

Tutorial: Fluid-Structure Interaction Module



## Inventium®Tutorial: Fluid-Structure Interaction Module

Copyright © 2010-2013 Engineering Technology Associates, Inc.

This material is a compilation of data and figures from various sources.

Engineering Technology Associates, Inc. assumes no liability or responsibility to any person or company for direct or indirect damages resulting from the use of any information contained herein.

Engineering Technology Associates, Inc., ETA, the ETA logo, and Inventium<sup>®</sup> and PreSys<sup>®</sup> logos are the registered trademarks of Engineering Technology Associates, Inc. All other trademarks or names are the property of their respective owners.

Inventium<sup>®</sup> and PreSys<sup>®</sup> are both registered trademarks of Engineering Technology Associates, Inc.

1133 E. Maple Road Troy, Michigan 48083 USA

www.eta.com

## 1 PreSys User Interface

The PreSys user interface can be configured in many different ways, for maximum flexibility. The default user interface is shown in Figure 1. The main areas of the interface are labeled, and will be referred to in this Tutorial.

The TOOLBAR area is located at the top of the window, and contains selectable icons for all of the commands available in PreSys.

The DROP DOWN MENUs are at the very top of the window allows users to access every command through an easy to use drop down menu system. When the Drop Test Module has been installed with a valid license option, an APPLICATIONS menu will appear in the Drop Down Menu. The Drop Test Module is accessed through this menu.

The DISPLAY AREA is where all graphical data is displayed.

The COMMAND WINDOW allows users to directly input commands and execute scripts.

The MESSAGE WINDOW displays information about the model and prompts the user if data is needed, as well as warning or error messages for the user.

The MODEL EXPLORER allows the user to access a great deal of information about the model, control the display of data, edit the model data and view non-graphical data via 'card' images.

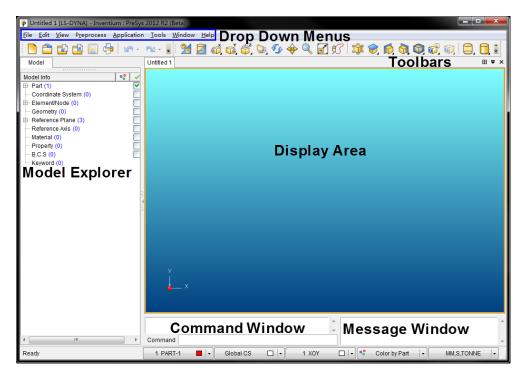


Figure 1 default user interface

## 2 Introduction

This tutorial was prepared to familiarize users with Inventium's pre and post processing and FSI module. This module provides a central location for all of the ALE-related options available in LS-DYNA. While all of the individual entities are available for creation in the various menus of PreSys, this module streamlines the creation of these models and provides modeling tools that are specific to the needs of Fluid-Structure Interaction simulation.

In this exercise, the finite element model preparation, specifying the section and material properties, defining coupling interface between the Eulerian and Lagrangian mesh, boundary condition definition, input or output requirements for analysis and post – processing of the results are described.

The model that will be created through this exercise is shown in the figure 2 below. The model is made up of 2 materials (fluids), which are excited by an explosive source located at the bottom of the model. These fluids will interact with a structure model. The fluids will be modeled using the Arbitrary Lagrangian Eulerian (ALE) method, while the structure will be modeled using traditional finite elements. Coupling will be defined for the fluids and the structure, in order to transmit the forces from the fluids to the structure.

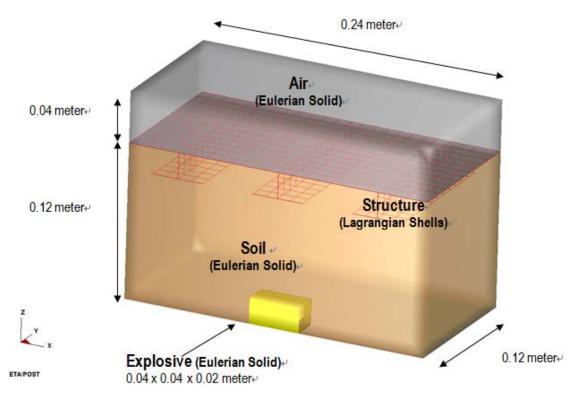
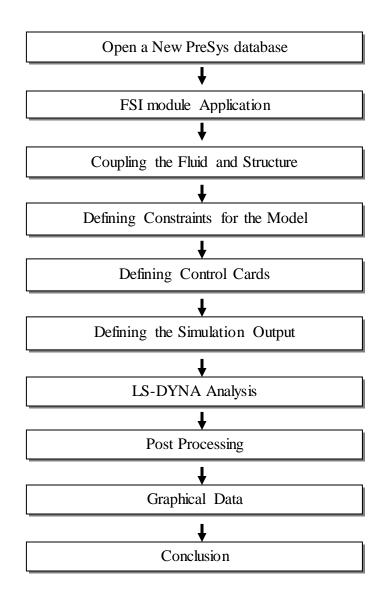


Figure 2 Model Dimensions (Symmetry about XZ plane)

### **Procedural Flowchart**



Inventium Tutorial: Fluid-Structure Interaction Module

# 3 Start PreSys

To start the process of modeling the materials, explosive and structure, we will open a new, empty model database. This can be done simply by starting PreSys from your desktop, doubleclicking the PreSys icon.

All actions that are to be done by the user for this exercise are displayed in Arial in this font.

Start a new PreSys database by opening the PreSys icon. PreSys will open, with a new, empty database.

