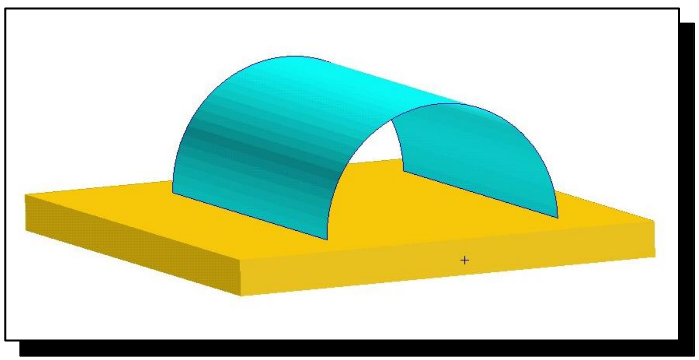
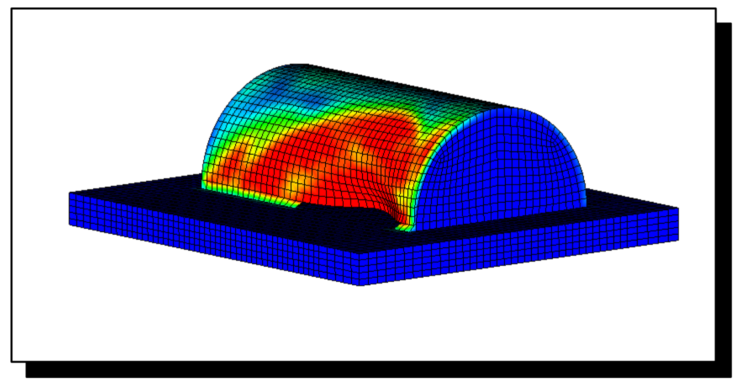
Workshop 12

Bunker Blast





- **Workshop Objectives**
  - Define 2 euler regions and connect 2 regions with PORFLCPL.
  - Activate interactive failure for shell elements.
  - Use 2 order euler solver.

- **Software Version**
  - Patran 2019
  - Dytran 2019

- **Files Required**
  - `geo.dat`
- **Unit**
  - inch/(lbf-s^2/inch)/sec/R

Problem Description

A blastwave hits a bunker shell.

The Dytran model is provided by geo.dat. The FEM is given in this file by dummy shell elements. When needed, dummy shells will be changed into real shells.

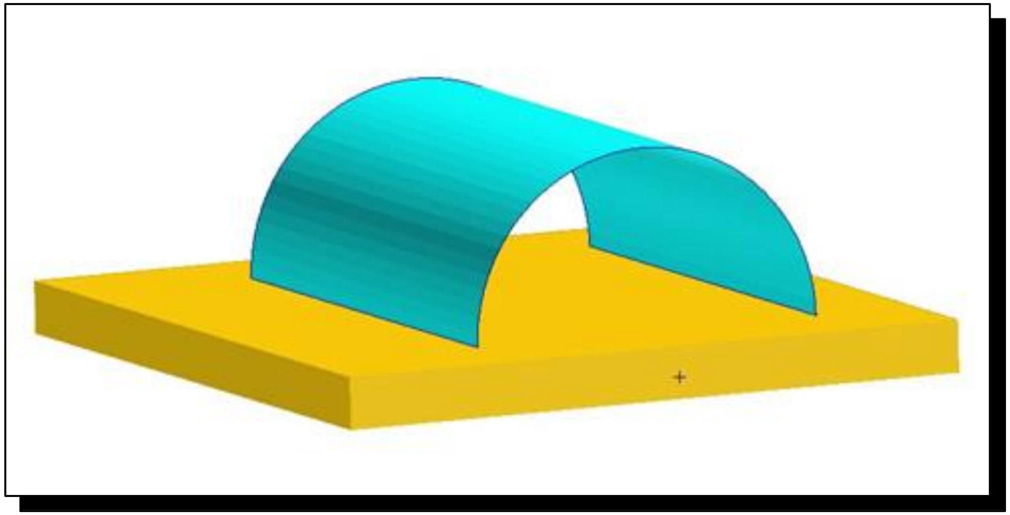
Eulerian Material = ideal gas

$\rho = 1.2e\text{-}7\ kg/m^3$

$\gamma = 1.4$

Lagrangian materials:

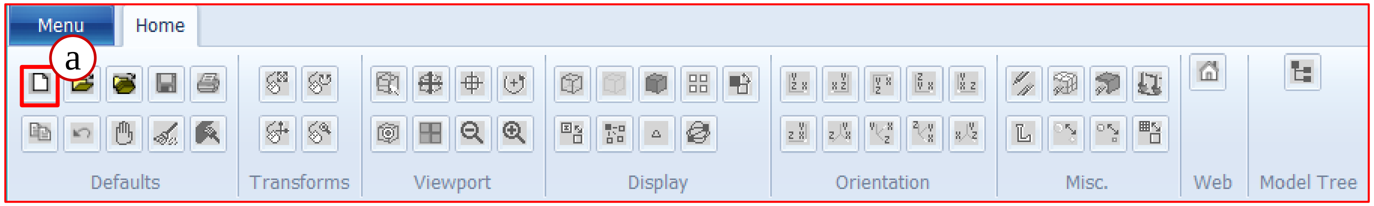
	Type shells	Material	Property ID as occurring in geo.dat
Bunker Shell	Shells of thick ness 0.15	DMATEP	1
Open Sides Bunker	Dummy shell elements	NA	2
Ground	Shells of thickness 0.1.	MATRIG	3



Problem Description (Cont.)

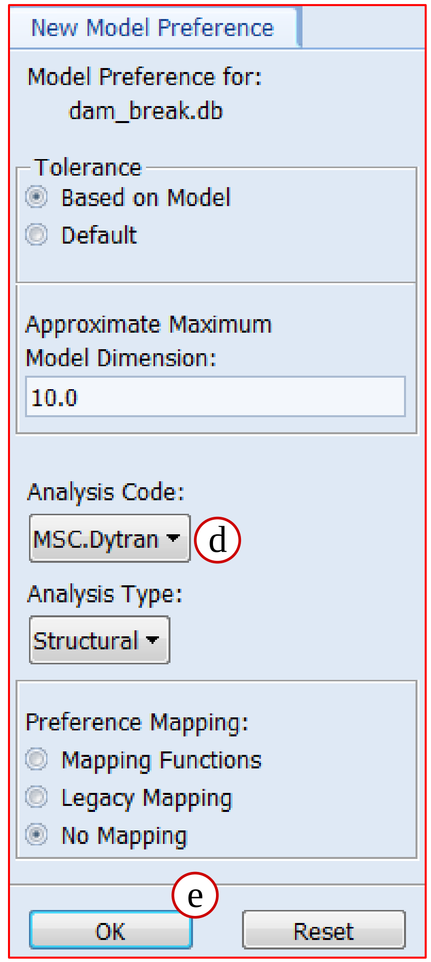
- Fluid-Structure Interaction
  - The gas inside the bunker is modeled by a separate Euler domain. This gas is contained by the first coupling surface. This surface consists of:
    - Shell
    - Two open sides
    - Ground within the bunker
  - The gas outside the bunker is modeled by an another Euler mesh. This gas is between the second coupling surface and the boundaries of the Euler mesh. The second coupling surface consists of:
    - Shell
    - Two open sides
    - Ground outside the bunker
  - The two open sides are each modeled by a fully porous subsurface. The porosity model porflcpl is used.
  - Flow of gas through failed shell elements is taken into account by activating interactive failure.
  - For simulations with coupling surfaces with failure, the Roe solver or MMHYDRO or MMSTREN has to be used. The second-order Roe solver is used to minimize diffusion of the blast wave.

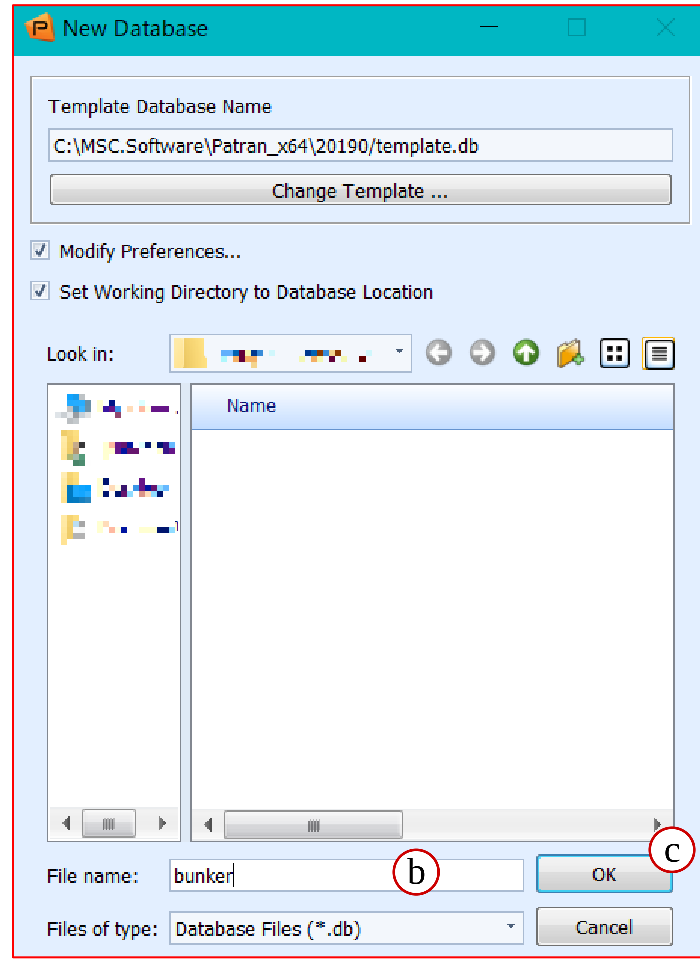
Step 1. Create New Database



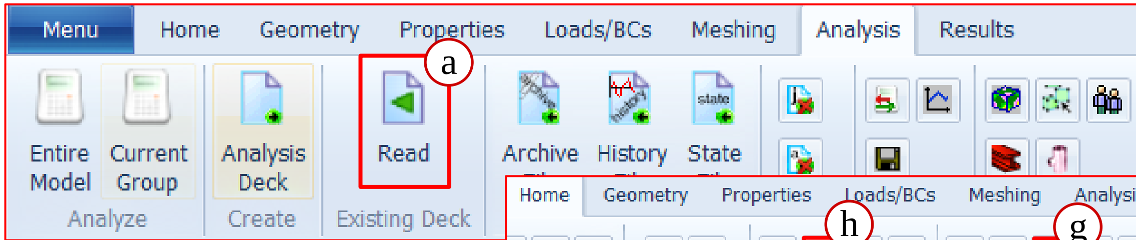
Create a new database named **bunker**.

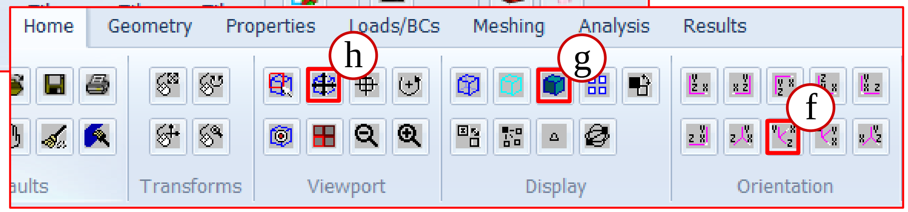
a. Under the *Home* tab, click **New** in the *Default* group.
b. Enter the *File name* **bunker**.
c. Click **OK**.
d. Select **MSC.Dytran** for *Analysis Code*.
e. Click **OK**.





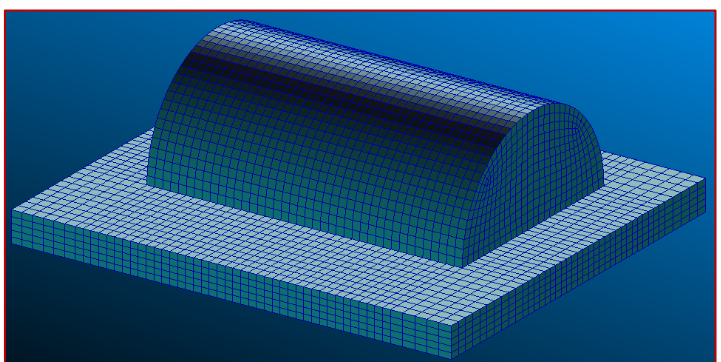
Step 2. Read the Input File

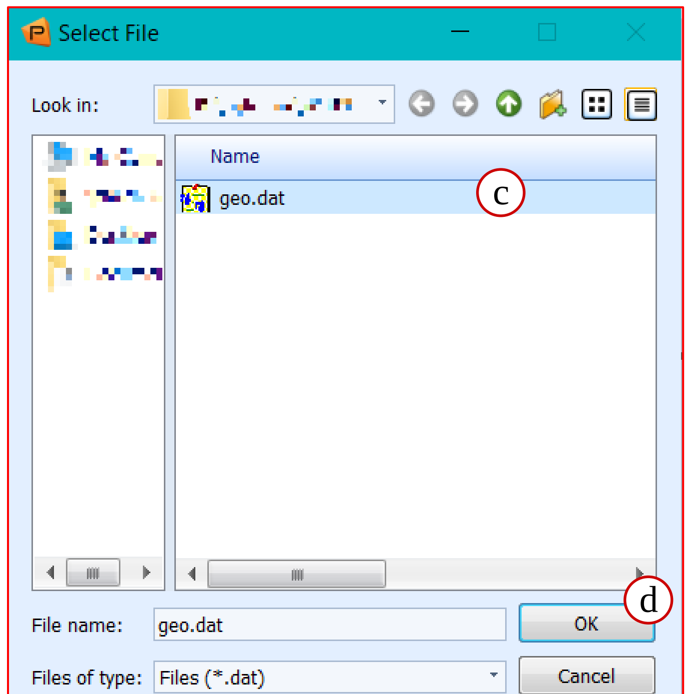


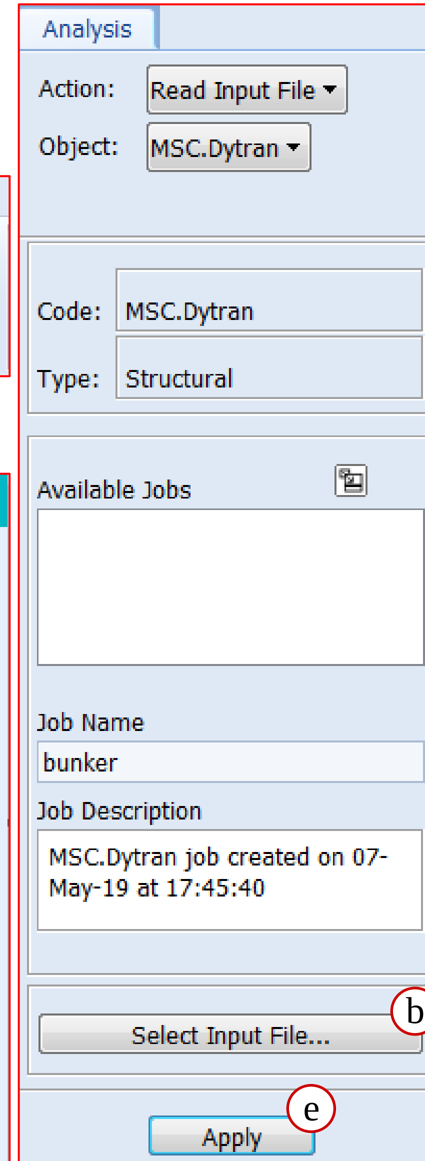


Read the geo.dat input file.

a. Under the *Analysis* tab, click **Read** in the *Existing Deck* group.
b. Click **Select Input File**.
c. Select `geo.dat`.
d. Click **OK**.
e. Click **Apply**.
f. Under the **Home** tab, click **Iso 2 View**.
g. Click **Smooth shaded**.
h. Click **Fit view**.





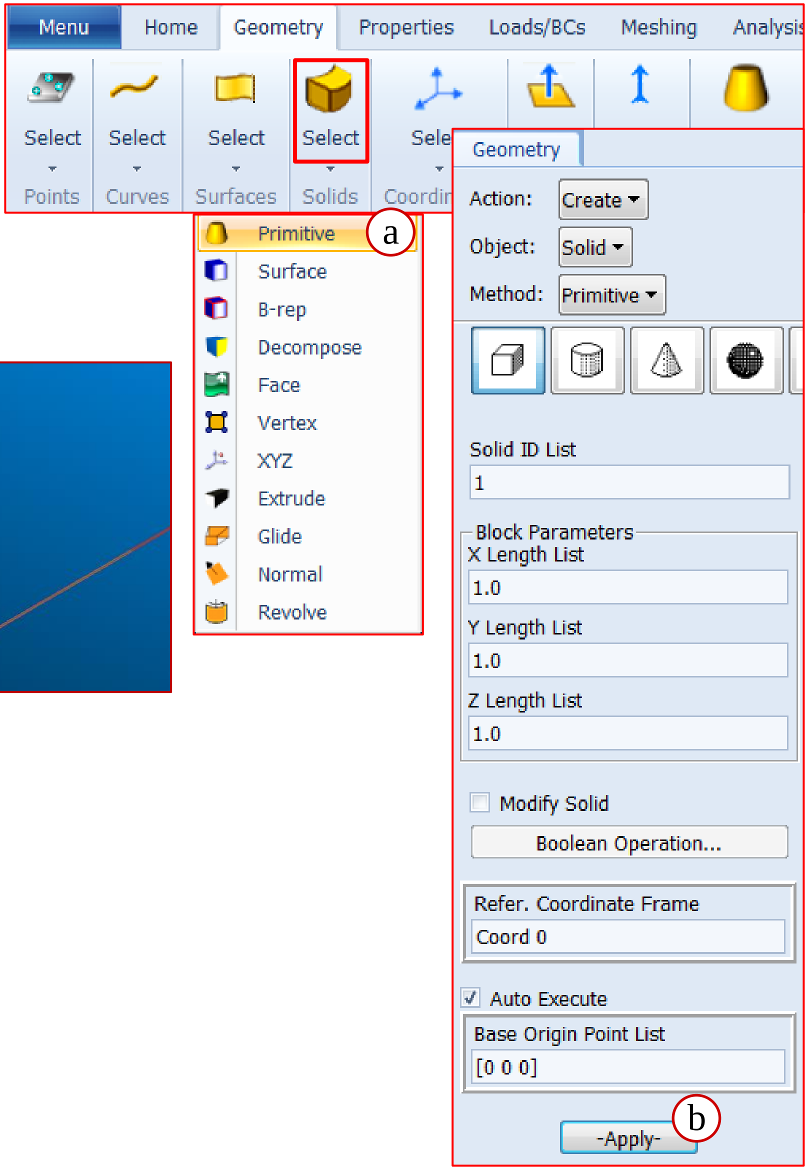


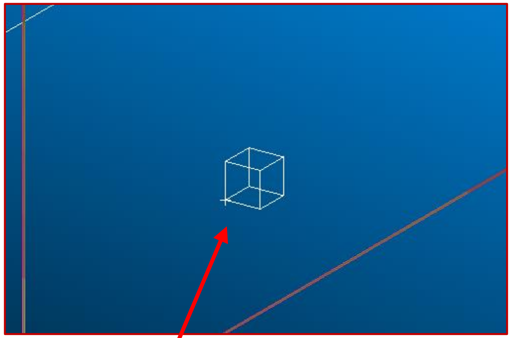
Step 3. Create Solids for 3D Property

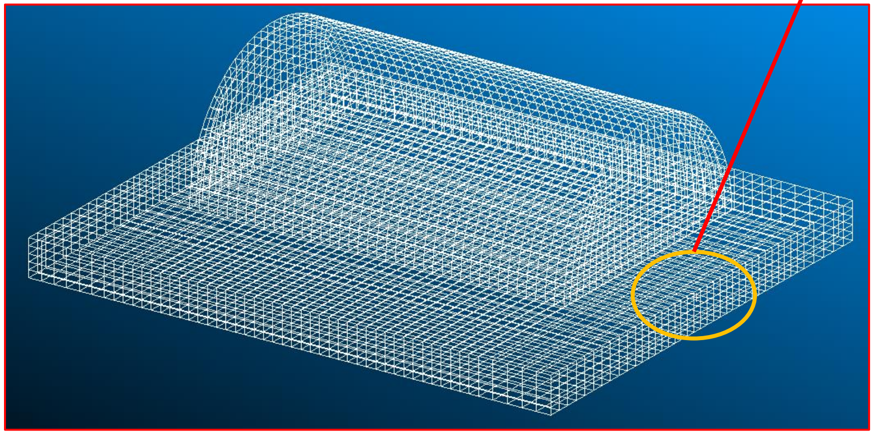
Create dummy solids that are used in creating 3D property sets. Create one solid for each couple surface. Any solid will suffice.

a. Under the *Geometry* tab, click **Select > Primitive** in the *Solids* group.

