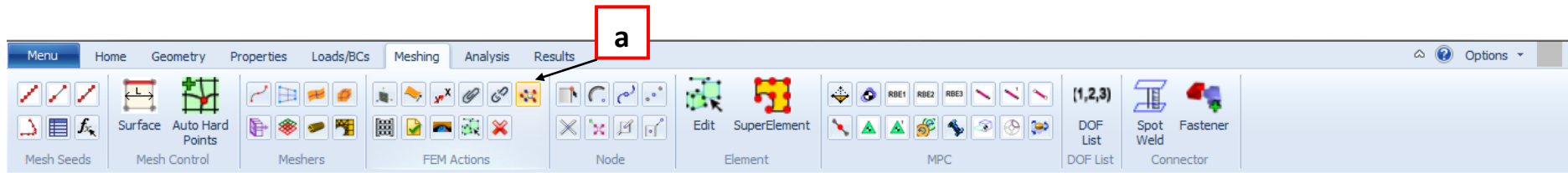
Prop. Name: - None -

Prop. Type: - N/A -

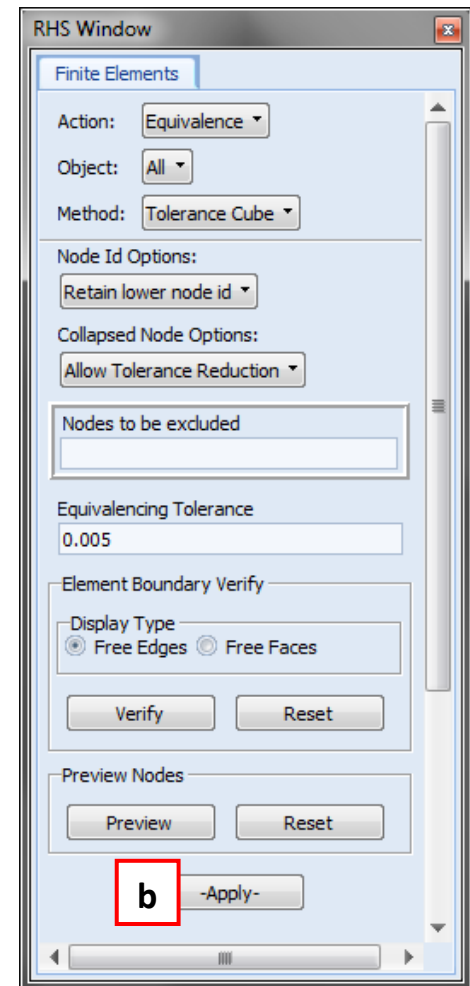
Select Existing Prop...

Create New Property...

-Apply-



Delete duplicate nodes:  
a. Meshing: **Equivalence** icon (*FEM Actions tab*)  
b. Click **Apply**



Apply the boundary conditions:

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



Menu Home Geometry Properties Loads/BCs Meshing Analysis Results

Displacement Force Temperature Velocity Acceleration Crack(VCCT)

Ex\_3.db - default\_viewport - default\_group - Entity

Conditions LBC Actions Load Cases LBC Fields

Point 4

Point 1

Patran® Student Edition

RHS Window

Load/Boundary Conditions

Action: Create

Object: Displacement

Type: Nodal

Option: Standard

Current Load Case: Default...

Type: Static

Existing Sets

New Set Name: pin

Input Data...

Select Application Region...

-Apply-

RHS Window

Conditions Select Application Region

Select: Geometry

Auto Select...

Application Region

Select Geometry Entities

Curve 1.1 3.2

Add Remove

Application Region

OK

- f. Click **Select Application Region...**
- g. Select: **Geometry**
- h. Click on the **Select Geometry Entities** panel
- i. Select **Point or Vertex** icon
- j. Select **Point 1** and **Point 4** (click on one of them, then press the **Shift Key** and holding it down click on the second point)
- k. Click **Add**
- l. Click **OK**
- m. Click **Apply**

Apply the force:

- Loads/BCs: **Force** icon
- Enter **load** as the New Set Name
- Click **Input Data...**
- Enter **<0,-20000,0>** for the Force
- Click **OK**

Ex\_3.db - default\_viewport - default\_group - Entity

20000.0000 Point 3

12345

Y  
X

12345

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

RHS Window

Conditions Select Application Region

Select: Geometry **g**

Auto Select...

Application Region

Select Geometry Entities

Point 3 **h,i**

**j** Add Remove

Application Region

**k** OK

RHS Window

Load/Boundary Conditions

Action: Create

Object: Force

Type: Nodal

Current Load Case: Default...

Type: Static

Existing Sets

New Set Name: load

Input Data... **f**

Select Application Region... **f**

**l** -Apply-

Define a **a** material:

- Click on the **Properties** icon/**Isotropic** icon
- Enter **mat** as the Material Name
- Click **Input Properties...**
- Enter **71.74e3** as the Elastic Modulus and **0.0** as the Poisson Ratio
- Click **OK**
- Click **Apply**

**Input Options**

Constitutive Model: Linear Elastic

Property Name	Value
Elastic Modulus =	71.74e3 <b>d</b>
Poisson Ratio =	0.0
Shear Modulus =	
Density =	
Thermal Expan. Coeff =	
Structural Damping Coeff =	
Reference Temperature =	

Temperature Dep/Model Variable Fields:

Current Constitutive Models:

**e** OK Clear Cancel

**RHS Window**

Materials

Action: Create

Object: Isotropic

Method: Manual Input

Existing Materials

Filter ON/OFF

Filter \*

Material Name **b** mat

Description

**c** Input Properties ...

Change Material Status ...

**f** Apply

Menu Home Geometry Properties Loads/BCs Meshing Analysis Results Options

Isotropic Orthotropic Anisotropic Fluid Cohesive Composite 0D Properties 1D Properties 2D Properties 3D Properties Property Actions Fields

**Input Properties**  
General Beam (CBEAM)

Property Name	Value	Value Type
[Section Name]	na:	Properties
Material Name		Mat Prop Name
Bar Orientation		Vector
[Offset @ Node 1]		Vector
[Offset @ Node 2]		Vector
[Pinned DOFs @ Node 1]		String
[Pinned DOFs @ Node 2]		String
[Warning Option]		

Create Sections  
ICL...  
Beam Library  
 Assoc. Beam Section

Enter the Section Name, select existing section using the icon, or use the create sections icon below to create a new section.