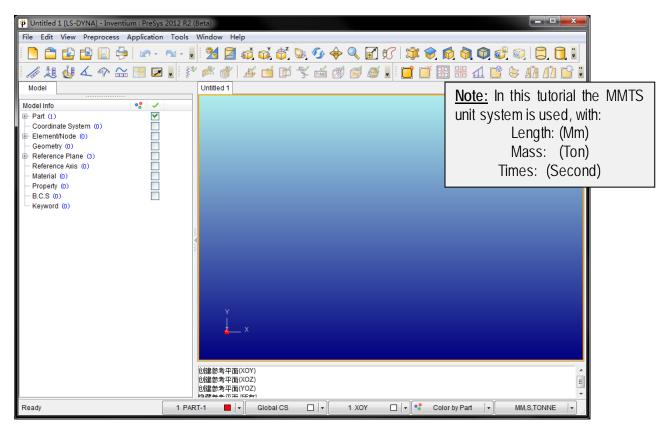
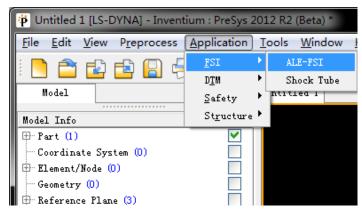


Figure 3 Empty Database

## 4 FSI module Application

Select the APPLICATION menu from the Drop Down Menus, and select the FSI Sub-menu.



When selected, the FSI Task Panel will be opened, with several tabs displayed at the top of the FSI Task Panel. Each of these tabs help guide the user to defined the needed data to define the FSI problem.

The FSI Task Panel Tabs are described in detail in the following sections. A brief overview of the Tabs are provided below:

ALE-FSI		
Setup Define	Cards	
Model	Half	Zone
Orientation	Z-axis 💌	Section
Shape		Zone2
Zone	Cube	
Explosive	Cube	Zone1
Mine Center	Below XY Plane 💌	20000

**Setup:** Describes the size, shape and number of ALE mesh zones to be created by PreSys. Allows the user to define mesh size for automeshing operations, and ¼ symmetry, ½ symmetry or full model.

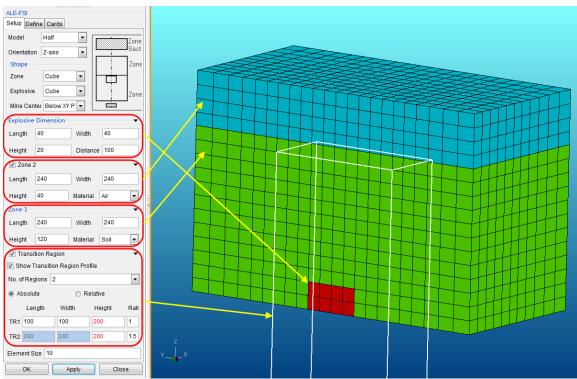
Define: Allows the definition of Material Properties and Sections.

Cards: FSI-specific control cards used by LS-DYNA to perform the analysis

### 4.1 Setup Tab

By default, the **Setup** Tab should be active when FSI is selected from the Application Menu. If it is not, make it active by selecting the Setup Tab with the mouse.

The Setup Tab is divided into 3 main sections, each providing a way to define the dimensions of explosive source, the 2 ALE mesh zones and the mesh size and transition mesh.



A graphic is displayed showing the relative position and orientation of the explosive and mesh zones.

#### 1. Define the ratio of the FIS model

The first option available is the Full / Half / Quarter model options, selectable from a drop down menu. This will create a full, half or quarter symmetry model. The symmetry will be in the YZ plane for the half model and the XZ / YZ plane for the quarter model.

For our example, use the Half model.

#### 2. Define the explosive

A general shape may be selected using the radio buttons. Cube, Spherical and Cylindrical shape explosives are supported.

Dimensions of the shape may be entered. Be sure to consider the model units when defining these values.

Our example model will use a 40mm by 40mm by 20mm cube-shaped explosive source.

The explosive source can be located Above Zone 1, Below Zone 1, or at a User defined location in the model. When these options are select, the schematic displayed in the Setup Tab will change, showing the configuration.

ALE-FSI									
Setup Define Cards									
Model	Half 💌	Zone							
Orientation	Z-axis 💌	Section							
Shape		Zone2							
Zone	Cube 🔻								
Explosive	Cube 💌	Zone1							
Mine Cente	r Below XY P 🔻	2000							

Explosive Dimension: Defines the dimensions and the location of the explosive source.

User can input 40, 40, 20 for the Length, Width, and Height of explosive; and the position is 100 mm below the XY plane, Orientation is Z-axis.

Explosive	Dimension		•
Length	40	Width	40
Height	20	Distance	100

#### 3. Set the Zone1 and Zone2

ALE Zone: The ALE zones may be defined with a Cube cross-section or a Cylindrical crosssection. This is selected using the Shape Drop Down menu. The schematic diagram will be updated to show the cross-section if this value is changed.

For our example, we will use a Cube cross-section.

Setup Define	Cards		
Model	Quarter 💌		Zone
Orientation	Z-axis 💌	i	Section
Shape			Zone2
Zone	Cube 🔻		
Explosive	Cube 💌		Zone1
Mine Center	Below XY Plan 💌	00000	

The Zone 1 is identified in the schematic, and is the lower section. By default, at least one ALE Zone is required for the model definition.

The user may select a material using the Material Drop Down menu, The available options are Soil, Air and Water, By selecting one of these options PreSys will automatically create a material and assign it to the elements contained in this ALE mesh zone.

For our exercise, please select Soil for this parameter.

The ALE mesh will be automatically created using the dimensions supplied by the user, entered into the Size field. *Enter 240,240, 120 into the Size field*. This is the FULL DIMENSION, not the size of the Half model. PreSys will automatically create the half model based on the full dimension.

A second ALE Mesh Zone, Zone 2, may be optionally defined in the model. This is active by default. If a second ALE Mesh Zone is not desired, uncheck the box next to the Zone 2 Dimension.

Zone 2 is located directly above Zone 1, and is required to have the same X and Y dimension when defining a Cube cross-section. When a Cylindrical cross-section is defined, Zone 2 must have the same Radius as Zone 1.

Similar to the Zone 1 definition, a material may be selected from the Material Drop Down menu.

For this exercise, please select Air for the material.

For the Size parameter, enter 240, 240 and 40 as the dimensions of the ALE Mesh Zone 2.