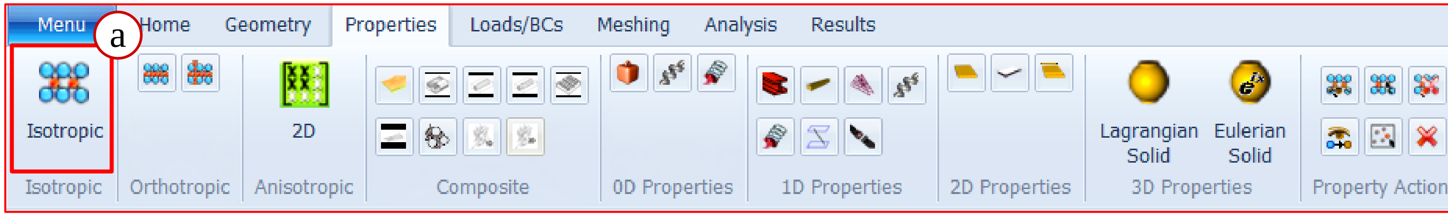
b. Click twice on **Apply**. This will create two solids. Solid 1 will be used by coupling surface1 and Solid 2 by coupling surface2.





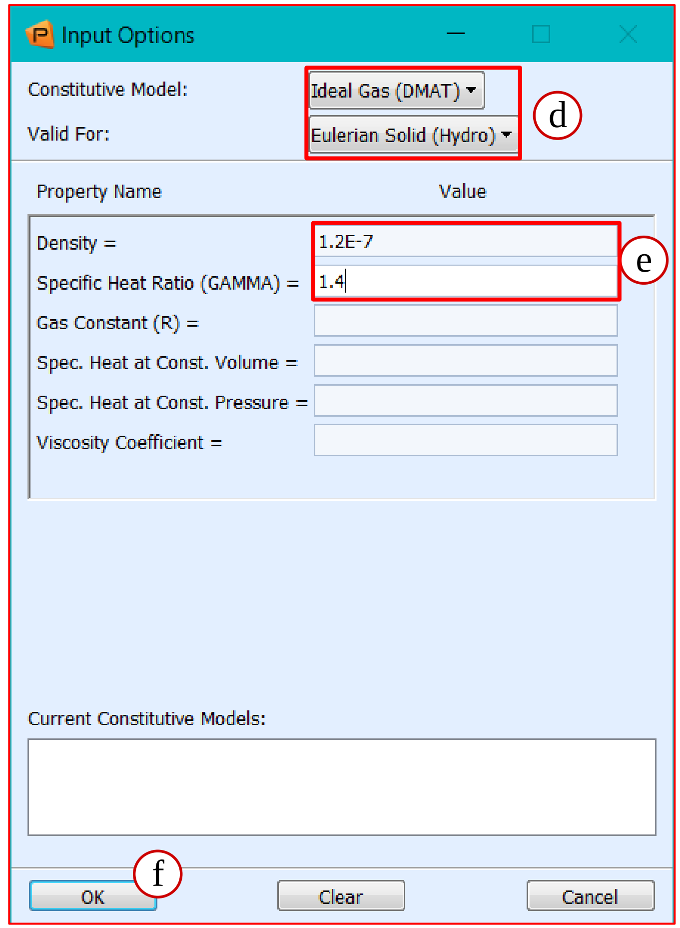


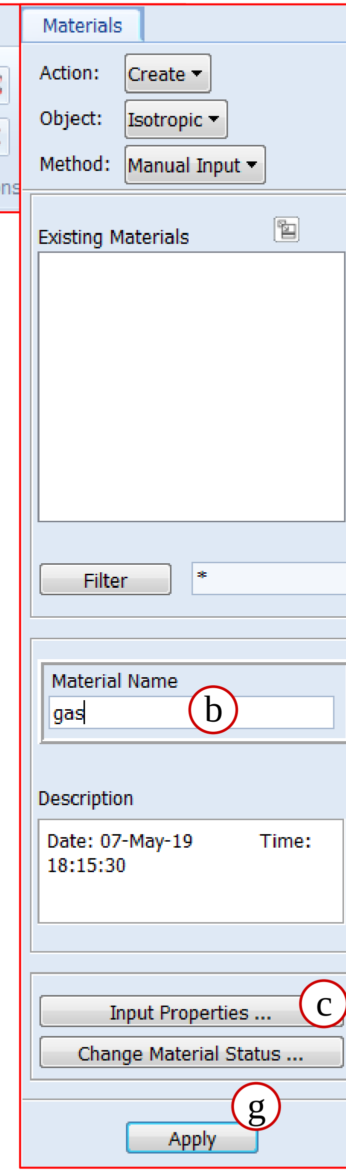
Step 4. Create Material Properties



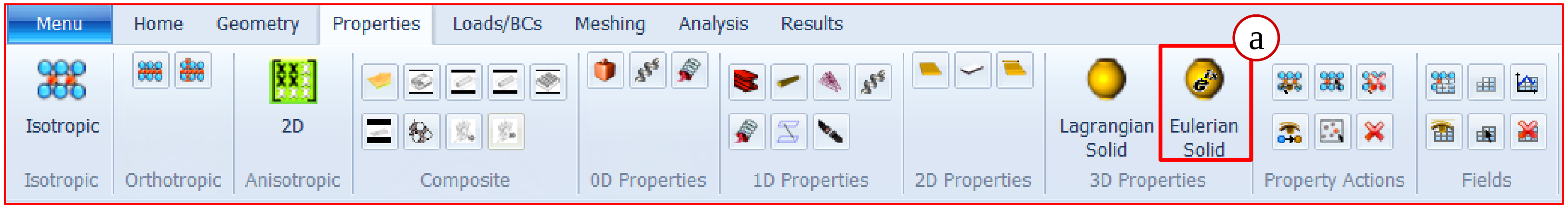
Create the material properties for gas.

a. Under the *Properties* tab, click **Isotropic**.
b. Enter **gas** for *Material name*.
c. Click **Input Properties**.
d. Set *Constitutive Model* **to Ideal Gas (DMAT)** and *Valid For* to **Eulerian Solid (Hydro)**.
e. Enter **1.2e-7** for *density*, **1.4** for *gamma*.
f. Click **OK**.
g. Click **Apply**.





Step 5. Create 3D Properties for the Solids

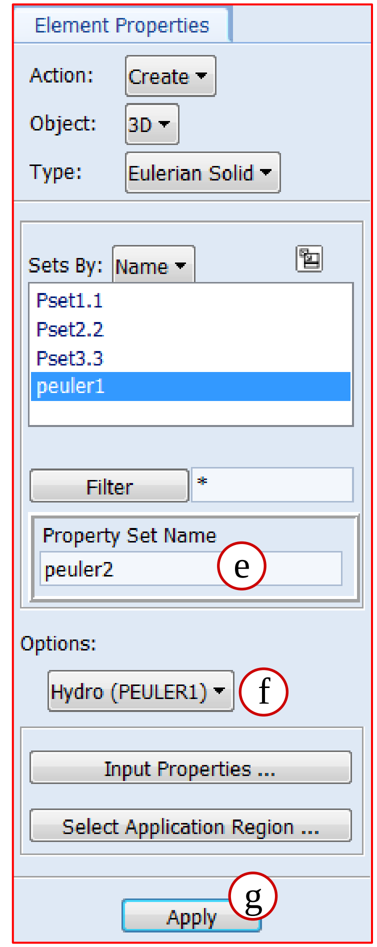


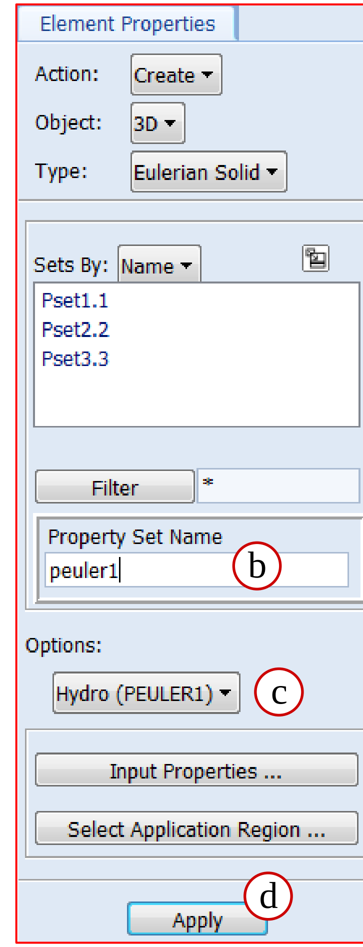
Create 3D properties for use in mesh generator and initialization.

a. Under the *Properties* tab, click **Eulerian Solid** in the *3D Properties* group.
b. Enter **peuler1** for *Property Set Name*.
c. Select **Hydro(PEULER1)** for *Options*.
d. Click **Apply.**

Do the same for the Solid 2 but now with property name peuler2.

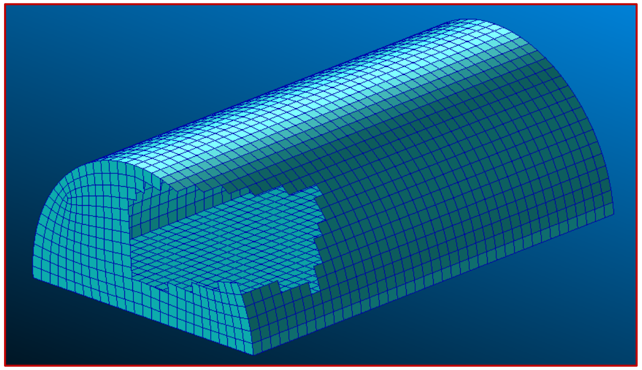
e. Enter **peuler2** for *Property Set Name*.
f. Select **Hydro(PEULER1)** for *Options*.
g. Click **Apply.**





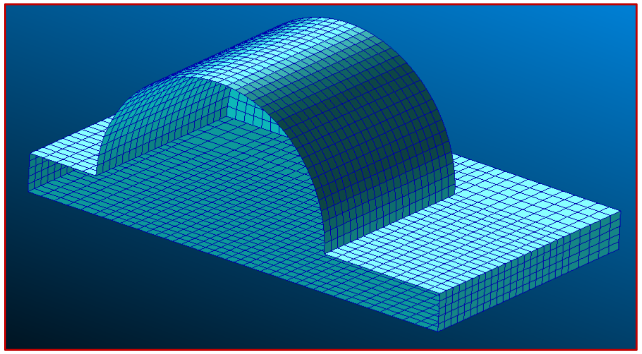
Step 6. Definition of Two Coupling Surfaces

	Is in Couple Surface 1?	Is in Couple Surface 2?	Elements
Bunker shell	Yes	Yes	1 thru 1600
Open Sides	Yes	Yes	1601 thru 2240
Ground within bunker	Yes	No	2241 thru 3280
Ground outside bunker	No	Yes	3413 thru 7904



Couple Surface 1

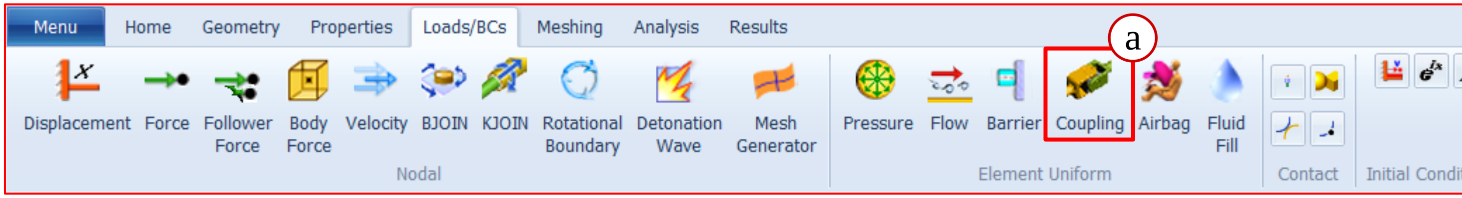
Partially hidden for viewing



Couple Surface 2

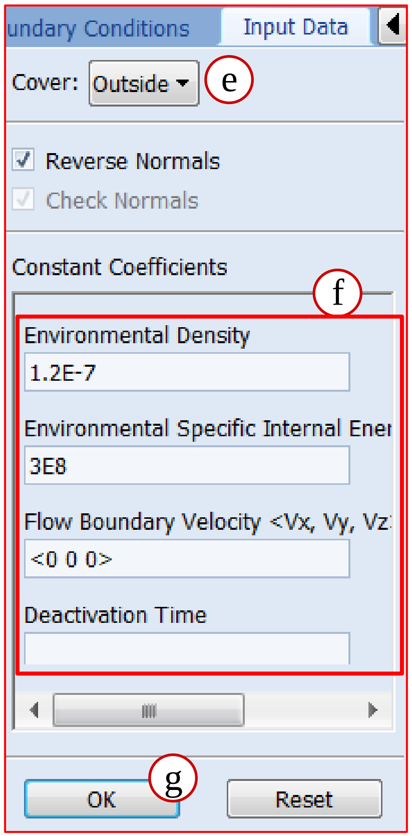
Hide half for viewing

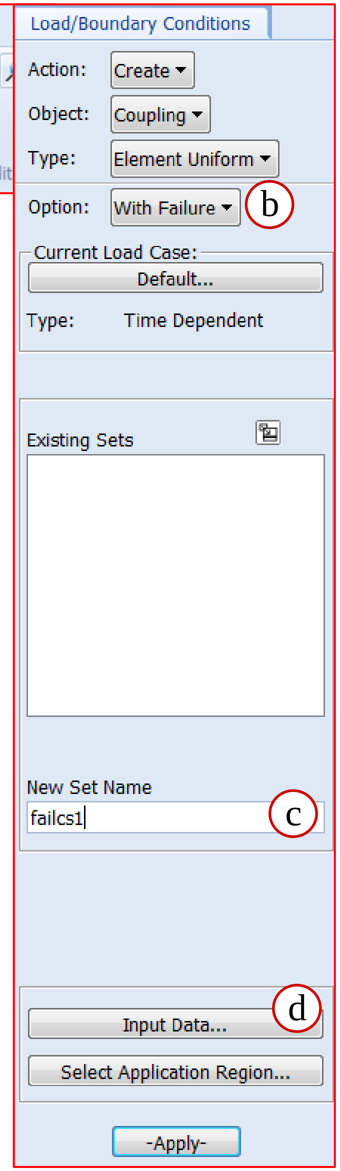
Step 7. Creating the First Coupling Surface



Create the first coupling surface, failcs1. Enter the input data.

a. Under the *Loads/BCs* tab, click **Coupling** in the *Element Uniform* group.
b. Set *Option* to **With Failure**.
c. Enter **failcs1** for the *New Set Name*.
d. Click **Input Data**.
e. Set *Cover* to **Outside**.
f. Enter **1.2e-7** for *Environmental Density* and **3e8** for *Environmental Specific Internal Energy* and **<0 0 0>** for *Flow Boundary Velocity*.
g. Click **OK**.

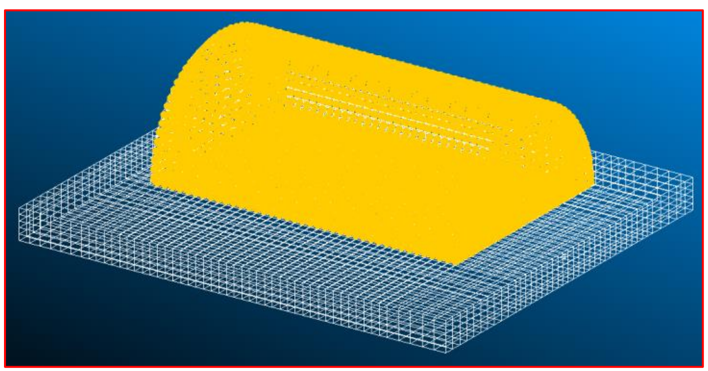


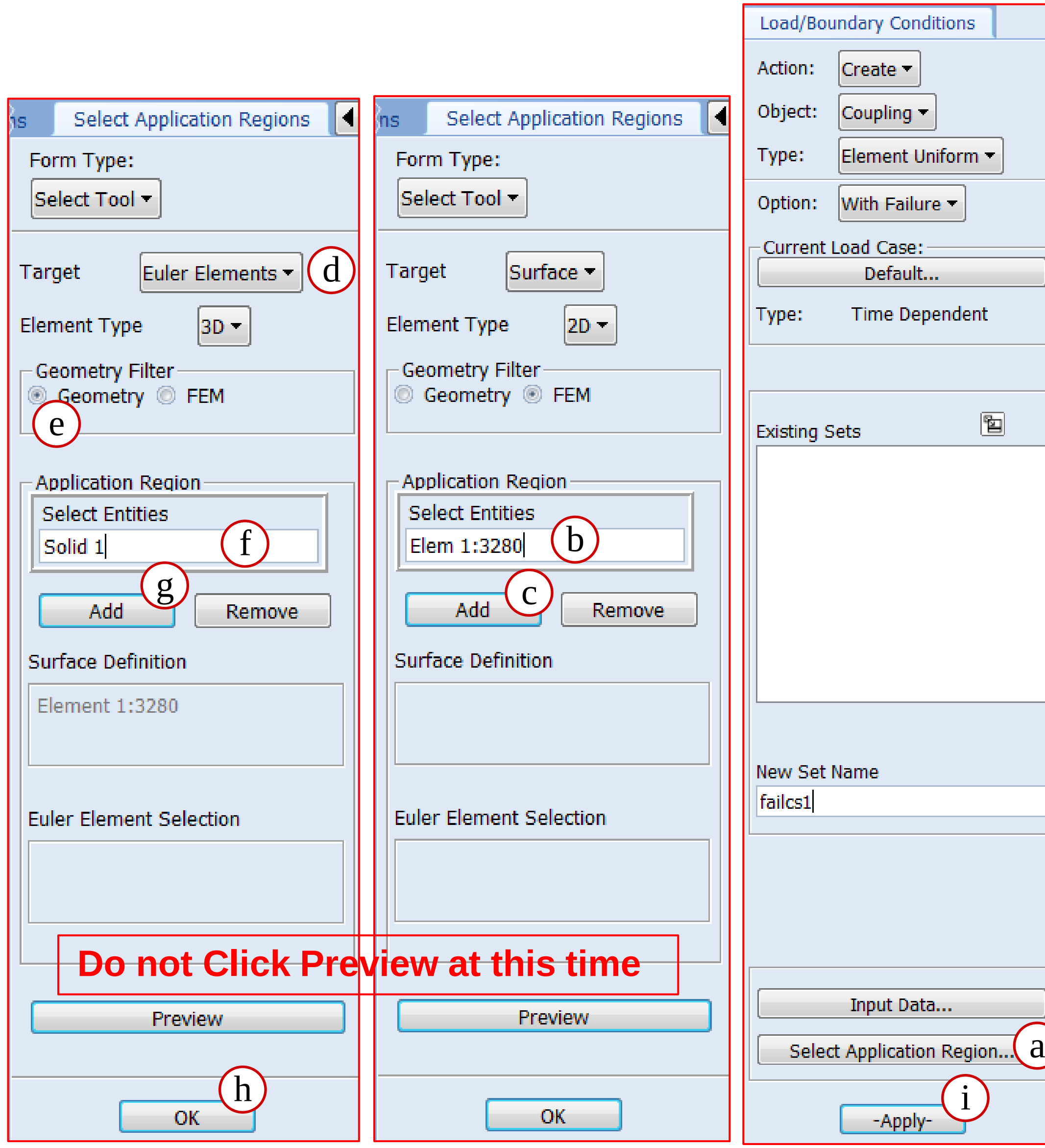


Step 7. Creating the First Coupling Surface (Cont.)