OK Clear Cancel

**RHS Window**  
Element Properties

Action: Create  
Object: 1D  
Type: Beam

Sets By: Name

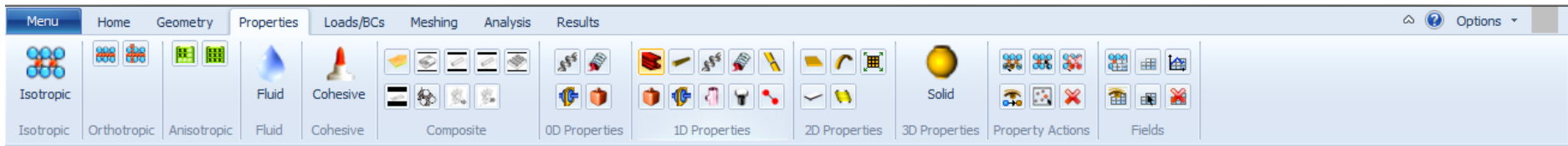
Filter ON/OFF  
Filter \*



Property Set Name  
frame

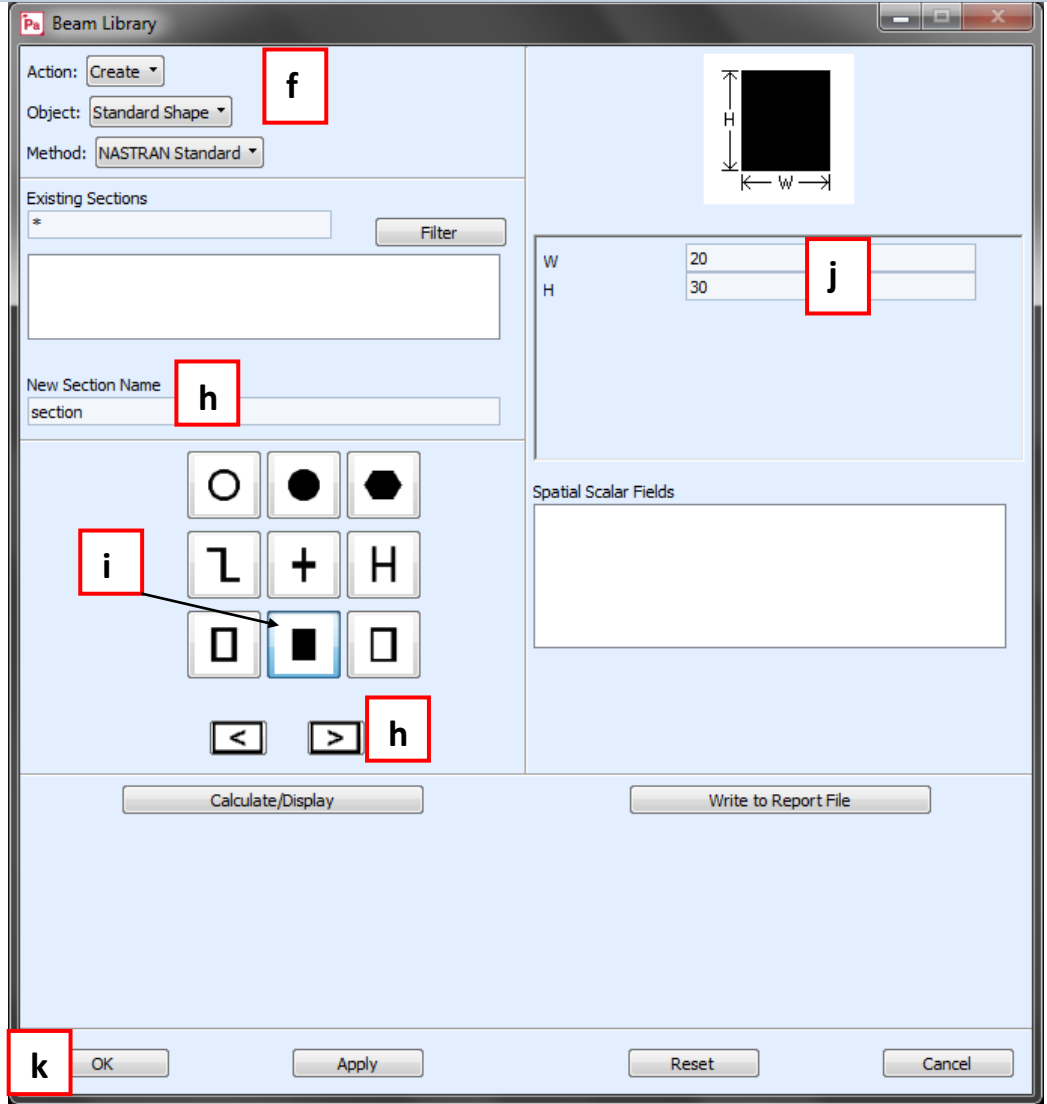
Options:  
General Section (CBEAM)  
Standard Formulation  
Input Properties ...  
Select Application Region ...  
Apply

