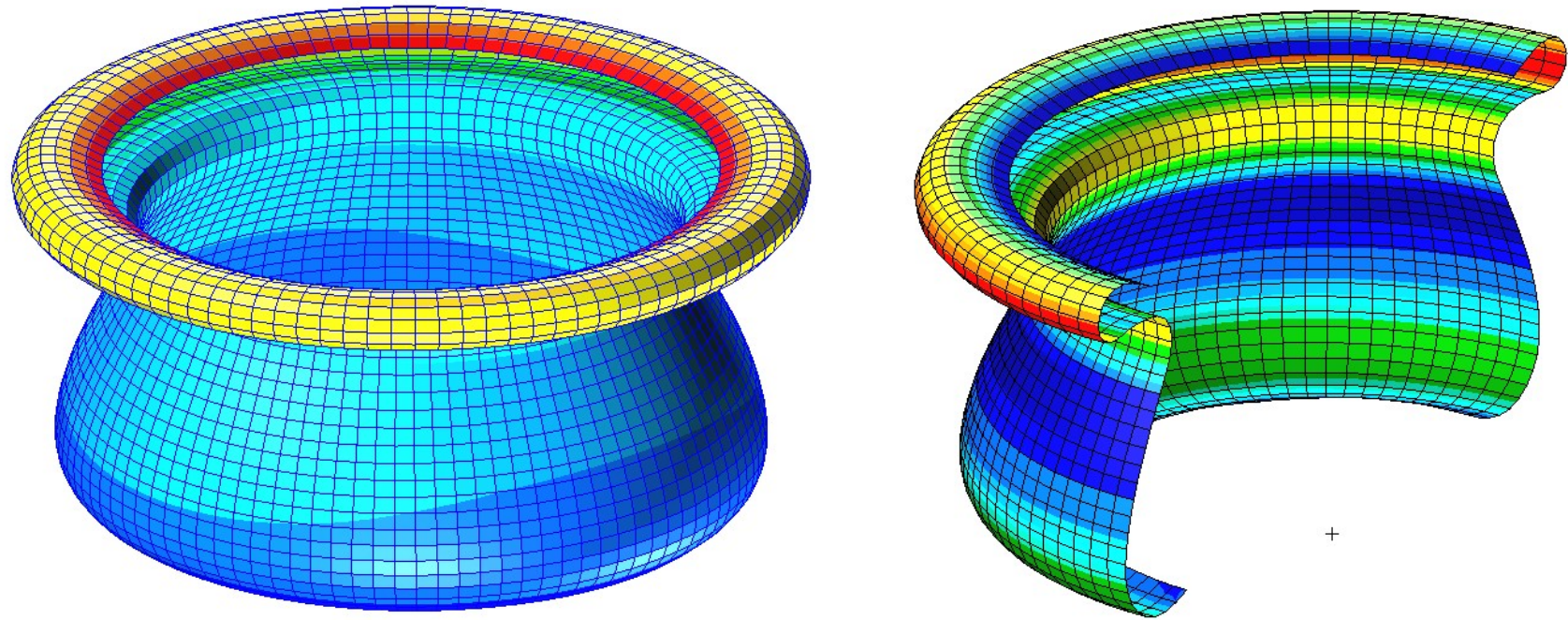
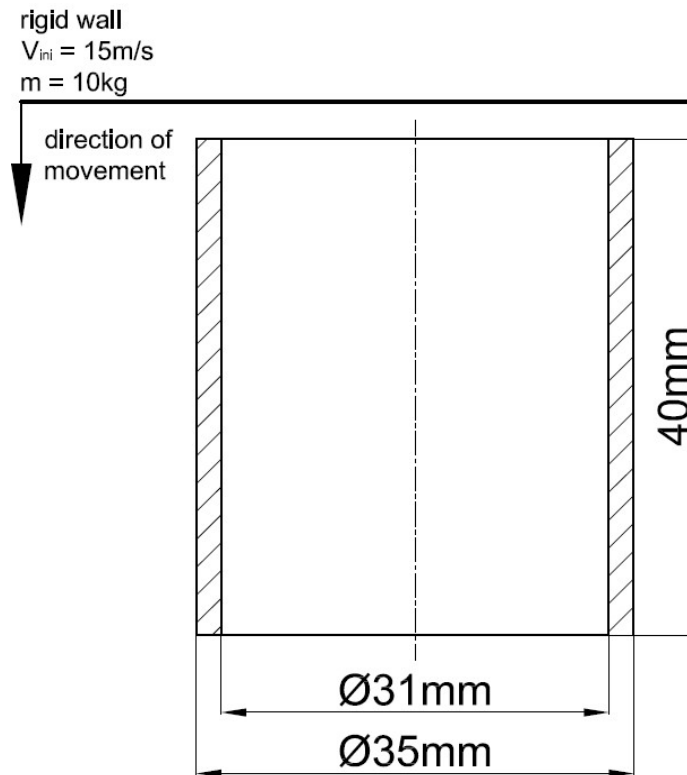


NONLINEAR MECHANICS OF STRUCTURES

EXERCISE 5

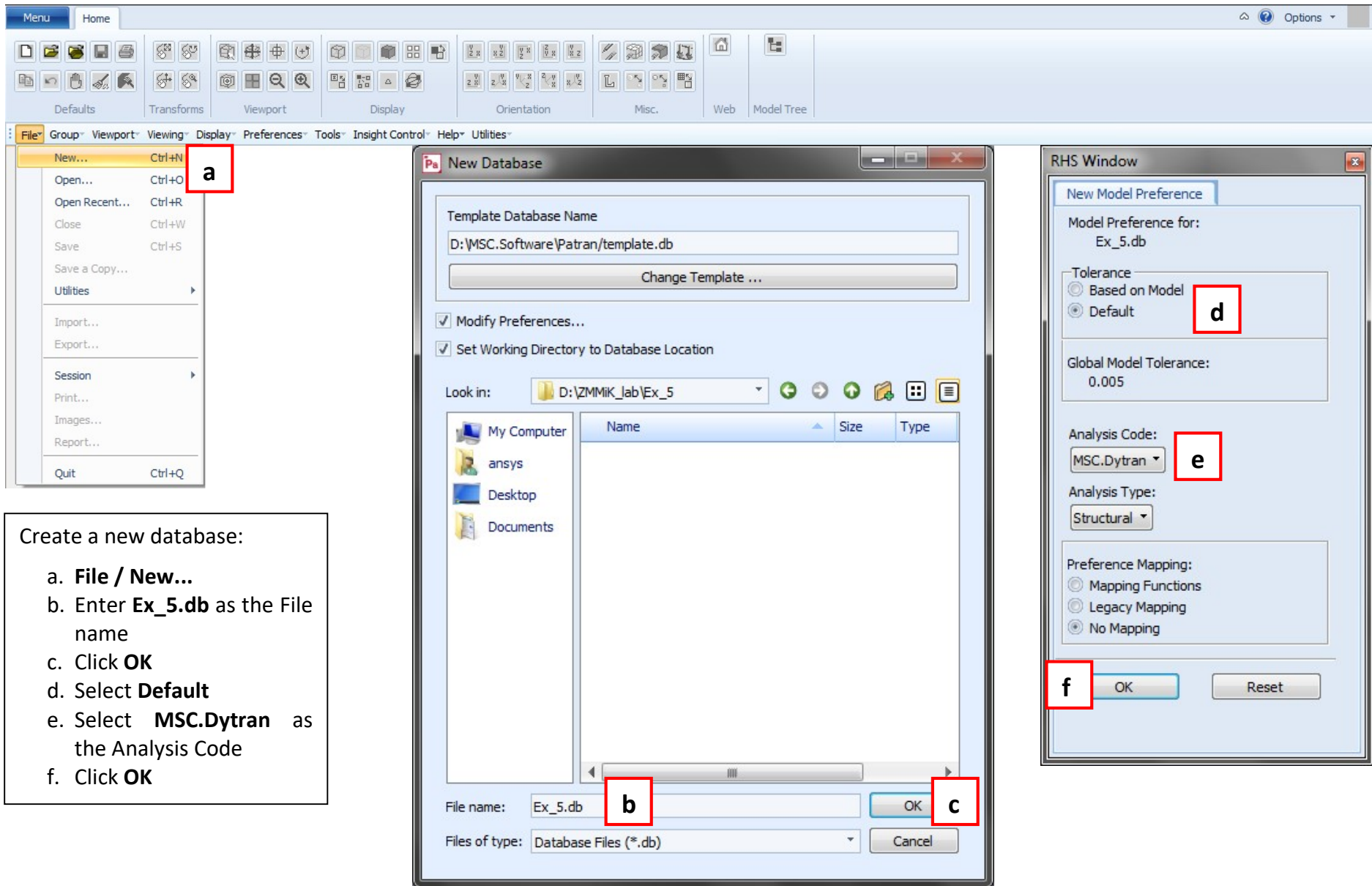


PROBLEM DESCRIPTION

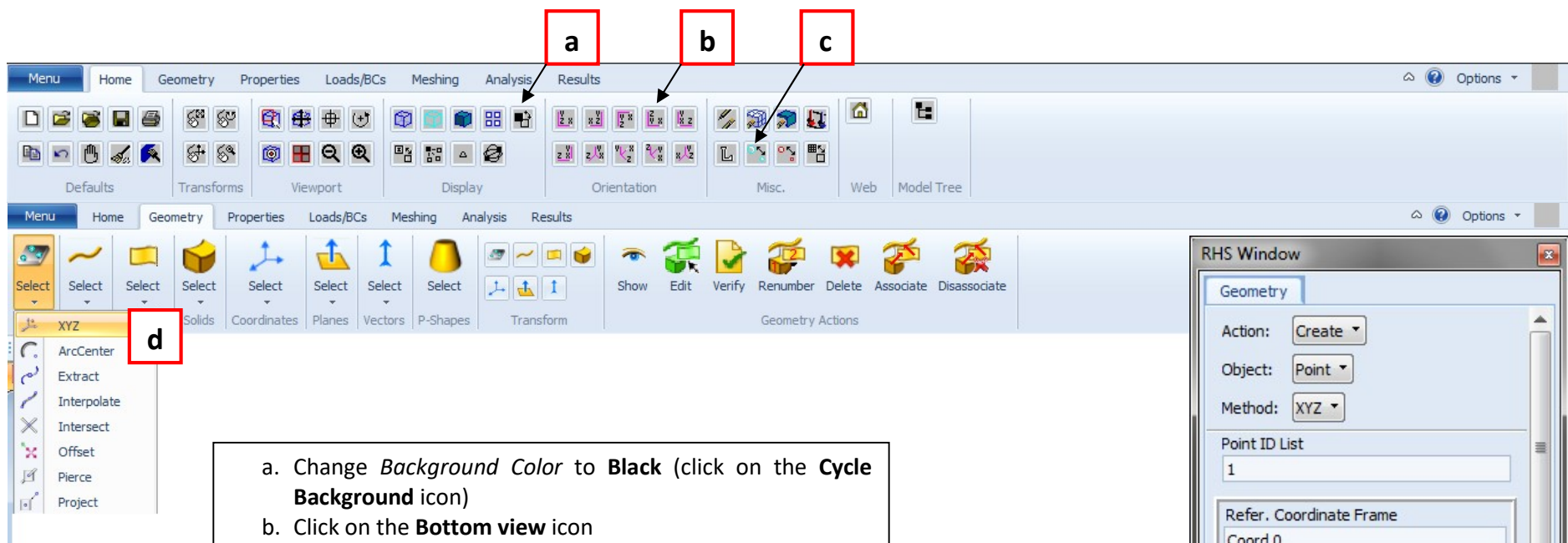


A thin-walled aluminum cylindrical element is struck by a rigid wall. The mass of the wall is 10kg and its initial velocity is equal to 15m/s. The goal of this exercise is to access the crashworthiness of the element (which can be used further e.g. in the crumple zone of a passenger car).

