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

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

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

Menu Home		
Defaults Transforms Viewport Di	splay Orientation	Misc. Web Model Tree
[New Model Preference	P New Database -
 Create a new database named bunker. a. Under the <i>Home</i> tab, click New in the <i>Default</i> group. b. Enter the <i>File name</i> bunker. 	Model Preference for: dam_break.db Tolerance Based on Model	Template Database Name C:\MSC.Software\Patran_x64\20190/template.db Change Template
c. Click OK.	© Default	 Modify Preferences Set Working Directory to Database Location
d. Select MSC.Dytran for <i>Analysis</i> <i>Code</i> .	Approximate Maximum Model Dimension:	Look in:
e. Click OK.	10.0	Name
	Analysis Code: MSC.Dytran ▼ d Analysis Type: Structural ▼	
	Preference Mapping: Mapping Functions Legacy Mapping No Mapping	File name: bunker
	OK Reset	Files of type: Database Files (*.db)

Step 2. Read the Input File

Menu Home Geometry Properties Loads/BCs Me	eshing Analysis Results	Analysis
		Action: Read Input File
Entire Current Analysis Read Archive History Sta	ate 👔 🔲 📚 🖑	Object: MSC.Dytran
Analyze Create Existing Deck Home Geometry	Properties Loads/BCs Meshing Analysis Results	
6 2 5 8 8		
) 🗸 💉 S ^a		Code: MSC.Dytran
Read the geo.dat input file.	ns Viewport Display Orientation	Type: Structural
a Under the Analysis tab, click Read in the		
Existing Deck group.		
h Click Select Input File	😢 Select File — 🗆 🛛	Available Jobs 🖺
c. Select appendiat		
	Look in: 📕 🕂 🕂 🕂 💶 🍸 ઉ 🛇 🚱 🙀 🖽 🔳	
	Name	
e. Click Apply.	🚦 🐜 🗤 🙀 geo.dat 🕐	
f. Under the Home tab, click Iso 2 View .	🚡 Satur	Job Namo
g. Click Smooth shaded.	Di secon	hunker
h. Click Fit view.		Jak Description
		Job Description
		MSC.Dytran job created on 07- May-19 at 17:45:40
		Select Input File
	File name: geo.dat OK	\frown

Files of type: Files (*.dat)

e

Apply

Cancel

Ŧ

Step 3. Create Solids for 3D Property

Home Geometry Properties Loads/BCs Meshing Analysis Menu ് T Select Select Select Select Sele Geometry Points Curves Surfaces Solids Coordi Create -Action: 0 a Primitive Create dummy solids that are used in Object: Solid 🔻 Surface D creating 3D property sets. Create one solid Method: Primitive -D for each couple surface. Any solid will suffice. B-rep Decompose a. Under the Geometry tab, click Select > Face Primitive in the Solids group. Ħ Vertex b. Click twice on **Apply**. This will create two Solid ID List t. XYZ solids. Solid 1 will be used by coupling 1 7 Extrude surface1 and Solid 2 by coupling Block Parameters 7 Glide surface2. X Length List 8 Normal 1.0 1th Revolve Y Length List 1.0 Z Length List 1.0 Modify Solid Boolean Operation... Refer, Coordinate Frame Coord 0 Auto Execute Base Origin Point List [0 0 0] b -Apply-

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.

1D Properties 2D Proper	Solid Solid	Property Action	Method: Manual Input 🕶
			Existing Materials
Pinput Options	—		
Constitutive Model:	Ideal Gas (DMAT) 🔻		
Valid For:	Eulerian Solid (Hydro) 🔻		
Property Name	Value		
Density =	1.2E-7	e	
Specific Heat Ratio (GAMMA) :	= 1.4		
Gas Constant (R) =			Filter *
Spec. Heat at Const. Volume =	=		
Spec. Heat at Const. Pressure	=		
Viscosity Coefficient =			gas b
1			
			Description
			Date: 07-May-19 Time: 18:15:30
Current Constitutive Models:			
			Input Properties C
			Change Material Status
ок	Clear	Cancel	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.

Element Properties	Element Properties
Action: Create -	Action: Create
Dbject: 3D ▼	Object: 3D -
ype: Eulerian Solid ▼	Type: Eulerian Solid ▼
ets By: Name ▼ 🖺 Pset1.1 Pset2.2 Pset3.3 poute1	Sets By: Name Pset1.1 Pset2.2 Pset3.3
Filter *	Filter *
Property Set Name peuler2 e	Property Set Name peuler1 b
ptions: Hydro (PEULER1) v f	Options: Hydro (PEULER1) V
Input Properties	Input Properties
Select Application Region	Select Application Region
Apply g	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.
- Enter failcs1 for the New Set Name. C.
- Click Input Data. d.
- Set Cover to Outside. е.
- Enter 1.2e-7 for Environmental Density and 3e8 for Environmental f. Specific Internal Energy and <0 0 0> for Flow Boundary Velocity.
- Click OK. q.

r Coupling Airbag Fluid 🥜 🔒	object:	Coupling -
Fill Contact Initial Condit	Type:	Element Uniform 🔻
· · · · · · · · · · · · · · · · · · ·	Option:	With Failure - b
	-Current	Load Case:
		Default
	Туре:	Time Dependent
undary Conditions Input Data 🚺		
Cover: Outside ▼ e	Existing S	Gets 🖺
 Reverse Normals Check Normals Constant Coefficients Environmental Density 1.2E-7 Environmental Specific Internal Ener 3E8 	New Set	Name
Flow Boundary Velocity <vx, vy,="" vz<br=""><0 0 0> Deactivation Time</vx,>		
OK g Reset	Selec	Input Data

С

d

Load/Boundary Conditions

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

Define the application region for failcs1.