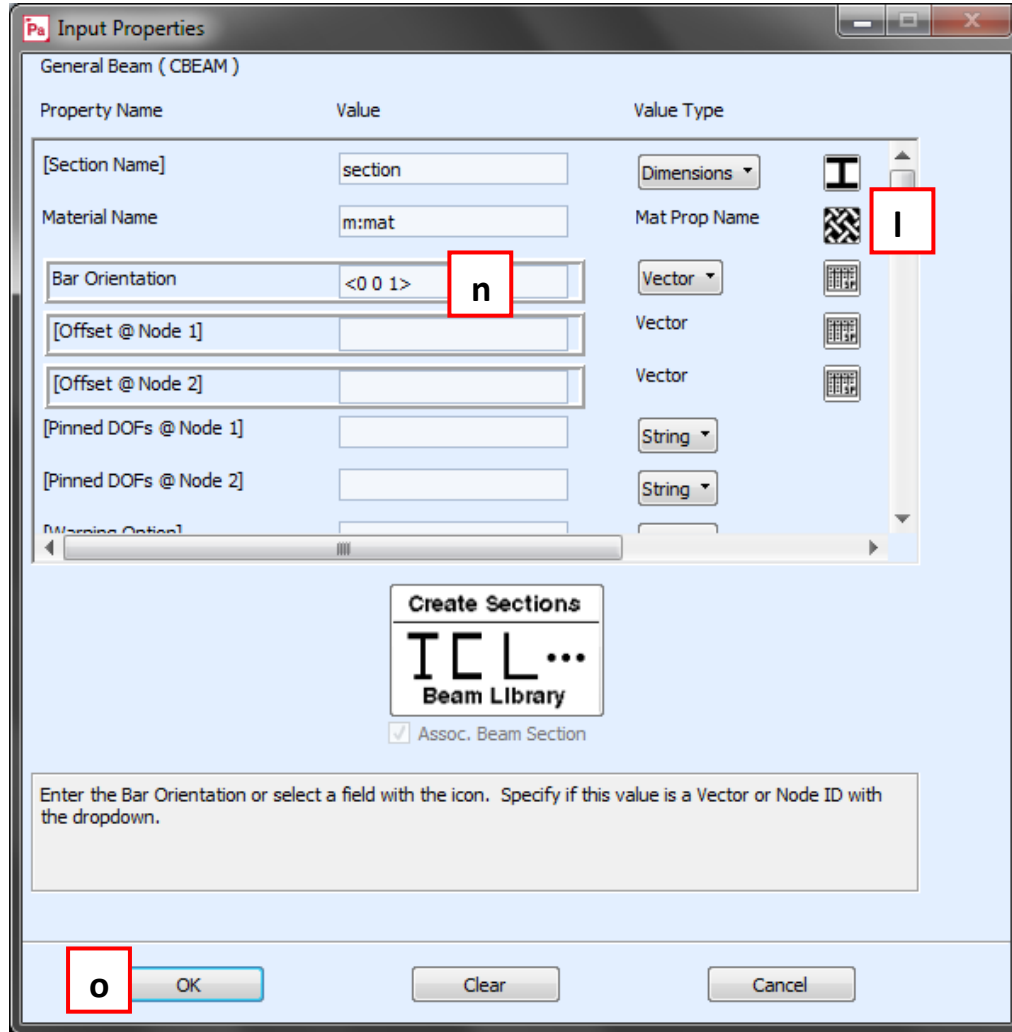
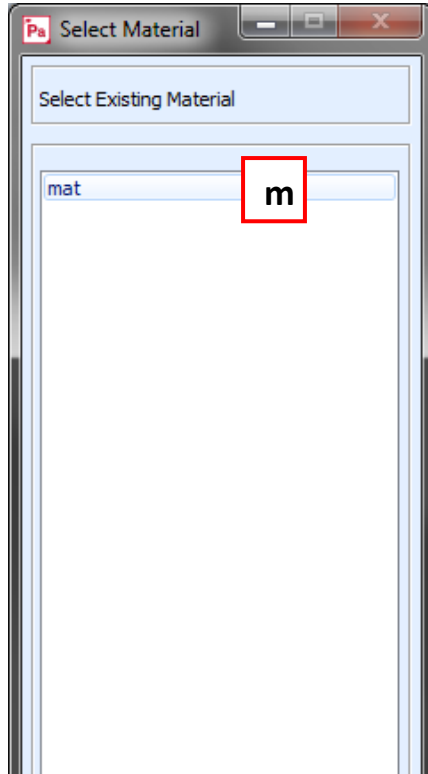
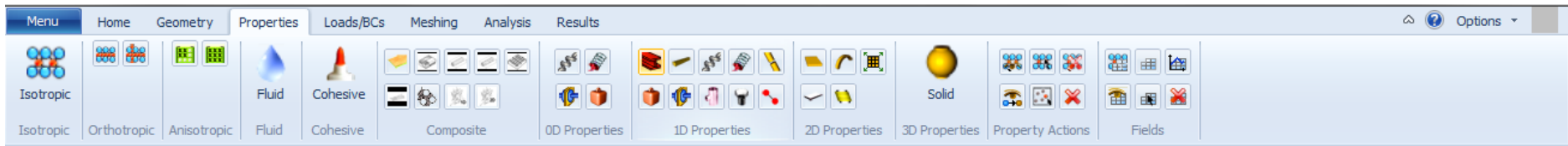
Assign the properties:

- Properties: Beam icon**
- Enter **frame** as the New Set Name
- Select options: **General Section (CBEAM)/Standard Formulation**
- Click **Input Properties...**
- Click on the **Create Sections** icon



- f. Create Sections: **Create/Standard Shape/NASTRAN Standard**
- g. Enter **section** as the New Section Name
- h. Click 
- i. Click 
- j. Enter **20** as the value of W and **30** as the value of H
- k. Click **OK**





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

p. Click **Select Application Region...**  
 q. Click on the **Select Members** panel  
 r. Select **Beam element** icon  
 s. Select all beam elements by clicking and dragging the mouse  
 t. Click **Add**  
 u. Click **OK**  
 v. Click **Apply**



The image shows the MSC.Nastran software interface with several dialog boxes open. The main window has a ribbon with tabs: Menu, Home, Geometry, Properties, Loads/BCs, Meshing, Analysis, and Results. The Analysis tab is active, showing icons for Analysis Deck, Read, Submit, HDF5, XDB, Read Output2, Attach Output2, t16/t19, d3plot, Delete, Job/View Output, and Actions.

Four dialog boxes are open:

- Results Output Format:** Contains checkboxes for OP2, XDB (checked), Print, and Punch. It also has checkboxes for HDF5 (checked), Compressed, and No Rot. DOFs. A dropdown for XDB Buffer Size is set to 1024. Buttons for OK, Defaults, and Cancel are at the bottom.
- Solution Parameters:** Contains sections for Nonlinear Static Solution Parameters (Automatic Constraints, Large Displacements checked, Follower Forces unchecked), Solution Sequence (SOL 106), Sol700 Parameters..., Shell Normal Tol. Angle, Mass Calculation (Lumped), Data Deck Echo (None), Plate Rz Stiffness Factor (100.0), Maximum Printed Lines, Maximum Run Time, Wt.-Mass Conversion (1.0), Node i.d. for Wt. Gener., Default Initial Temperature, Default Load Temperature, and Rigid Element Type (LINEAR). Buttons for Results Output Format..., OK, Defaults, and Cancel are at the bottom.
- RHS Window (Solution Type):** Shows MSC.Nastran Solution Type with radio buttons for Solution Type: LINEAR STATIC, NONLINEAR STATIC (selected), NORMAL MODES, BUCKLING, COMPLEX EIGENVALUE, FREQUENCY RESPONSE, TRANSIENT RESPONSE, NONLINEAR TRANSIENT, IMPLICIT NONLINEAR, and DDAM Solution. Buttons for Select ASET/QSET..., Solution Parameters..., and OK/Cancel are at the bottom.
- RHS Window (Analysis):** Shows Analysis settings: Action (Analyze), Object (Entire Model), Method (Analysis Deck), Code (MSC.Nastran), and Type (Structural). It also has fields for Job Name (Ex\_3) and Job Description (TITLE). Buttons for Translation Parameters..., Solution Type..., Direct Text Input..., Select Superelements..., Subcases..., and Subcase Select... are at the bottom.

Numbered callouts (a-j) point to specific elements in the interface:

- a:** Analysis Deck icon in the ribbon.
- b:** Solution Type... button in the RHS Window (Analysis).
- c:** NONLINEAR STATIC radio button in the RHS Window (Solution Type).
- d:** Solution Parameters... button in the RHS Window (Solution Type).
- e:** Large Displacements checkbox 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 (Solution Type).