Define the application region for failcs1.

a. Click **Select Application Region**.
b. Enter **Elm 1:3280** for *Select Entities*.
c. Click **Add**.
d. Set *Target* to **Euler Elements**.
e. Select **Geometry**.
f. Enter **Solid 1** for *Select Entities*.
g. Click **Add**.
h. Click **OK**.
i. Click **Apply**.

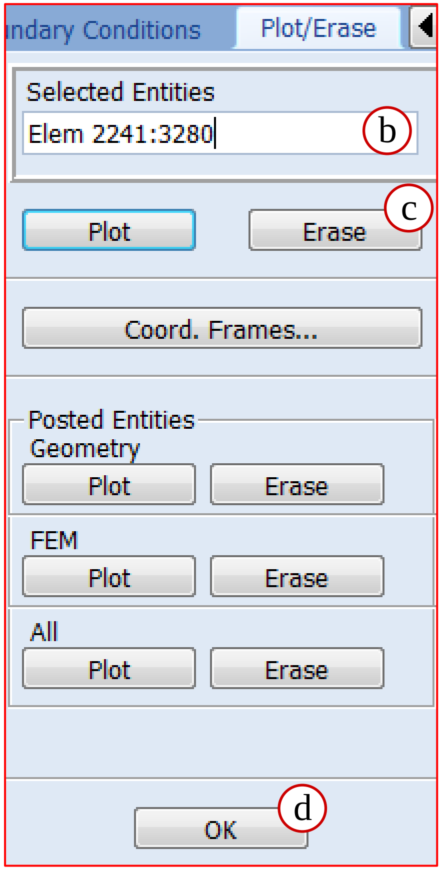




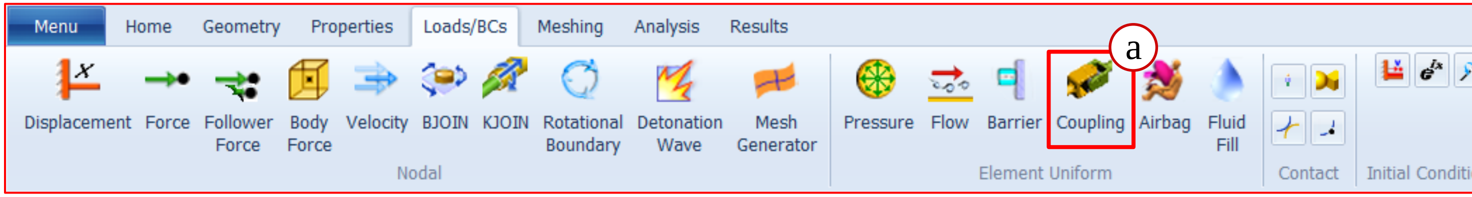
Step 8. Creating the Second Coupling Surface

The second coupling surface is given by all elements except the elements that model the ground within the bunker. First, erase these ground elements.

a. Under the *Home* tab, click **Plot/Erase** in the *Misc.* group.
b. Enter **Elm 2241:3280**.
c. Click **Erase**.
d. Click **OK**.



Step 8. Creating the Second Coupling Surface (Cont.)



Define the cover and application region.

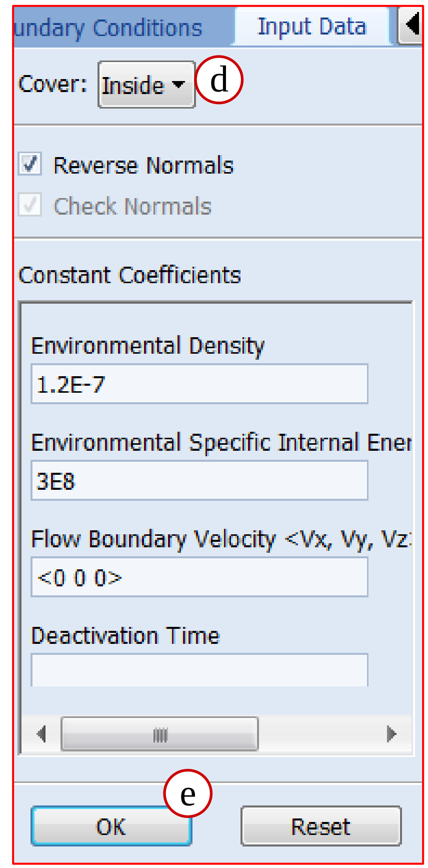
a. Under the *Loads/BCs* tab, click **Coupling** in the *Element Uniform* group.

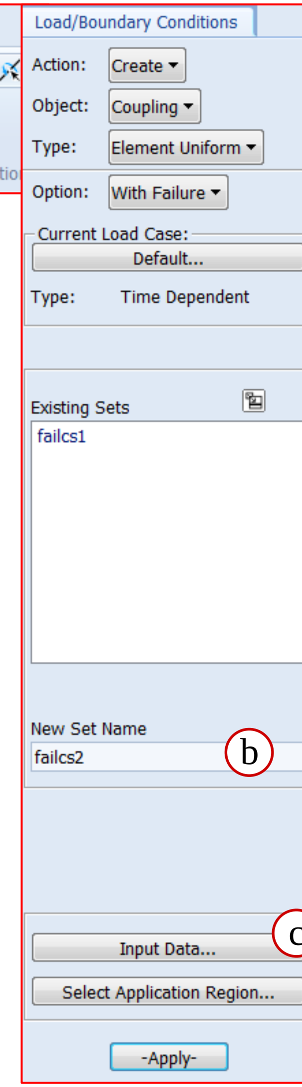
b. Enter **failcs2**.

c. Click **Input Data**.

d. Select **Inside**.

e. Click **OK**.

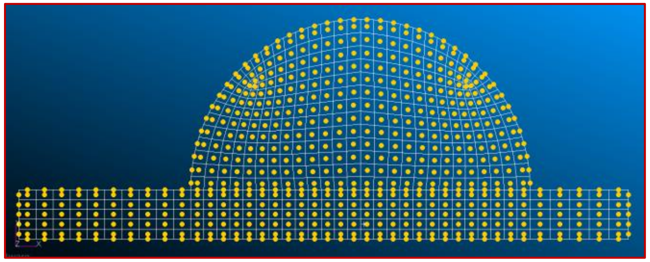


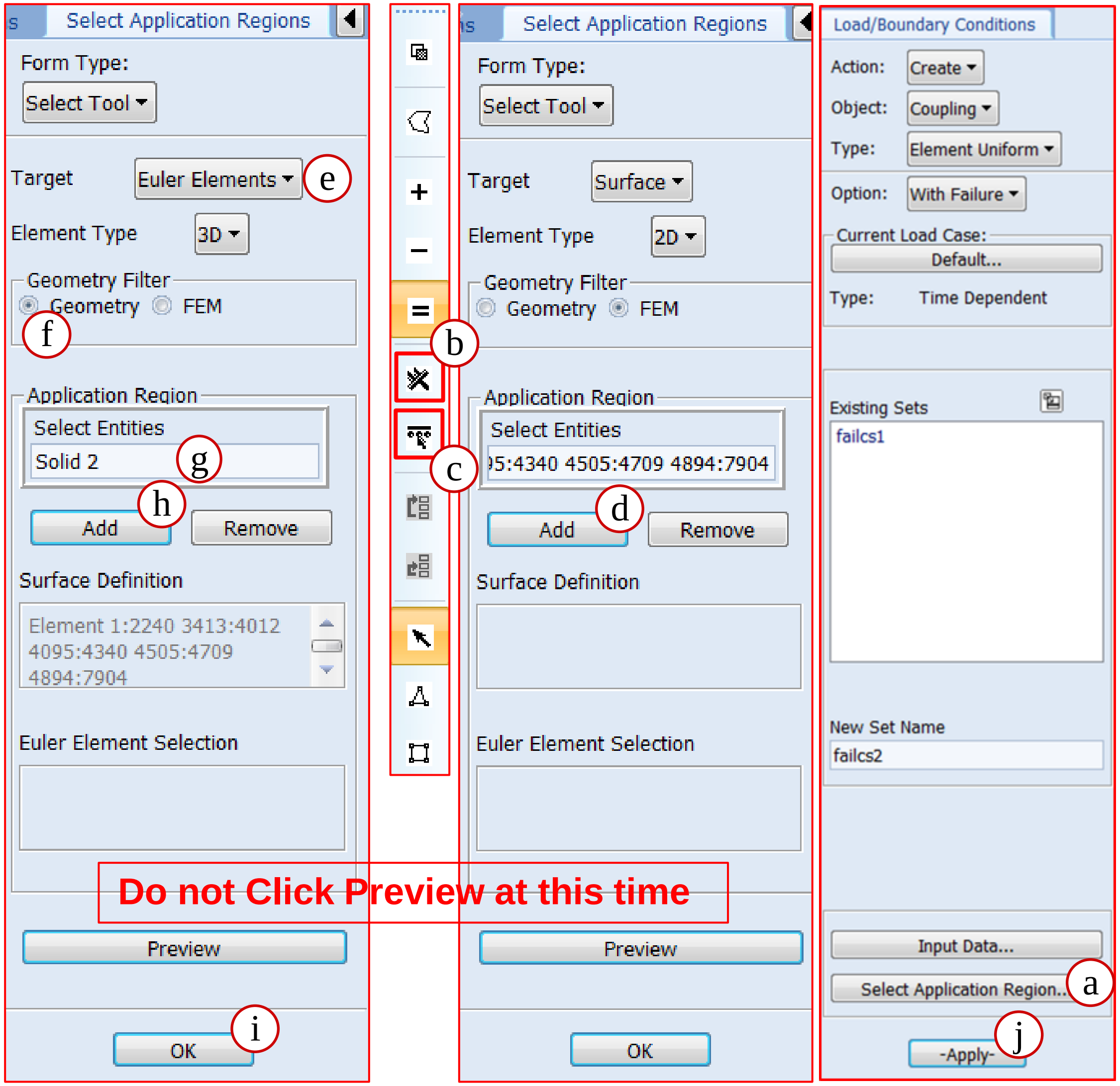


Step 8. Creating the Second Coupling Surface (Cont.)

Define the application region.

a. Click **Select Application Region**.
b. Click **Pick Clear**.
c. Click **Pick All**.
d. Click **Add**.
e. Set *Target* to **Euler Elements**.
f. Select **Geometry**.
g. Enter **Solid 2** for *Select Entities*.
h. Click **Add**.
i. Click **OK**.
j. Click **Apply**.

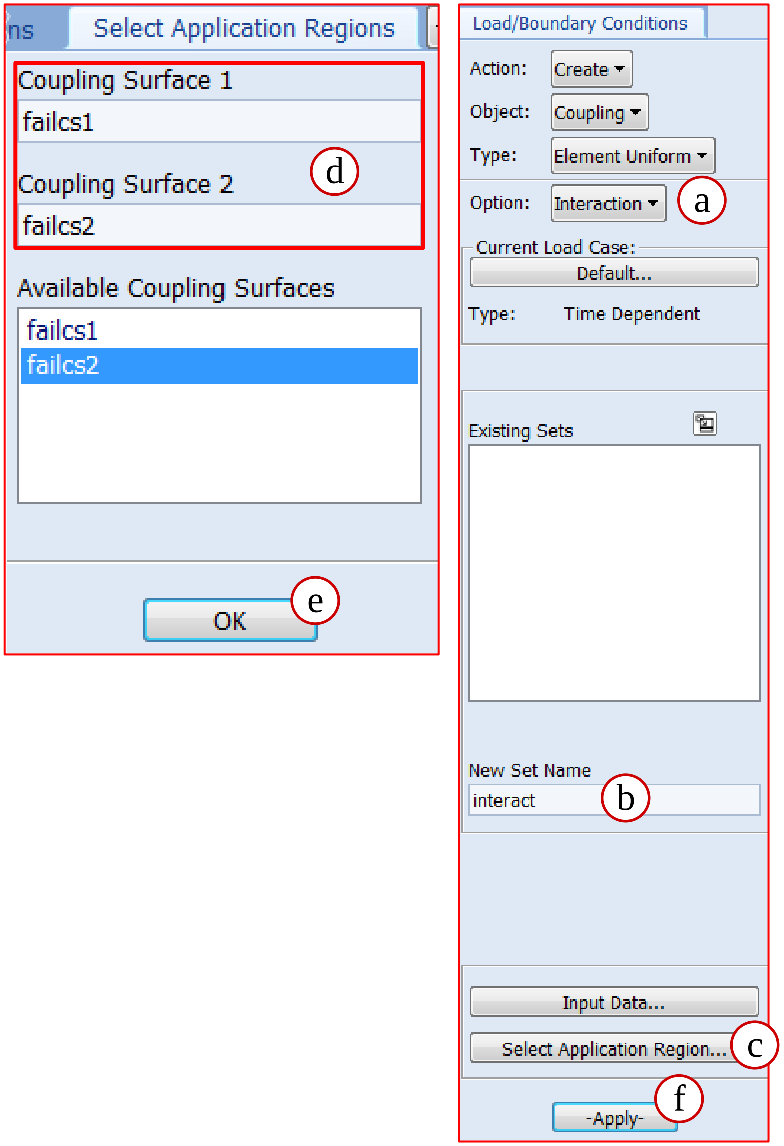




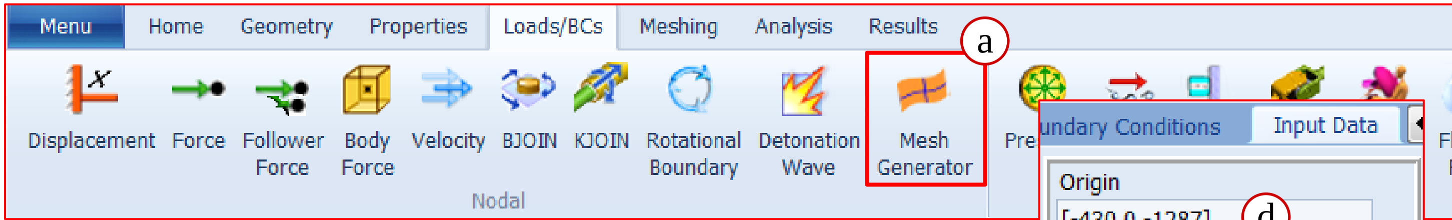
Step 9. Creating Interactive Failure

Create interactive failure for the coupling surfaces.

a. Select **Interaction** for *Option*.
b. Enter **interact** as *the New Set Name.*
c. Click **Select Application Region**.
d. Select **failcs1** and **falilcs2** for *Coupling Surface 1* and Coupling Surface *2.*
e. Click **OK**.
f. Click **Apply**.

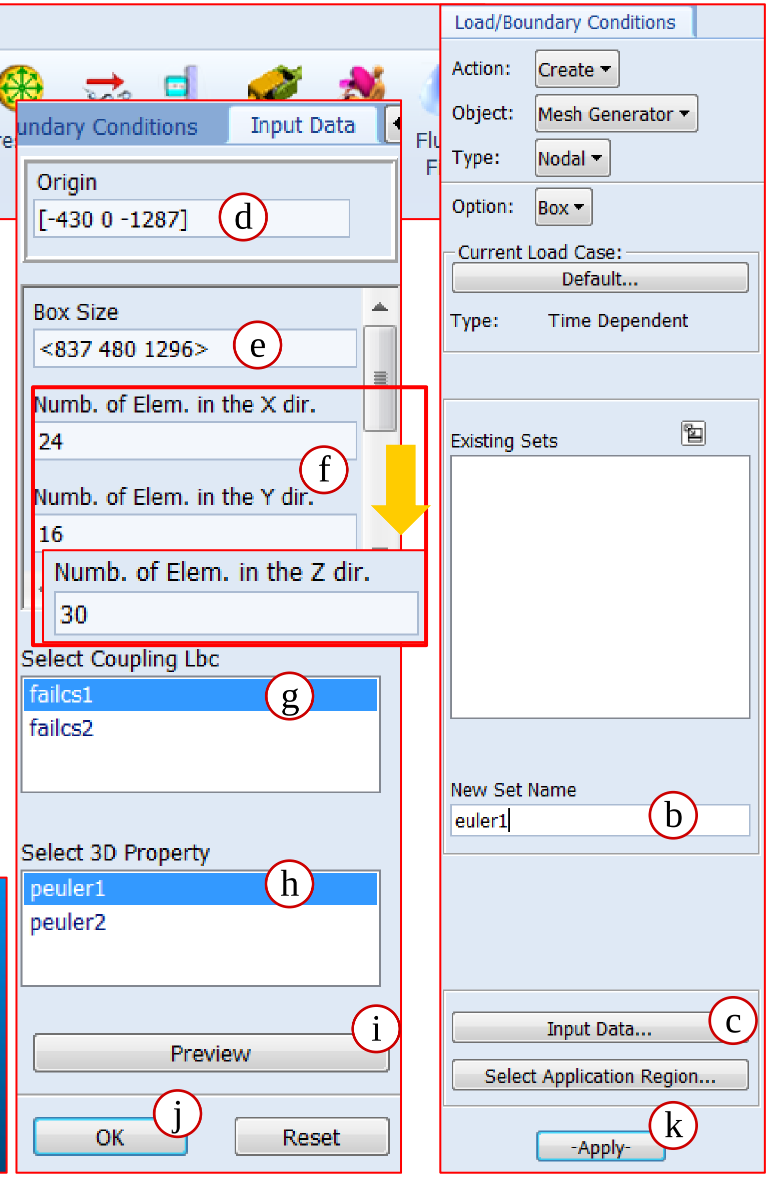


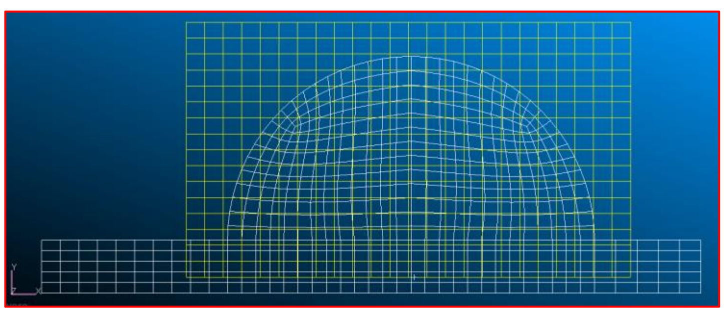
Step 10. Euler Mesh For the First Coupling



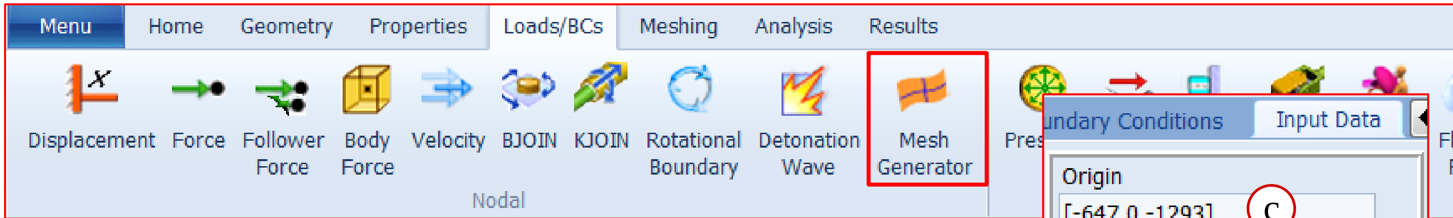
Create the euler mesh for the first coupling surface.