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



		Action: Create -
s Select Application Regions	ns Select Application Regions	Object: Coupling ▼
Form Type:	Form Type:	Type: Element Uniform -
Select Tool 🔻	Select Tool 🔻	Option: With Failure -
		Current Load Case:
Target Euler Elements $-$ (d)	Target Surface 🔻	Default
Element Type 3D -	Element Type 2D 🔻	Type: Time Dependent
- Geometry Filter	Geometry Filter	
Geometry SEM	Geometry I FEM	
(e)		Existing Sets
Application Region Select Entities Solid 1 f Add Remove Surface Definition Element 1:3280 Euler Element Selection	Application Region Select Entities Elem 1:3280 Add C Remove Surface Definition Euler Element Selection	New Set Name failcs1
Do not Click Pre Preview	view at this time Preview OK	Input Data Select Application Regiona



Step 8. Creating the Second Coupling Surface

Menu Home Ge	eometry Prope	erties Loads/BCs	Meshing Analysis	Results	\frown		
D 🖻 🍯 🖬 🖨	8° 87	€ # ⊕ ⊕	1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1	YX XZ YX VX			
🖻 🖻 🖉 🌠	8t 8t	💿 🗄 Q Q	B A		L 🔨 😘 🗳		
Defaults	Transforms	Viewport	Display	Orientation	Misc.	Web	Model Tree

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.

ndary Conditions	Plot/Erase
Selected Entities	
Elem 2241:3280	b
Plot	Erase
Coord. Fr	ames
Posted Entities Geometry	
Plot	Erase
FEM	
Plot	Erase
All	
Plot	Erase
ОК	d

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

Menu Home Geometry Properties Loads/BCs Meshing Analysis Results	Load/Boundary Conditions
	Action: Create
│ 14 → 🐳 🖳 ⇒ ≫ 🖉 😳 💆 🟴 🐯 💁 ¶ 🐼 🕒 ല ≝ ≝	Object: Coupling
Displacement Force Follower Body Velocity BJOIN KJOIN Rotational Detonation Mesh Force For	Type: Element Uniform
Nodal Element Uniform Contact Initial Condition	
	- Option: With Failure -
undary Conditions Input Data	Current Load Case: Default
Define the cover and application region.	Type: Time Dependent
a. Under the Loads/BCs tab, click Coupling in the Element	
Uniform group.	Evicting Cote
b. Enter failcs2.	failcs1
c Click Input Data	
d Select Inside	
C. Select Inside.	
e. Click OK.	
Environmental Specific Internal Ener	
3E8	New Set Name
Flow Boundary Velocity <vx, td="" vy,="" vz<=""><td>failcs2 b</td></vx,>	failcs2 b
<0 0 0>	
Deactivation Time	
	Input Data
	Select Application Region
OK Reset	Apply

-Apply-

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.