Zone	2			•
Length	240	Width	240	
Height	40	Material	Air	•
Zone 1				•
Length	240	Width	240	
Height	120	Material	Soil	•

#### 4. Set the transition Region

An optional Transition Region will be defined to allow for a locally refined mesh near the explosive component. The Max ratio defines the ratio of the Element Size to the Transition region mesh size.

Click the Expand arrow in the right, and set the transition region parameters.

For our example, the no. of regions is 2, and the size of transition region 1 is 100 by 100 by 200, the Ratio value is 1.0;

The size of transition region 2 is 240 by 240 by 200, and the Ratio value is 1.5.

✓ Transition Region										
Show Transition Region Profile										
No. of Regions 2										
At	osolute	C								
	Length	Width	Height	Ratio						
TR1	100	100	200	1						
TR2	240	240	200	1.5						

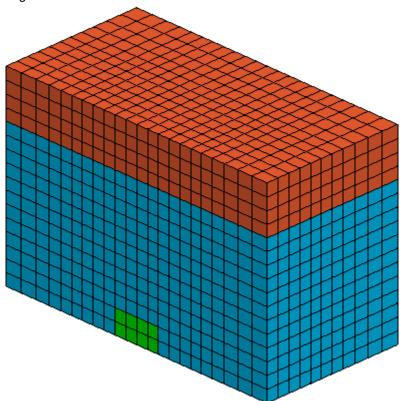
### 5. Set the element size

The mesh size can be defined for the ALE mesh zones by using the Element Size parameter. This controls the nominal element size for mesh. PreSys applies constraints on this parameter to avoid creating models which are unreasonably large. The minimum element size is limited to 1.0 mm.

Enter a value of 10 as the Element Size for our example.

### 6. Select APPLY from the bottom of the Setup Tab.

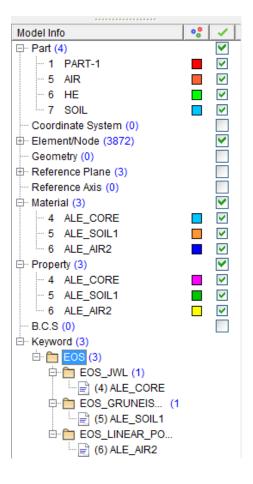
The resulting model will be displayed. Note that the Shade option is active and the Edges option is active in the image below.



The user may inspect the mesh by rotating the model and by viewing the individual parts created.

Select the Model tab and turn the newly created parts on/off. Do not close the FSI Task tab at this time.

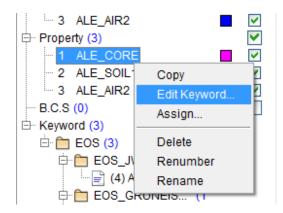
When looking at the Model Tab, the user will also note that several material, properties and Equations of State (EOS) were defined along with the elements.



### 4.2 <u>Review the Materials and Properties.</u>

The default sections and material properties of all part generated are automatically defined in the Setup tab.

Navigate to the Model Explorer (Model Tab) and expand the Property section, clicking on the + next to the word Property. Three properties will be listed:



ALE\_CORE: the property of the explosive ALE\_SOIL1: The property of the ALE Zone 1 ALE\_AIR2: The property of the Air in ALE Zone 2

To view the properties of the explosive, right click on the ALE\_CORE and select the Edit Keyword option.

The KEYWORD EDITOR view of the entity will be displayed. The user may edit any of the parameters in the Keyword Editor. Fields that are left blank are typically the default values and need not be modified for this exercise.

🖗 Keyword	Editor				_	x
*SECTION_	SOLID					
OPTIONS	Choose	OPTIONS	Choose		✓	
SECID	ELFORM AET				*	
1	11 🛛 🖳					
COMMENT						
				*		
				Ŧ		
					OK	
					- Cancel	
Description	of Fields					đ

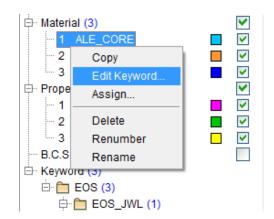
Note that each field will have a tool tip displayed at the lower left of the keyword Editor window, providing the description of the field.

The Setup Tab operation created a SECTION\_SOLID entity for the explosive, and ALE Zones 1 and 2. The ELFORM (element formulation) for the explosive part is set to 11, which is the 1 Point Integration ALE Multi-Material element.

Select CANCEL to close the Keyword editor for the property.

The user is invited to do the same operation for the SOIL and AIR properties. Note that the same section properties are defined for each of the components of the model.

Next navigate to the Material section of the Model Explorer and right click on the ALE\_CORE material.



A Keyword Editor will open for the MAT\_HIGH\_EXPLOSIVE\_BURN material used to define the explosive. The options for that material definition may be modified to

P Keyword	Editor									<b>**</b>
*MAT_HIGH	LEXPLOSIVE_BUR									
*MAT_HIGH OPTIONS	TITLE									
TITLE									^	
ALE_CORI MID 1 COMMENT	E									
MID	RO	D	PCJ	BETA	К	G	SIGY			
1	1.654e-009	6.94e+006	25500							
COMMENT										
								*		
								-		
										OK
									-	Cancel
Description	of Fields									th.

Select CANCEL to close the Keyword Editor for the MAT\_HIGH\_EXPLOSIVE\_BURN and open the Keyword Editor for the SOIL and AIR materials.

Note that the SOIL and AIR are defined with MAT\_NULL, and have common properties of soil and air. Users may wish to modify these parameters to model specific types of soil or air of certain humidity.

Change the default values to those shown in the fields above. Select OK to save the data and then close the Keyword Editor windows.

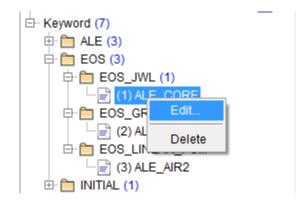
### 4.3 EOS Properties

A thermodynamic state of a homogeneous material, not undergoing any chemical reaction or phase change, may be defined by two state variables. This relation is generally called an Equation of State (EOS).