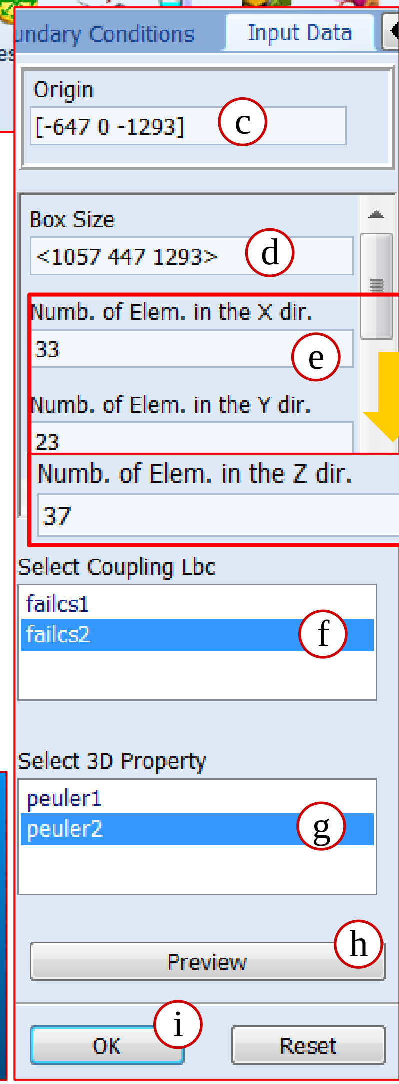
a. Under the *Loads/BCs* tab, click **Mesh Generator** in the *Nodal* group.
b. Enter **euler1** for the *New Set Name*.
c. Click **Input Data**.
d. Enter **[-430, 0, -1287]** for *Origin*.
e. Enter **<837, 480, 1296>** for *Box Size*.
f. For *Number of Elements in X, Y and Z dir.,* enter **24, 16** and **30**.
g. Select **failcs1**.
h. Select **peuler1**.
i. Click **Preview**.
j. Click **OK**.
k. Click **Apply**.

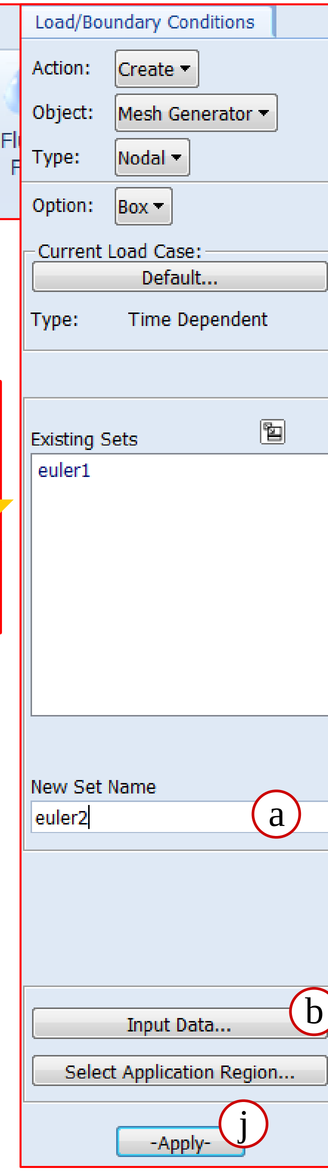




Step 11. Euler Mesh for the Second Coupling

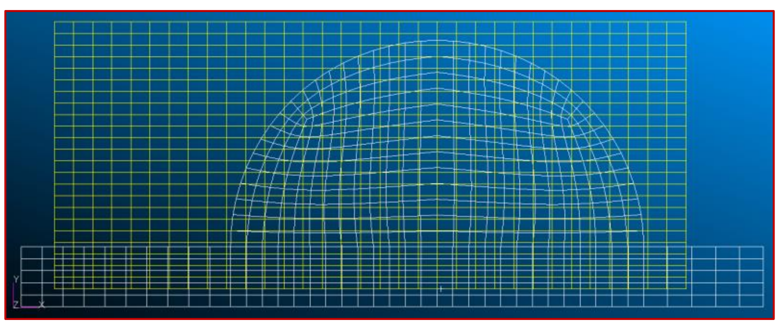




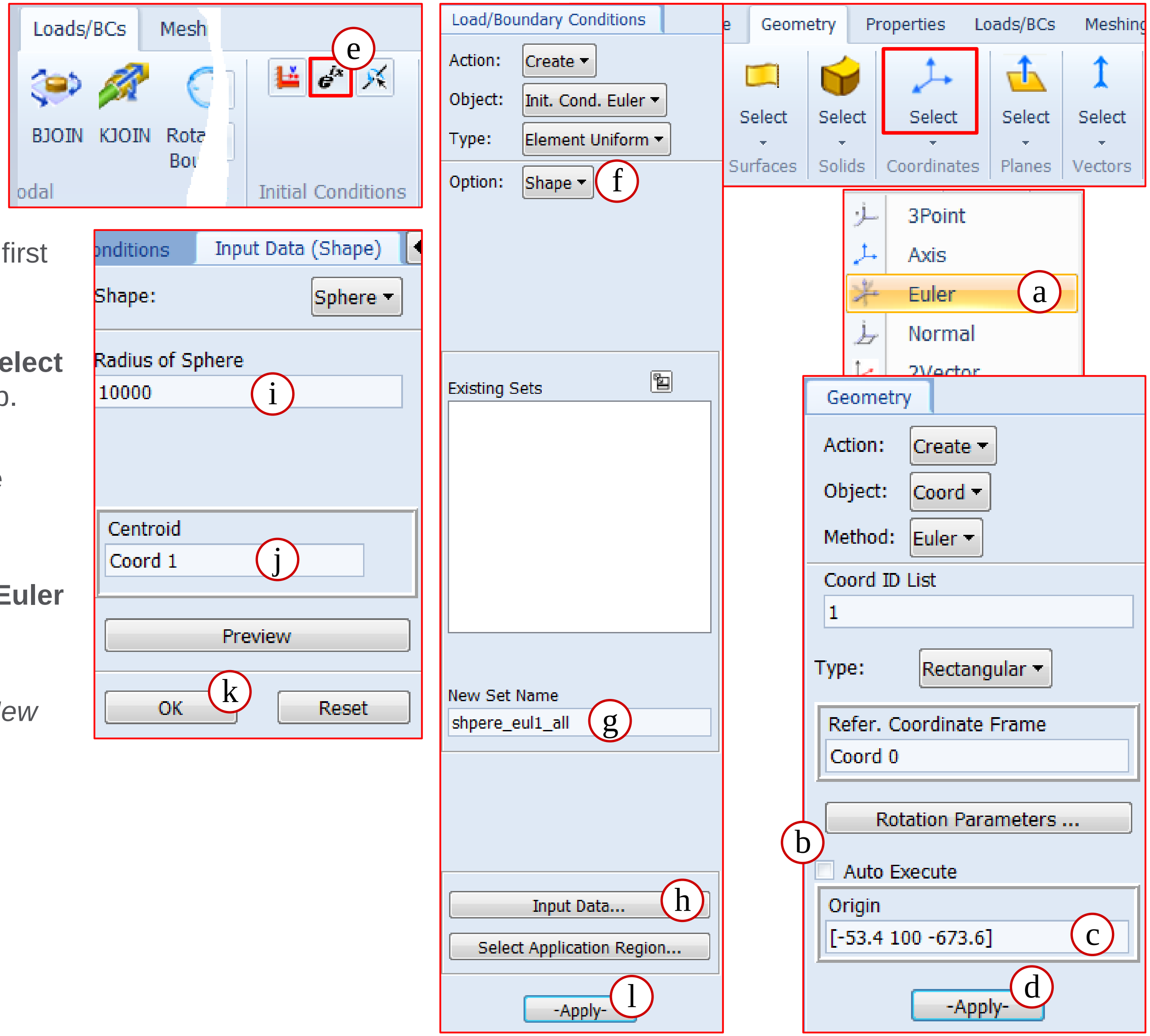


Create the euler mesh for the second coupling surface.

a. Enter **euler2** for the *New Set Name*.
b. Click **Input Data**.
c. Enter **[-647, 0, -1293]** for *Origin*.
d. Enter **<1057, 447, 1293>** for *Box Size*.
e. For *Number of Elements in X, Y and Z dir.,* enter **33, 23** and **37**.
f. Select **failcs2**.
g. Select **peuler2**.
h. Click **Preview.**
i. Click **OK**.
j. Click **Apply**.



Step 12. Initialization of Euler Mesh/Shapes



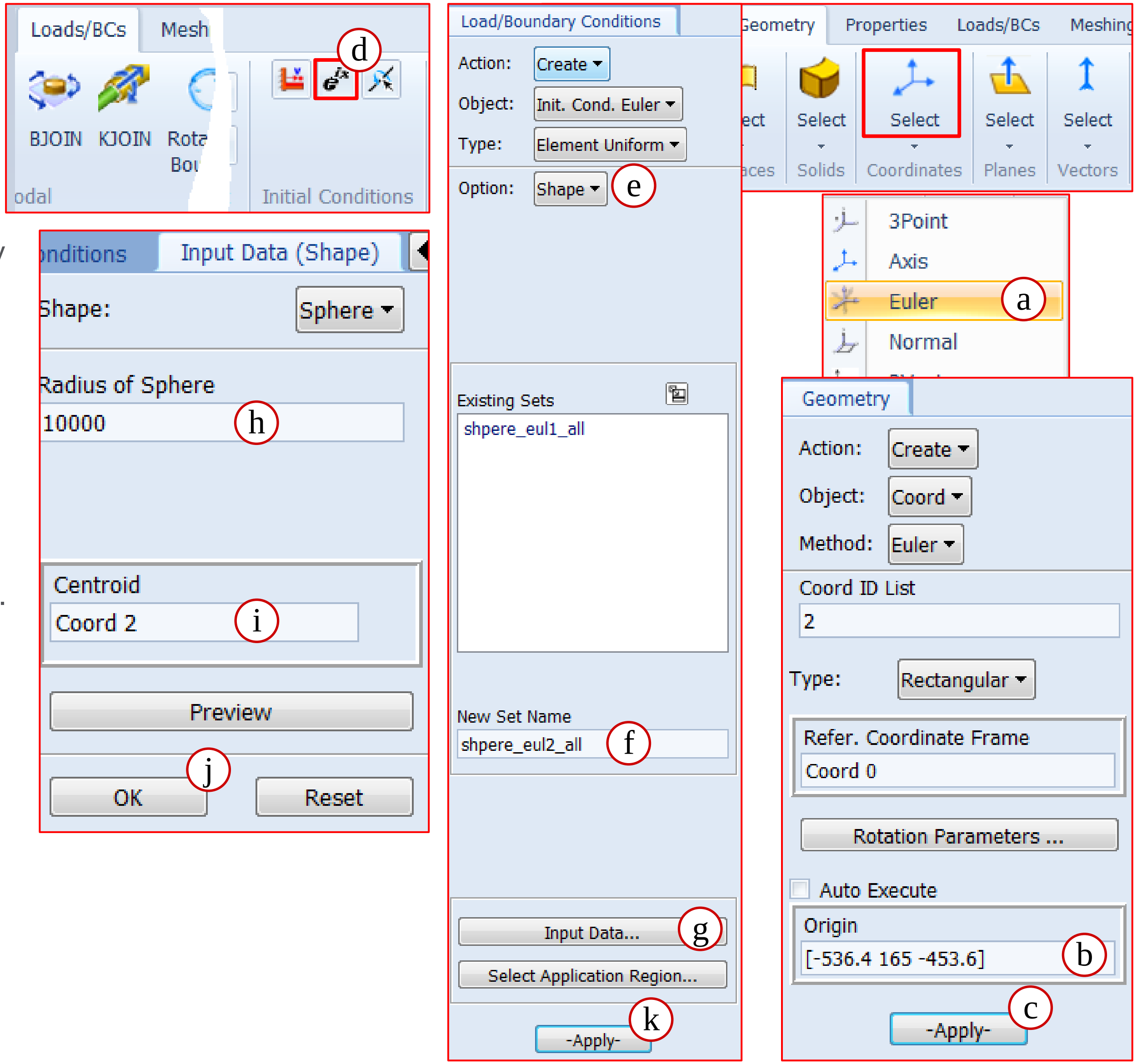
Create the coordinate frame for the first sphere. Then, set the boundary conditions.

a. Under the *Geometry* tab, click **Select > Euler** in the *Coordinates* group.
b. Uncheck **Auto Execute**.
c. Enter **[-53.4, 100, -673.6]** for the coordinates at the *Origin*.
d. Click **Apply**.
e. Under the *Loads/BCs* tab, click **Euler** in the *Initial Conditions* group.
f. Set *Option* to **Shape**.
g. Enter **sphere_eul1_all** for the *New Set Name*.
h. Click **Input Data**.
i. Enter *Radius* = **10000**.
j. Select **Coord1**.
k. Click **OK**.
l. Click **Apply**.

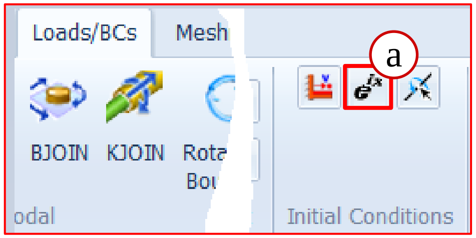
Step 12. Initialization of Euler Mesh/Shapes (Cont.)

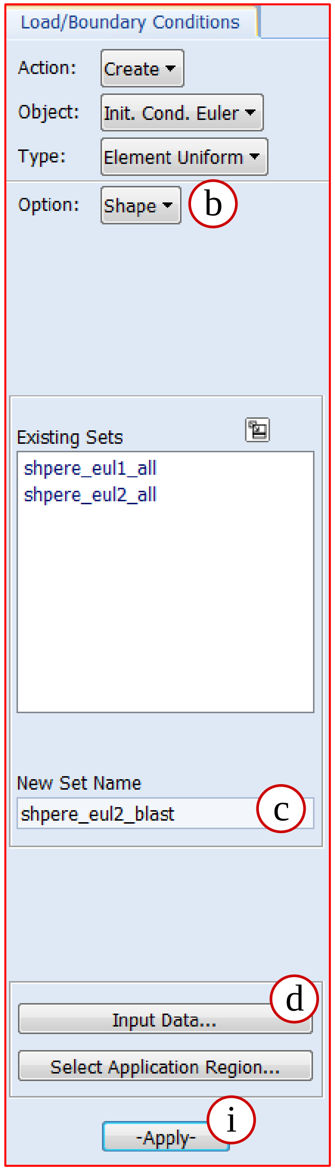
Create the coordinate frame for the second sphere. Then, set the boundary conditions.

a. Under the *Geometry* tab, click **Select > Euler** in the *Coordinates* group.
b. Enter **[-536.4, 165, -453.6]** for the coordinates at the *Origin*.
c. Click **Apply**.
d. Under the *Loads/BCs* tab, click **Euler** in the *Initial Conditions* group.
e. Set *Option* to **Shape**.
f. Enter **sphere_eul2_all** for the *New Set Name*.
g. Click **Input Data**.
h. Enter *Radius* = **10000**.
i. For *Centroid* select **Coord 2**.
j. Click **OK**.
k. Click **Apply**.



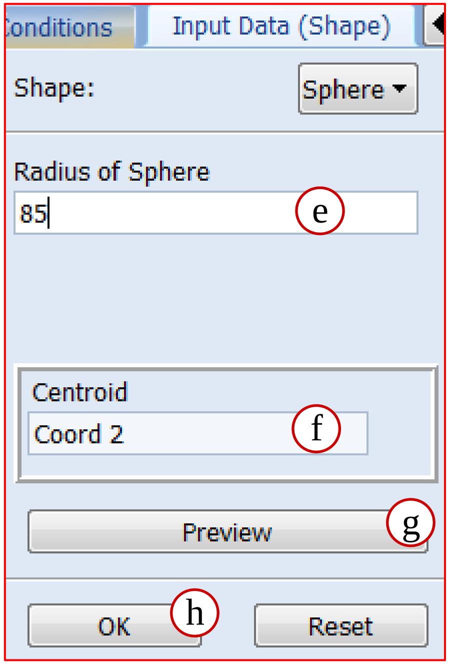
Step 12. Initialization of Euler Mesh/Shapes (Cont.)

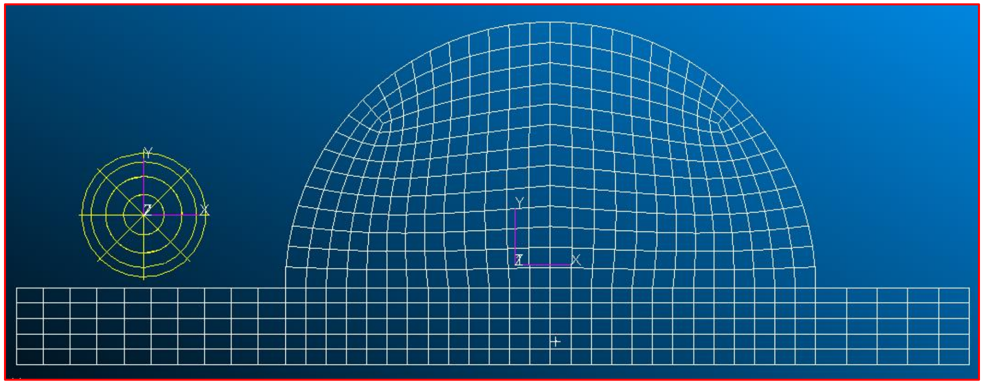




Set the boundary conditions for the sphere containing the blast.

a. Under the *Loads/BCs* tab, click **Euler** in the *Initial Conditions* group.
b. Set *Option* to **Shape**.
c. Enter **sphere_eul2_blast** for the *New Set Name*.
d. Click **Input Data**.
e. Enter *Radius* = **85**.
f. For *Centroid* select **Coord 2**.
g. Click **Preview**.
h. Click **OK**.
i. Click **Apply**.

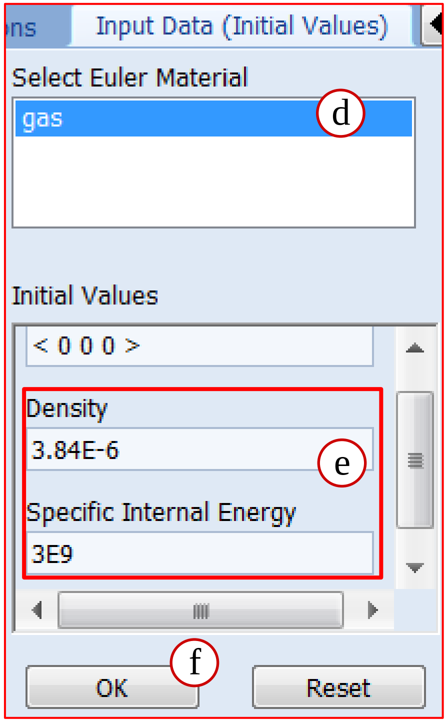


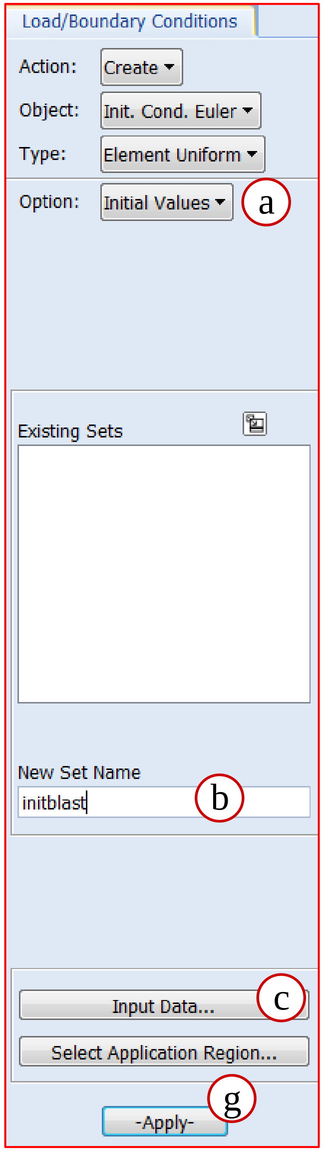


Step 13. Initialization of Euler Mesh/Initial Values

Create the initial values for the blastwave.

a. Set *Option* to **Initial Value**.
b. Enter **initblast** for the initial value set name.
c. Select **Input Data**.
d. Select **gas** for *Select Material*.
e. Enter *Density* = **3.84e-6**, *Specific Internal Energy* = **3e9**
f. Click **OK**.
g. Click **Apply**.