Form Type: Select Tool * Action: create * Select Tool * Geometry Filter Object: Coupling * Target Euler Elements * E F Target Surface * Option: Wth Failure * Cometry Filter Geometry Filter Geometry Filter Current Load Case: Default * Geometry Filter Geometry Filter Type: Time Dependent * Select Entities Select Entities Select Entities Failes1 Solid 2 ge Add Remove Select Entities Failes1 Surface Definition Surface Definition Surface Definition New Set Name Failes2 Element Selection I Euler Element Selection New Set Name Failes2 Do not Click Preview at this time Preview Preview Input Data Select Application Region	s Select Application Regions		s Select Application Regions	Load/Boundary Conditions
Select Tool • Object: Coupling • Target Euler Elements • (e) Element Type 3D • Geometry Filter - Geometry Filter Geometry (inter) Geometry (inter) FEM Female Filter Select Entities Select Entities Solid 2 (f) Add Remove Surface Definition Surface Definition Element 1:2240 3413:4012 (f) Add Remove Surface Definition Surface Definition Element Selection New Set Name Fallcs2 Input Data Do not Click Preview at this time Preview Preview	Form Type:	Les I	Form Type:	Action: Create -
Target Euler Elements • • • Element Type 3D • Geometry Filter Geometry Filter • Geometry © FEM © Geometry @ FEM • Application Region Select Entities Select Entities Select Entities Solid 2 g • Add Remove Surface Definition Surface Definition Element 1:2240 3413:4012 N • Ag94:7904 A Euler Element Selection New Set Name fallcs2 New Set Name fallcs2 New Set Name fallcs2 New Set Name fallcs2 New Set Name fallcs2 Select Application Region Euler Element Selection i <	Select Tool -	Ø	Select Tool 🔻	Object: Coupling -
Corrent Load Case: Element Type Bow Geometry Filter Geometry Filter Geometry @ FEM Current Load Case: Default Current Load Case: Default Type: Time Dependent Preview Surface Definition Element 1:2240 3413:4012 Add Remove Surface Definition Element Selection Interpreting Preview Preview Preview Input Data Select Application Region			Target Surface	Type: Element Uniform ▼
Element Type D Geometry Filter Geometry Filter Files Select Entities Solid 2 g Add Remove Surface Definition Element 1:2240 3413:4012 Add Euler Element Selection Do not Click Preview at this time Preview Preview Preview Input Data Select Application Region Current Load Case: Default Type: Time Dependent Existing Sets Failcs1 New Set Name Failcs2 Input Data Select Application Region Current Load Case: Default Type: Time Dependent Existing Sets Failcs1 New Set Name Failcs2 Input Data Select Application Region Current Load Case: Default Type: Time Dependent Existing Sets Failcs1 New Set Name Failcs2 Input Data Select Application Region Current Load Case: Default Type: Time Dependent Existing Sets Failcs1 New Set Name Failcs2 Input Data Select Application Region Current Load Case: Default Type: Time Dependent Existing Sets Failcs1 New Set Name Failcs2 Input Data Select Application Region Current Load Case: Default Type: Time Dependent Existing Sets Failcs1 New Set Name Failcs2 Input Data Select Application Region Current Load Case: Default Type: Time Dependent Failcs1 Select Application Region Current Load Case: Default Type: Time Dependent Failcs1 Select Application Region Current Load Case: Type: Time Dependent Failcs1 Select Application Region Current Load Case: Sele		+		Option: With Failure ▼
Geometry Filter Geometry Filter Geometry Filter Type: Time Dependent Type: Time Dependent Type: Time Dependent Type: Time Dependent Type: Time Dependent Failcs1 Evisting Sets Failcs1 Evisting Sets Failcs2 Failcs2 Evisting Sets Failcs1 Evisting Sets Failcs1 Evisting Sets Failcs2 Failcs2 Evisting Sets Failcs2 Fail	Element Type 3D 🔻	-	Element Type 2D 🔻	Current Load Case:
Application Region Select Entities Solid 2 Add Add Remove Surface Definition Element 1:2240 3413:4012 4894:7904 Euler Element Selection Do not Click Preview at this time Preview Input Data Select Application Region Existing Sets Failcs1 Existing Sets Failcs1 New Set Name Failcs2 Input Data Select Application Region C Surface Definition Element Selection Input Data Select Application Region C Surface Definition Euler Element Selection Input Data Select Application Region C Surface Definition Element Selection Input Data Select Application Region C Surface Definition Euler Element Selection Input Data Select Application Region C Surface Definition Element Selection Input Data Select Application Region C Select Application Region C Select Application Region C Surface Definition Element Selection Input Data Select Application Region C Select Application Region Select	Geometry Filter Geometry © FEM	=	⊂Geometry Filter ◎ Geometry ⑧ FEM	Type: Time Dependent
Application Region Existing Sets Select Entities Select Entities Solid 2 g Add Remove Surface Definition Add Element 1:2240 3413:4012 Surface Definition Euler Element 3election Ad Image: Select Entities Surface Definition Euler Element Selection New Set Name Image: Select Entities Failes2 Do not Click Preview at this time Input Data Preview Preview			<u>ب</u>	
Select Entities Solid 2 g Add Remove Surface Definition Element 1:2240 3413;4012 4095:4340 4505;4709 4894:7904 Ad Euler Element Selection II Euler Element Selection Preview Preview Preview Preview Preview Preview Preview	Application Region	Ľ	Application Region	Existing Sets
Solid 2 g c 15:4340 4505:4709 4894:7904 Add Remove Surface Definition Surface Definition Surface Definition Element 1:2240 3413:4012 Image: Surface Definition Add Remove Surface Definition Image: Surface Definition Euler Element Selection Image: Surface Definition Do not Click Preview at this time Failcs2 Input Data Select Application Region Image: Surface Definition Image: Surface Definition	Select Entities	-	Select Entities	failcs1
Add Remove Surface Definition Surface Definition Element 1:2240 3413:4012 Surface Definition 4894:7904 A Euler Element Selection Euler Element Selection II Euler Element Selection Preview Preview II Preview II Preview II II	Solid 2 g		15:4340 4505:4709 4894:7904	
Surface Definition Image: Surface Definition Element 1:2240 3413:4012 Image: Surface Definition 4095:4340 4505:4709 Image: Surface Definition A Image: Surface Definition Euler Element Selection New Set Name failcs2 Do not Click Preview at this time Image: Surface Definition Preview Preview Image: Surface Definition Image: Surface Definition Image: Surface Definition New Set Name failcs2 Image: Do not Click Preview at this time Image: Surface Definition Image: Preview Preview Image: Surface Definition Image: Surface Definition Image: Surface Definition Image: Surface Definition Image: Surface Definition New Set Name Image: Surface Definition Image: Surface Definition Image: Surface Definition Image: Surface Definition <td< td=""><td>Add Remove</td><td>C 🗄</td><td>Add d Remove</td><td></td></td<>	Add Remove	C 🗄	Add d Remove	
Surface Definition Element 1:2240 3413:4012 4095:4340 4505:4709 4894:7904 A II Euler Element Selection II Euler Element Selection Preview Preview II Preview II II Euler Element Selection III Euler Element Selection IIIII Euler Element Selection IIIIIIIIIIIIIIIIIIIIIIIIIIIIIIIIIIII	Surface Definition	₫	Curferer Definition	
Element 1:2240 3413:4012 4095:4340 4505:4709 4894:7904 Euler Element Selection Do not Click Preview at this time Preview Input Data Select Application Region ()			Surface Definition	
4894:7904 A Euler Element Selection Euler Element Selection Do not Click Preview at this time Preview Preview Image: Select Application Region	Element 1:2240 3413:4012	×		
Euler Element Selection New Set Name Do not Click Preview at this time Input Data Preview Preview Input Data	4894:7904	х		
Image: Construction Image: Construction Image: Construction Image: Construction Do not Click Preview at this time Preview Preview Image: Construction Image: Construction Image: Construction Image: Construction </td <td>Eular Element Selection</td> <td></td> <td>Fular Element Colortion</td> <td>New Set Name</td>	Eular Element Selection		Fular Element Colortion	New Set Name
Do not Click Preview at this time Preview i		耳	Euler Element Selection	failcs2
Do not Click Preview at this time Preview Preview i i				
Do not Click Preview at this time Preview Input Data Select Application Region				
Preview Preview Input Data Select Application Region	Do not Click P	revie	w at this time	
i) Select Application Region.	Preview		Preview	Input Data
(i)				Select Application Region
	i			(j)