Units: mm, kg, ms, kN, GPa



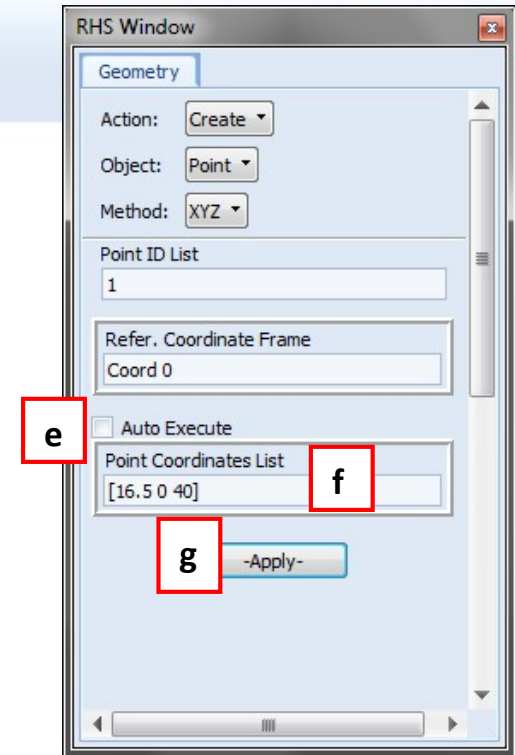
- Create a new database:
- a. **File / New...**
 - b. Enter **Ex_5.db** as the File name
 - c. Click **OK**
 - d. Select **Default**
 - e. Select **MSC.Dytran** as the Analysis Code
 - f. Click **OK**

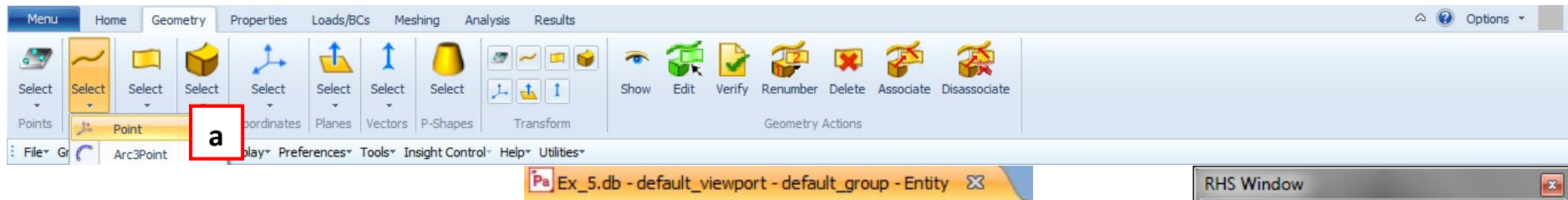


- a. Change *Background Color* to **Black** (click on the **Cycle Background** icon)
- b. Click on the **Bottom view** icon
- c. Click on the **Point size** icon

Create geometry points:

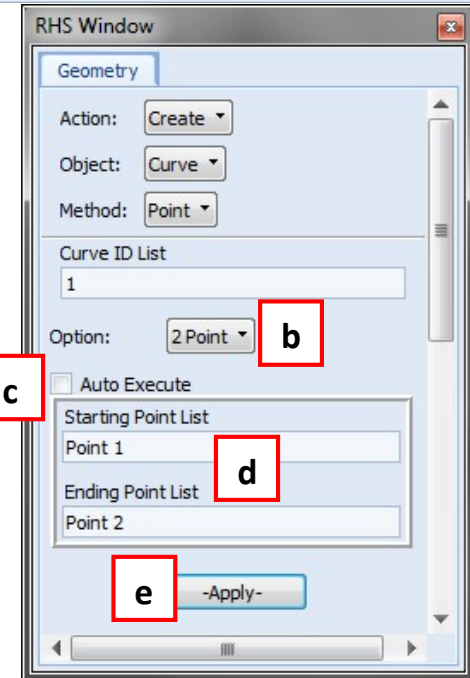
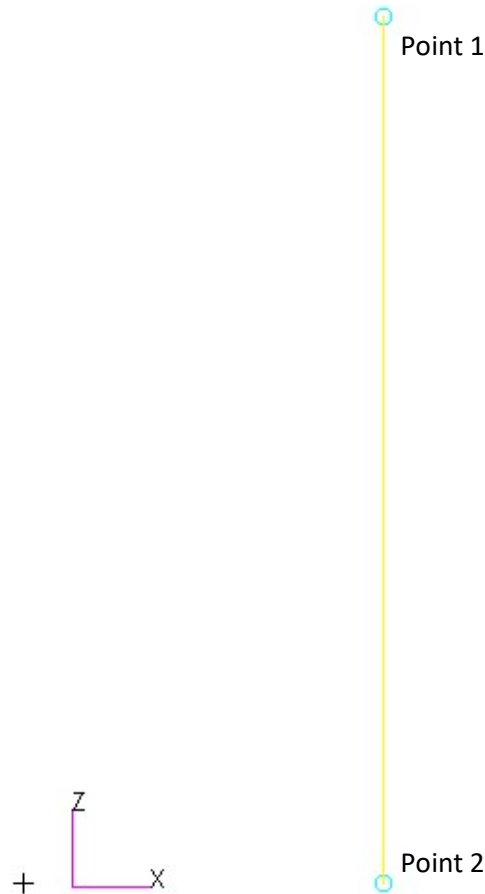
- d. Click on the **Geometry icon/Points icon/Select/XYZ**
- e. Uncheck **Auto Execute**
- f. Enter **[16.5 0 40]** as the Point Coordinates List
- g. Click **Apply**
- h. Create one more point using coordinates: **[16.5 0 0]**

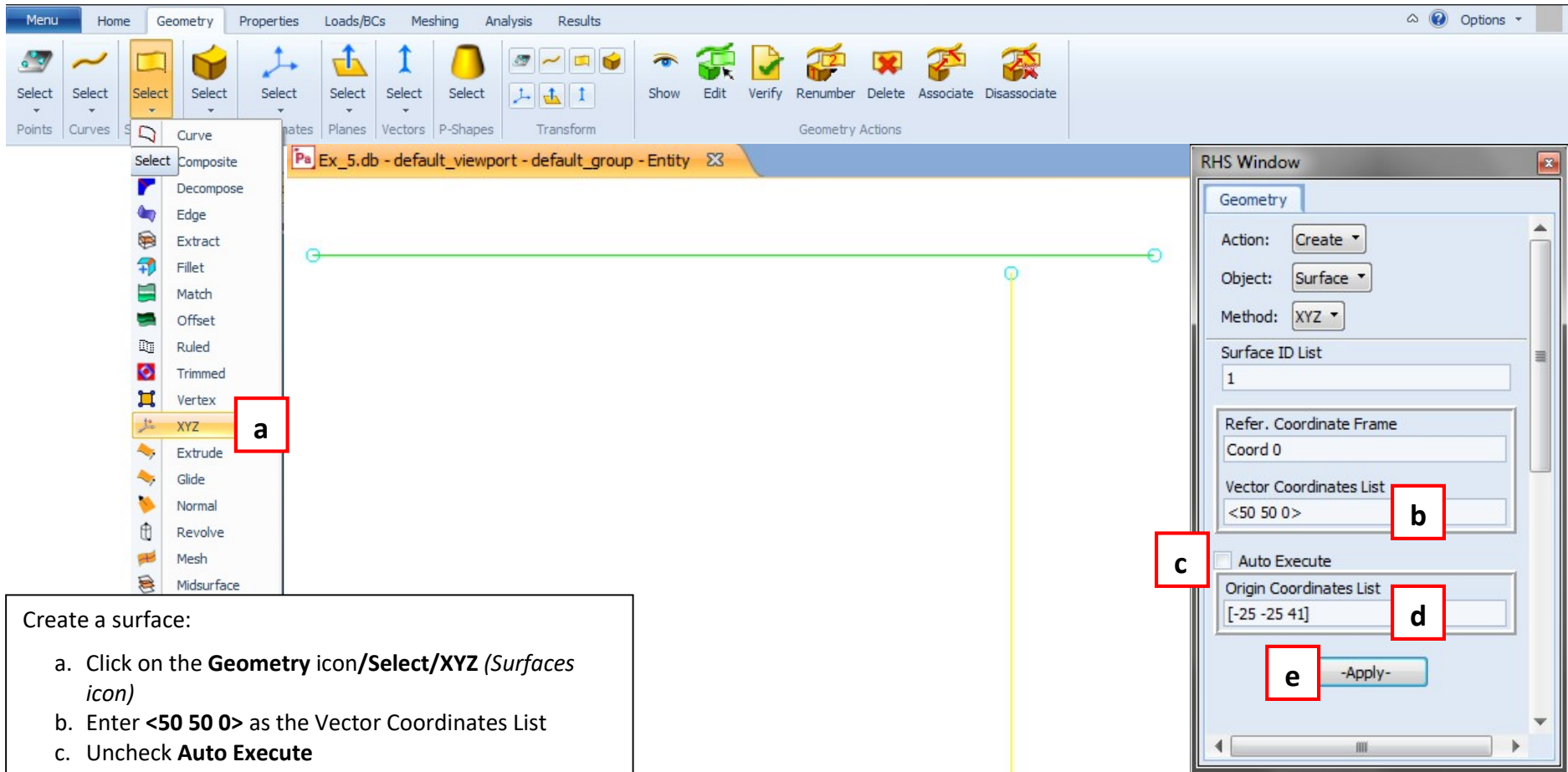




Create a curve:

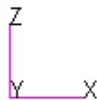
- a. Click on the **Geometry** icon: **Curves** icon/**Select/Point**
- b. Option: **2 Point**
- c. Uncheck **Auto Execute**
- d. Select **Point 1** as the starting point and **Point 2** as the ending point
- e. Click **Apply**





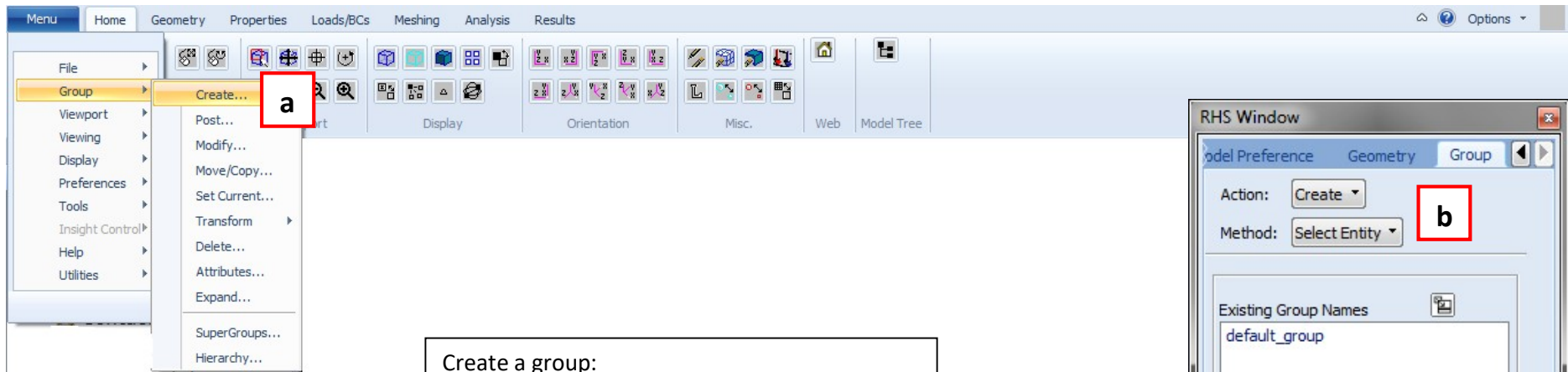
Create a surface:

- Click on the **Geometry** icon/**Select/XYZ** (*Surfaces icon*)
- Enter **<50 50 0>** as the Vector Coordinates List
- Uncheck **Auto Execute**
- Enter **[-25 -25 41]** as the Origin Coordinates List
- Click **Apply**