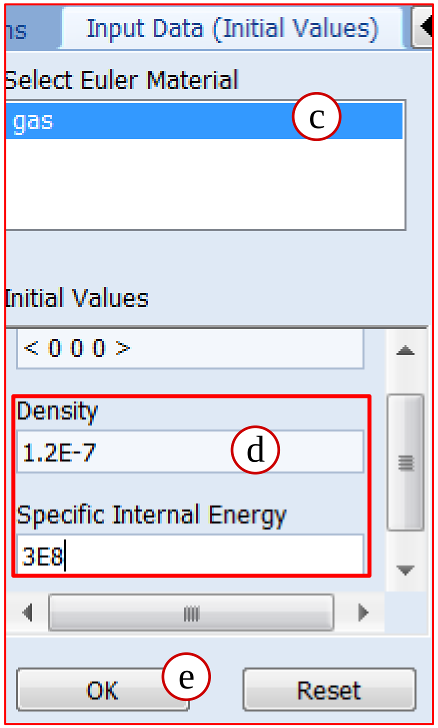


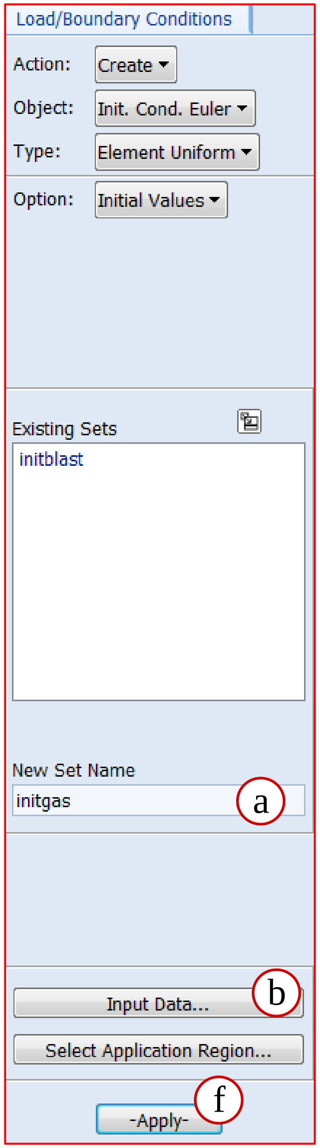


Step 13. Initialization of Euler Mesh/Initial Values (Cont.)

Create initial values for the gas outside the initial blast.

a. Enter **initgas** for the initial value set name.
b. Select **Input Data**.
c. Select **gas** for *Select Material*.
d. Enter *Density* = **1.2e-7**, *Specific Internal Energy* = **3e8**
e. Click **OK**.
f. Click **Apply**.

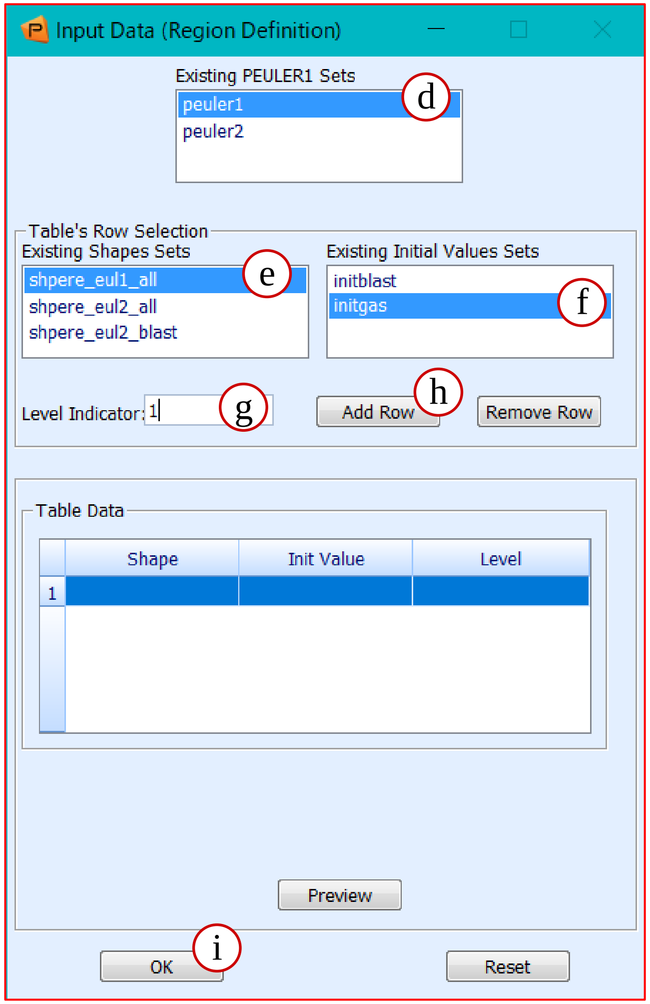


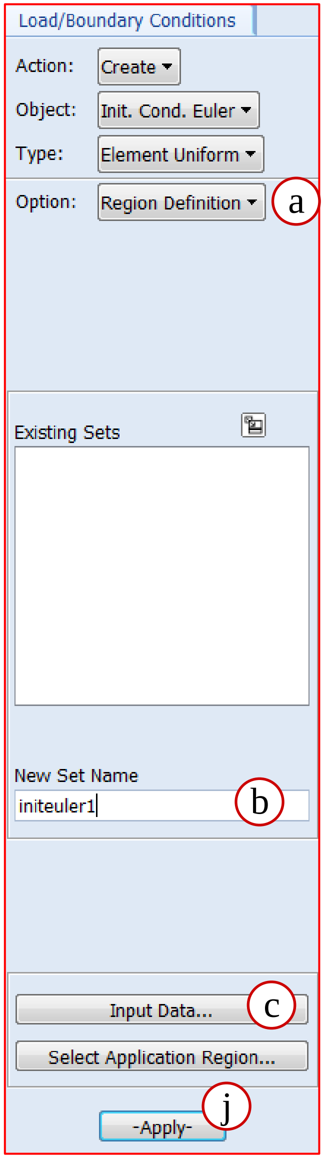


Step 14. Initialization of the First Euler Mesh

Initialize the euler mesh that references peuler1. This is the euler mesh for the first coupling surface.

a. Select **Region Definition** for *Option.*
b. Enter **initeuler1** for *New Set Name*.
c. Click **Input Data**.
d. Select **peuler1** for *Existing PEULER1 Sets.*
e. Select **sphere_eul1_all** for *Existing Shapes Sets.*
f. Select **initgas** for *Existing Initial Value Sets*.
g. Enter **Level = 1.0**.
h. Click **Add Row**.
i. Click **OK**.
j. Click **Apply**.

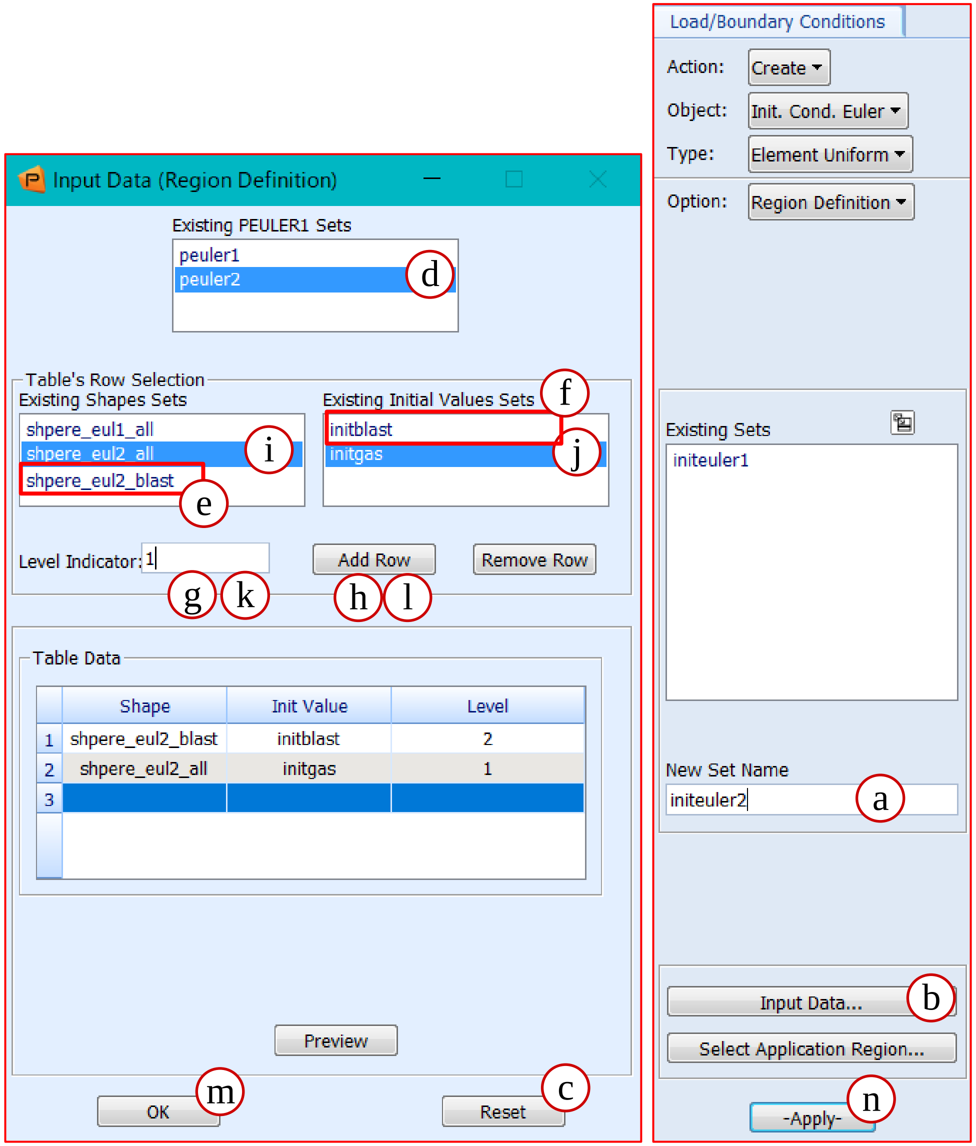




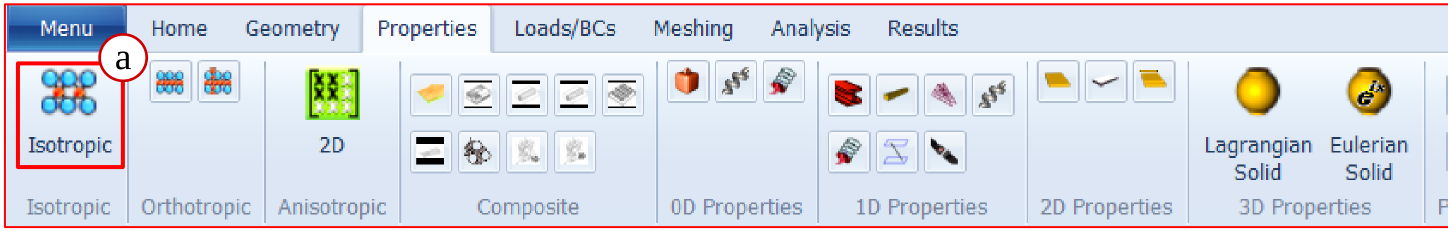
Step 15. Initialization of the Second Euler Msesh

Initialize the Euler mesh that references peuler2. This is the euler mesh for the second coupling surface.

a. Enter **initeuler2** for *New Set Name*.
b. Click **Input Data**.
c. Click **Reset**.
d. Select **peuler2** for *Existing PEULER1 Sets.*
e. Select **sphere_eul2_blast** in the *Existing Shapes Sets*.
f. Select **initblast** in the *Existing Initial Values Sets*.
g. Enter **Level = 2**.
h. Click **Add Row**.
i. Select **sphere_eul2_all** in the *Existing Shapes Sets*.
j. Select **initgas** in the *Existing Initial Values Sets*.
k. Enter **Level = 1**.
l. Click **Add Row**.
m. Click **OK**.
n. Click **Apply**.

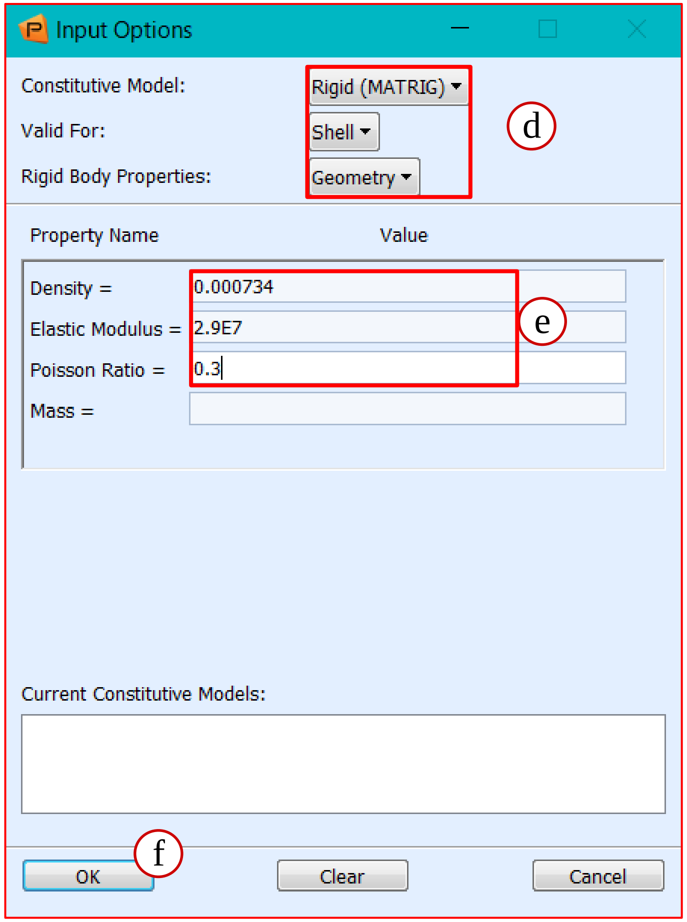


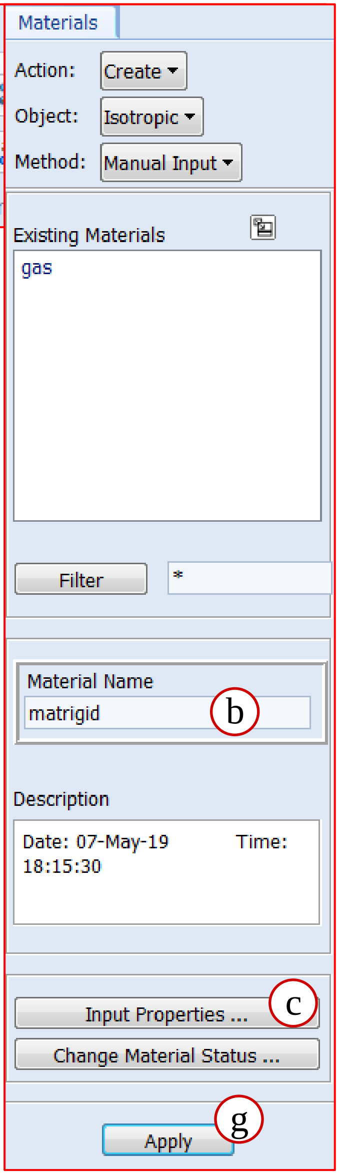
Step 16. Create Material for Ground



Create the material for the ground.

a. Under the *Properties* tab, click **Isotropic**.
b. Enter **matrigid** for *Material Name*.
c. Click **Input Properties**.
d. Set *Constitutive Model* to **Rigid (MATRIG)**, *Valid For* to **Shell** and *Rigid Body Properties* to **Geometry**.
e. Enter **0.000734** for *Density*, **2.9e7** for *Elasticity Modulus* and **0.3** for *Poisson Ratio*.
f. Click **OK**.
g. Click **Apply**.





Step 17. Create Material for Bunker Shell

Create the material for the bunker shell.

a. Enter **steel** for *Material Name*.

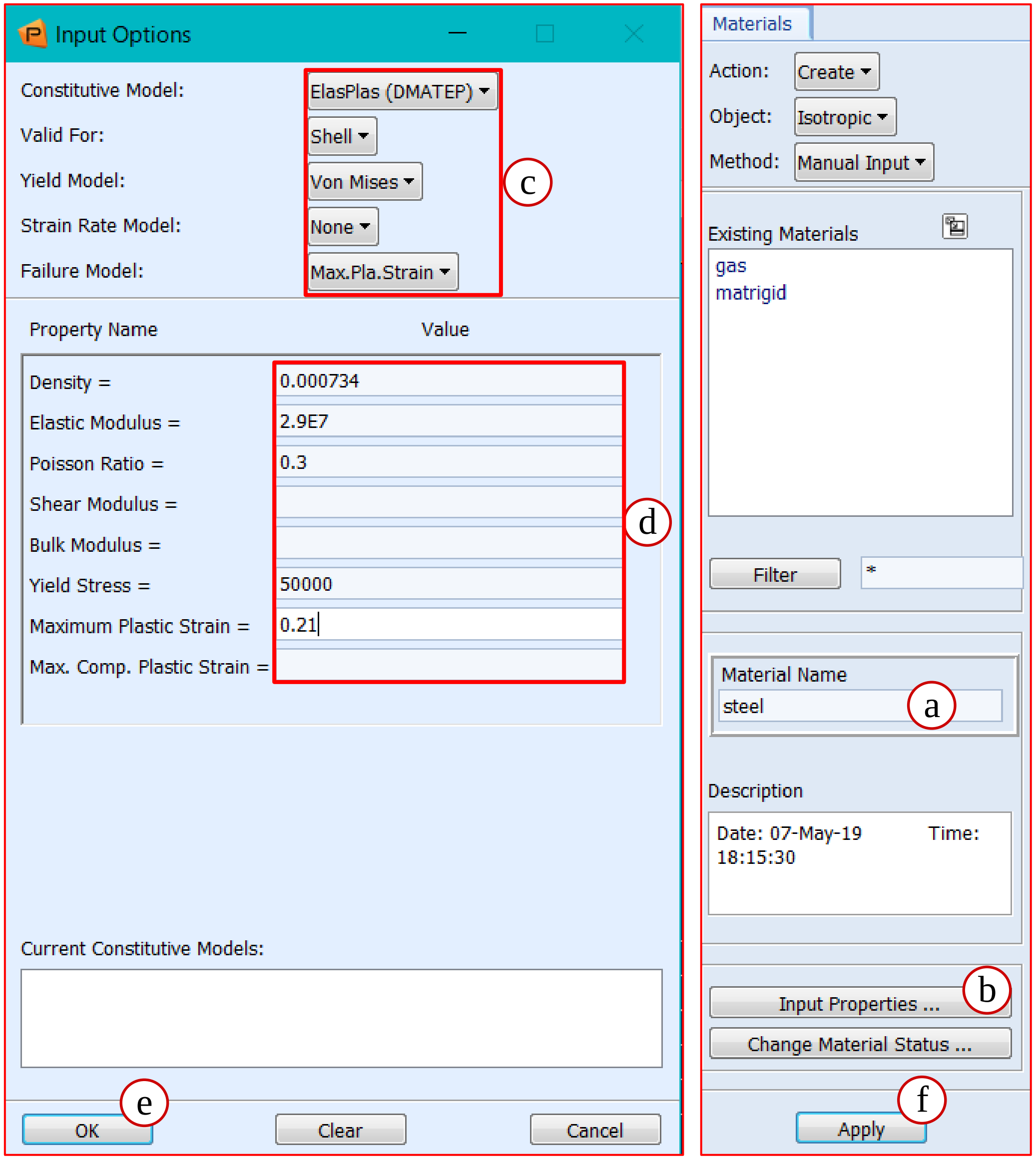
b. Click **Input Properties**.

c. Set the following:
Constitutive Model: **ElasPlas (DMATEP)**
Valid For: **Shell**
Yield Model: **Von Mises**
Strain Rate Model: **None**
Failure Model: **Max. Pla. Strain**.

d. Enter **0.000734** for *Density*, **2.9e7** for *Elasticity Modulus,* **0.3** for *Poisson Ratio*, **50000** for *Yield Stress* and **0.21** for *Maximum Plastic Strain*.

e. Click **OK**.