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.

ns Select Application Regions	Load/Boundary Conditions
Coupling Surface 1	Action: Create Object: Coupling
failcs1	Type: Element Uniform
Coupling Surface 2 d	
failcs2	
Available Coupling Surfaces	Default
failcs1	Type: Time Dependent
failcs2	
	Existing Sets
оке	
	New Set Name
	Input Data
	Select Application Region
	-Apply-

Step 10. Euler Mesh For the First Coupling



Step 11. Euler Mesh for the Second Coupling



Step 12. Initialization of Euler Mesh/Shapes

odal

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.
- Enter Radius = 10000. 1.
- Select Coord1. Ι.
- k. Click OK.
- Click Apply. I.

					_					
Loads/BCs Mesh		Load/Bo	undary Conditions		е	Geom	etry	Properties	Loads/BCs	Meshing
ا رې 🔦 🤃	≝ € * ×	Action: Object:	Create		C		6	2.	1	1
BJOIN KJOIN Rotz		Type:	Element Uniform 🔻		Se	elect •	Selec	t Select	Select	Select •
Bot -	tial Conditions	Option:	Shape • (f)		Sur	faces	Solid	s Coordina	tes Planes	Vectors
							2	J 3Poin	t	
St onditions Input Da	ata (Shape)							斗 🗛		
Shape:	Sphere -						2	⊁ Euler	a	
								上 Norm	al	
ect Radius of Sphere		Existing	Sets				1	21/00	tor	
							Geo	meury		
							Acti	on: Creat	te 🔻	
							Obje	ect: Coor	d 🕶	
Centroid							Met	hod: Euler	-	
Coord 1						·	Coo	rd ID List		
Proviou							1			
Freview	v						Туре	: Red	tangular 🔻	
V OK K	Reset	New Set	Name				Def	ion. Coondin	ata Evana	
		snpere_					Coc	er. Coordin ord 0	ate Frame	
								Rotation	Parameters	
								uto Execute		
			Input Data	h			Orig	gin		
		Sele	ct Application Regior				[-5	3.4 100 -67	3.6]	(c)
			-Apply-					-	Apply-d	

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.

Loads/BCs Mesh	Load/Boundary Conditions	Seometry Properties Loads/BCs Meshing
Image: Second system Image: Second system BJOIN KJOIN Rotz Bot Initial Conditions	Action: Create Object: Init. Cond. Euler Type: Element Uniform Option: Shape e	ect Select Select Select Select Select Vectors
Shape: Sphere -		↓ Axis ★ Euler ↓ Normal
Radius of Sphere	Existing Sets 🖺	Geometry Action: Create Object: Coord Method: Euler
Centroid Coord 2 i Preview OK Reset	New Set Name shpere_eul2_all f	Coord ID List 2 Type: Rectangular Refer. Coordinate Frame Coord 0 Rotation Parameters
	Input Data g Select Application Region	Auto Execute Origin [-536.4 165 -453.6] b -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

Se

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.

Input Data (Initial Values)	Load/Boundary Conditions
elect Euler Material	Action: Create -
as d	Object: Init. Cond. Euler ▼
	Type: Element Uniform -
	Option: Initial Values 🕶 (a)
itial Values	
< 0 0 0 >	
ensity	Evisting Sets
3.84E-6 (e) ■	
Specific Internal Energy	
3E9	
OK f Reset	
Neset	
	New Set Name
	initblast b
	Input Data
	Select Application Region
	-Apply-
	-Арріу-

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.
- I. Click Add Row.
- m. Click OK.
- n. Click Apply.



Materials Home Geometry Properties Loads/BCs Meshing Analysis Menu Results a Action: Create -858 P 386 XX. A 155 ~ Þ Ś Ċ Ċ Object: Isotropic 🔻 Isotropic 2D 戴戴 🖉 🖂 💊 Lagrangian Eulerian 8 Method: Manual Input -Solid Solid Orthotropic Anisotropic Composite **OD** Properties **1D** Properties 2D Properties **3D Properties** Isotropic 5 Existing Materials gas ៧ Input Options Create the material for the ground. a. Under the *Properties* tab, click **Isotropic**. Constitutive Model: Rigid (MATRIG) 🔻 (d) Enter matrigid for Material Name. Valid For: Shell 🔻 Click Input Properties. **Rigid Body Properties:** Geometry -Set Constitutive Model to Rigid (MATRIG), Property Name Value Valid For to Shell and Rigid Body Properties to 0.000734 Density = Geometry. e Elastic Modulus = 2.9E7 * Filter e. Enter 0.000734 for Density, 2.9e7 for Elasticity 0.3 Poisson Ratio = Modulus and 0.3 for Poisson Ratio. Mass = Click OK. Material Name b matrigid Click **Apply**. Description Date: 07-May-19 Time: 18:15:30 Current Constitutive Models: С Input Properties ... Change Material Status ... f g OK Clear Cancel Apply

Step 16. Create Material for Ground

b.