The PreSys FSI module creates appropriate default EOS entities for the explosive, soil and air.

#### 1. Review or Edit the EOS

Expand the KEYWORD section of the Model Explorer and further expand the EOS section. Three EOS types will be displayed. *Right-click on the EOS name ALE\_CORE, and select the Edit option.* 



The explosive uses the JWL Equation of State, which is described in the *LS-DYNA User's Manual.* Since there are no defaults values for the EOS, PreSys provides the values for each of the EOS keywords, shown below.

🏟 Keywoi	rd Editor											x
*EOS_JWL_ OPTIONS	TITLE											
	TITLE								$\sim$	ID	Title	*
TITLE									*	1	ALE_CORE	
ALE_CORE												
EOSID	A	В	R1	R2	OMEGA	EO	VO					-
1	540900	9370	4.5	1.1	0.35	8000	1			•	III	P.
COMMENT											New	
								~			Delete	
								-			Save	
TITLE ALE_CORE EOSID 1 COMMENT									-		Close	
Descriptio	on of Fields											

Review the values there and select CLOSE to close the keyword editor.

The Soil material uses an EOS\_GRUNEISEN form, which is a relationship between pressure and volume. See the *LS-DYNA User's Manual* for more information on the EOS\_GRUNEISEN definition.

Right-click on the EOS\_GRUNEISEN entry in the Model Explorer to view the parameters automatically defined by PreSys. Review the values there and select CLOSE to close the keyword editor.

🖗 Keywor	d Editor											x
*EOS_GRUNN OPTIONS	EISEN_TITLE											
OPTIONS	TITLE								~	ID	Title	*
TITLE									*	2	ALE_SOIL1	
ALE_SOIL1	L											
EOSID	С	S1	S2	S3	GAMAO	A	EO					-
2	1.6e+006	1.9								•	111	P.
VO									E		New	
1											<b>D</b> 1 .	$\equiv$
COMMENT											Delete	
								*			Save	
TITLE ALE_SOILJ EOSID 2 VO 1 COMMENT									-		Close	
Descriptio	on of Fields											н

The Air material uses an EOS\_LINEAR\_POLYNOMIAL equation of state. Please refer to the *LS*-DYNA User's Manual for more information on the parameters used in this EOS

Right-click on the EOS\_LINEAR\_POLYNOMIAL entry in the Model Explorer to view the parameters automatically defined by PreSys. Review the values there and select CLOSE to close the keyword editor.

🖗 Keywor	d Editor											x
*EOS_GRUN	EISEN_TITLE											
OPTIONS	TITLE								~	ID	Title	-
TITLE									*	2	ALE_SOIL1	
ALE_SOIL:	1											
EOSID	С	S1	S2	S3	GAMAO	A	EO					-
2	1.6e+006	1.9								•	III	P .
VO									Ξ		New	
1											Delete	
COMMENT								_			Detere	
								*			Save	
									-		Close	
Descriptio	on of Fields											

#### 2. Assign EOS to \*Part.

Model In	fo	•	-
	4)		~
- 1	PART-1		<b>~</b>
2	AIR		<b>~</b>
- 3	Move To		<b>~</b>
- 4	Edit Keyword		
Coor	Current	968	
Elem Geor	Wireframe		
⊕ Refe	Transparency	F	
	Edit Part		

Right click the Part: AIR to select Edit Keyword... on the model tree.

Click EOSID on the popped \*PART Keyword Editor window to select EOS LINEAR POLYNOMIAL.

ᠹ Keywor	rd Editor							×
*PART								
OPTIONS	Choose			OPTIONS	Choose			
OPTIONS	Choose			OPTIONS	Choose		~	
OPTIONS	Choose				L		~	
HEADING							*	
AIR								
PID	SECID	MID	EOSID	HGID GR/		TMID		
2	3	⊻ 3	3	EOS_GASKET				
COMMENT				EOS_GRUNEISEN	T			
				EOS_IDEAL_GAS				
				EOS_IGNITION_	AND_GROWTH_OF_REACTI	ON_IN_HE		OK
				EOS_JWL				
				EOS_JWLB			~	Cancel
HOURGLASS	ID			EOS_LINEAR_PO	LYNOMIAL			±

Select EOS Type

Select the created ALE\_AIR2 in the popped \*EOS LINEAR POLYNOMIAL, and then select Accept to return to \* PART Keyword Editor window and select OK.

🖗 Keywor	d Selection										l	x
*EOS_LINE/ OPTIONS	AR_POLYNOMIA	L_TITLE										
OPTIONS	TITLE								~	ID	Title	~
TITLE									*	3	ALE_AIR2	
ALE_AIR2												
EOSID	CO	C1	C2	C3	C4	C5	C6					
3					0.4	0.4						_
EO	VO								=	•		•
0.253313	1										W	
COMMENT								_			New	
								<u>^</u>			Save	
ALE_ATR2 EOSID 3 EO 0.253313 COMMENT									-		Accept	
Descriptio	n of Fields											H

Select EOS ID

Relate the ID of EOS to the Part HE and SOLI respectively according to the above steps.

Please note that the EOS type for the Part HE is EOS JWL, and SOLI corresponds to EOS GRUNEISEN.

### 4.4 Define Tab

The Define Tab of the FSI Task Panel allows the user to create additional Materials and Properties and assign those to parts which were not automatically defined in the Setup Tab. For this exercise, no additional materials or properties are needed. We'll move on to the Cards Tab.

ALE-FSI	
Setup Define Cards	
Octup Ourus	
Define Material	Dofine Bronarty
Denne Material	Define Property

## 4.5 Cards Tab

The Cards Tab provides an easy way for the user to create the Control Cards and Keywords that are needed to complete the set-up of the FSI model. All of the entities that are commonly used for fluid-structure interaction modeling are included in the Cards Tab.