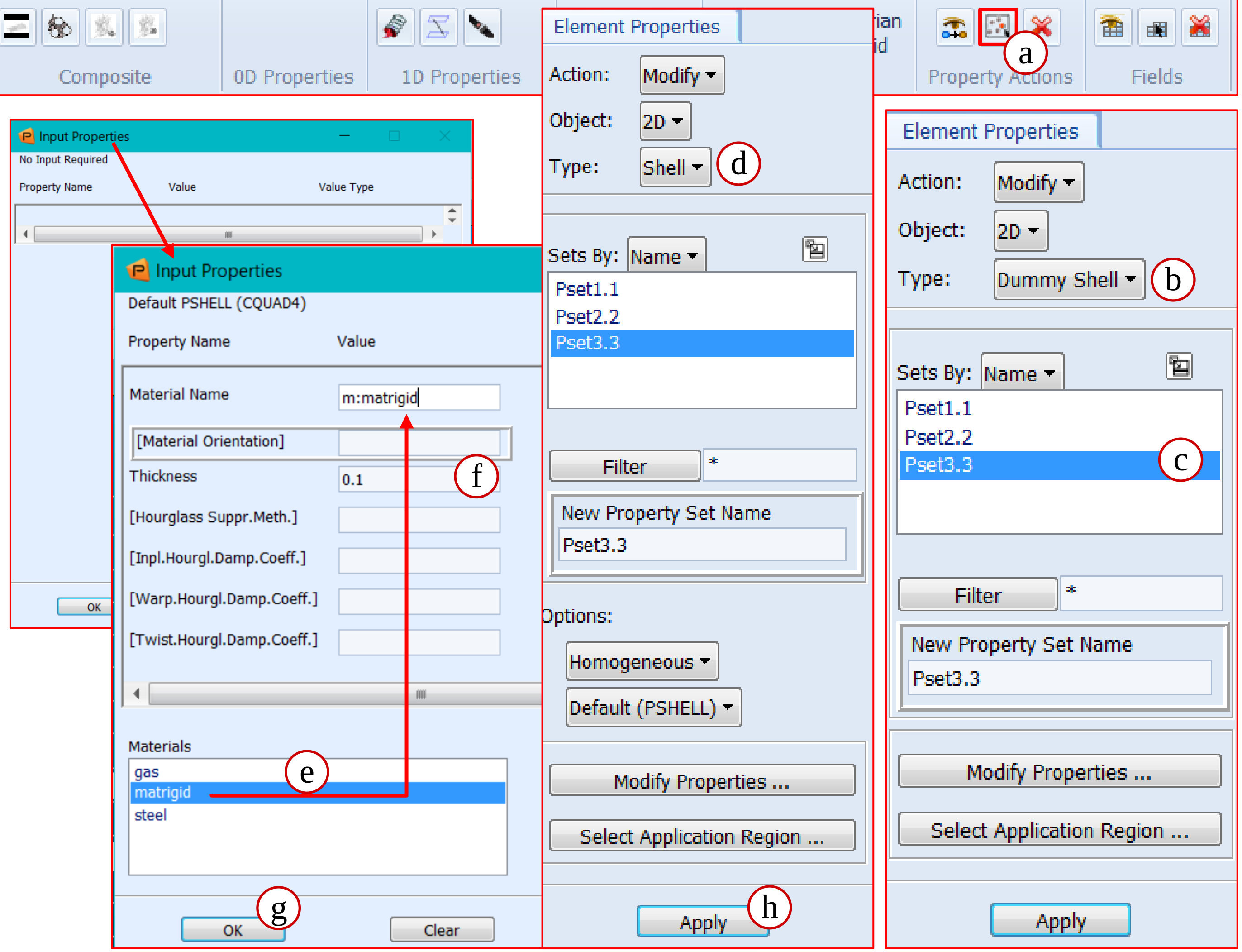
f. Click **Apply**.



Step 18. Modify Properties for Ground

Change the dummy shells of Property 3 into real shells.

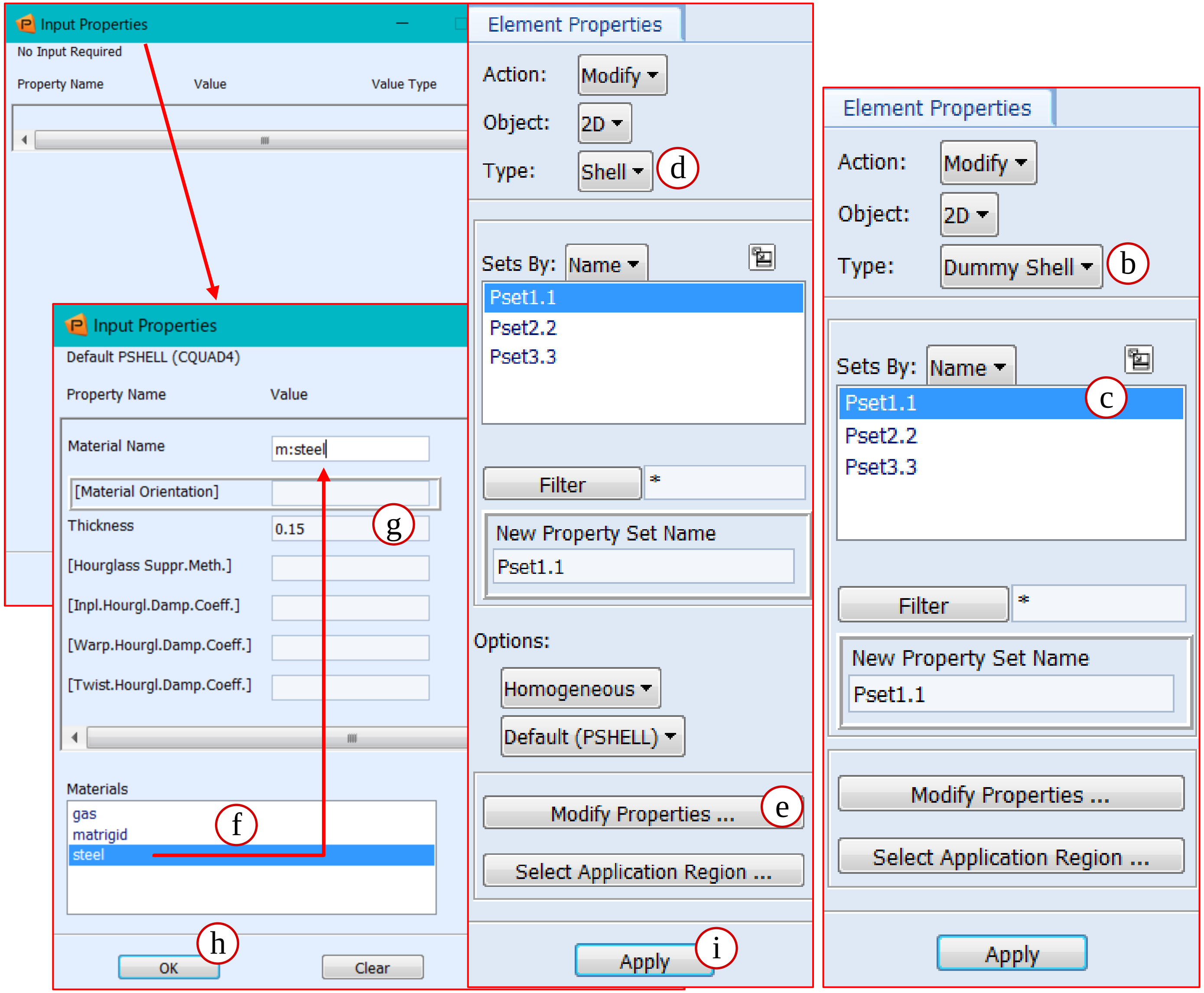
a. Click **Modify Property** on the *Property Actions group*.
b. Pull down *Type* to **Dummy Shell**.
c. Select **Pset3.3** and click **OK** to close *Input Properties*.
d. Pull down *Type* to **Shell**.
e. Click **matrigid**.
f. Enter a *thickness* of **0.1**.
g. Click **OK**.
h. Click **Apply**.



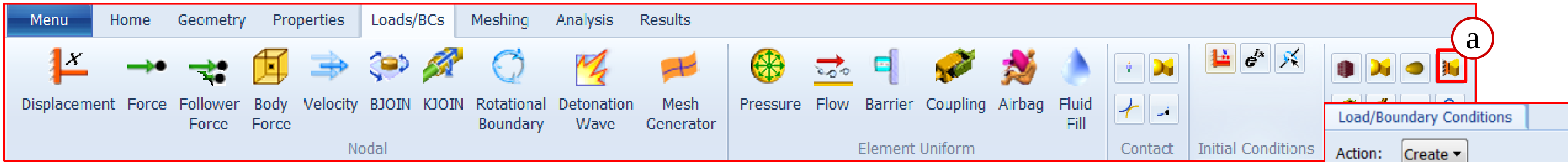
Step 19. Modify Properties for Bunker Shell

Change the dummy shells of Property 1 into real shells.

a. Click **Modify Property** on the *Property Actions group*.
b. Pull down *Type* to **Dummy Shell**.
c. Select **Pset1.1** and click **OK** to close *Input Properties*.
d. Pull down *Type* to **Shell**.
e. Click **Modify Properties**.
f. Click **steel**.
g. Enter a *thickness* of **0.15**.
h. Click **OK**.
i. Click **Apply**.

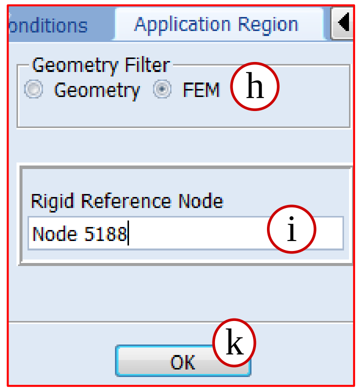


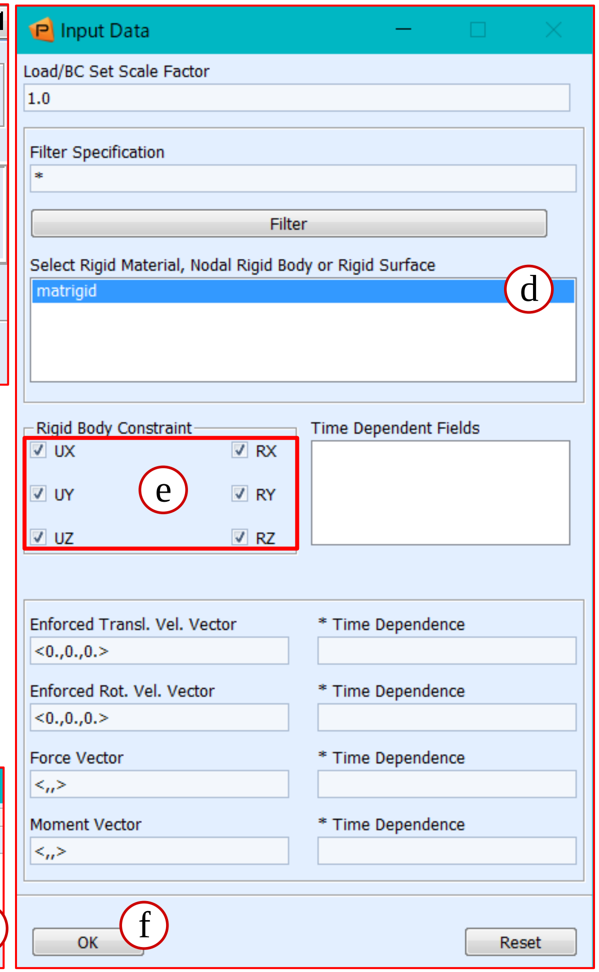
Step 20. Create Rigid for Ground

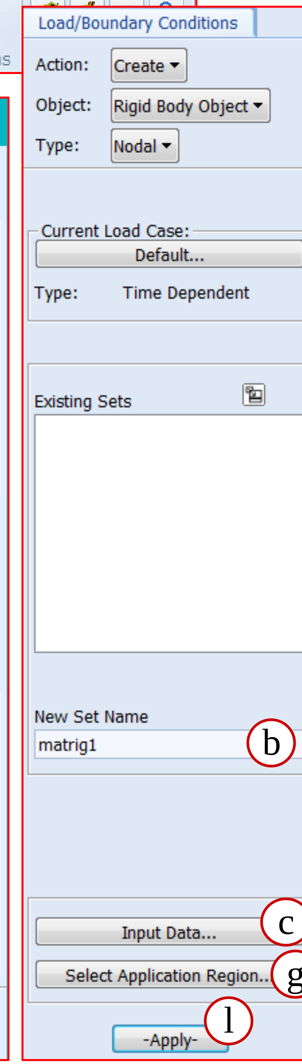


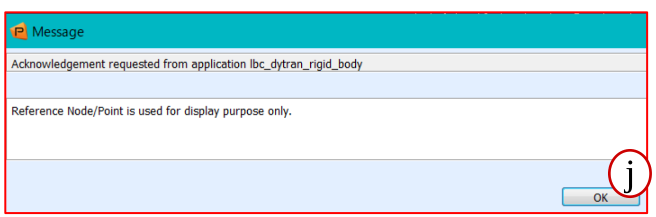
Create the rigid body object for the ground.

a. Under the *Loads/BCs* tab, click **Body Object** in the *Rigid* group.
b. Enter the *New Set Name* **matrig1**.
c. Click **Input Data**.
d. Select **matrigid**.
e. Constrain all degrees of freedom.
f. Click **OK**.
g. Click **Select Application Region**.
h. Set *Geometry Filter* to **FEM**.
i. Select any Node of the Ground
j. Click **OK** in the *Message dialogue*.
k. Click **OK**.
l. Click **Apply**.

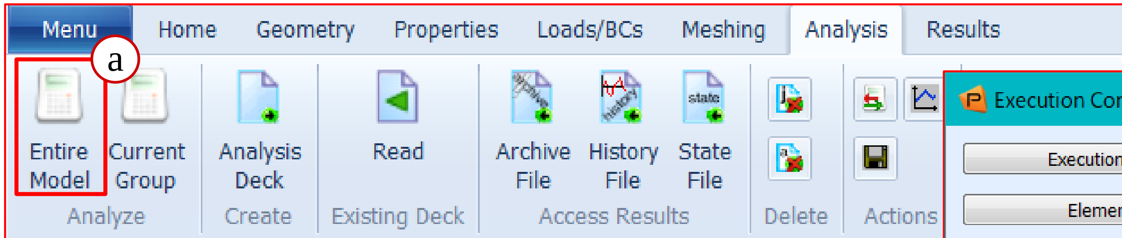






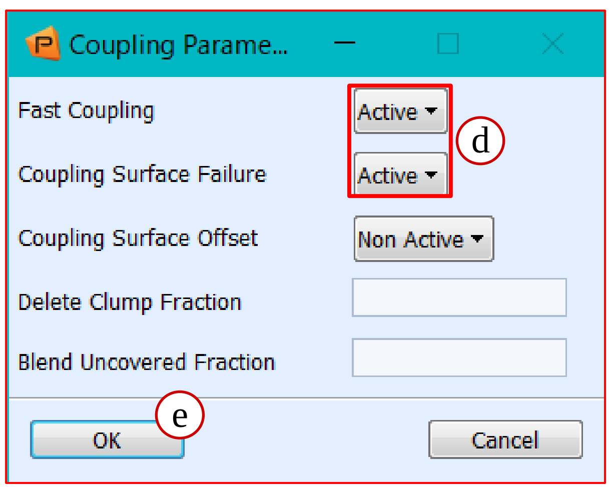


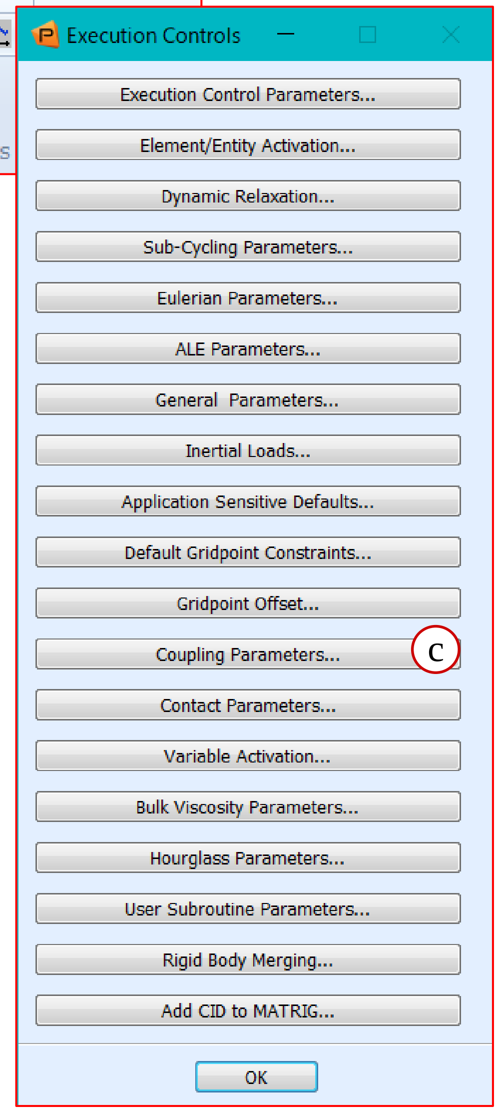
Step 21. Set Up the Analysis

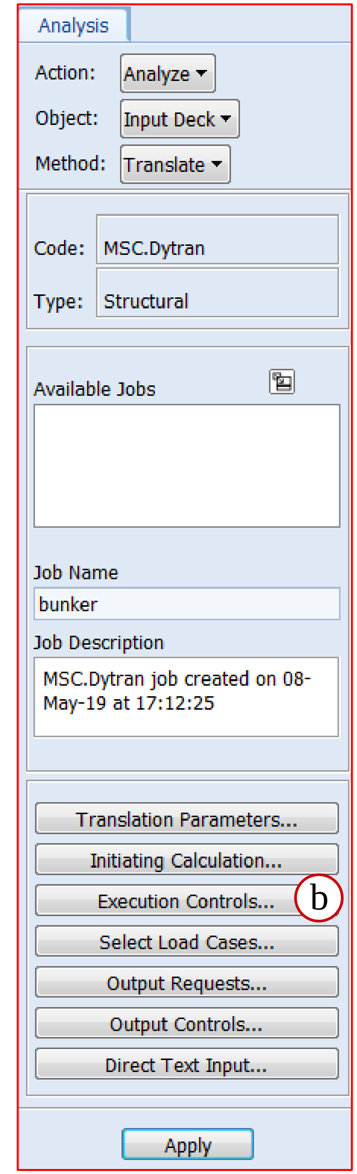


Set the execution controls and coupling parameters for the job.

a. Under the *Analysis* tab, click **Entire Model** in the *Analyze* group.
b. Click **Execution Controls**.
c. Click **Coupling Parameters**.
d. Set *Fast Coupling* and *Coupling Surface Failure* to **Active**.
e. Click **OK**.