+

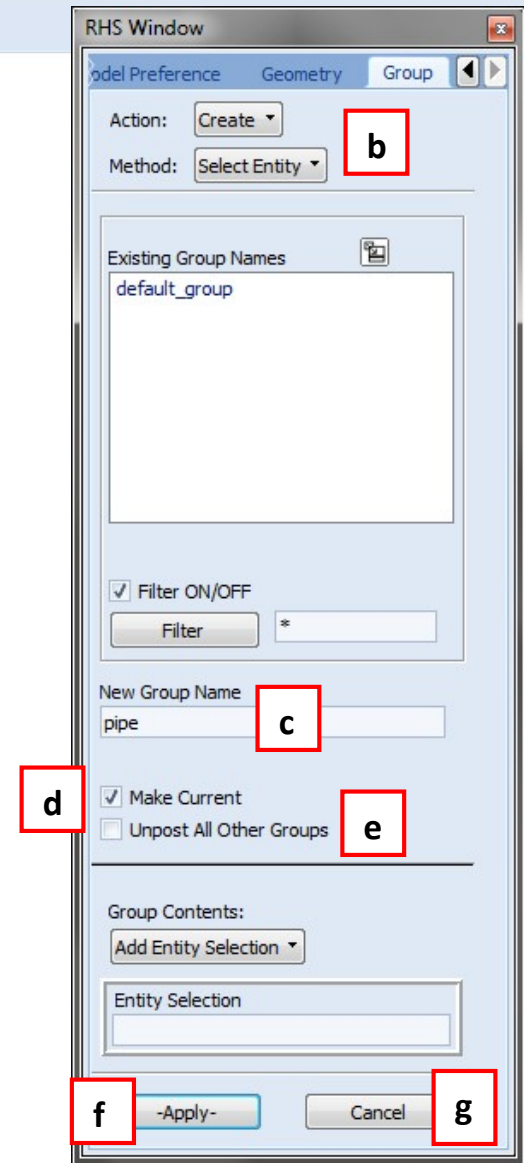


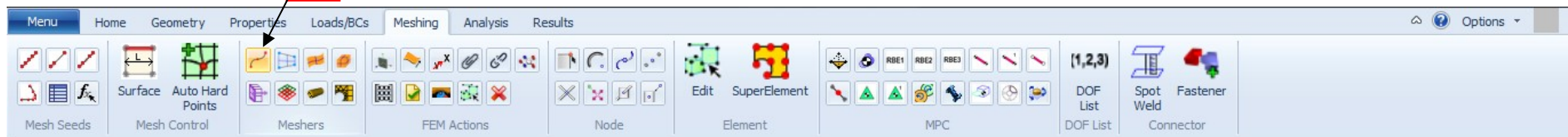


Create a group:

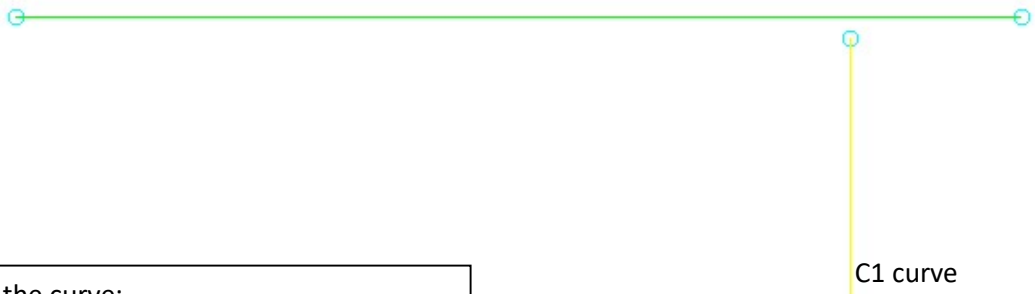
- a. **Group / Create...**
- b. Group: **Create/Select Entity**
- c. Enter **pipe** as the New Group Name
- d. Check **Make Current**
- e. Uncheck **Unpost All Other Groups**
- f. Click **Apply**
- g. Click **Cancel**

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

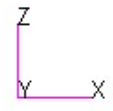




Ex_5.db - default_viewport - default_group - Entity



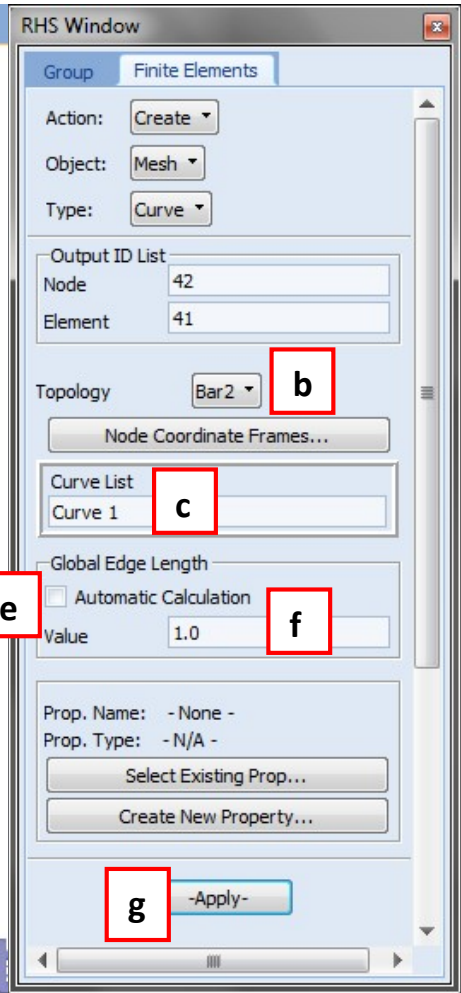
- Mesh the curve:
- Click on the **Meshing** icon/**Curve** icon (Meshers tab)
 - Topology: **Bar2**
 - Click on the **Curve List** panel
 - Select the **C1** (vertical) curve
 - Uncheck **Automatic Calculation**
 - Enter **1** as the Value of the Global Edge Length
 - Click **Apply**



+

d

C1 curve



Pat
Student

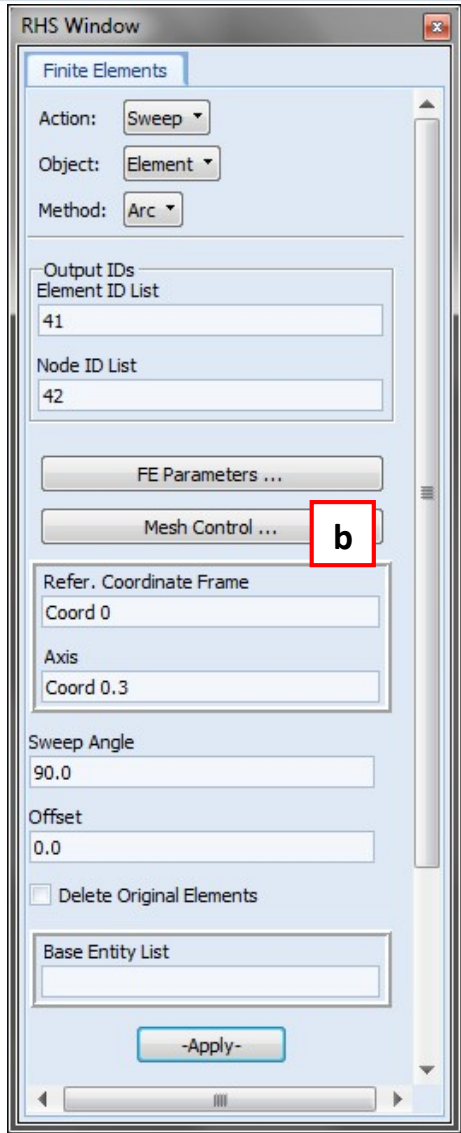
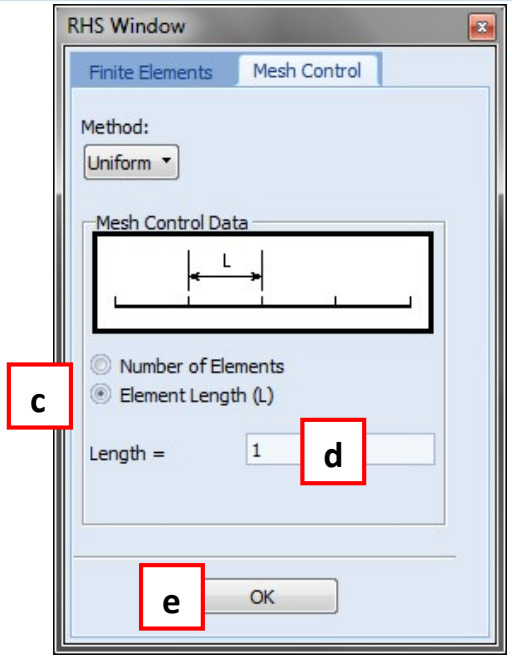
Post only the **pipe** group:

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



Sweep the elements:

- a. Meshing: **Sweep** icon (FEM Actions tab)
- b. Click **Mesh Control...**
- c. Select **Element Length (L)**
- d. Enter **1** as the Length
- e. Click **OK**



Ex_5.db - default_viewport - default_group - Entity

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

RHS Window

Finite Elements

Action: Sweep
 Object: Element
 Method: Arc

Output IDs
 Element ID List
 41
 Node ID List
 42

FE Parameters ...
 Mesh Control ...

Refer. Coordinate Frame
 Coord 0
 Axis
 Coord 0.3

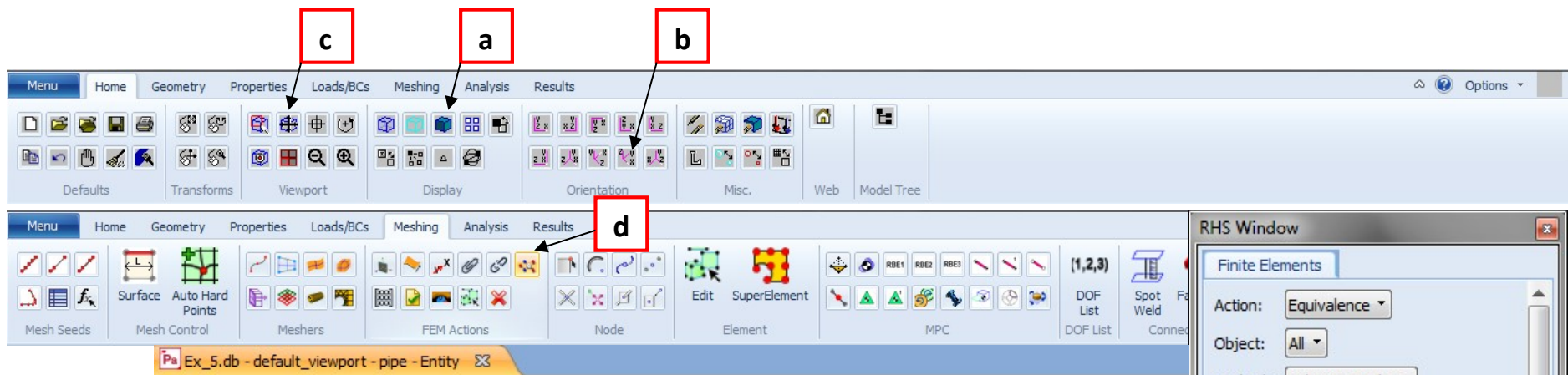
Sweep Angle **f**
 360

Offset
 0.0

Delete Original Elements **g**

Base Entity List **h**
 Elm 1:40

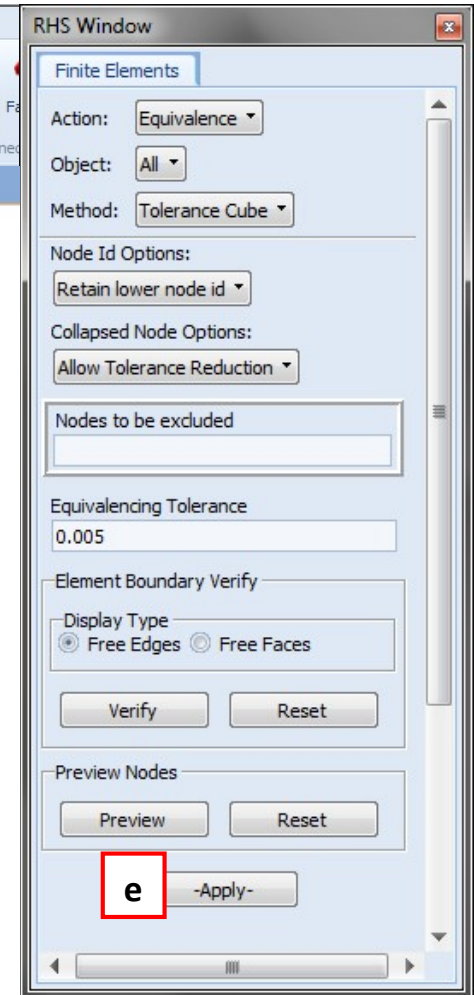
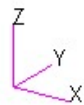
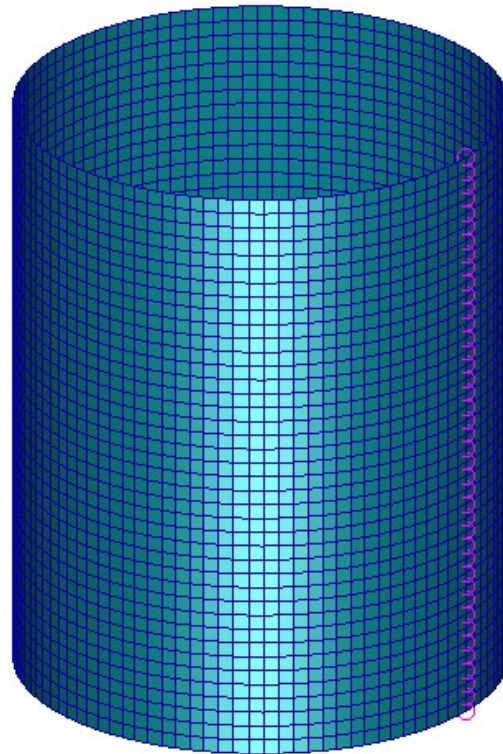
l -Apply-