C.

d.

f.

q.

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.

P Input Options	– 🗆 🗙	Materials
Constitutive Model: Valid For: Yield Model: Strain Rate Model: Failure Model: Property Name	ElasPlas (DMATEP) • Shell • Von Mises • None • Max.Pla.Strain •	Action: Create Object: Isotropic Method: Manual Input Existing Materials gas matrigid
Density = Elastic Modulus = Poisson Ratio = Shear Modulus = Bulk Modulus = Yield Stress = Maximum Plastic Strain = Max. Comp. Plastic Strain =	0.000734 2.9E7 0.3 50000 0.21	Filter * Material Name steel a Description
Current Constitutive Models:	Clear Cancel	Date: 07-May-19 Time: 18:15:30 Time: Input Properties b Change Material Status

Menu Home Geometry Properties L	oads/BCs Meshing Analysis Results		
Isotropic 2D 🗾 🏠		Element Properties	ian 🏔 🔯 💥 🖀 📾 😹 🛛
Isotropic Orthotropic Anisotropic Comp	osite 0D Properties 1D Properties	Action: Modify -	Property Actions Fields
Change the dummy challe of eliput Prope	t <u>i</u> es — 🗆 X	Object: 2D ▼	Element Properties
Property 3 into real shells.	Value Value Type	Type: Shell • d	Action: Modify
a. Click Modify Property on	*		Object: 2D -
the Property Actions	Properties	Sets By: Name ▼ 🖺	Type: Dummy Shell - (b)
group.	Default PSHELL (CQUAD4)	Pset2.2	
Dummy Shell.	Property Name Value	Pset3.3	Sets By: Name 🔻 🖺
c. Select Pset3.3 and click	Material Name m:matrigid		Pset1.1
OK to close Input	[Material Orientation]	Filter *	Pset2.2 Pset3.3 C
Properties.	Thickness 0.1 I	New Property Set Name	
d. Pull down <i>Type</i> to Shell .	[nourgiass suppr.mem.]	Pset3.3	
e. Click matrigid.	[Warp.Hourgl.Damp.Coeff.]		Filter *
Click OK	[Twist.Hourgl.Damp.Coeff.]	Options:	New Property Set Name
h Click Apply	4	Homogeneous •	Pset3.3
		Default (PSHELL) 🔻	
	gas e	Modify Properties	Modify Properties
	steel		Select Application Region
		Select Application Region	
	OK Clear	Apply	Apply

Step 18. Modify Properties for Ground

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.

🖻 Input Properties 🛛 🚽 🗆	Element Properties	
No Input Required Property Name Value Value Value Type	Action: Modify -	Element Descertion
	Object: 2D ▼	Element Properties
	Type: Shell • d	Action: Modify Object: 2D
\mathbf{I}	Sets By: Name 🕶 🖺	Type: Dummy Shell • b
D Input Properties	Pset1.1	
Default PSHELL (COUAD4)	Pset3.3	
Property Name Value	1.500.5	Sets By: Name
		Pset1.1
Material Name m:steel		PSet2.2
[Material Orientation]	Filter *	1360.5
Thickness 0.15 g	New Property Set Name	
[Hourglass Suppr.Meth.]	Pset1.1	
[Inpl.Hourgl.Damp.Coeff.]		Filter *
[Warp.Hourgl.Damp.Coeff.]	Options:	New Prenerty Cet Name
[Twist.Hourgl.Damp.Coeff.]	Homogeneous	New Property Set Name
		PSet1.1
	Default (PSHELL)	
Materials gas (f)	Modify Properties e	Modify Properties
steel	Select Application Region	Select Application Region
OK Clear	Apply	Apply



Step 20. Create Rigid for Ground

Step 21. Set Up the Analysis

	Menu Ho	me Geom	etry Properti	es Loads/	BCs Mesh	ing Ar	nalysis	Re	esults			Analysis	1	
				×	state		5	Ŀ.	Execution Cont	trols — 🗆 🗙		Action:	Analyze ▼ Input Deck ▼	
	Entire Current Model Group	Analysis Deck	Read	Archive Hi File	istory State File File				Execution	Control Parameters		Method:	Translate 🔻	
	Analyze	Create	Existing Deck	Access	s Results	Delete	Act	ions	Element	/Entity Activation				
									Dyna	mic Relaxation		Code: M	SC.Dytran	
S a	Set the execution controls and coupling parameters for the job.						Sub-Cy	cling Parameters		Type: St	ructural			
group.						Eulerian Parameters			- h -	5				
b. Click Execution Controls.						ALE	Parameters		Avallable J	IODS				
C.	Click Coup	ling Par	ameters.						Gener	al Parameters				
d. Set Fast Coupling and Coupling Surface Failure to Active.							ertial Loads							
e	e. Click OK.							Application	n Sensitive Defaults		Job Name			
									Default Gr	idpoint Constraints		bunker		
									Grid	Inoint Offset		Job Descri	ption	
									GIII	apoint on octin		MCC Dute	an ich created	1 on 00



Analys	is						
Action:	Analyze 🔻						
Object:	Input Deck 🔻						
Method	I: Translate ▼						
Code:	MSC.Dytran						
Туре:	Structural						
Availab	le Jobs 🖺						
Job Nar	me						
bunker	·						
Job Des	scription						
MSC.D May-1	Oytran job created on 08- 9 at 17:12:25						
Tr	anslation Parameters						
I	initiating Calculation						
Execution Controls (b)							
Select Load Cases							
Output Requests							
	Output Controls						
	Direct Text Input						
	Apply						

c

Coupling Parameters...