To create any Keyword listed on the Cards Tab, the user may double click on the name of the entity. A Keyword Editor for that entity will be displayed. Where default data is provided by LS-DYNA, it will be displayed. The user is again referred to the LS-DYNA User's Manual for details on each of the parameters used in the various keyword entities.

ALE-FSI		
Setup Define Cards		
ALE Card Define		
Card	Defined	
*ALE_REFERENCE_SYSTEM_GR	NO	
*ALE_SMOOTHING	NO	
*ALE_TANK_TEST	NO	
*ALE_REFERENCE_SYSTEM_CU	NO	
*ALE_REFERENCE_SYSTEM_NO	NO	
*ALE_REFERENCE_SYSTEM_SW	NO	
*ALE_MULTI-MATERIAL_GROUP	NO	
*SET_MULTI-MATERIAL_GROUP	NO	
*ALE_FSI_SWITCH_MMG	NO	Ξ
*ALE_UP_SWITCH	NO	
*ALE_FSI_PROJECTION	NO	
*CONSTRAINED_LAGRANGE_IN	NO	
*CONTROL_ALE	NO	
*DATABASE_FSI	NO	
*EOS_LINEAR_POLYNOMIAL	NO	
*EOS_JWL	NO	
*EOS_JWLB	NO	
*EOS_GRUNEISEN	NO	
*EOS_IGNITION_AND_GROWTH	NO	
*EOS_IDEAL_GAS	NO	
*INITIAL_DETONATION	NO	
*INITIAL_VOID	NO	
	NO F	Ŧ
OK Apply	Close	

#### 1. Define Detonation

The detonation of the explosive material is controlled by an initial condition, applied to the explosive material. To create this, the user must scroll down the list on the Cards Tab and find the \*INITIAL\_DETONATION entity. *Double-click the entity name*.

The Keyword Editor for the INITIAL\_DETONATION entity will be displayed. The user must specify the part of the model which is the explosive, and the node location at which the detonation will be initiated.

ip Keyword Editor			×
*INITIAL_DETONATION			
PID X, Y, Z LT	^	ID Title	^
3 🖸 0.000000, 0.000000, -10.000000 0		1	
-1: AN ACOUSTIC BOUNDARY, ALSO, *BOUNDARY_USA_SURFACE 0: ALL HIGH EXPLOSIVE MATERIALS ARE CONSIDERED			
PART		< III	
*		New	
		Delete	
		Save	
	~	Close	
X, Y, Z			

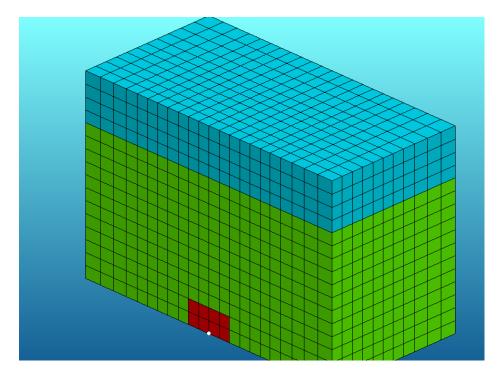
Select the drop down below the PID field on the INITAIL\_DETONATION keyword. The user may select from various options.

Select PART from the drop down list, and then PART from the sub-menu. A Part List will be displayed, where the user may select from the list of select an element of the Part from the Display Area. Select the part HE from the part list.

Select Pa	art : 0		
-By Curs	or		
🔽 Trac	e	Front	
-By List			
Name F	ïlter *		
ID	Name		Col ^
2	AIR		
3	HE		
4	SOIL		
			Þ
-By Othe	r Methods		
	New	Ву	
	OK	Can	cel

The detonation center can be identified by an X, Y, Z location. *Click on the button beneath the X, Y*, *Z field*. This will open a Node/Point selection tool that will allow the user to select an existing node or point in the model or define a location using an x, y, z coordinate.

Select the center node of the explosive (HE part), at the center of the symmetry plane, at the bottom.



The x, y, z coordinate of that node will be applied to the field on the INITAIL\_DETONATION keyword. *Select SAVE and then CLOSE to complete the detonation definition.* 

### 2. Define Multi-Material Group

The user needs to also define which ALE materials interact with each other during the simulation. LS-DYNA accomplishes this definition by defining ALE Multi-Material Groupings, defined on the ALE\_MULTI\_MATERIAL\_GROUP keyword.

To create this keyword the user may scroll down the Cards Tab list to find the entry. *Double-click on the entry.* A Keyword Editor for the ALE\_MULTI\_MATERIAL\_GROUP will be displayed.

Keyword Editor     *ALE_MULTI-MATERIAL_GROUP		
SID IDTYPE 2 V 1 COMMENT 0: PART SET ID V 1: FART ID	*	ID Title   I  I  I  I  I  I  I  I  I  I  I  I  I
SET TYPE		

Select the IDTYPE drop down and select the PART option.

Then, select the SID drop down, selecting the PART ID option. Select the HE part from the list provided. The PID will be used in the SID field.

Perform this same task for the other ALE materials, using the same procedure for the SOIL and AIR materials.

Close the FSI Task Panel by selecting OK.

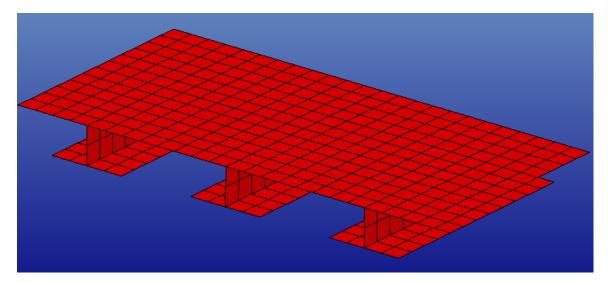
# 5 Coupling the Fluid and Structure

The structure that the explosive blast will interact with is a simple plate structure, which will be made of a material which will have nonlinear behavior. This structure is contained in another model which will be imported into the current model

### 1. Import the Structure

Go to the FILE drop down menu and Select IMPORT. Select the file plate.dyn, from the tutorial folder. The folder lie in the installation directory: ...\PreSys 2012\e-Manual\Inventium-FSI Training Manual.