- a. Click on the **Smooth shaded** icon
- b. Click on the **Iso 3 View** icon
- c. Click on the **Fit view** icon

Delete duplicate nodes:

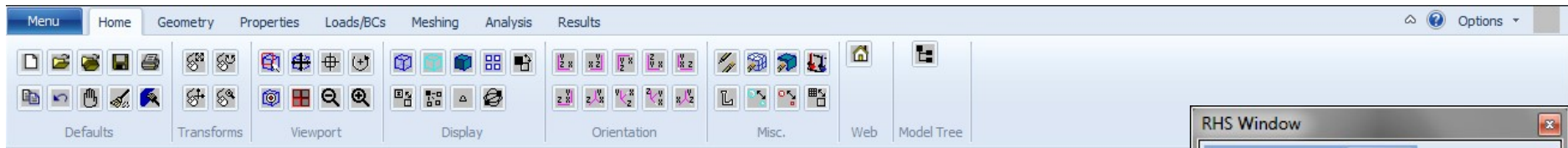
- d. Meshing: **Equivalence** icon (*FEM Actions tab*)
- e. Click **Apply**



Patran®
Student Edition

Post only the **default_group**:

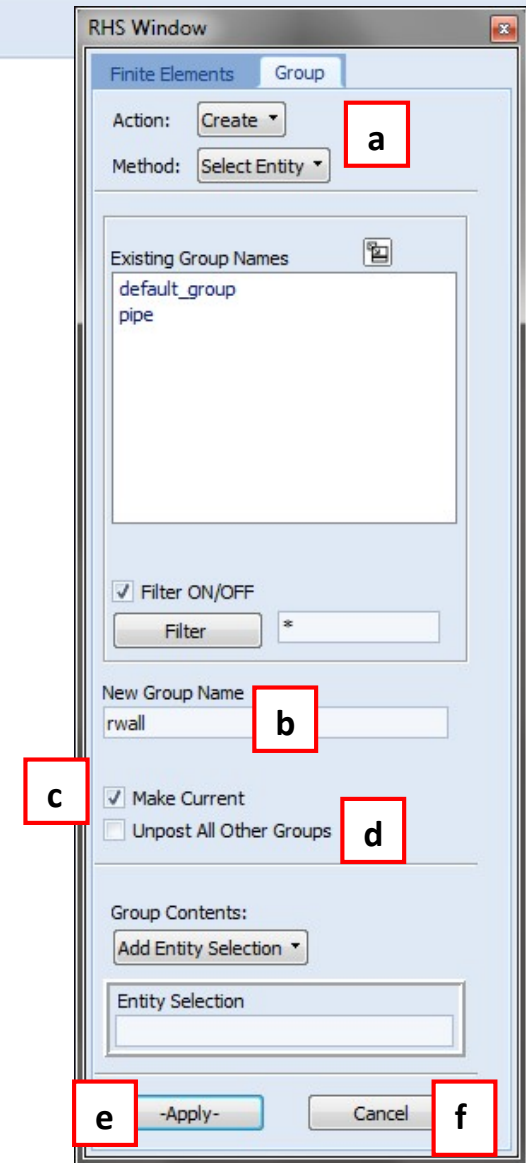
- Group / Post...
- Select **default_group**
- Click **Apply**



Create one more group:

- a. Change *Action* to **Create**
- b. Enter **rwall** as the New Group Name
- c. Check **Make Current**
- d. Uncheck **Unpost All Other Groups**
- e. Click **Apply**
- f. Click **Cancel**

Remark: Mesh of the moving rigid wall will be assigned to this group.



Menu Home Geometry Properties Loads/BCs Meshing Analysis Results Options

Mesh Seeds Surface Auto Hard Points Meshers FEM Actions Node Element MPC

Ex_5.db - default_viewport - rwall - Entity

a

d

S1 surface

b

c

e

Finite Elements

Action: Create

Object: Mesh

Type: Surface

Output ID List

Node 4265

Element 4161

Elem Shape Quad

Mesher IsoMesh

Topology Quad4

IsoMesh Parameters...

Node Coordinate Frames...

Surface List

Global Edge Length

Automatic Calculation

Value 0.1

Prop. Name: - None -

Prop. Type: - N/A -

Select Existing Prop...

Create New Property...

-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 **S1** surface
- Click **Apply**

Patran® Student Edition

Define materials:

- Click on the **Properties** icon/**Isotropic** icon
- Enter **aluminum** as the Material Name
- Click **Input Properties...**
- Constitutive Model: **ElasPlas (DYMAT24)**; Element Type: **Shell**; Yield Model: **Bilinear**; Strain Rate Model & Failure Model: **None**
- Enter **2.8e-6** as Density, **72** as Elastic Modulus, **0.33** as Poisson Ratio, **0.345** as Yield Stress and **0.765** as Hardening Modulus
- Click **OK**
- Click **Apply**

h. Enter **rigid** as the Material Name

i. Click **Input Properties...**

j. Constitutive Model: **Rigid (MATRIG)**; Valid For: **Shell**; Rigid Body Properties: **Geometry**

k. Enter **72** as Elastic Modulus, **0.33** as Poisson Ratio and **10** as Mass (of the wall)

l. Click **OK**

m. Click **Apply**

b

Menu Home Geometry Properties **Loads/BCs** Meshing Analysis Results

Displacement Force Follower Force Body Force Velocity BJOIN KJOIN Rotational Boundary Detonation Wave Mesh Generator

Nodal Element Uniform Contact Initial Conditions Rigid LBC Actions

a. Post only the **rwall** group:
Menu/Group/Post.../select only rwall
 Apply initial velocity to the rigid wall:

b. Click on the **Loads/BCs** icon/**Initial Velocity** icon (*Initial Conditions* tab)

c. Enter **ini_vel** as the New Set Name

d. Click **Input Data...**

e. Enter **<,,-15>** for the Trans Veloc

f. Click **OK**

Input Data

Load/BC Set Scale Factor
1.

Trans Veloc <v1 v2 v3> **e**
<,,-15>

Rot Veloc <w1 w2 w3>
< >

Spatial Fields

FEM Dependent Data...

Analysis Coordinate Frame
Coord 0

f OK Reset

RHS Window

Load/Boundary Conditions

Action: Create

Object: Initial Velocity

Type: Nodal

Current Load Case:
Default...

Type: Time Dependent

Existing Sets

New Set Name **c**
ini_vel

d Input Data...
Select Application Region...

-Apply-

g. Click **Select Application Region...**

h. Select **FEM**

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

j. Select the nodes belonging to the rigid wall by clicking and dragging the mouse

k. Click **Add**

l. Click **OK**

m. Click **Apply**

b

a. Post only the **pipe** group
Menu/Group/Post.../select only pipe
 Define a planar rigid wall which will be used as the bottom boundary/barrier for the cylindrical element:

b. Click on the **Loads/BCs** icon/**Planar Rigid Wall** icon (*Rigid tab*)

c. Enter **pwall** as the New Set Name

d. Click **Input Data...**

e. Select **Penalty**

f. Enter **0.3** as the Static Friction Coefficient and **0.15** as the Kinematic Friction Coefficient

g. Centroid and Z-Orientation: **Coord 0**

h. Click **OK**

The image shows the Patran software interface with a 3D mesh of a cylinder. Two dialog boxes are open. The 'Select Application Region' dialog has buttons labeled i, j, k, m, n. The 'Load/Boundary Conditions' dialog has buttons labeled i, o. A list of steps is provided in a box at the bottom left.

Patran® Student Edition

Load/Boundary Conditions

Action: Create
 Object: Planar Rigid Wall
 Type: Nodal

Current Load Case: Default...
 Type: Time Dependent

Existing Sets

New Set Name: pwall

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

Select Application Region

Select: FEM j
 Auto Select...
 Application Region
 Select Nodes
 Node 1:4264 k
 m Add Remove
 Application Region
 n OK

Steps:

- Click **Select Application Region...**
- Select **FEM**
- Click on the **Select Nodes** panel
- Select the nodes belonging to the cylindrical element by clicking and dragging the mouse
- Click **Add**
- Click **OK**
- Click **Apply**

Define a self contact for the cylindrical element:

- Loads/BCs: **Self Contact** icon (*Contact tab*)
- Enter **self** as the New Set Name
- Click **Input Data...**
- Select **Advanced** as the Form Type
- Search Algorithm: **Full**;
Damping: **Yes**;
Weight Factor: **Both**
- Enter **0.3** as the Static and Kinematic Friction Coefficient
- Click **OK**

h. Click **Select Application Region...**
 i. Select **Select Tool**
 j. Element Type: **2D**
 k. Check **FEM**
 l. Click on the **Select Entities** panel
 m. Select the elements by clicking and dragging the mouse
 n. Click **Add**
 o. Click **OK**
 p. Click **Apply**

a. Post two groups: **rwall** and **pipe**:
Menu/Group/Post.../Select rwall and pipe

b. Click on the **Bottom view** icon

c. Click on the **Fit view** icon

Define the contact between the rigid wall and the cylindrical element:

d. Loads/BCs: **Master-Slave Surface** icon (*Contact tab*)