Contact Parameters...

Variable Activation...

Bulk Viscosity Parameters...

Hourglass Parameters... User Subroutine Parameters...

Rigid Body Merging...

Add CID to MATRIG ...

OK

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.

Recution Contro	- 🗆 🗙			e Execution Controls — 🗌
- Limits CPU Time				Execution Control Parameters
Integer Memory Size		el Eulerian Paramet	– 🗆 🗙	Element/Entity Activation
Float Memory Size		General Controls		Dynamic Relaxation
		Gas Fraction Update	Default 🔻	Sub-Cycling Parameters
Time-Step Control		Multi-Mat. Trans. Scheme	Default 🔻	Eulerian Parameters
End Step	9999999	Multi-Material Array Size		
End Time	0.01 (1)	Initial Condition Accuracy		
Fime-Step Size at Start	1E-7 (g)	Minimimum Velocity		General Parameters
Minimum Time Step		Maximimum Velocity		Inertial Loads
/laximum Time Step		Small Mass Removal	Default	Application Sensitive Defaults
ime-Sten Scale Factor		Sinai Mass Removal	Derault	Default Gridpoint Constraints
interstep scale Pactor		Universal Gas Constant		Cridpoint Offset
.agr. Time Step Sc. Fact.		Minimum Densities for Eurler	ian Elements	
Lisses Control		All Eulerian Elements		Coupling Parameters
Job Queuing (Minutes)		Single Material Elements		Contact Parameters
		Single Mats with Strength		Variable Activation
Mass Scaling		Multi-Material Elements		Bulk Viscosity Parameters
Activate Mass Scaling	No 🔻			Hourolass Parameters
Min. Allowable Time Step		Roe Solver Scheme	Active - b	
Max. Perc. of Mass Incr.		Spatial Accuracy	2nd Order	User Subroutine Parameters
Steps for Freq. Checks			(c)	Rigid Body Merging
		I ime-integration Scheme	2nd Order 🔻	Add CID to MATRIG
(h)		(d)		

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.

	Eulerian Parameters
	ALE Parameters
🖨 Default Cridnain — 🗆 🗸	General Parameters
	Inertial Loads
Single Point Constraint	Application Sensitive Defaults
Coord. Sys. of Gridpoints	Default Gridpoint Constraints a
	Gridpoint Offset
	Coupling Parameters
🗆 UY 🔲 RY	Contact Parameters
	Variable Activation
	Bulk Viscosity Parameters
	Hourglass Parameters
	User Subroutine Parameters
Auto Constrain Failed Node Yes 🗸 🕑	Rigid Body Merging
	Add CID to MATRIG
OK Cancel	ок

e Execution Controls

Execution Control Parameters...

Element/Entity Activation...

Dynamic Relaxation...

Sub-Cycling Parameters...

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.
- I. Click OK.