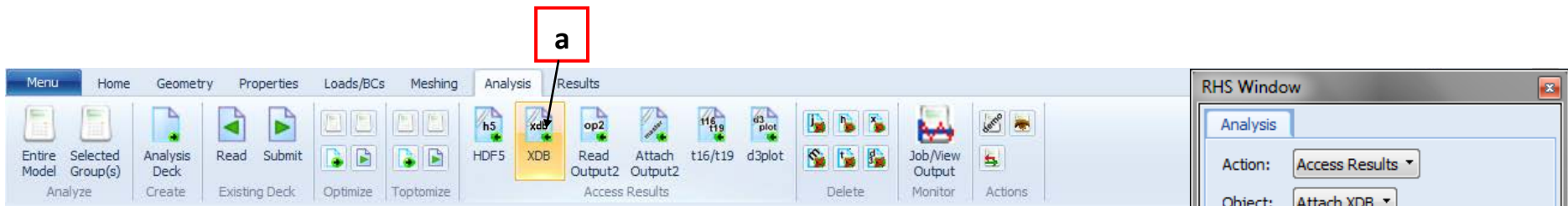
- Run a nonlinear analysis:
- Click on the **Analysis/Analysis Deck** icon
  - Click **Solution Type...**
  - Select **NONLINEARSTATIC** as the Solution Type
  - Click **Solution Parameters...**
  - Uncheck **Follower Forces**
  - Click **Results Output Format...**
  - Uncheck **Print** and check **XDB**
  - Click **OK**
  - Click **OK**
  - Click **OK**

**Analysis Deck Configuration Steps:**

- k. Click **Subcases...**
- l. Select **Default**
- m. Click **Subcase Parameters...**
- n. Enter **20** as the Number of Load Increments
- o. Click **Arc-Length Method**
- p. Select **CRIS** and check **Use Arc-Length Method**
- q. Enter **50** as the Convergence Iterations and **100** as the Max. Controlled Increment Steps
- r. Click **OK**
- s. Click **OK**

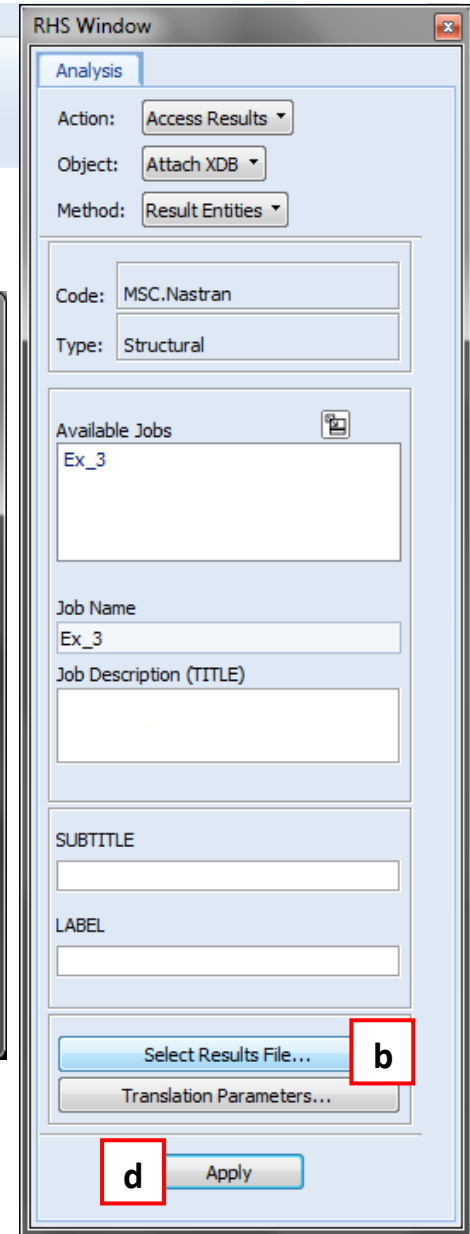
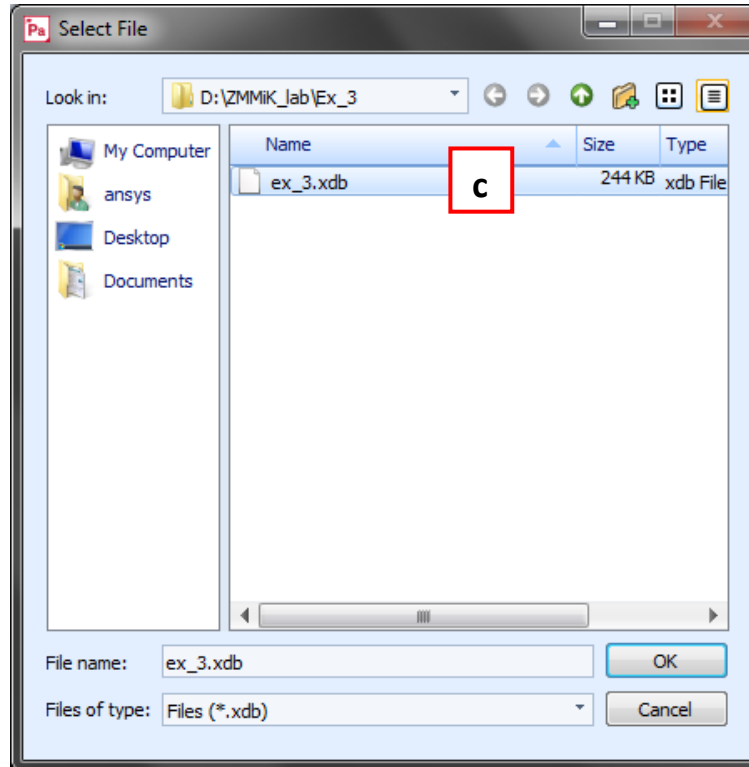
The screenshot displays the MSC Nastran software interface. The top ribbon includes tabs for Menu, Home, Geometry, Properties, Loads/BCs, Meshing, Analysis, and Results. The Analysis Deck is active, showing a list of output requests for the 'Default' subcase. The 'Output Requests' dialog is open, with 'Advanced' selected for the Form Type and 'Yes' for the Intermediate Output Option. The 'Subcases' dialog is also open, showing the 'Default' subcase. The 'RHS Window' is visible on the right, showing the analysis settings for 'MSC.Nastran' and 'Structural' type. A list of available jobs is shown, with 'Ex\_3' selected. The 'SUBTITLE' field is empty. The 'Apply' button at the bottom of the RHS Window is highlighted with a red box labeled 'z'.