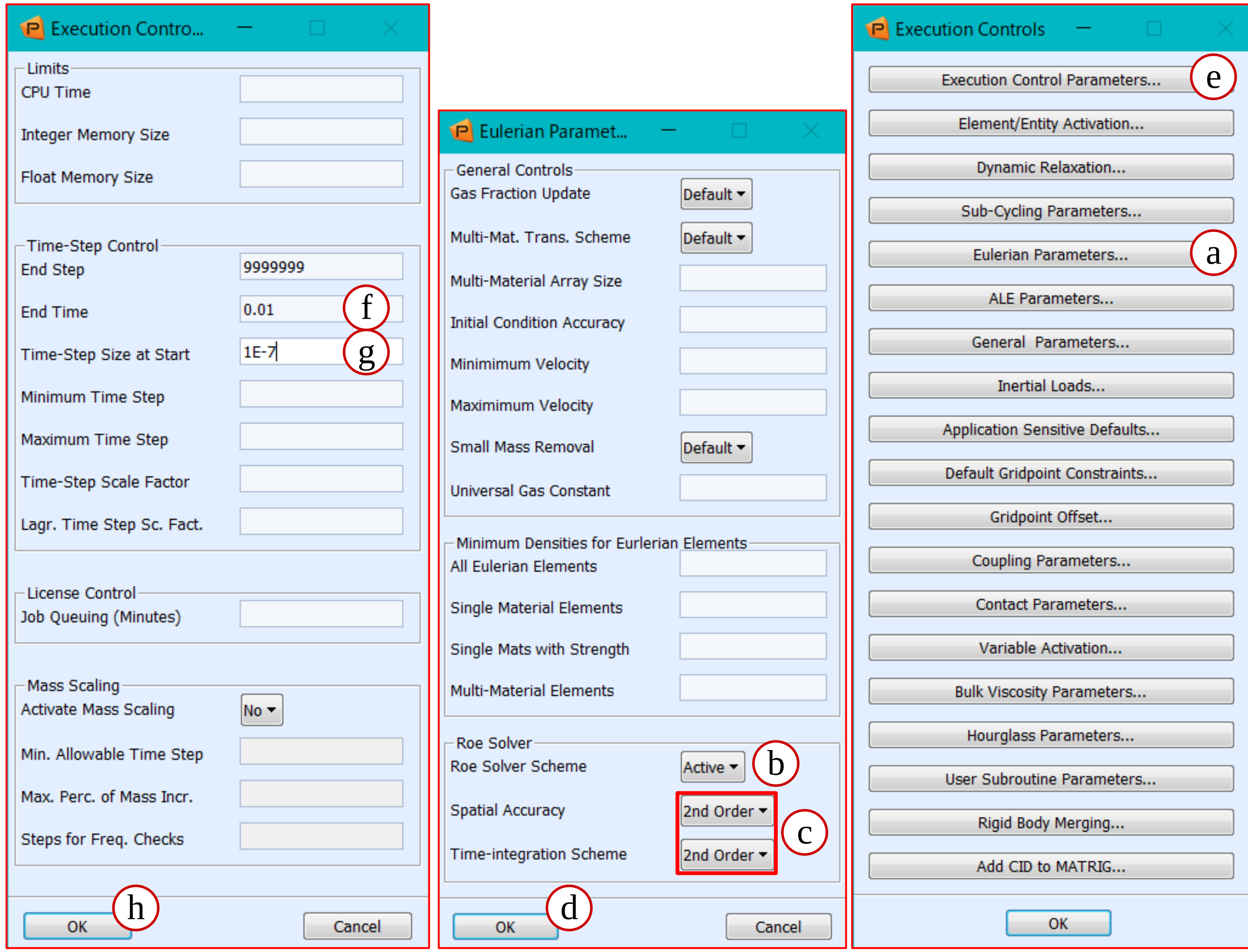




Step 21. Set Up the Analysis (Cont.)

Coupling Surfaces with failure require the Roe solver. Activate the second order Roe solver.

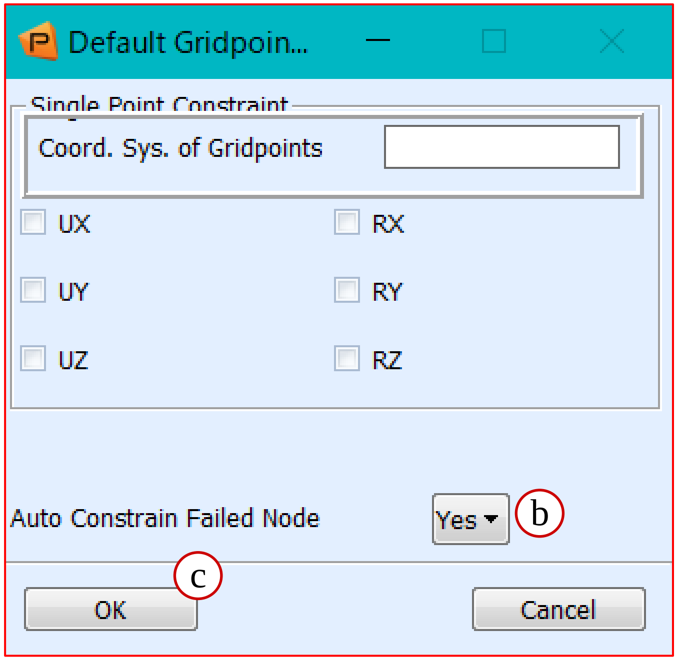
a. Click **Eulerian Parameters**.
b. Set *Roe Solver Scheme* to **Active**.
c. Set both *Spatial Accuracy* and *Time-integration Scheme* to **2nd Order**.
d. Click **OK**.
e. Click **Execution Control Parameters**.
f. Enter **0.01** for *End Time*.
g. Enter **1e-7** for *Time-Step Size at Start*.
h. Click **OK**.

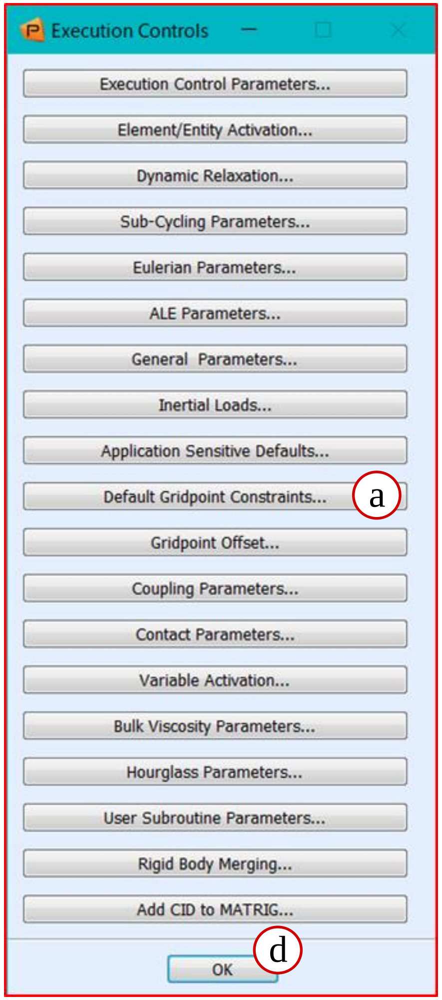


Step 21. Set Up the Analysis (Cont.)

Enable automatic constraints for failed nodes.

a. Click **Default Gridpoint Constraints**.
b. Set *Auto Constrain Failed Node* to **Yes**.
c. Click **OK**.
d. Click **OK**.

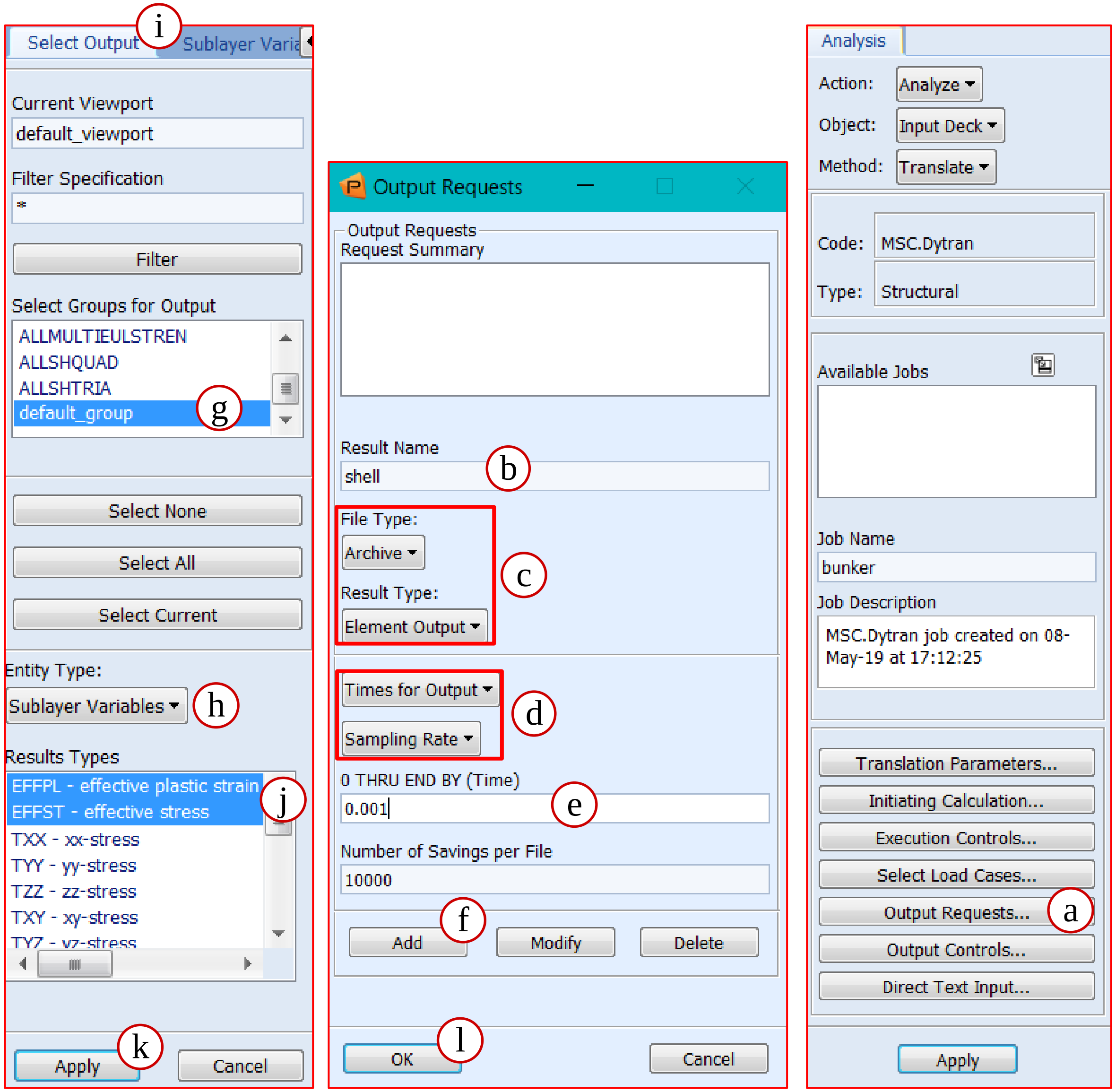




Step 21. Set Up the Analysis (Cont.)

Set the output requests for shell elements.

a. Click **Output Requests**.
b. Enter the *Result Name* **shell**.
c. Set *File Type* to **Archive** and *Result Type* to **Element Output**.
d. Set to **Times for Output** and **Sampling Rate**.
e. Enter **0.001** for *0 THRU END BY (Time)*.
f. Click **Add**.
g. Select the **default_group**.
h. Select **Sublayer Variables**.
i. Click **Select Output** to return to the Select Output tab.
j. Select **EFFPL** and **EFFST**.
k. Click **Apply**.
l. Click **OK**.



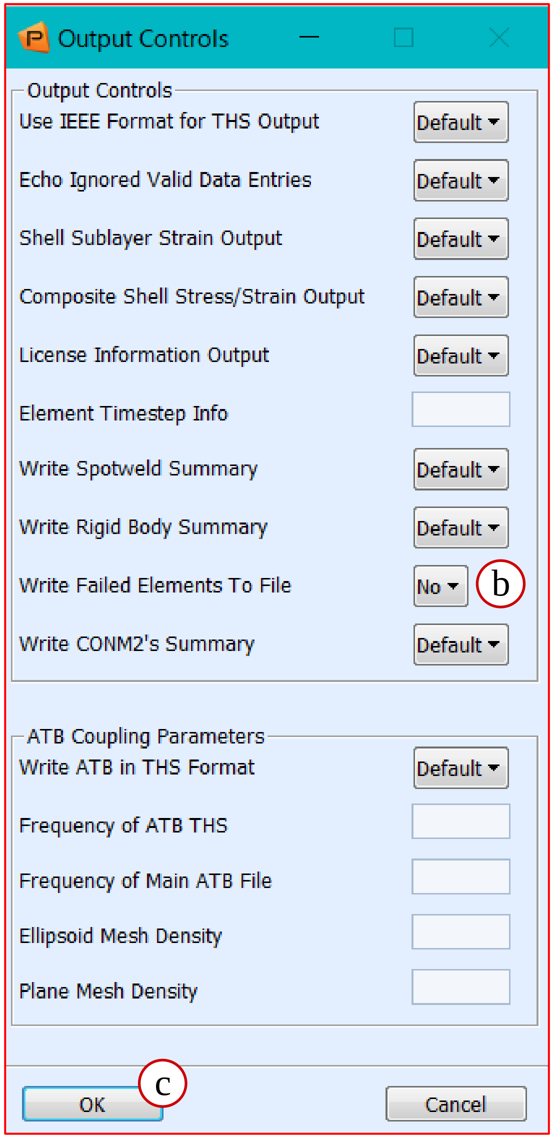
Step 21. Set Up the Analysis (Cont.)

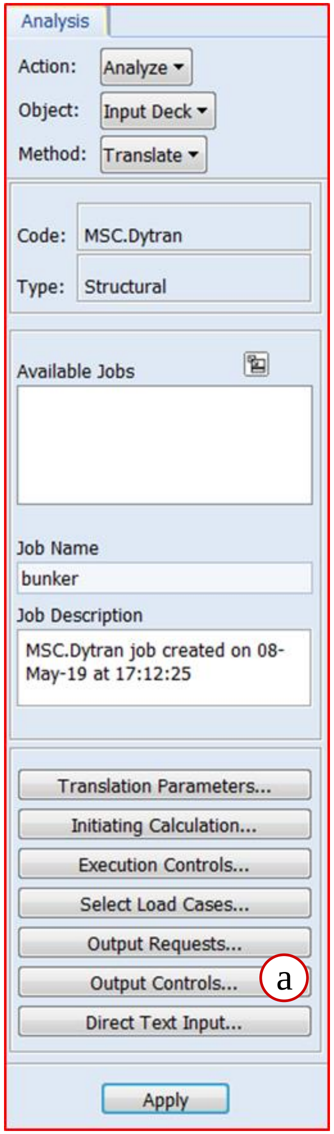
Failed elements are not be written to the archives.

a. Click **Output Controls**.

b. Set *Write Failed Elements to File* to **No**.

c. Click **OK**.





Step 21. Set Up the Analysis (Cont.)

Enter the output request for euler elements:

a. Click **Direct Text Input**.

b. Select **File Management Section**.

c. Enter as shown:

TYPE (AIR) = ARCHIVE

ELEMENTS (AIR) = 14

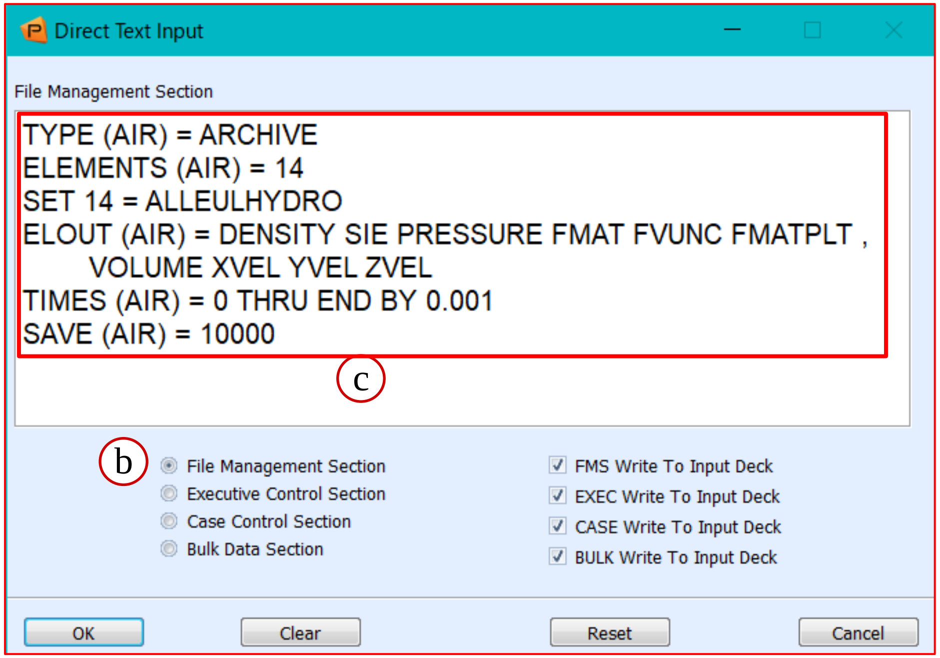
SET 14 = ALLEULHYDRO

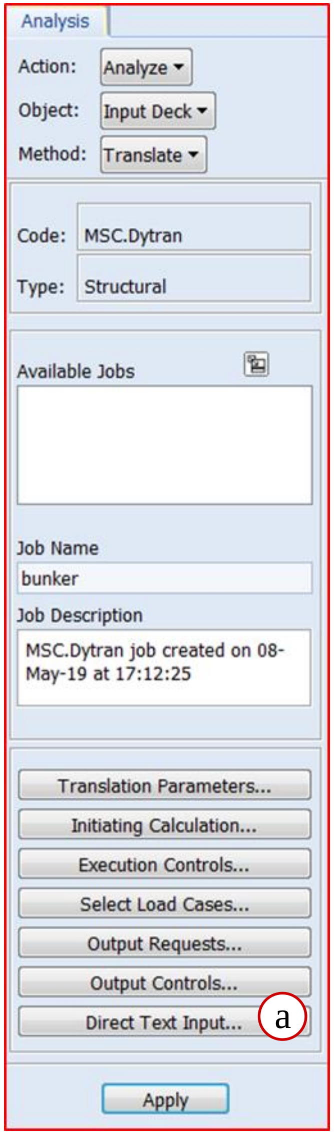
ELOUT (AIR) = DENSITY SIE PRESSURE FMAT FVUNC FMATPLT ,

VOLUME XVEL YVEL ZVEL

TIMES (AIR) = 0 THRU END BY 0.001

SAVE (AIR) = 10000





Step 21. Set Up the Analysis (Cont.)

Enter the porosity for the sides of the bunker and FLOWDEF. The sides are fully porous.