The structure, shown below, is a single component, modeled with shell elements.



It is always good practice to review the materials and properties of models which are imported, to assure that all units are properly imported.

To view the properties and materials associated with the imported plate, *navigate to the Model Explorer and right click on the property titled "plate"*. The SECTION\_SHELL for the plate will be displayed in the Keyword Editor.

🖗 Keywor	d Editor										<b>x</b>
	HELL_TITLE										
OPTIONS	Choose			○ 0P	TIONS	TITI	E			<u> </u>	
TITLE											
Plate											
Plate SECID 101001	ELFORM	SHRF	NIP	PROPT	(	R/IRID	ICOMP				
101001	2	✓ 1	3	0		)	0				
T1	T2	ТЗ	T4	NLOC		AREA	IDOF	EDGSET		=	
2	2	2	2	0	$\sim$	)	0				
COMMENT											
									<u>~</u>		OK
											Cancel
										-	Cancer
Description	n of Fields										

The user should note the material thickness (2 mm), and confirm the element formulation.

Select Cancel to close the Keyword Editor.

Navigate to the Material section of the Model Explorer and right-click on the material associated with the plate (named 'plate').

The Keyword Editor for the MAT\_PLASTIC\_KINEMATIC will be displayed. Confirm that the units for the density (RO), Young's Modulus(E) an Yield Stress (SIGY). The constants provided are those for a steel material. Users can find more detailed information on this material model by looking at the LS-DYNA User's Manual.

🖗 Keywor	rd Editor								×
	TIC_KINEMATIC_T	ITLE							
OPTIONS	TITLE							<ul> <li>Image: A start of the start of</li></ul>	
TITLE								*	
Plate									
MID	RO	E	PR	SIGY	ETAN	BETA			
101001	7.83e-009	207000	0.3	800	0	0			
SRC	SRP	FS	VP					=	
0	0	0	0	$\sim$					
COMMENT									
							*		OK
								_	Cancel
								*	
Descriptio	on of Fields								

Close the Keyword Editor by selecting CANCEL.

The model now has all of the ALE and Lagrangian meshes defined, properties and materials defined, and is ready to define the coupling between the fluid and structural meshes.

#### 2. Coupling the Fluid and Structure Meshes

To define the interaction between the ALE (fluid) and Lagrangian (structure) meshes, This interaction will be defined using a CONSTRAINED\_LAGRANGE\_IN\_SOLID entity.

The user is able to define this from the FSI menu, using the Cards Tab.

Open the Keyword Editor by double-clicking the CONSTRAINED\_LAGRANGE\_IN\_SOLID entry. The keyword editor for the entity will open. Enter/select the parameters shown in the image below.

*CONSTRA	INED_LAGRANG	E_IN_SOLID											
OPTIONS	Choose					ONS Choos	se			~	ID	Title	
SLAVE	MASTER	SSTYP	MSTYP		NQUAD	CTYPE		DIREC	MCOUP	-			
		☑ 0	☑ 0	~	0	2		1	0				
START	END	🗸 0: PAR	T SET ID		FRCMIN	NORM		NORMTYP	DAMP				
0	1e+010	1: PAR	TID		0.5	0		0	· 0	=			
cq	HMIN	2: SEG	MENT SET ID		PLEAK	LCIDPOR	_	NVENT	IBLOCK	=	•		Þ
0			0	~	0.1			0	0			New	
IBOXID	IPENCHK	INTFORC	IALESOF		LAGMUL	PFACMM	_	THKF					
0	✓ 0	0	<ul><li>✓ 0</li></ul>	~	0	0		0				Delete	
A1	B1	A2	B2		A3	B3	_	GRADVF	POREINI			Save	
0	0	0	0		0	0		0	0			Close	

The key parameters to define are the Master and Slave entities for the interaction. All other parameters are default values.

Select the SSTYP drop down and select the option for PART SET. Next, select the SLAVE drop down and the SET\_PART option. Once selected, the user can create a SET\_PART which contains the LAGRANGIAN part; the plate, by using the PART button under the SET field.

🖗 Keyword	d Selection								×
*SET_PART									
OPTIONS	Choose			✓ OPTION	IS Choose		<ul> <li>Image: A start of the start of</li></ul>	ID Title	*
CARD2_COLU	MN 1			E CARD2_	LIST_GENERATE 1	1		1 2	
SID	DA1	DA2	DA3	DA4	SOLVER		*	2	
3	0	0	0	0	MECH				
SET									
0 PART									-
COMMENT								٠ III	•
						<u>^</u>		New	
						-		Save	
							-	Accept	
PART SET									

The user selects SAVE and CLOSE to end that SLAVE definition.

The MASTER definition is the ALE mesh parts. This is defined by selecting the MASTER drop down and selecting the SET\_PART option. As with the Slave definition, select the SET button and select the 2 ALE mesh parts ALE\_SOIL and ALE\_AIR. Again, SAVE and CLOSE the keyword editor for the Part Set creation.

Select SAVE on the CONSTRAINED\_LAGRANGE\_IN\_SOLID keyword editor and the coupling definition is completed.

## 6 Defining Constraints

The model needs constraints on each of the exterior faces of the ALE meshes. That is to say, the free surfaces of the model need to have constraints applied to represent the symmetry boundary and the exterior boundaries of the model.

We will assign the boundary conditions through Node Sets which will be defined for each of the exterior faces of the model.