- t. Click **Output Requests...**
- u. Form type: **Advanced**
- v. Intermediate Output Option: **Yes**
- w. Click **OK**
- x. Click **Apply**
- y. Click **Cancel**
- z. Click **Apply**
- aa. Run **Nastran** analysis using **Ex\_3a.bdf** file



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

- a. Analysis: **XDB** icon
- b. Click **Select Results File...**
- c. Select **Ex\_3.xdb** file and click **OK**
- d. Click **Apply**



**a.** Click on the **Plot/Erase Geometry** icon

Post-process the results:

**b.** Change background color to white

**c.** Click on the **Results** icon:

**Fringe/Deformation (Quick Plot)**

**d.** Select **all** steps

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

**f.** Quantity: **Magnitude**

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

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

**i.** Select **True Scale** with the Scale Factor equals to **1.0**

**j.** Click **Apply**

Remark: To capture the plot use **File / Images...**

Patran 2020 (Student Edition) 03-Apr-21 14:41:22  
 Fringe: Default, A1:Non-linear: 100. % of Load, Displacements, Translational, Magnitude, (NON-LAYERED)  
 Deform: Default, A1:Non-linear: 100. % of Load, Displacements, Translational, Magnitude, (NON-LAYERED)

Max 1.36+03 @Nd 11  
 Min 0. @Nd 1

default\_Fringe :  
 Max 1.36+03 @Nd 11  
 Min 0. @Nd 1

default\_Deformation :  
 Max 0.61+03 @Nd 11  
 Min 0. @Nd 1

The image shows the ANSYS Workbench software interface. The top ribbon is set to the 'Results' tab. A red box labeled 'a' points to the 'Graph' icon in the 'Result Plots' section. Below the ribbon, the 'Defaults' section contains a 'Reset Graphics' icon, highlighted with a red box labeled 'h'. The 'Select Result Cases' dialog box is open, showing a list of result cases. A red box labeled 'c' points to the 'Filter Method' dropdown menu, which is set to 'All'. A red box labeled 'd' points to the 'Filter' button. A red box labeled 'e' points to the list of selected result cases. A red box labeled 'f' points to the 'Apply' button at the bottom left of the dialog. A red box labeled 'g' points to the 'Close' button at the bottom right of the dialog. On the right side, the 'RHS Window' is visible, showing the 'Results' tab with a 'Graph' object selected. A red box labeled 'b' points to the 'Default, 0 of 75 subcases; -MSC.NASTRAN' entry in the 'Select Result Cases' list within the RHS window.

Create a graph (an equilibrium path):

- Results: **Graph** icon
- Select and click on the **SC1: 0 of 75 subcases;**
- Filter Method: **All**
- Click **Filter**
- Select **all** steps
- Click **Apply**
- Click **Close**
- Click **Reset Graphics**

**n**

**p**

**o**

**m**

**i**

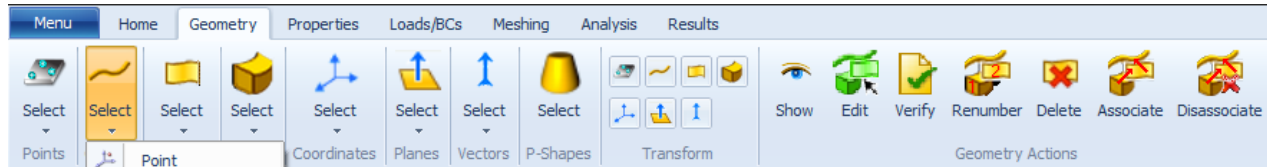
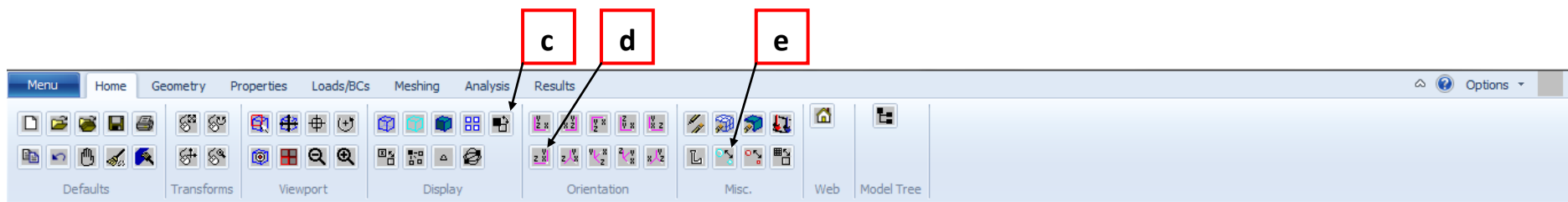
**j**

**k**

**l**

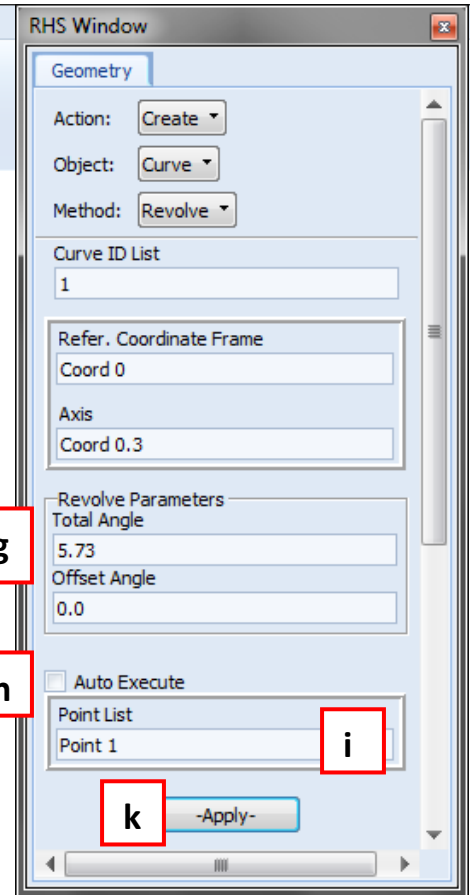
**q**

- i. Y: **Result**
- j. Select **Displacements, Translational** as the Y Result
- k. Quantity: **Y Component**
- l. X: **Global Variable**; Variable: **Percent of Load**
- m. Click on the **Target Entities** icon
- n. Select the node to which the force was applied
- o. Click on the **Display Attributes** icon
- p. Check **Show Symbol** and Uncheck **Sort Data By X Coordinate**
- q. Click **Apply**