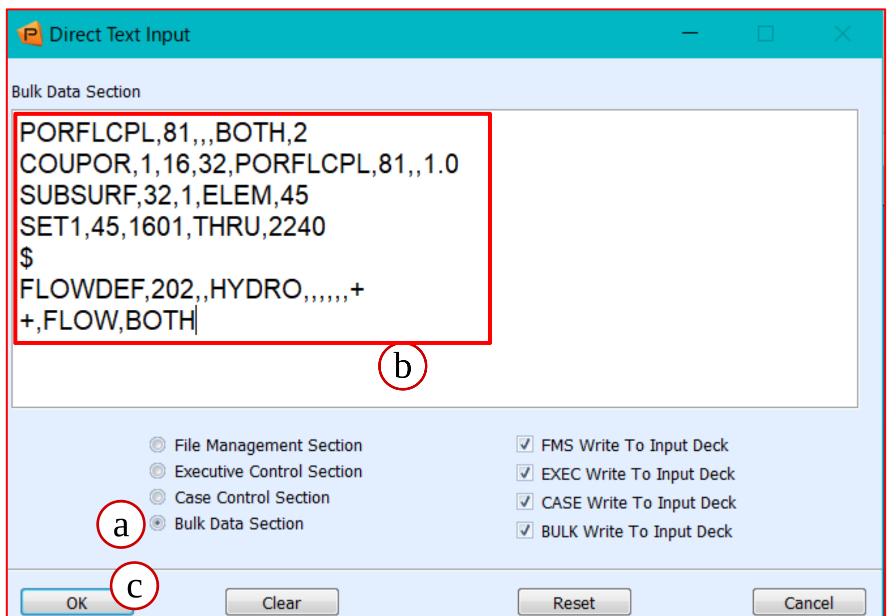
a. Select **Bulk Data Selection**.

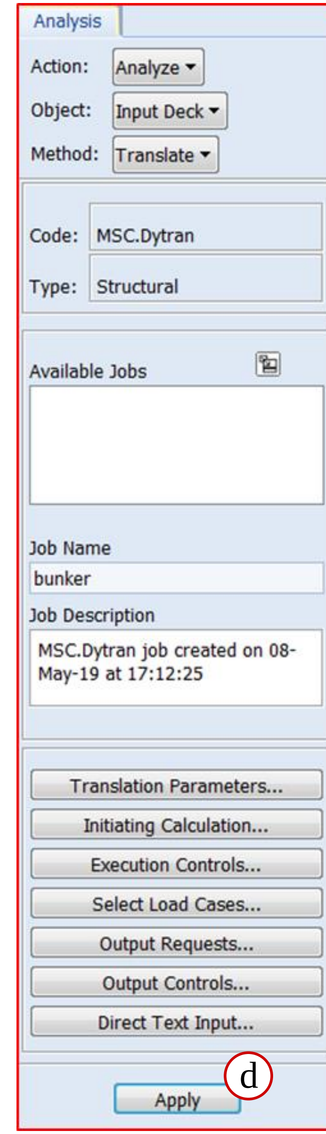
b. Enter as shown:

    PORFLCPL,81,,,BOTH,2
    COUPOR,1,16,32,PORFLCPL,81,,1.0
    SUBSURF,32,1,ELEM,45
    SET1,45,1601,THRU,2240
    $
    FLOWDEF,202,,HYDRO,,,,,,+
    +,FLOW,BOTH

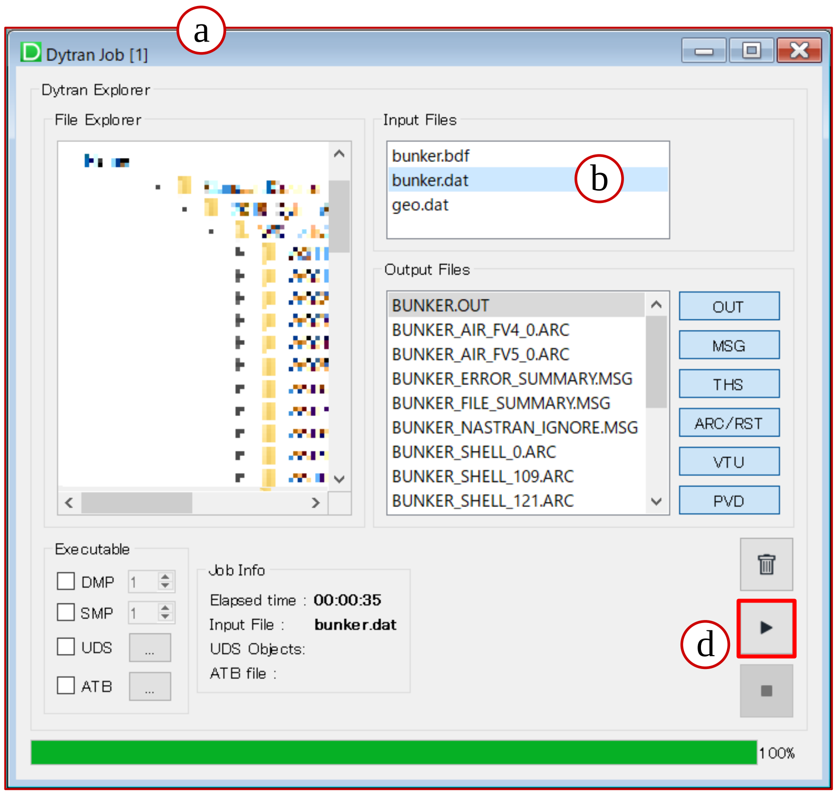
c. Click **OK**.

d. Click **Apply**.





Step 22. Dytran Analysis



Open Dytran Explorer, modify the .dat file and execute the job.

a. Open **Dytran Explorer**.

b. Browse to your working directory and double-click on **bunker.dat**. This will open bunker.dat in a text editor.

c. Add a value of **16** to the *PORID* field in the *Couple1: interact* entry. Save and close **bunker.dat**.

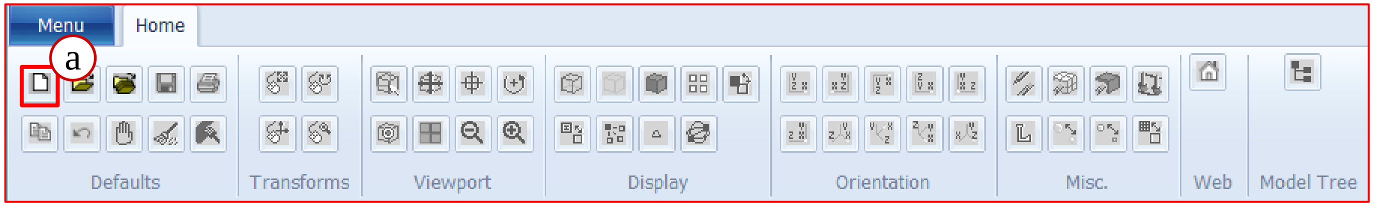
d. Click Play to run the analysis.

```
$ ------- Couple1: interact
$
COUPLE          1         1 OUTSIDE       ON        ON                          +
+                                                                               +
+                         4               1
$
```

c

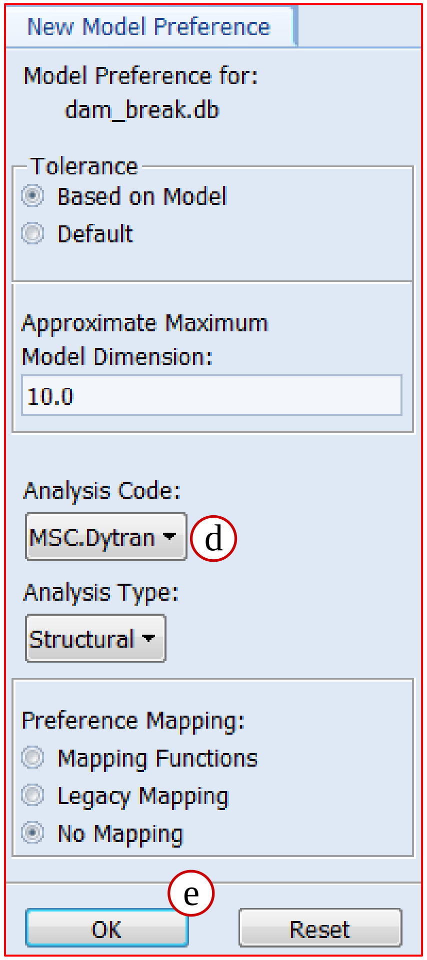
```
$ ------- Couple1: interact
$
COUPLE          1         1 OUTSIDE       ON        ON        16                +
+                                                                               +
+                         4               1
$
```

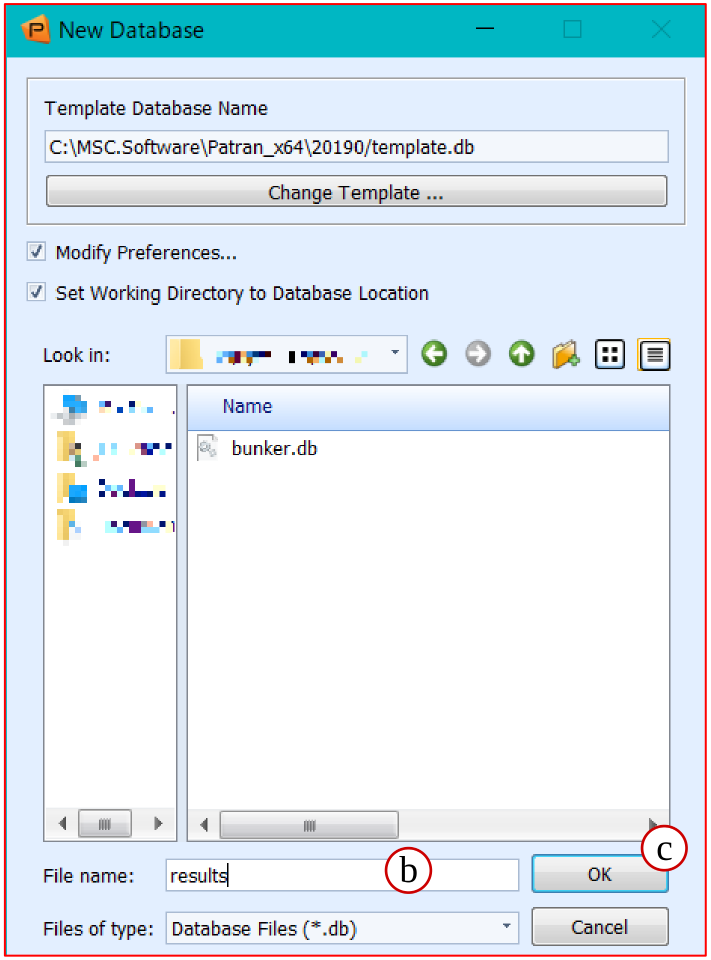
Step 23. Post-Processing in Patran



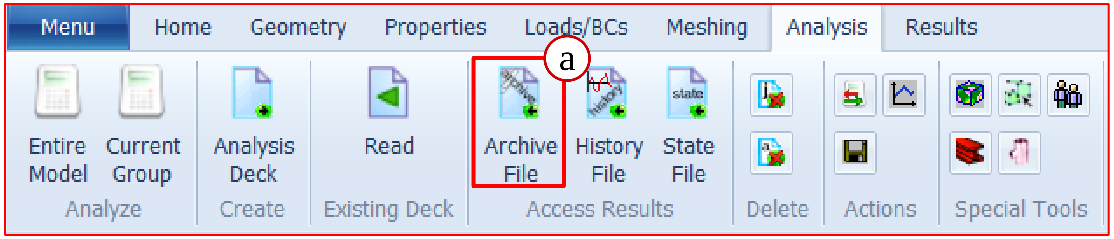
Create a new database to import the results into.

a. Under the *Home* tab, click **New** in the *Default* group.
b. Enter the *File name* **results**.
c. Click **OK**.
d. Select **MSC.Dytran** for *Analysis Code*.
e. Click **OK**.



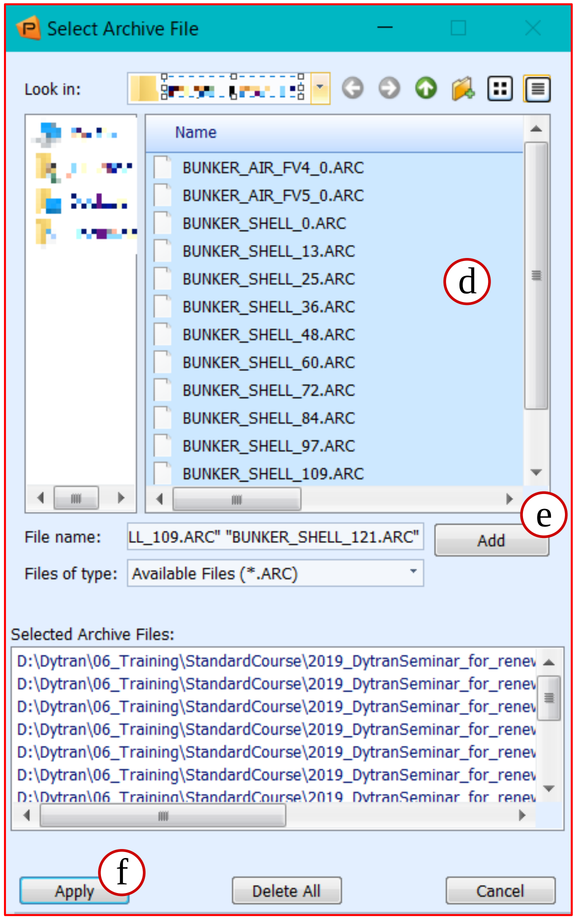


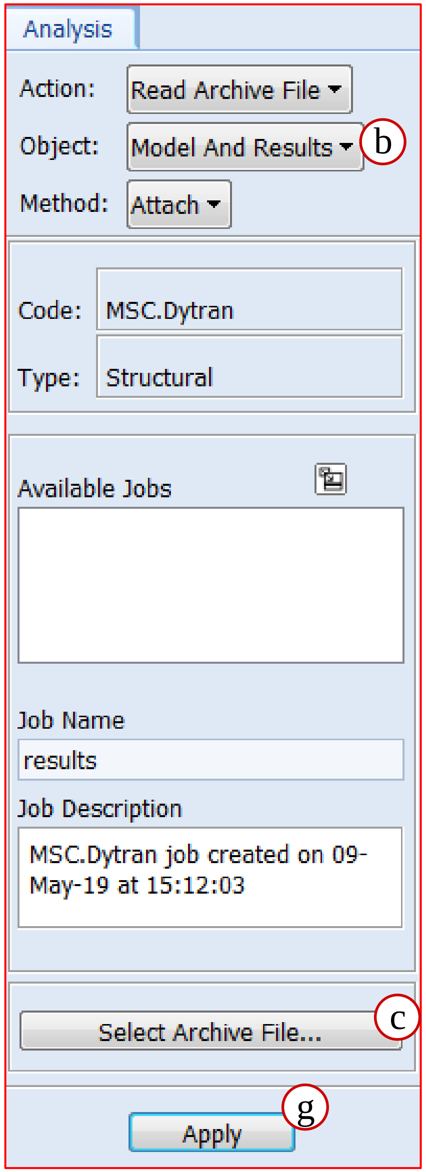
Step 23. Post-Processing in Patran (Cont.)



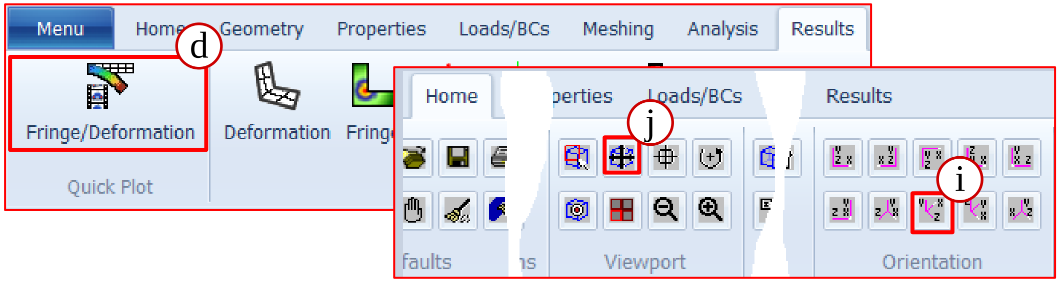
Import the model and the archive files into Patran.

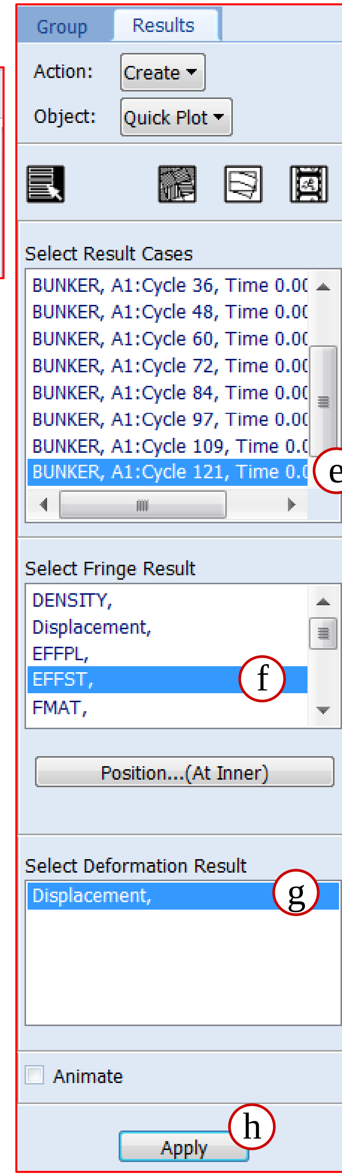
a. Under the *Analysis* tab, click **Archive File** in the *Access Results* group.
b. Set *Object* to **Model and Results**.
c. Click **Select Archive File**.
d. Select all the .ARC files.
e. Click **Add**.
f. Click **Apply**.
g. Click **Apply**.

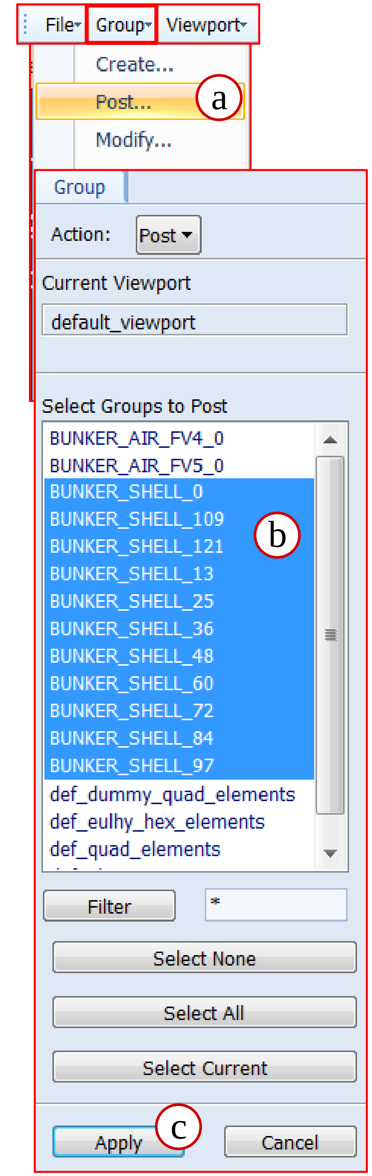




Step 23. Post-Processing in Patran (Cont.)







a. Click **Post** under *Group menu*.
b. Select **BUNKER_SHELL_*** in the tree of the Group.
c. Click **Apply**. Then Click **OK** in the dialogue for *Current Group Selection*.
d. Under the *Results* tab, click **Fringe/Deformation** on the Quick Plot.
e. Select last result case in the **Select Result Cases.**
f. Select EFFST in the **Select Fringe Result**.
g. Select **Displacement** in the Select Deformation Result.
h. Click **Apply.**
i. Click **Iso 2 View**.
j. Click **Fit view**.