Content receptor default_viewport Filter Specification * Filter Select Groups for Output ALLSHUTIEULSTREN ALLSHTRIA default_group Select None Select All Select Current intity Type: Sublayer Variables • h tesults Types EFFFL - effective plastic strain TXX - xx-stress TYY - yy-stress TXY - xy-stress TXY - xy-stress Y - vy-stress	nt Viewport		
Filter Specification * Filter Filter Filter Gelect Groups for Output ALLMULTIEULSTREN ALLSHQUAD ALLSHTRIA default_group Select None Select All Select Current filte Type: Select Current filte	It_viewport		
* Filter Select Groups for Output ALLMULTIEULSTREN ALLSHTRIA default_group Select None Select All Select All Select Current htty Type: sublayer Variables • (h) esults Types FFFL - effective plastic strain FFST - effective stress TX - xx-stress TY - xy-stress TY - xy-stress TY - xy-stress TY - xy-stress TY - xy-stress TX - xx-stress TY - xy-stress TX - xx-stress TY - xy-stress TX - xx-stress C Add Modify Delete Output Requests Request Summary Result Name shell b File Type: Archive • Result Name shell b File Type: Archive • C Element Output • (d) 0 THRU END BY (Time) 0.001 (e) Number of Savings per File 1000 1000 (f) Modify Delete	Specification	P Output Requests - 🗆 🗙	
Filter Select Groups for Output ALLSHQUAD ALLSHTRIA default_group Select None Select All Select Current Select Current titty Type: ublayer Variables < h			
adetect Groups for Output ALLSHTRIA ALLSHTRIA default_group Select None Select All Select Current titty Type: ublayer Variables FFPL - effective plastic strain Yr - yr-stress Yr - yr-stress Yr - yr-stress Yr - yr-stress	Filter	Request Summary]
select Groups for Output ALLSHTRIA default_group Select None Select All Select All Select Current titty Type: ublayer Variables • h seults Types FFPL - effective plastic strain FFST - effective stress XX - xx-stress YY - yy-stress ZZ - zz-stress YY - xy-stress	T inter		
ALLMULTIEULSTREN ALLSHTRIA default_group Select None Select All Select Current tity Type: ublayer Variables The sults Types FFPL - effective plastic strain FFST - effective stress XX - xx-stress YY - yy-stress ZZ - zz-stress YY - yy-stress YY - yy-stress	: Groups for Output		
ALLSHQUAD ALLSHTRIA default_group g Select None Select All Select Current tity Type: ublayer Variables • h esults Types FFPL - effective plastic strain FFST - effective stress Y - yy-stress ZZ - zz-stress YY - yy-stress Y - yy-stres	ULTIEULSTREN		
ALLSHINDA default_group g Select None Select All Select All Select Current tity Type: ublayer Variables • h esults Types FFPL - effective plastic strain FFST - effective stress XX - xx-stress YY - yy-stress ZZ - zz-stress XY - xy-stress YZ - vz-stress YY - yy-stress YZ - vz-stress YY - stress YZ - vz-stress	HQUAD		
Select None Select All Select Current Select Current titty Type: ublayer Variables • h esults Types FFPL - effective plastic strain FFST - effective stress Y' - yy-stress ZZ - zz-stress Y' - vy-stress	ult group (g)		
Select None Select All Select Current Select Current tity Type: ublayer Variables • h esults Types FFPL - effective plastic strain FFPL - effective plastic strain J O THRU END BY (Time) 0.001 e Number of Savings per File 10000 Y - yy-stress		Result Name	
Select None Select All Select Current tity Type: ublayer Variables • h esults Types FFPL - effective plastic strain FFST - effective stress YY - yy-stress ZZ - zz-stress YY - vy-stress YY - with the plastic strain YY - vy-stress YY - vy-stress YY - vy-stress YY - vy-stress YY - with the plastic strain		shell (b)	
Select All Select Current tity Type: ublayer Variables • h ssults Types FFPL - effective plastic strain FFST - effective stress YY - yy-stress ZZ - zz-stress YY - vy-stress	Select None		
Select All Select Current Archive Result Type: Element Output Times for Output Sampling Rate Times for Output Sampling Rate O THRU END BY (Time) 0.001 e Number of Savings per File 10000 XX - xx-stress YY - yy-stress ZZ - zz-stress YY - vy-stress YY - vy-str		File Type:	
Select Current ntity Type: ublayer Variables • h esults Types FFFL - effective plastic strain FFST - effective stress YY - yy-stress ZZ - zz-stress YY - vy-stress YY - with the plane in the	Select All	Archive	
Select Current ntity Type: ublayer Variables • h esults Types FFFL - effective plastic strain FFST - effective plastic strain Y - yy-stress YZ - xx-stress YY - yy-stress YZ - vx-stress YY - yy-stress		Result Type:	
httiy Type: ublayer Variables Types FFPL - effective plastic strain FFST - effective stress XX - xx-stress YY - yy-stress ZZ - zz-stress XY - xy-stress Y7 - yy-stress Y7 - yy-stress	Select Current	Element Output 🔻	
ublayer Variables h esults Types FFPL - effective plastic strain FFST - effective stress XX - xx-stress YY - yy-stress ZZ - zz-stress XY - xy-stress Y7 - vz-stress Y7 - v	Гуре:		Ì
Sampling Rate Sampling Rate Sampling Rate Contraction Sampling Rate Sampling Rate Contraction Sampling Rate Sampling R	ver Variables Th	Times for Output	
esults Types FFPL - effective plastic strain FFST - effective stress XX - xx-stress YY - yy-stress ZZ - zz-stress XY - xy-stress Y7 - vz-stress Y7 - vz-stress		Sampling Rate	
FFPL - effective plastic strain j FFST - effective stress j XX - xx-stress j YY - yy-stress 0.001 ZZ - zz-stress v YY - vz-stress j Y7 - vz-stress Add Modify Delete	Types		
FFST - effective stress 0.001 XX - xx-stress Number of Savings per File YY - yy-stress 10000 XX - xx-stress Add Modify Delete	- effective plastic strain	0 THRU END BY (Time)	
Number of Savings per File YY - yy-stress ZZ - zz-stress YY - vy-stress YY - vz-stress Add Modify Delete	- effective stress		
ZZ - zz-stress XY - xy-stress Y7 - vz-stress Add Modify Delete	vv-stress	Number of Savings per File	
XY - xy-stress Y7 - vz-stress Add Modify Delete	zz-stress	10000	
Add Modify Delete	xy-stress	(f)	ĺ
	vz-stress	Add Modify Delete	
			1

Analysi	S			
Action:	Analyze 🔻			
Object:	Input Deck 🔻			
Method	I: Translate ▼			
Code:	MSC.Dytran			
Type:	Structural			
Availab	le Jobs 🖺			
Job Nar	ne			
bunker				
Job Des	scription			
MSC.D May-1	ytran job created on 08- 9 at 17:12:25			
Tr	anslation Parameters			
I	nitiating Calculation			
	Execution Controls			
	Select Load Cases			
Output Requests (a)				
Output Controls				
	Direct Text Input			
	Apply			

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.

Output Controls		Analysis
Culput controls		Action: Analyze -
Output Controls		Object: Input Dock -
Use IEEE Format for THS Output	Default 🔻	Mothod: Translate =
Echo Ignored Valid Data Entries	Default 🔻	Interiou.
Shell Sublayer Strain Output	Default 🔻	Code: MSC.Dytran
Composite Shell Stress/Strain Output	Default ▼	Type: Structural
License Information Output	Default 🔻	Available Jobs
Element Timestep Info		
Write Spotweld Summary	Default 🔻	
Write Rigid Body Summary	Default 🔻	Job Name
Write Failed Elements To File	No - (b)	bunker
		Job Description
Write CONM2's Summary	Default 🔻	MSC.Dytran job created on 08- May-19 at 17:12:25
⊢ATB Coupling Parameters		
Write ATB in THS Format	Default ▼	Translation Parameters
Frequency of ATB THS		Initiating Calculation
Frequency of Main ATB File		Execution Controls
		Select Load Cases
Ellipsoid Mesh Density		Output Requests
Plane Mesh Density		Output Controls
		Direct Text Input
C	Cancel	Apply
	Cancel	Арріу