## PART 2: Snap-through of a shallow, cylindrical roof under a point load

- a. Save and close the previous database
  - b. Create a new database (**Ex\_3b.db**)
  - c. Change *Viewport Color* to **Black** (click on the **Cycle Background** icon)
  - d. Click on the **Point size** icon
  - e. Create a point using coordinates: **[2540 0 0]**
- Create a curve:
- f. Click on the **Geometry** icon: **Curves** icon/**Select/Revolve**
  - g. Enter **5.73** as the Total Angle
  - h. Uncheck **Auto Execute**
  - i. Click on the **Point List** panel
  - j. Select the point created in step e
  - k. Click **Apply**





**a**

**b**

**c**

**d**

**e**

**f**

**g**

Menu Home Geometry Properties Loads/BCs Meshing Analysis Results

Menu Home Geometry Properties Loads/BCs Meshing Analysis Results

Geometry Actions

Ex\_3b.db - default\_viewport - default\_group - Entity

RHS Window

Geometry

Action: Create

Object: Surface

Method: Extrude

Surface ID List

1

Refer. Coordinate Frame

Coord 0

Origin of Scale and Rotate

[0 0 0]

Translation Vector

<0 0 254>

Sweep Parameters

Scale Factor

1.0

Angle

0.0

Auto Execute

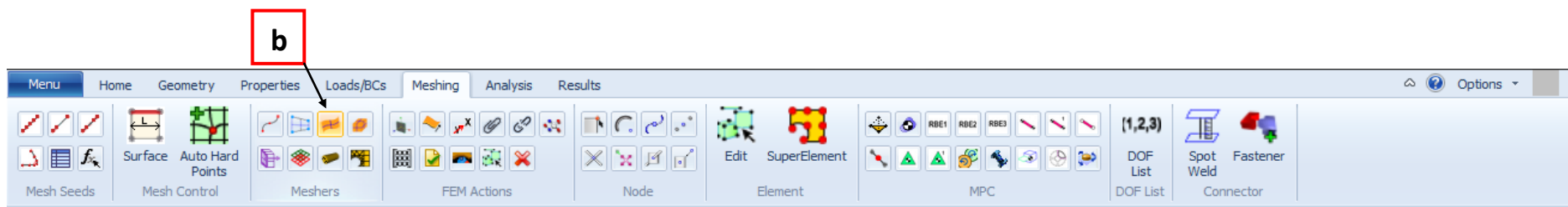
Curve List

-Apply-

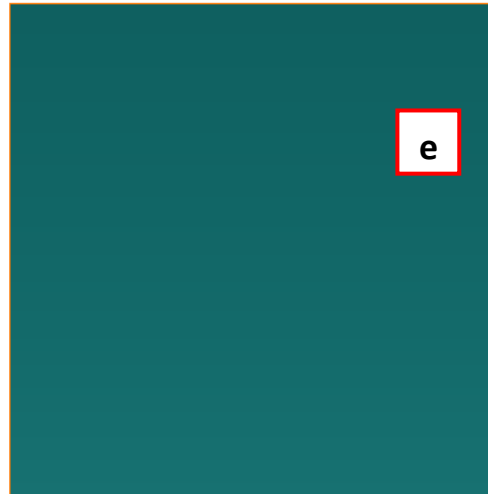
Y

Z

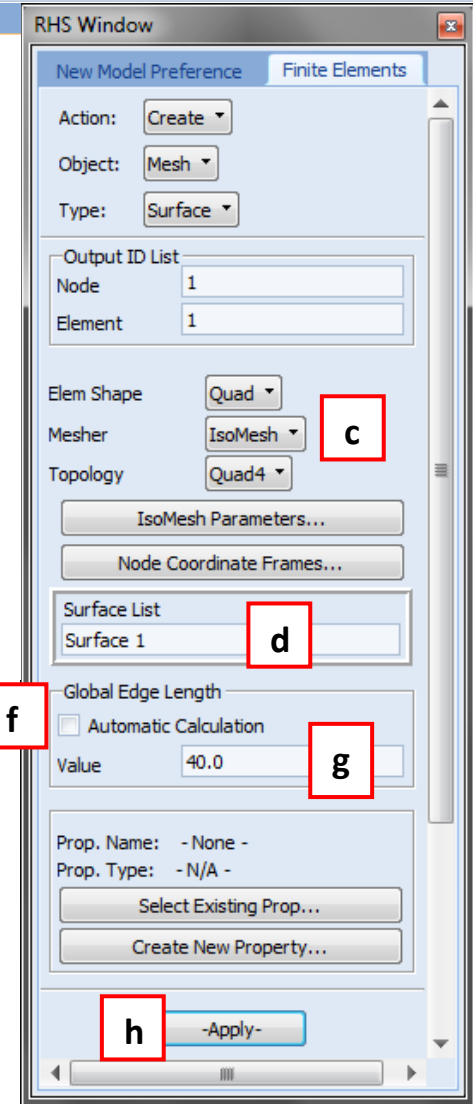
- a. Click on the **Right side view** icon
- Create a surface:
- b. Click on the **Geometry** icon: **Surface** icon/**Select/Extrude**
- c. Enter **<0 0 254>** as the Translation Vector
- d. Uncheck **Auto Execute**
- e. Click on the **Curve List** panel
- f. Select the curve
- g. Click **Apply**



- a. Click on the **Smooth shaded** icon  
 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 **40** as the Value of the Global Edge Length
  - Click **Apply**



Patran®  
Student Edition



The image shows the ANSYS Workbench software interface. The top ribbon includes tabs for Menu, Home, Geometry, Properties, Loads/BCs, Meshing, Analysis, and Results. The 'Loads/BCs' tab is active, showing various icons for applying loads and boundary conditions. A red box labeled 'a' points to the 'Plot/Erase Geometry' icon, and another red box labeled 'b' points to the 'Fit view' icon. Below the ribbon, the 'Displacement Constraint' icon is highlighted with a red box labeled 'c'. The 'Input Data' dialog box is open, showing the 'Load/BC Set Scale Factor' as 1. The 'Translations <T1 T2 T3>' field contains '<,,0>' (highlighted with a red box 'f'), and the 'Rotations <R1 R2 R3>' field contains '<0,0, >'. The 'Spatial Fields' section is empty. The 'Analysis Coordinate Frame' is set to 'Coord 0'. The 'OK' button is highlighted with a red box labeled 'g'. The 'RHS Window' on the right shows the 'Load/Boundary Conditions' panel with 'Action: Create', 'Object: Displacement', 'Type: Nodal', and 'Option: Standard'. The 'New Set Name' field contains 'sym\_xy' (highlighted with a red box 'd'), and the 'Input Data...' button is highlighted with a red box labeled 'e'.