e. Enter **rwall_to_pipe** as the New Set Name

f. Click **Input Data...**

g. Select **Advanced** as the Form Type

h. Search Algorithm: **Full**;
Damping: **Yes**;
Weight Factor: **Slave**

i. Enter **0.3** as the Static and Kinematic Friction Coefficient

j. Click **OK**

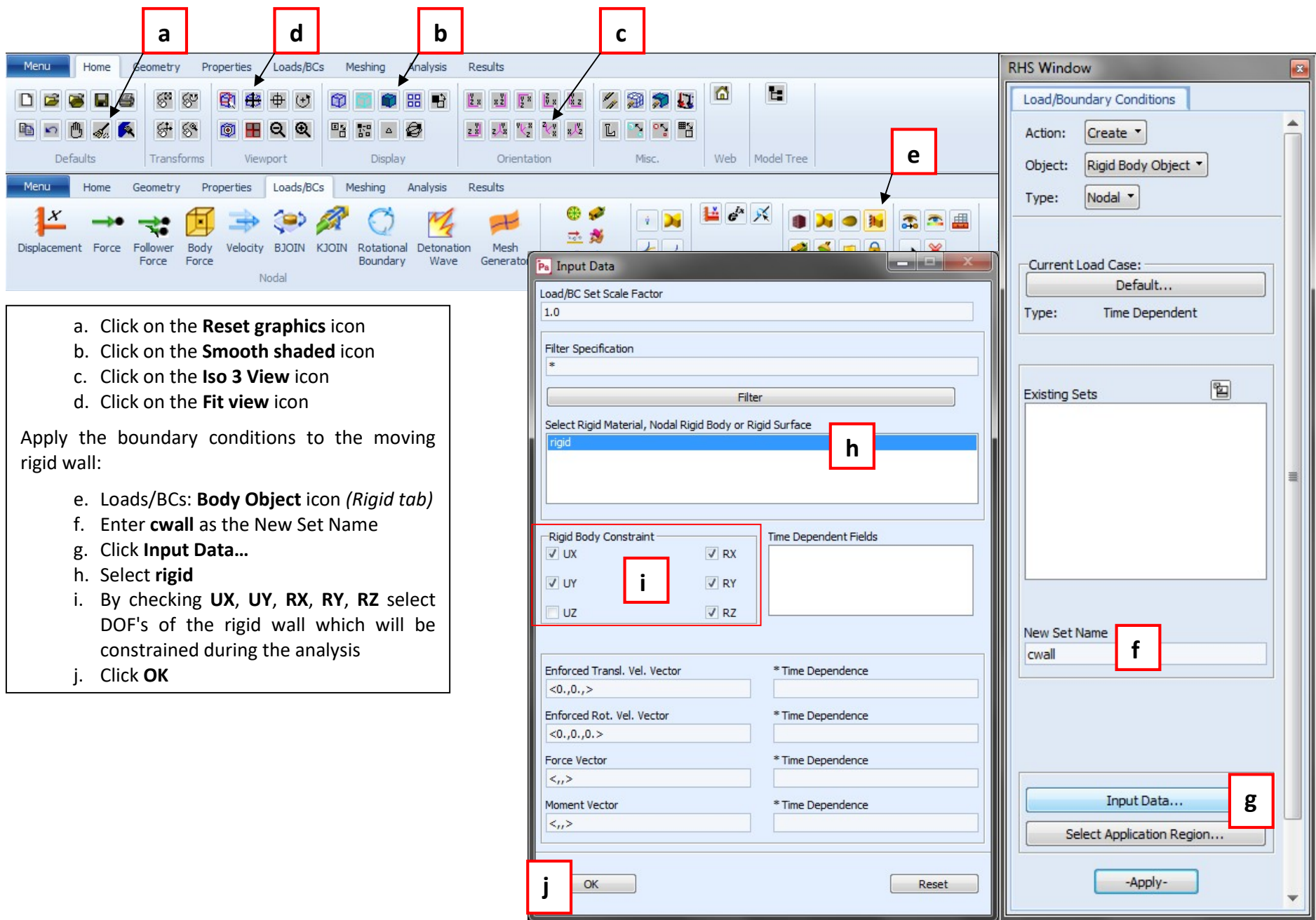
The image shows the Patran software interface with a meshed pipe model. Two dialog boxes are open: 'Explicit Application Tool' and 'Load/Boundary Conditions'. The 'Explicit Application Tool' dialog has the following settings: Form Type: Select Tool, Type: Master, Element Type: 2D, Contact Side: Both, Geometry Filter: FEM, and Application Region: Elm 4161:4185. The 'Load/Boundary Conditions' dialog has Action: Create, Object: Contact, Type: Element Uniform, Option: Master-Slave Surface, and New Set Name: rwall_to_pipe. A list of instructions (k-q) is provided in a box at the bottom left.

Instructions:

- k. Click **Select Application Region...**
- l. Select **Select Tool**
- m. Type: **Master**
Element Type: **2D**;
Contact Side: **Both**
- n. Check **FEM**
- o. Click on the **Select Entities** panel
- p. Select only the elements belonging to the rigid wall – by clicking and dragging the mouse
- q. Click **Add**

The image shows the Patran software interface with a meshed cylindrical element. Two dialog boxes are open: 'Explicit Application Tool' and 'RHS Window'. The 'Explicit Application Tool' dialog is configured for a 'Slave' type contact. The 'RHS Window' dialog is configured for a 'Contact' object. Red boxes highlight specific UI elements: 't' on the mesh, 'r' on the 'Type' dropdown, 's' on the 'Select Entities' input, 'u' on the 'Add' button, 'v' on the 'OK' button, and 'w' on the '-Apply-' button.

- r. Change *Type* to **Slave**
- s. Click on the **Select Entities** panel
- t. Select only the elements belonging to the cylindrical element – by clicking and dragging the mouse
- u. Click **Add**
- v. Click **OK**
- w. Click **Apply**



- a. Click on the **Reset graphics** icon
 - b. Click on the **Smooth shaded** icon
 - c. Click on the **Iso 3 View** icon
 - d. Click on the **Fit view** icon
- Apply the boundary conditions to the moving rigid wall:
- e. Loads/BCs: **Body Object** icon (*Rigid tab*)
 - f. Enter **cwall** as the New Set Name
 - g. Click **Input Data...**
 - h. Select **rigid**
 - i. By checking **UX, UY, RX, RY, RZ** select DOF's of the rigid wall which will be constrained during the analysis
 - j. Click **OK**

Ex_5.db - default_viewport - pipe - Entity

Menu Home Geometry Properties Loads/BCs Meshing Analysis Results

Displacement Force Follower Force Body Force Velocity BJOIN KJOIN Rotational Boundary Detonation Wave Mesh Generator

Nodal Element Uniform Contact Initial Conditions Rigid LBC Actions

n

o OK

Message

Acknowledgement requested from application lbc_dytran_rigid_body

Reference Node/Point is used for display purpose only.

p OK

l

m

o OK

RHS Window

Boundary Conditions Application Region

Geometry Filter
 Geometry FEM

Rigid Reference Node

o OK

RHS Window

Load/Boundary Conditions

Action: Create

Object: Rigid Body Object

Type: Nodal

Current Load Case: Default...

Type: Time Dependent

Existing Sets

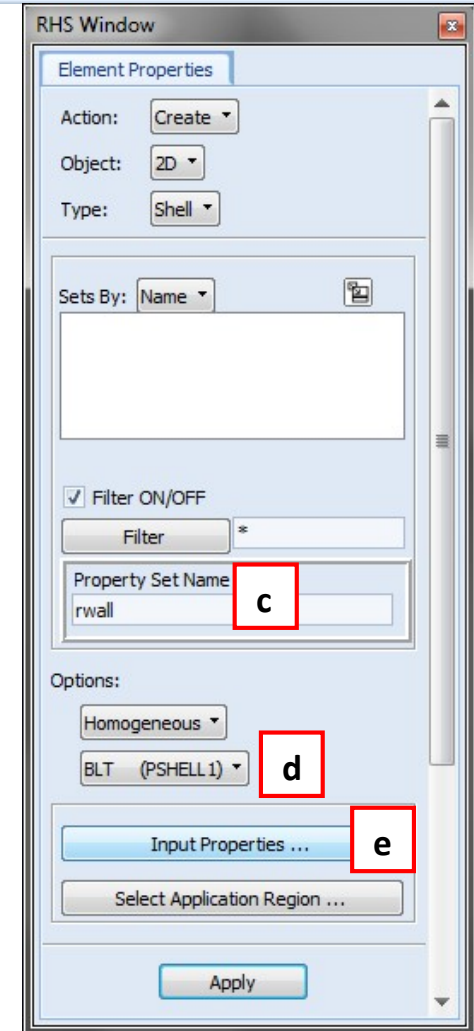
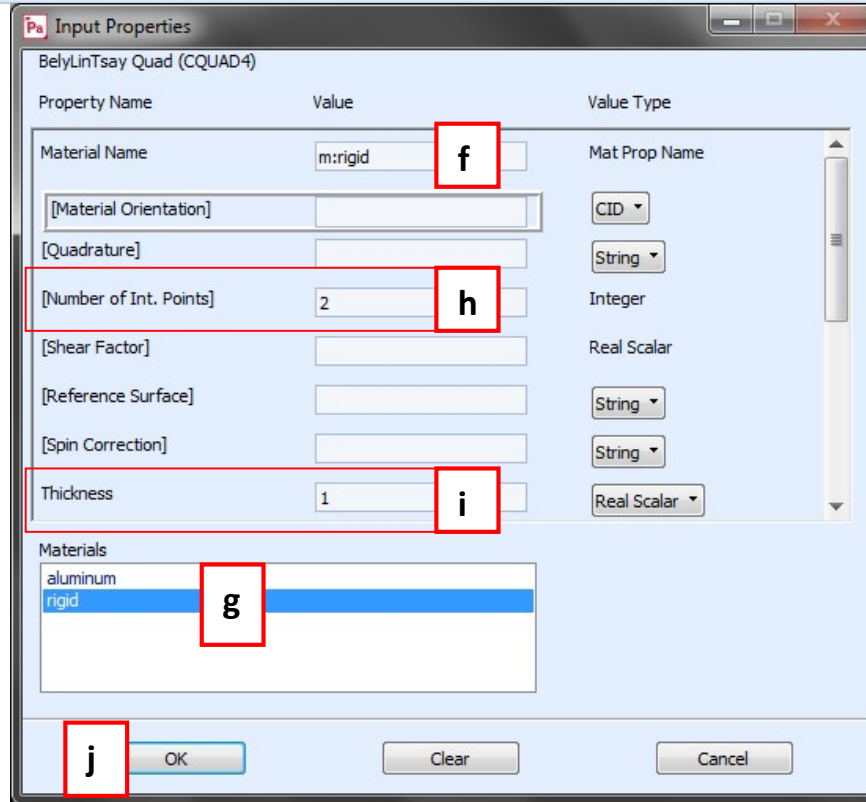
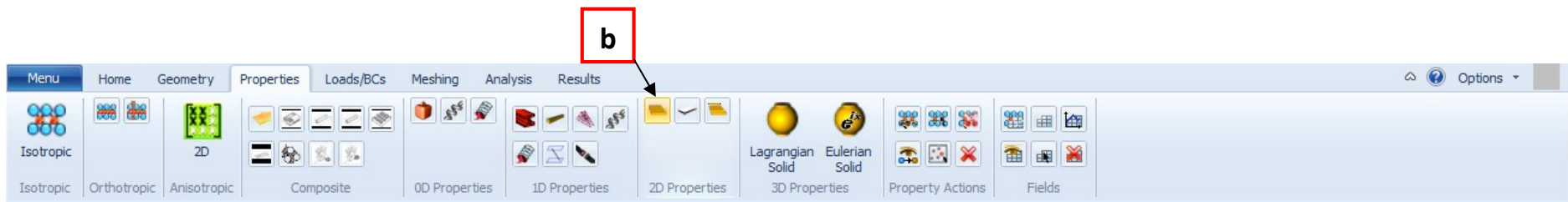
New Set Name: cwall

Input Data...

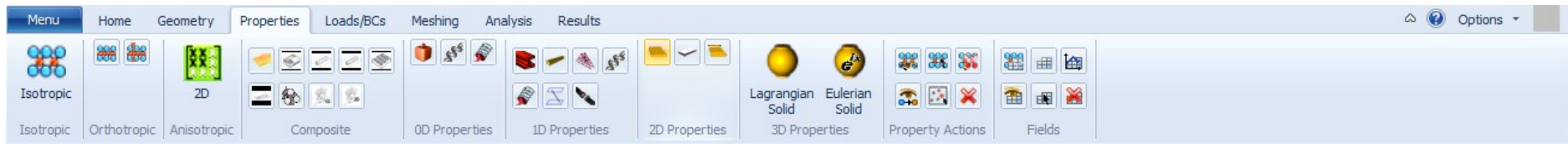
Select Application Region... k

q -Apply-

k. Click **Select Application Region...**
 l. Select **FEM**
 m. Click on the **Rigid Reference Node** panel
 n. Select any node belonging to the rigid wall
 o. Click **OK**
 p. Click **OK**
 q. Click **Apply**



- a. Post only the **rwall** group
Assign the properties
- b. Properties: **Shell** icon (*2D Properties tab*)
- c. Enter **rwall** as the New Set Name
- d. Select **BLT (PSHELL1)**
- e. Click **Input Properties...**
- f. Click on the **Material Name** panel
- g. Select **rigid**
- h. Enter **2** as the Number of Points
- i. Enter **1** as the Thickness
- j. Click **OK**



- r. Post only the **pipe** group
- s. Change *the Property Set Name* to **pipe**
- t. Click **Input Data...**
- u. Click on the **Material Name** panel
- v. Select **aluminum**
- w. Enter **4** as the Number of Points
- x. Enter **2** as the Thickness
- y. Click **OK**

Input Properties
BelyLinTsay Quad (CQUAD4)

Property Name	Value	Value Type
Material Name	m:aluminum u	Mat Prop Name
[Material Orientation]		CID
[Quadrature]		String
[Number of Int. Points]	4 w	Integer
[Shear Factor]		Real Scalar
[Reference Surface]		String
[Spin Correction]		String
Thickness	2 x	Real Scalar

Materials **v**

- aluminum
- rigid

y OK Clear Cancel

RHS Window

Element Properties

Action: Create

Object: 2D

Type: Shell

Sets By: Name

rwall

Filter ON/OFF

Filter *

Property Set Name **s**

pipe

Options:

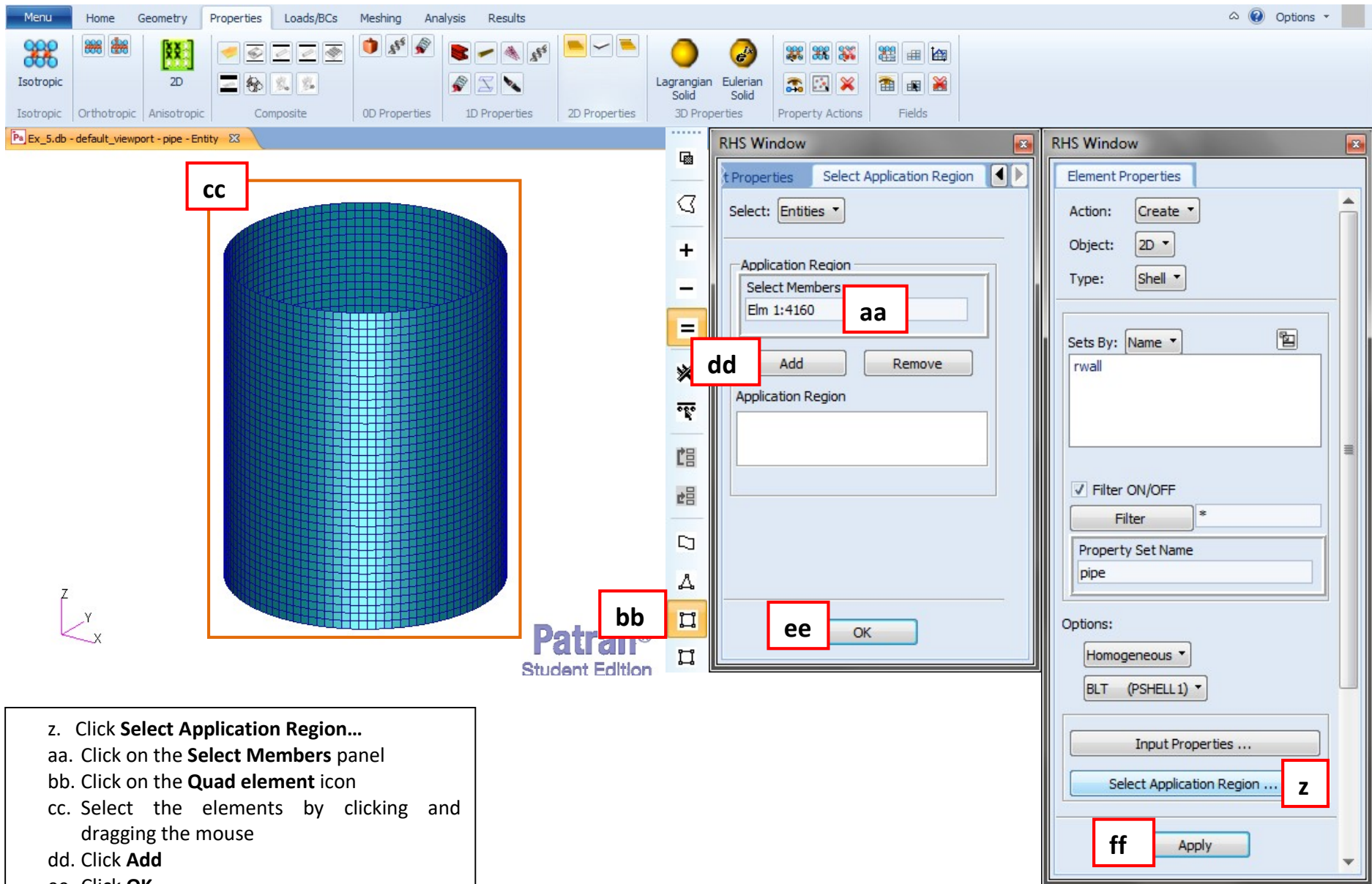
Homogeneous

BLT (PSHELL 1)

t Input Properties ...

Select Application Region ...

Apply



- z. Click **Select Application Region...**
- aa. Click on the **Select Members** panel
- bb. Click on the **Quad element** icon
- cc. Select the elements by clicking and dragging the mouse
- dd. Click **Add**
- ee. Click **OK**
- ff. Click **Apply**

Set analysis parameters:

- Click on the **Analysis** icon
- Analysis: **Analyze/Input Deck/Translate**
- Enter **ex_5** as the Job Name
- Click **Execution Controls..**
- Click **Execution Control Parameters**
- Enter **5** as the End Time and **1e-7** as the Time-Step Size at Start
- Enter **0.9** as the Time-Step Scale factor
- Click **OK**

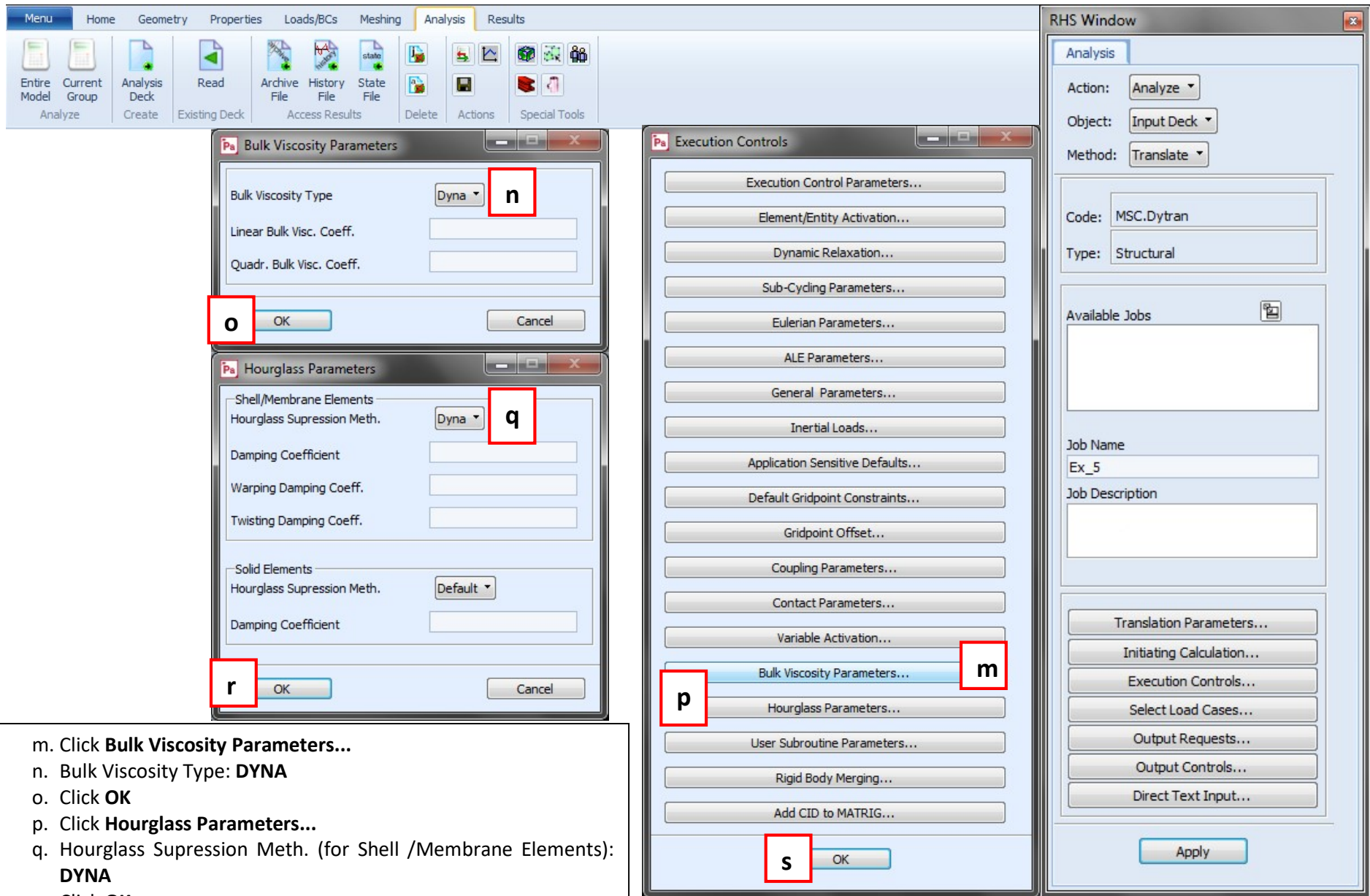
The screenshot shows the ANSYS software interface with the following components and annotations:

- Menu Bar:** Menu, Home, Geometry, Properties, Loads/BCs, Meshing, **Analysis** (labeled 'a'), Results.
- Toolbar:** Entire Model, Current Group, Analysis Deck, Read, Archive File, History File, State File, Delete, Actions, Special Tools.
- Execution Control Parameters Dialog:**
 - Limits: CPU Time, Integer Memory Size, Float Memory Size.
 - Time-Step Control: End Step (9999999), End Time (5, labeled 'f'), Time-Step Size at Start (1e-7, labeled 'f'), Minimum Time Step, Maximum Time Step.
 - Time-Step Scale Factor (0.9, labeled 'g').
 - Lagr. Time Step Sc. Fact.
 - License Control: Job Queuing (Minutes).
 - Mass Scaling: Activate Mass Scaling (No), Min. Allowable Time Step, Max. Perc. of Mass Incr., Steps for Freq. Checks.
 - Buttons: OK (labeled 'h'), Cancel.
- Execution Controls Dialog:**
 - Buttons: Execution Control Parameters... (labeled 'e'), Element/Entity Activation..., Dynamic Relaxation..., Sub-Cycling Parameters..., Eulerian Parameters..., ALE Parameters..., General Parameters..., Inertial Loads..., Application Sensitive Defaults..., Default Gridpoint Constraints..., Gridpoint Offset..., Coupling Parameters..., Contact Parameters..., Variable Activation..., Bulk Viscosity Parameters..., Hourglass Parameters..., User Subroutine Parameters..., Rigid Body Merging..., Add CID to MATRIG..., OK.
- RHS Window:**
 - Analysis: Action (Analyze, labeled 'b'), Object (Input Deck, labeled 'b'), Method (Translate).
 - Code: MSC.Dytran, Type: Structural.
 - Available Jobs: (Empty list).
 - Job Name (Ex_5, labeled 'c'), Job Description.
 - Buttons: Translation Parameters..., Initiating Calculation..., Execution Controls... (labeled 'd'), Select Load Cases..., Output Requests..., Output Controls..., Direct Text Input..., Apply.