а

			Analysis
Ente	er the output request for euler ele	ements:	Action: Analyze -
a. (Click Direct Text Input.		Object: Input Deck -
b. S	Select File Management Sectio	n.	Method: Translate -
c. E	Enter as shown:		
	TYPE (AIR) = $ARCHIVE$		Code: MSC.Dytran
	FI FMENTS (AIR) = 14		Type: Structural
	SET $14 = AU FULHYDRO$		
		E DRESSLIRE EMATEVI INC EMATRI T	Available Jobs
	TIMES(AIR) = O TARO END	BY 0.001	
	SAVE (AIR) = 10000	Pirect Text Input	Job Name
			bunker
		File Management Section	Job Description
		TYPE (AIR) = ARCHIVE	MSC.Dytran job created on 08- May-19 at 17:12:25
		ELEMENTS (AIR) = 14	May 19 dt 17.12.29
		ELOUT (AIR) = DENSITY SIE PRESSURE FMAT FVUNC FMATPLT .	
		VOLUME XVEL YVEL ZVEL	Translation Parameters
		TIMES (AIR) = 0 THRU END BY 0.001	Initiating Calculation
		SAVE (AIR) = 10000	Execution Controls
		C	Select Load Cases
			Output Requests
		● ● File Management Section ✓ FMS Write To Input Deck ● Executive Control Section ✓ EXEC Write To Input Deck	Output Controls
		© Case Control Section ✓ CASE Write To Input Deck	Direct Text Input a
		Bulk Data Section V BULK Write To Input Deck	
			Analy
			Apply

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.



Analys	is				
Action:	Analyze 🔻				
Object:	Input Deck 🕶				
Method	I: Translate 🔻				
Code:	MSC.Dytran				
Type	Structural				
Type.	Structural				
Availab	le Jobs 📔				
Job Nar	ne				
bunker	·				
Job Des	scription				
MSC.D May-1	Oytran job created on 08- 9 at 17:12:25				
Tr	andation Parameters				
	alisiation Farameters				
	Finantian Controls				
	Execution Controls				
	Select Load Cases				
Output Requests					
Output Controls					
	Direct Text Input				
	Apply				

Bunker Blast

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.

)		
Dytran Job [1]			
Dytran Explorer			
File Explorer	Inpu	ut Files	
Prime 	e e e e e e e e e e e e e e e e e e e	nker.bdf nker.dat o.dat	
¢		INKER.OUT INKER_AIR_FV4_0.ARC INKER_AIR_FV5_0.ARC INKER_ERROR_SUMMARY.MSG INKER_ILE_SUMMARY.MSG INKER_ILE_SUMMARY.MSG INKER_SHELL_0.ARC INKER_SHELL_109.ARC INKER_SHELL_121.ARC	OUT MSG THS ARC/RST VTU PVD
Executable DMP 1 SMP 1 UDS UU ATB	ib Info apsed time : 00:00:35 iput File : bunker.dat DS Objects: TB file :		d 100%



Step 23. Post-Processing in Patran

Menu Home		
Defaults Transforms Viewport D	Visplay Orientation	Misc. Web Model Tree
	New Model Preference	New Database — 🗆
 Create a new database to import the results into. a. Under the <i>Home</i> tab, click New in the <i>Default</i> group. b. Enter the <i>File name</i> results. 	Model Preference for: dam_break.db Tolerance Based on Model Default	Template Database Name C:\MSC.Software\Patran_x64\20190/template.db Change Template ✓ Modify Preferences
 c. Click OK. d. Select MSC.Dytran for Analysis Code. e. Click OK. 	Approximate Maximum Model Dimension: 10.0 Analysis Code: MSC.Dytran V	Set Working Directory to Database Location
	Analysis Type: Structural Preference Mapping: Mapping Functions Legacy Mapping No Mapping	Image: File name: File name: File of terms Cancel

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

Menu	Home	Geome	etry Propertie	es Loa	ds/BCs	Meshir	ng Ana	ilysis	Results
				No.	a)	state		5	1 🐼 🐺 🐝
Entire Cur Model Gr	rrent	Analysis Deck	Read	Archive File	History File	State File			S
Analyze	e	Create	Existing Deck	Acc	ess Resu	lts	Delete	Action	s Special Tools

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.)

Menu Hom Geometry Prope	rties Loads/BC	Cs Meshing Analysis	Results	
	Home	perties Loads/BCs	Resu	ilts
Fringe/Deformation Deformation Fring				
Quick Plot				
				2/4 K2 K8 8/2
	faults ins	Viewport		Orientation

- 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.





File≁	Group [*] Viewport*	
	Create	
	Post (a)	
	Modify	
Gro	up	
Actio	on: Post 🔻	
Curre	ent Viewport	
defa	ault_viewport	
Selec	ct Groups to Post	
BUN	KER_AIR_FV4_0	*
BUN	KER_AIR_FV5_0	
BUN	KER_SHELL_U	
BUN	KER_SHELL_121	
BUN	KER_SHELL_13	
BUN	KER_SHELL_25	
BUN	KER_SHELL_36	≡
BUN	KER_SHELL_48	
BUN	KER_SHELL_00	
BUN	KER SHELL 84	
BUN	KER_SHELL_97	
def_	_dummy_quad_elements	
def_	eulhy_hex_elements	
def_	_quad_elements	•
	Filter *	
	Select None	
	Select All	
	Select Current	
	Apply C Cance	