- a. Click on the **Plot/Erase Geometry** icon  
 b. Click on the **Fit view** icon
- Apply the boundary conditions:
- c. Click on the **Loads/BCs icon/Displacement Constraint** icon  
 d. Enter **sym\_xy** as the New Set Name  
 e. Click **Input Data...**  
 f. Enter **<,,0>** for the Translations and **<0,0, >** for the Rotations  
 g. Click **OK**

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

i. Select: **FEM**

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

k. Select the nodes (belonging to the right free edge of the model) by clicking and dragging the mouse

l. Click **Add**

m. Click **OK**

n. Click **Apply**

o. Enter **sym\_xz** as the New Set Name

p. Click **Input Data...**

q. Enter **<,0,>** for the Translations and **<0,,0>** for the Rotations

r. Click **OK**

The image shows the ANSYS software interface with a meshed model and two dialog boxes. The 'Select Application Region' dialog is open, showing 'FEM' selected and 'Node 1:43:7' entered. The 'Load/Boundary Conditions' dialog is also open, showing 'Displacement' as the object and 'Nodal' as the type. A list of steps (s-y) is provided in a box at the bottom left.

**Steps:**

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

z. Enter **pinned\_edge** as the New Set Name

aa. Click **Input Data...**

bb. Enter **<0,0,0>** for the Translations and **<0,0, >** for the Rotations

cc. Click **OK**

dd. Click **Select Application Region...**

ee. Select: **FEM**

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

gg. Select the nodes (belonging to the top free edge of the model) by clicking and dragging the mouse

hh. Click **Add**

ii. Click **OK**

jj. Click **Apply**



Apply the force:

- Loads/BCs: **Force** icon
- Enter **load** as the New Set Name
- Click **Input Data...**
- Enter **<-3000,,>** for the Force
- Click **OK**

The image shows a software interface with a meshed rectangular domain. The domain is divided into a grid of elements. A coordinate system is shown in the bottom left corner with axes labeled Z and Y. Two dialog boxes are open. The left dialog box is titled "Select Application Region" and has the following fields and buttons: "Select" (FEM), "Select Nodes" (Node 1), "Add", "Remove", and "OK". The right dialog box is titled "Load/Boundary Conditions" and has the following fields and buttons: "Action" (Create), "Object" (Force), "Type" (Nodal), "Current Load Case" (Default...), "Type" (Static), "Existing Sets" (empty list), "New Set Name" (load), "Input Data...", "Select Application Region...", and "-Apply-".

Ex\_3b.db - default\_viewport - default\_group - Entity

RHS Window

reference Load/Boundary Conditions

Action: Create

Object: Force

Type: Nodal

Current Load Case:

Default...

Type: Static

Existing Sets

New Set Name

load

Input Data...

Select Application Region...

-Apply-

RHS Window

Conditions Select Application Region

Select: FEM

Auto Select...

Application Region

Select Nodes

Node 1

Add Remove

Application Region

OK

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

g. Select: **FEM**

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

i. Select the node

j. Click **Add**

k. Click **OK**

l. Click **Apply**

**Define a material:**

- Click on the **Properties** icon/**Isotropic** icon
- Enter **mat** as the Material Name
- Click **Input Properties...**
- Enter **3103** as the Elastic Modulus and **0.3** as the Poisson Ratio
- Click **OK**
- Click **Apply**

**Assign the properties:**

- Properties: **Shell** icon
- Enter **roof** as the New Set Name
- Click **Input Properties...**
- Click on the **Mat Prop Name** icon
- Select **mat**
- Enter **6.35** 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**

**Run a nonlinear analysis:**

- Click on the **Analysis/Analysis Deck** icon
- Click **Solution Type...**
- Select **NONLINEAR STATIC** as the Solution Type
- Click **Solution Parameters...**
- Uncheck **Follower Forces**
- Click **Results Output Format...**
- Uncheck **Print**
- Click **OK**
- Click **OK**
- 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, Topoptimize, Access Results, Delete, Monitor, and Actions.

Three dialog boxes are open:

- Arc-Length Method Parameters:** Contains fields for 'Use Arc-Length Method' (checked), 'Constraint type' (CRIS), 'Min. Adjust. ratio (MINALR)' (0.1), 'Max. Adjust. ratio (MAXALR)' (1), 'Scale Factor (W)' (0.00), 'Convergence Iterations' (12), and 'Max. Controlled Increment Steps' (20). Buttons include 'OK', 'Defaults', and 'Cancel'.
- Subcase Parameters:** Contains fields for 'Static Nonlinear Iterations' (25), 'Total Time', 'Matrix Update Method' (Automatic), 'Number of Iterations per Update' (5), and 'Allowable Iterations per Increment' (25). It also has checkboxes for 'Convergence Criteria' (Displacement Error, Load Error, Work Error) and 'Normal Modes' (Normal Modes, Buckling). Buttons include 'OK', 'Cancel', and 'Arc-Length Method ...'.
- Subcases:** Shows 'Solution Sequence: 106' and 'Action: Create'. It lists 'Available Subcases' (Default) and 'Available Load Cases' (Default). Buttons include 'Subcase Parameters...', 'Output Requests...', 'Direct Text Input...', 'Select Explicit MPCs...', 'Apply', and 'Cancel'.

The **RHS Window** on the right shows 'Analysis' settings: 'Action: Analyze', 'Object: Entire Model', 'Method: Analysis Deck', 'Code: MSC.Nastran', 'Type: Structural', 'Job Name: Ex\_3b', and 'Job Description (TITLE)'. It also has fields for 'SUBTITLE' and 'LABEL'. Buttons include 'Translation Parameters...', 'Solution Type...', 'Direct Text Input...', 'Select Superelements...', 'Subcases...', and 'Subcase Select...'.

Red boxes with letters mark the following elements:

- p:** 'Use Arc-Length Method' checkbox
- q:** 'Min. Adjust. ratio (MINALR)' field
- r:** 'OK' button in Arc-Length Method Parameters
- n:** 'Number of Load Increments' field
- o:** 'Arc-Length Method ...' button
- s:** 'OK' button in Subcase Parameters
- l:** 'Default' subcase in Subcases
- m:** 'Subcase Parameters...' button in Subcases
- k:** 'Solution Type...' button in RHS Window

- k. Click **Subcases...**
- l. Select **Default**
- m. Click **Subcase Parameters...**
- n. Enter **25** as the Number of Load Increments
- o. Click **Arc-Length Method**
- p. Select **CRIS** and check **Use Arc-Length Method**
- q. Enter **0.1** as the Min. Adjust. ratio and **1** as the Max. Adjust. ratio
- r. Click **OK**
- s. Click **OK**

**Output Requests**

SUBCASE NAME: Default  
SOLUTION SEQUENCE: 106

Form Type: **Advanced** (u)

Select Result Type

- Displacements
- Element Stresses
- Constraint Forces
- Multi-Point Constraint Forces
- Element Forces
- Applied Loads
- Element Strain Energies
- Element Strains

Output Requests

```
DISPLACEMENT(SORT 1,REAL)=All FEM
STRESS(SORT 1,REAL,VONMISES,BILIN)=All FEM;PARAM,NOCO
SPCFORCES(SORT 1,REAL)=All FEM
```

Select Group(s)/SET

- All FEM
- default\_group

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

Intermediate Output Option: **Yes** (v)

**w** OK

**x** Apply

**y** Cancel

**Subcases**

Solution Sequence: 106

Action: **Create**

Available Subcases

- Default

Subcase Name: Default

Available Load Cases

- Default

Subcase Options

- Subcase Parameters...
- Output Requests...** (t)
- Direct Text Input...
- Select Explicit MPCs...