Preprocess		_	
<u>P</u> art	۲		
Line/Point	۲		
<u>S</u> urface	۲		
Reference <u>G</u> eometry	۲		
<u>M</u> esher	۲		
<u>E</u> lement	۲		
Node	۲		
Model <u>C</u> heck	۲		
Coor <u>d</u> inate System	۲		
M <u>a</u> terial	۲		
P <u>r</u> operty	۰.		
<u>B</u> .C.S	۲	Create B.C.S 🕨	Force
Connection		Delete B.C.S	Moment
			Pressure
			SPC
			InitVel

Using the Drop Down Menu, navigate to PreProcess, B.C.S, and select the SPC entry.

When selected, a SPC Task Panel will be displayed that allows the user to define the degrees of freedom (DOFS) that will be constrained.

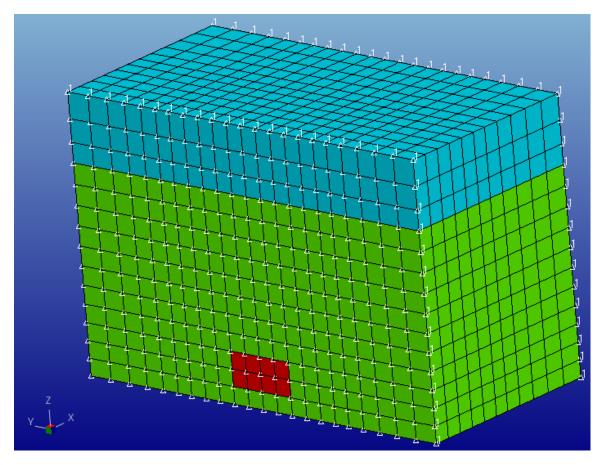
Create S	PC						
Туре	BOUNDARY_SPC_NODE						
Name	SPC-1						
ID	1						
D.O.F							
D.0.F	Define D.O.F		•				
X-Translation X-Rotation							
Y-Translation Y-Rotation							
📃 Z-Tra	anslation	Z-Rotation					
-Local C	oordinate Syste	em					
	Define LC	S (Global)	•				
Entity							
Node	e C	Geometry					
Select Node [0]							
ОК	Apr	oly Close					

We will first define the constraints for the front and rear surface of the model. Select the Type as BOUNDARY\_SPC\_NODE.

Tick on the X-Translation to constrained the X freedom. Then click Select Node button.

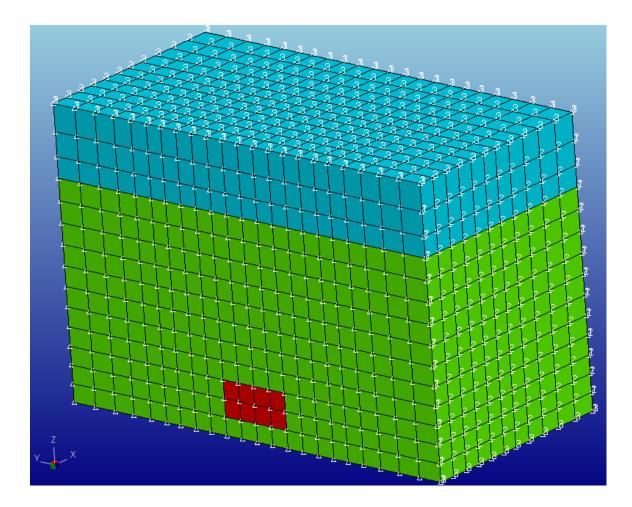
Select Node : 0 🛛 ស
By Cursor
+ 🗆 🔛 🔿 🖍
🔽 Trace 📃 Front
V Spread 15.0 🔳 Stop Angle 80.0
-By Other Methods
All Displayed By
Selection Options
Exclude Consider Feature Line
Angle > 30.0
Reject Reset Close

Input the Spread as 15 in the Select Nodes window, and then pick up any one of nodes in the two surfaces vertical to the X-axis.



Click Apply button to constrain the X-DOF. The result is shown as the following figure.

Repeat the steps above, and constrain the Z-DOF to the nodes in top and bottom; Constrain the Y-DOF to the nodes in the left and right. The result is show as the following figures.



# 7 Defining Control Cards

LS-DYNA simulations require that several control cards be defined. These control cards define the simulation time, simulation time steps, methods used to deal with model data, such as hourglass energy. As mentioned earlier, some of the necessary control cards may be defined using the FSI Cards Tab. There are more general purpose Control Cards that also need to be defined, which do not appear in the Cards Tab.

To define the control card, *right-click the Keyword entry on the Model Explorer, and navigate down to the CONTROL entry, and select the submenu items.* 

- 3 A	CONTROL		CONTROL FORMING STONING
- 1010			CONTROL_FORMING_STONING
□ Property	00111102(111121011)		CONTROL_FORMING_TEMPLATE
- 1 A	CONTROL(MPP)	•	CONTROL_FORMING_TIPPING
-2 A	DAMPING	•	CONTROL_FORMING_TRAVEL
- 3 A	DATABASE		CONTROL_FORMING_USER
- 1010	DATABASE(ASCII)		CONTROL_HOURGLASS
⊕ B.C.S (3	DATABASE(BINARY)		CONTROL_NONLOCAL
E Keyword	DEFINE		CONTROL_OUTPUT
	DEFORMABLE		CONTROL_PARALLEL
	EF		CONTROL_PORE_AIR
⊕ · 🛅 B ⊕ · 🛅 C	ELEMENT		CONTROL_PORE_FLUID
	EOS		CONTROL_REMESHING
	FREQUENCY_DOMAIN		CONTROL_RIGID
¢.€	HOURGLASS		CONTROL_SHELL
	INCLUDE		CONTROL_SOLID
Ė.Ę	INITIAL		CONTROL_SOLUTION
	INTEGRATION		CONTROL_SPH
	INTERFACE		CONTROL_SPOTWELD_BEAM
	LOAD		CONTROL_STAGED_CONSTRUCTON
⊞ <mark>È⊐ II</mark> ⊞ <mark>È⊐ S</mark>	MATERIAL(OTHERS)		CONTROL_START
	NODE		CONTROL_STEADY_STATE_ROLLING
	PART	•	CONTROL_STRUCTURED
	PERTURBATION	•	CONTROL_SUBCYCLE
	RAIL	•	CONTROL_TERMINATION

Select CONTROL\_TERMINATION and enter a termination time of 200e-06 in the ENDTIM field.