The screenshot displays a software interface with a menu bar (Menu, Home, Geometry, Properties, Loads/BCs, Meshing, Analysis, Results) and a toolbar. Three windows are open:

- Contact Parameters:** A dialog box with the following settings:
 - Contact Parameter Defaults: MSC/DYNA (labeled 'j')
 - Max. Cubes used in Sorting: [empty]
 - Contact Thickness: 1.0
 - Contact Gap: [empty]
 - Default Contact Damping: On (labeled 'k')
 - Activate Contact Porosity: Default
 - Grid Contact Info: [empty]
 - Buttons: OK (labeled 'l'), Cancel
- Execution Controls:** A panel with a list of buttons. The 'Contact Parameters...' button is highlighted in blue and labeled 'i'. Other buttons include Execution Control Parameters..., Element/Entity Activation..., Dynamic Relaxation..., Sub-Cycling Parameters..., Eulerian Parameters..., ALE Parameters..., General Parameters..., Inertial Loads..., Application Sensitive Defaults..., Default Gridpoint Constraints..., Gridpoint Offset..., Coupling Parameters..., Variable Activation..., Bulk Viscosity Parameters..., Hourglass Parameters..., User Subroutine Parameters..., Rigid Body Merging..., and Add CID to MATRIG... The OK button is at the bottom.
- RHS Window:** A window with the following settings:
 - Analysis: Analyze
 - Object: Input Deck
 - Method: Translate
 - Code: MSC.Dytran
 - Type: Structural
 - Available Jobs: [empty list]
 - Job Name: Ex_5
 - Job Description: [empty text area]
 - Buttons: Translation Parameters..., Initiating Calculation..., Execution Controls..., Select Load Cases..., Output Requests..., Output Controls..., Direct Text Input..., and Apply

- i. Click **Contact Parameters...**
- j. Contact Parameter Defaults: **MSC/DYNA**
- k. Default Contact Damping: **On**
- l. Click **OK**



- m. Click **Bulk Viscosity Parameters...**
- n. Bulk Viscosity Type: **DYNA**
- o. Click **OK**
- p. Click **Hourglass Parameters...**
- q. Hourglass Suppression Meth. (for Shell /Membrane Elements): **DYNA**
- r. Click **OK**
- s. Click **OK**

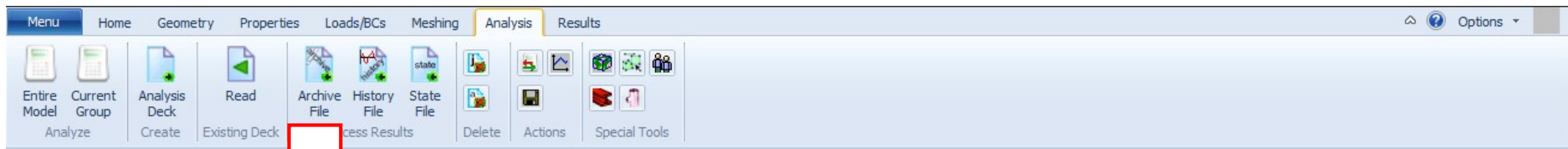
Define the output:

- Click **Output Requests...**
- Enter **element_output** as the Result Name
- File Type: **Archive**
- Result Type: **Element Output**
- Select **Times for Output** and **Sampling Rate**
- Enter **0.2** as the archive time step
- Click **Add**
- Select **ALLELEMENTS**
- Change *Entity Type* to **Lagrangian**
- Select Results Types:
 - Stress: **TXX, TYY, TZZ, TXY, TYZ, TZX**
 - EFFSTS** (effective stress)
 - EFFPLS** (effective plastic strain)
 - Centroidal Strain: **EPSXX, EPSYY, EPSZZ**
- Click **Apply**

The image shows a screenshot of the ANSYS software interface with three dialog boxes open. Red boxes highlight specific elements in each dialog, corresponding to steps i through w in the instructions.

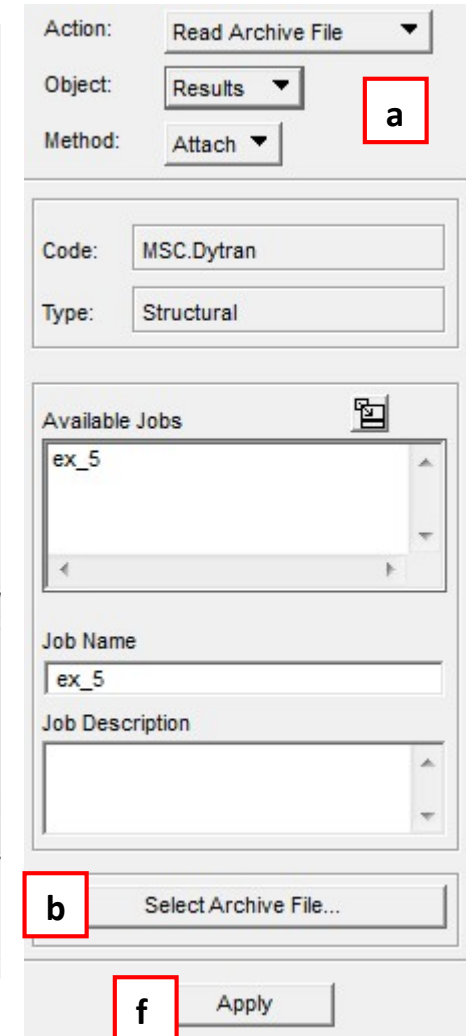
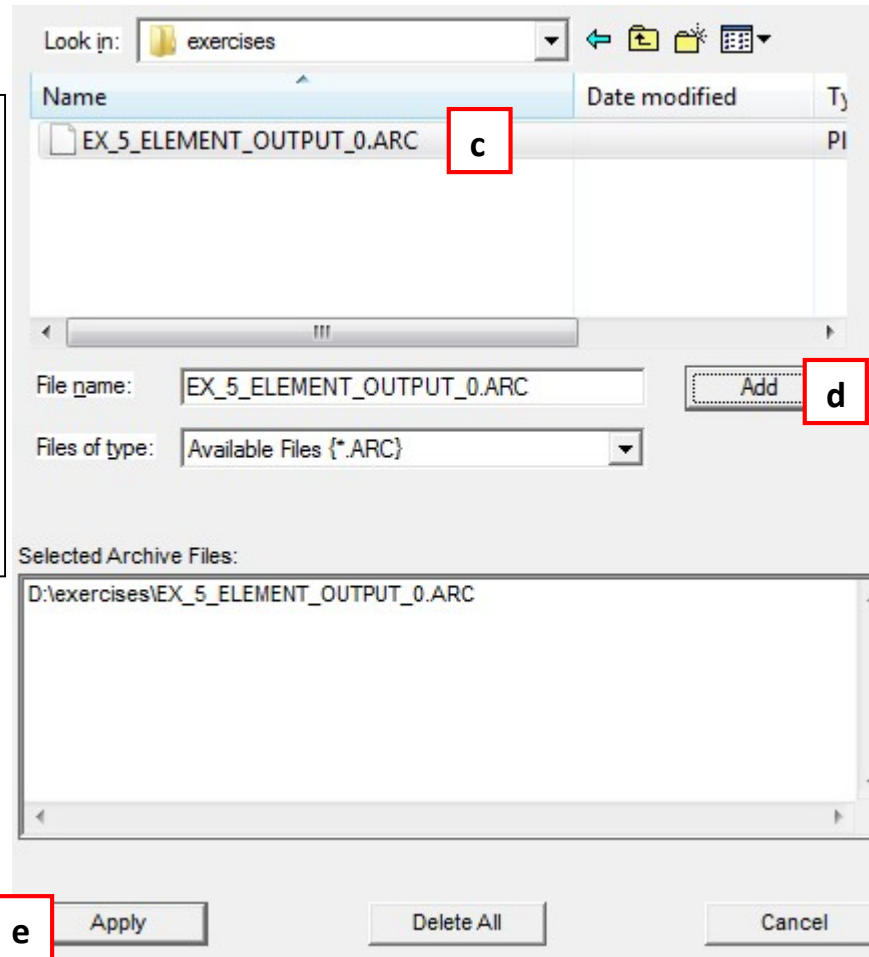
- Step i:** In the 'RHS Window' (left), 'contact_output' is entered in the 'Result Name' field.
- Step m:** In the 'RHS Window' (left), 'Time History' is selected in the 'File Type' dropdown.
- Step n:** In the 'RHS Window' (left), 'Contact Surface Output' is selected in the 'Result Type' dropdown.
- Step o:** In the 'RHS Window' (left), '0.01' is entered in the 'Sampling Rate' field.
- Step p:** In the 'RHS Window' (left), '0.01' is entered in the '0 THRU END BY (Time)' field.
- Step q:** In the 'Output Requests' dialog (middle), the 'Add' button is highlighted.
- Step r:** In the 'RHS Window' (left), 'rwall_to_pipe' is selected in the 'Select Contacts for Output' list.
- Step s:** In the 'RHS Window' (left), 'XFORCE' is selected in the 'Results Types' list.
- Step t:** In the 'RHS Window' (left), the 'Apply' button is highlighted.
- Step u:** In the 'Output Requests' dialog (middle), the 'OK' button is highlighted.
- Step v:** In the 'Output Requests' dialog (middle), the 'OK' button is highlighted.
- Step w:** In the 'RHS Window' (right), the 'Apply' button is highlighted.

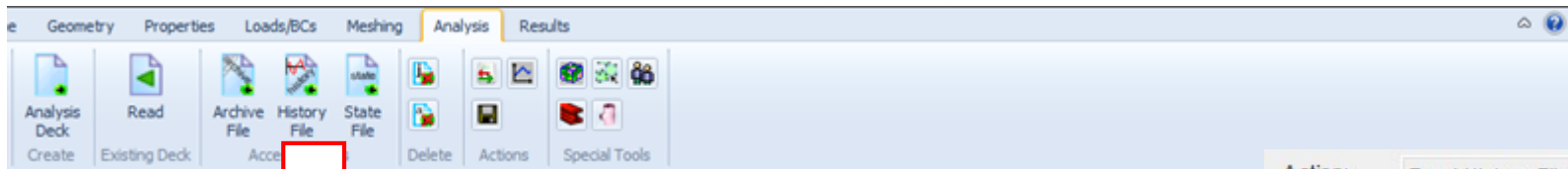
- i. Enter **contact_output** as the Result Name
- m. File Type: **Time History**
- n. Result Type: **Contact Surface Output**
- o. Select **Times for Output** and **Sampling Rate**
- p. Enter **0.01** as the archive time step
- q. Click **Add**
- r. Select **rwall_to_pipe**
- s. Select Results Types: **XFORCE**, **YFORCE**, **ZFORCE**, **FMAGN**
- t. Click **Apply**
- u. **Additional task:** Using the **Time History** and **Grid Point Output** options, try to define an output for the rigid wall
- v. Click **OK**
- w. Click **Apply**
- x. Run **Dytran** analysis using **Ex_5.dat** file



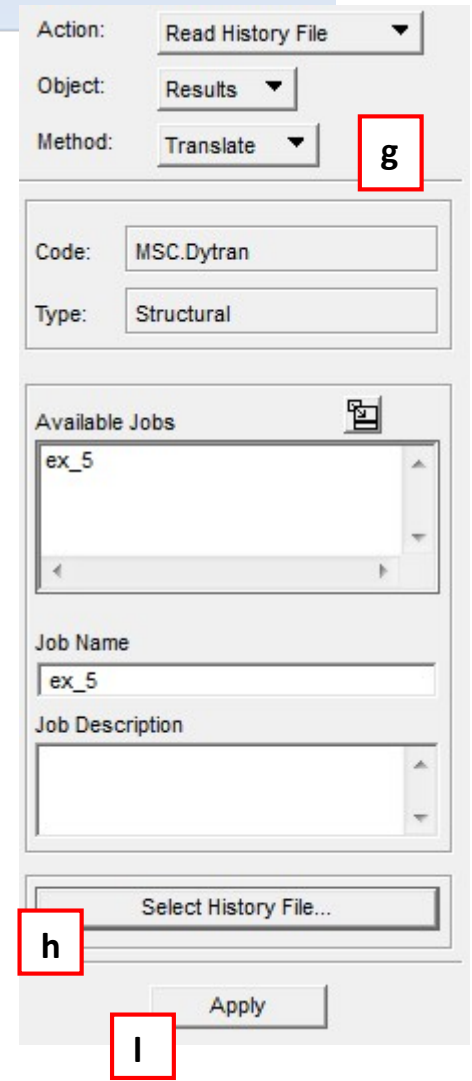
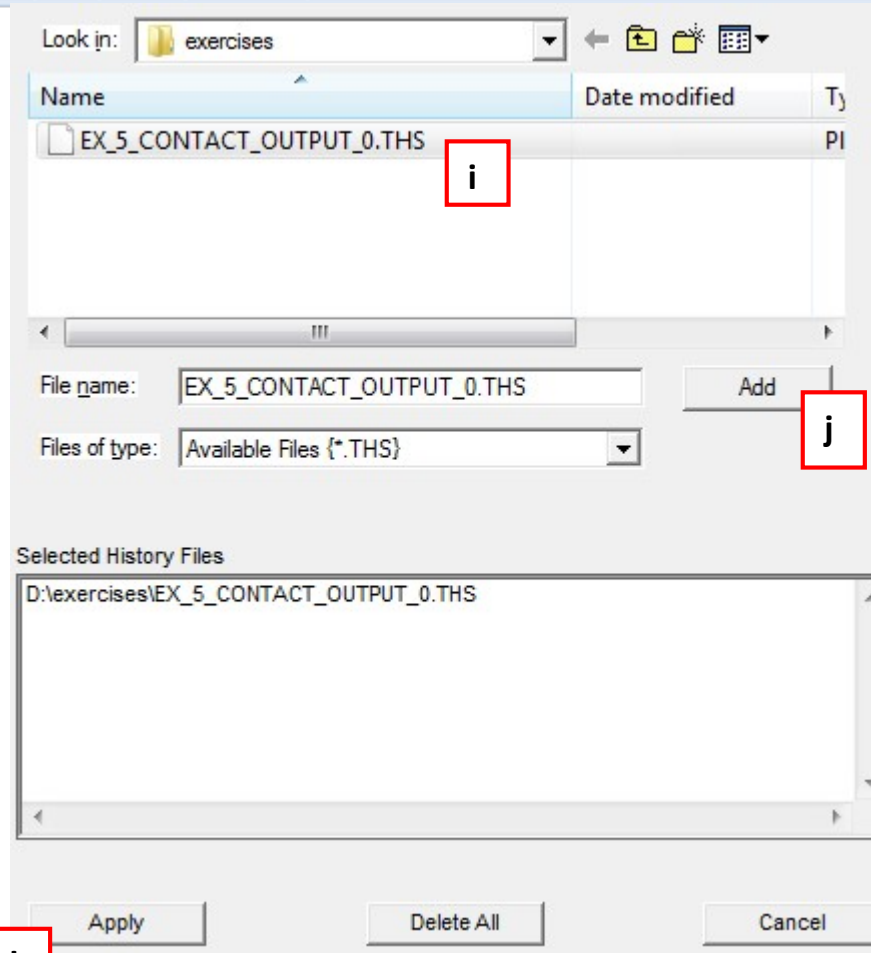
Attach the results files, when the analysis job is completed:

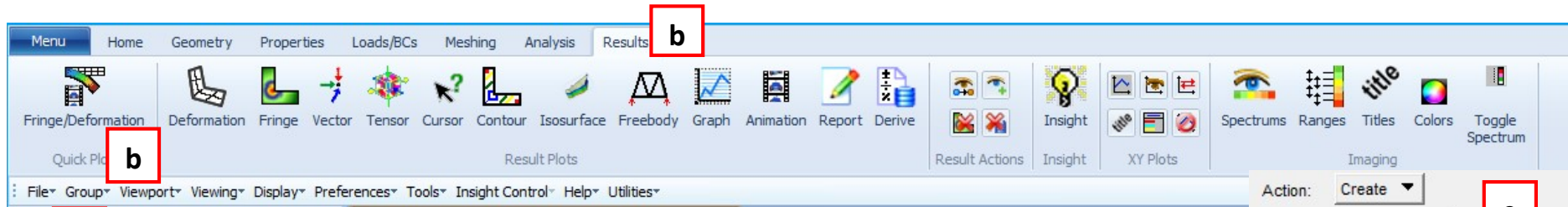
- a. Analysis:
Read Archive File/Results/Attach
- b. Click **Select Archive File...**
- c. Select the
EX_5_ELEMENT_OUTPUT_0.ARC file
- d. Click **Add**
- e. Click **Apply**
- f. Click **Apply**



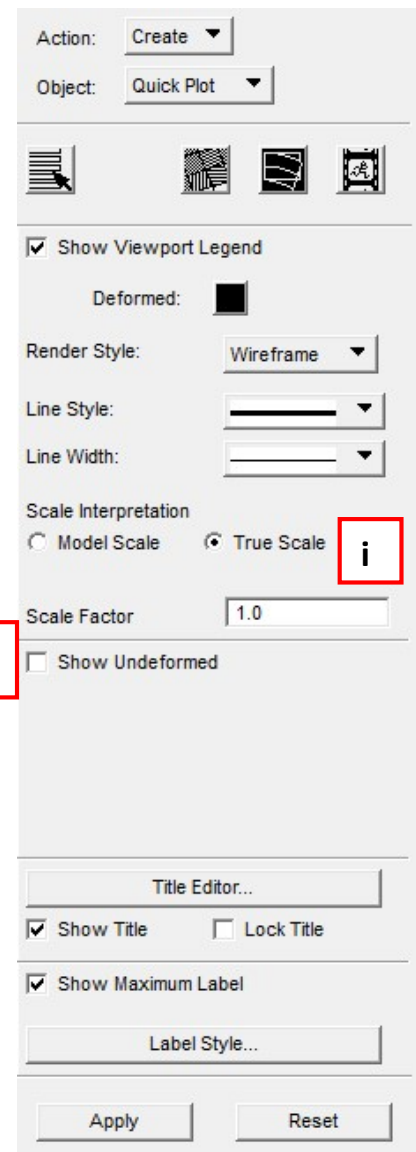
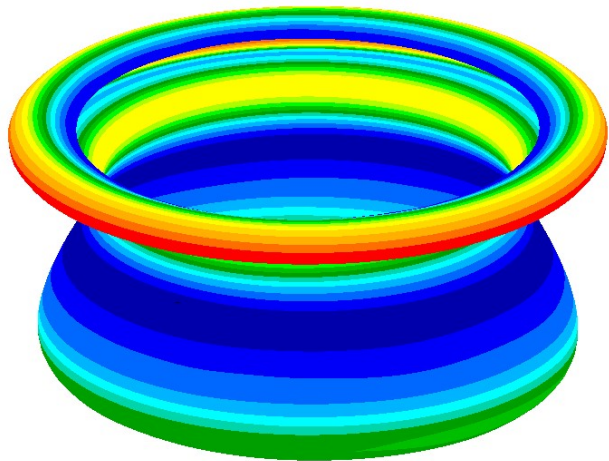


- g. Analysis: **Read History File /Results/Attach**
- h. Click **Select History File...**
- i. Select the **EX_5_CONTACT_OUTPUT_0.THS** file
- j. Click **Add**
- k. Click **Apply**
- l. Click **Apply**

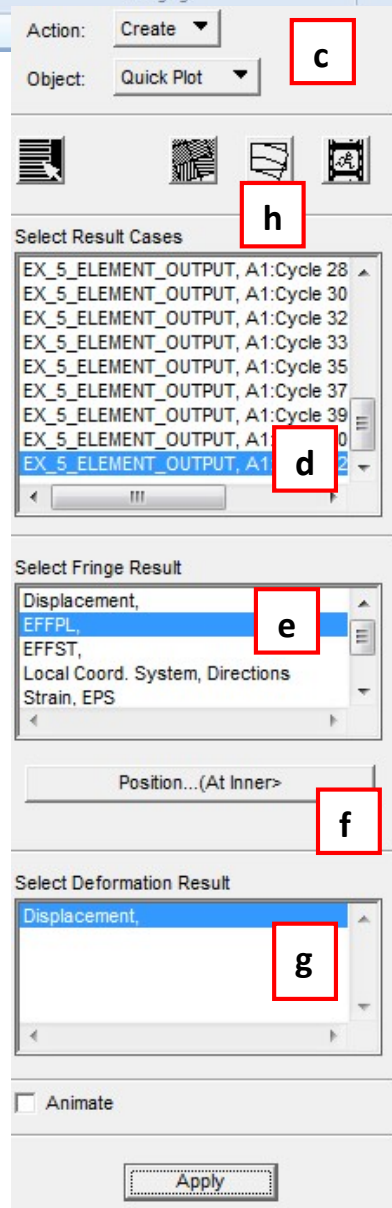




a



j



- a. Post only the **pipe** group (from menu: **Group/Post....**)
 - Post-process the results:
 - b. Click on the **Results** tab in the Ribbon (*Fringe/Deformation* icon)
 - c. Results: **Create/Quick Plot**
 - d. Select *the last step*
 - e. Select Fringe Result: **EFFPL** (EFFective PLastic strain)
 - f. Position: select **At Inner, At Middle, At Outer**
 - g. Select Deformation Result: **Displacement**
 - h. Click on the **Deform Attributes** icon
 - i. Select **True Scale** with the Scale Factor equals to **1.0**
 - j. Uncheck **Show Undeformed**
 - k. Click **Apply**
- Remark: To capture the plot use **File / Images...**

b

b

c

h

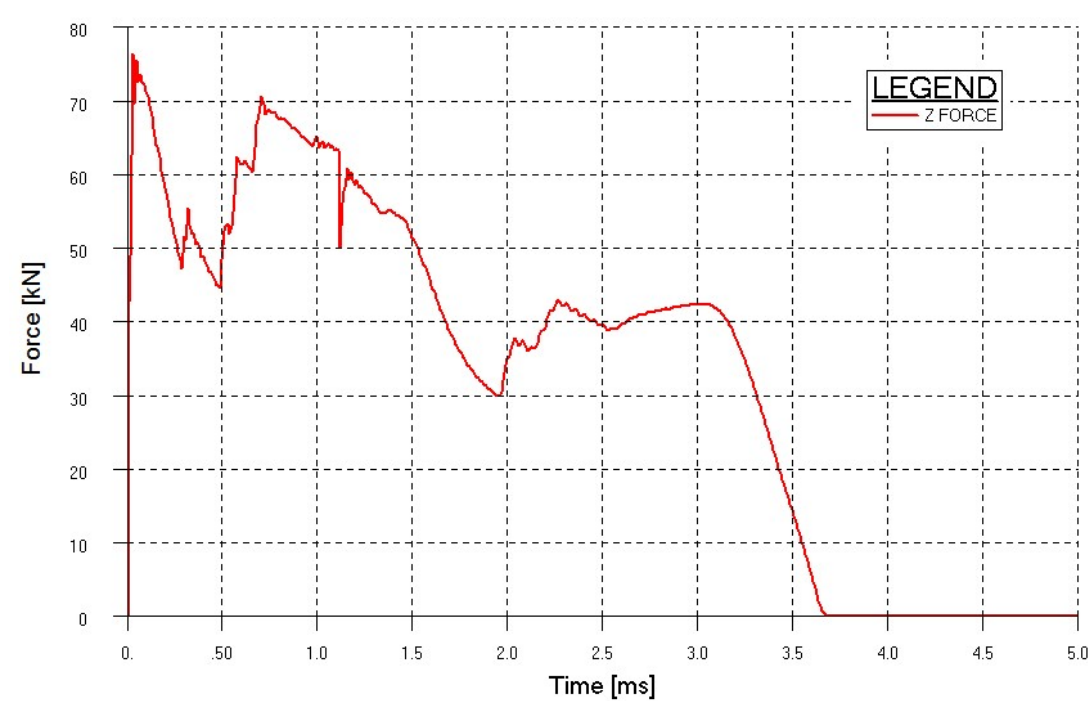
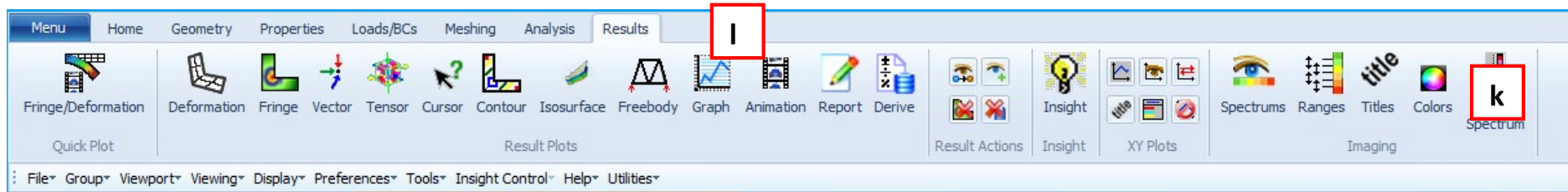
d

i

e

f

g



Action: **p**

Object: **p**

Select Current XYWindow

q

Post/Unpost Curves

FMAGN_CONTACT_4_CONTACT_O
 FMAGN_COSLAVE_4_CONTACT_O
 FMAGN_COTOTAL_4_CONTACT_O
 XFORCE_CONTACT_4_CONTACT_O
 XFORCE_COSLAVE_4_CONTACT_O
 XFORCE_COTOTAL_4_CONTACT_O
 YFORCE_CONTACT_4_CONTACT_O
 YFORCE_COSLAVE_4_CONTACT_C
 YFORCE_COTOTAL_4_CONTACT_O
ZFORCE_CONTACT_4_CONTACT_O
 ZFORCE_COSLAVE_4_CONTACT_O
 ZFORCE_COTOTAL_4_CONTACT_O

s

Action: **m**

Object: **m**

Existing XYWindows

Enter XYWindow Name

n

o

- l. Click on the **Graph** icon
- m. XY Plot: **Create/XYWindow**
- n. Enter **graphs** as the XYWindow Name
- o. Click **Apply**
- p. Change **Action** to **Post** (Object: **Curve**)
- q. Select **graphs**
- r. Select **ZFORCE_CONTACT**
- s. Click **Apply**