**z** Apply

**RHS Window**

Analysis

Action: Analyze

Object: Entire Model

Method: Analysis Deck

Code: MSC.Nastran

Type: Structural

Available Jobs

Job Name: Ex\_3b

Job Description (TITLE)

SUBTITLE

LABEL

Translation Parameters...

Solution Type...

Direct Text Input...

Select Superelements...

Subcases...

Subcase Select...

**z** Apply

**t.** Click **Output Requests...**

**u.** Form type: **Advanced**

**v.** Intermediate Output Option: **Yes**

**w.** Click **OK**

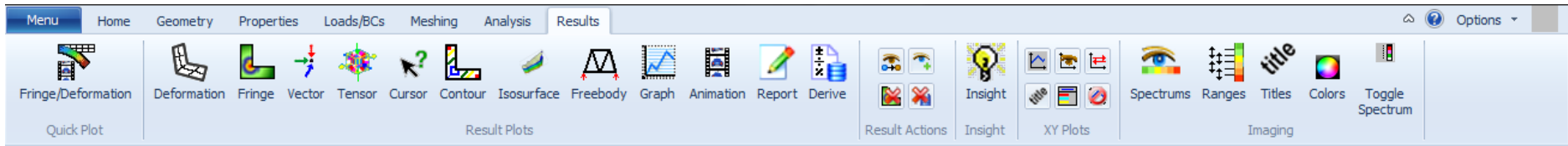
**x.** Click **Apply**

**y.** Click **Cancel**

**z.** Click **Apply**

**aa.** Run **Nastran** analysis using **Ex\_3b.bdf** file



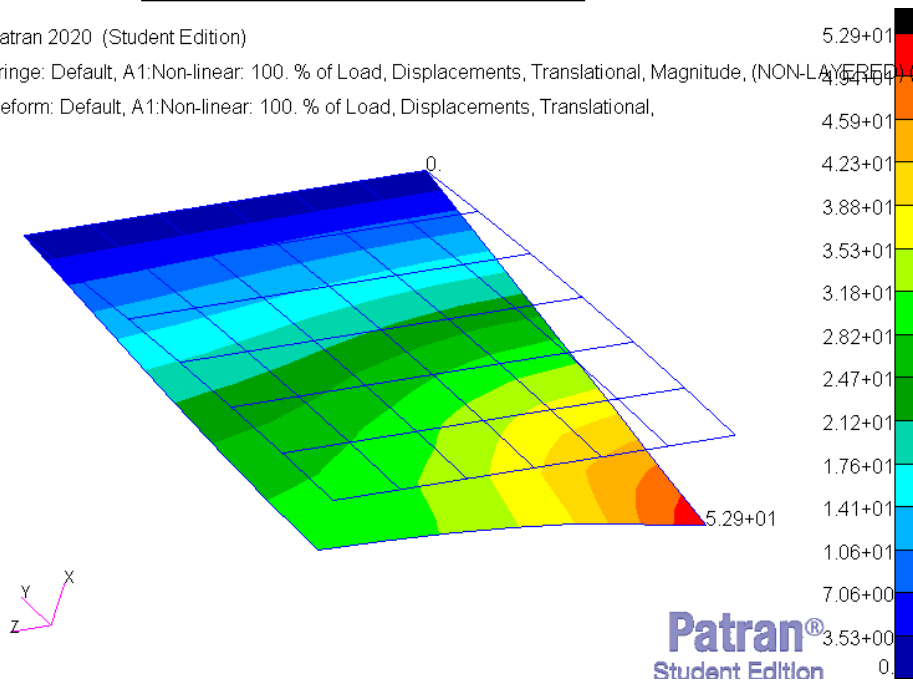


- a. Attach the results file
- b. Post-process the results

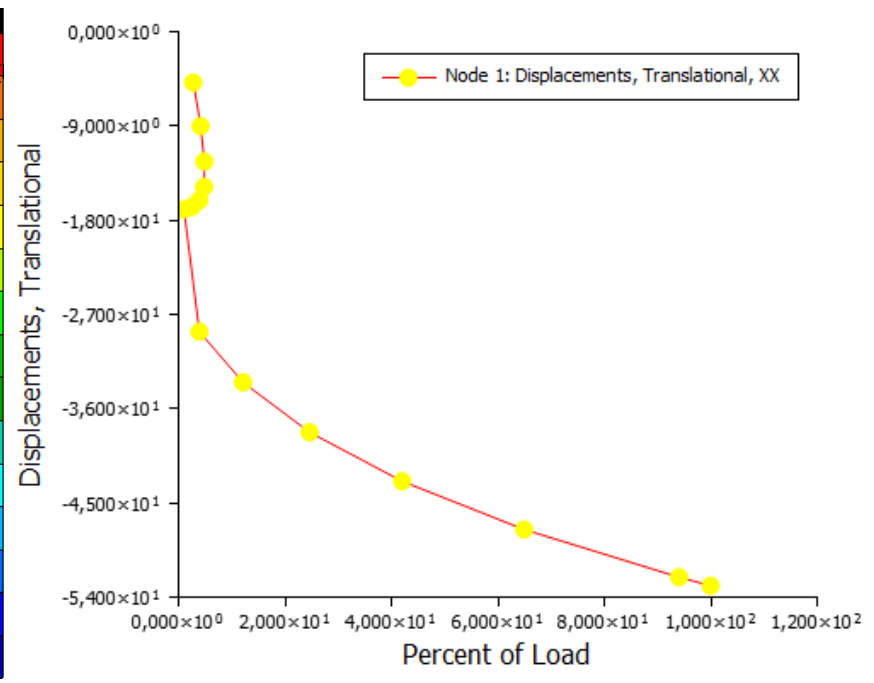
Patran 2020 (Student Edition)

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

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



100% of Load; magnitude displacements



Equilibrium path