🖗 Keyword	Editor								×
*CONTROL_T	ERMINATION								
ENDTIM	ENDCYC	DTMIN	ENDENG	ENDMAS	NOSOL		*	ID Title	*
0.0002	0	0	0	0	0			1	
COMMENT									-
						<u>~</u>		<ul> <li>III</li> </ul>	- F
*CONTROL_T ENDIM 0.0002 COMMENT Description								New	
						$\nabla$		Delete	
								Save	
							-	Close	
Description	v of Fields								

Select Save and Close to close the Control\_Termination Keyword Editor.

Again, from the CONTROL list, *select the CONTROL\_TIMESTEP* entry. The associated Keyword Editor will open. This Control Card defines a timestep for the calculation and a scale factor for reduction in timestep, to achieve a stable solution.

Enter 0.6 in the TSSFAC field, and select SAVE and CLOSE, to close the CONTROL\_TIMESTEP Keyword Editor.

p Keyword	l Editor										×
*CONTROL_T	IMESTEP										
DTINIT	TSSFAC	ISDO	TSLIMT	DT2MS	LCTM	ERODE	MS1ST		~	ID Title	^
	0.6	0	✓ 0	0	0	✓ 0	0				
DT2MSF	DT2MSLC	IMSCL	BLANK1	BLANK2	RMSCL						-
		☑ 0			0					•	- F
*CONTROL_T DTINIT DT2MSF COMMENT								*		New	
										Delete	
								-		Save	
									-	Close	
MASS SCALI	IG LIMIT										н

The ALE elements require a special control card, which defines the methods used to make the ALE fluid calculations, and assure a reasonable solution. These parameters are discussed in the LS-DYNA User's Manual.

Carefully enter the values shown in the Keyword Editor image below. The fields that have been modified from the default values are DCT, NADV, METH, AFAC, and VFACT.

ip	Keyword E	ditor										l	x
•	CONTROL_ALE												
	DCT	NADV	METH	AFAC	BFAC		CFAC	DFAC	EFAC	*	ID	Title	~
	2	1	2	-1	0		0	0	0		3		
	START	END	AAFAC	 VFACT	PRIT		EBC	PREF	NSIDEBC				-
	0	1e+020	1	0.0001	0		0 🕑	0			•	III	•
	NCPL	NBKT	IMASCL	CHECKR	BEAMIN		INIJWL	PDIFMX		Ξ		New	
	1	50	0	0	0		0 🔽	0			_		_
ļ	COMMENT											Delete	
									<u>^</u>			Save	
										-		Close	
	11 1.1	to eat ROIMPO	TRUTTLE C	 11 WEET/100	1 6 1.	-	(0)						

Select SAVE and CLOSE to complete the CONTRAL\_ALE card definition.

## 8 Defining Output

The output from this exercise will be in 2 basic forms – binary data that can be displayed in PreSys for post processing, and graphical data that will provide us detailed insight into the results of the simulation.

To request this data using the input file, we will define several DATABASE cards in the model These DATABASE cards may be defined by accessing the Keyword entry on the Model Explorer. The primary parameter defined on the DATABSE cards is the time interval at which the requested data will be written to the file. For each of the DATABASE cards selected, there will be a file created, containing that specific type of data.

To define the binary data, we will right-click on the Keyword entry in the Model Explorer and navigate to the DATABASE (ASCII) entry, and select the DATABASE\_GLSTAT entry. The Keyword Editor for the GLSTAT control card will be displayed.

1 3 7			-
- 1010	DATABASE(ASCII)	•	DATABASE_DEFORC
🕀 B.C.S (3	DATABASE(BINARY)	•	DATABASE_DISBOUT
🖻 Keyword	DEFINE	→	DATABASE_ELOUT
🖻 🛅 A	DEFORMABLE	→	DATABASE_GCEOUT
<u></u> <u></u> <u></u> <u></u> <u></u> <u></u> <u></u> <u></u> <u></u> <u></u>	EF	→	DATABASE_GLSTAT
	ELEMENT	→	DATABASE_H3OUT

Enter a time interval (DT field) of 1E-6 seconds. The GLSTAT global energy data will be written every 1E-6 seconds for the duration of the simulation. Select SAVE and CLOSE to complete the definition of the DATABASE\_GLSTAT card.

In a similar manner, create a DATABASE\_ELOUT card, again using the DT value of 1E-6 second for the output interval.

🏺 Keyword	Editor											×
*DATABASE_E	LOVT											
DT	BINARY	LCUR	IOOPT	OPTION1	OPTION2	OPTION3	OPTION4		*	ID	Title	*
1e-006	1		✓ 1	✓ 0	0	0	0			2		
COMMENT								_				-
								^		•	111	P.
											New	
								Ŧ			Delete	
											Save	
									-		Close	
Double clici	k to set F	LAG TO CONTR	OL THE FREQUE	NCY OF PLOTTING	to default v	value (1)						

Select SAVE and CLOSE to create the DATABASE\_ELOUT card.

The ELOUT database will contain all of the element stresses and strain values for the elements in the model. Note that this data is created only for elements which are contained in Element Sets.

Both the DATABASE\_TRHIST and the DATABASE\_TRACER cards are used to define the displacement, velocity and stresses of history of a particle in the model.

Select DATABASE\_TRHIST from the DATABASE\_ASCII menu and enter the DT value of 1E-6 on the DATABASE\_TRHIST card.

🖗 Keyword Editor		×
*DATABASE_TRHIST DT BINARY LCUR IOOPT 1e-006 1 v v 1 v COMMENT		
DT BINARY LCUR IOOPT	*	ID Title ^
1e-006 1 V V 1 V		3
COMMENT		Ψ.
		< <u> </u>
		New
		Delete
		Save
	-	Close
Description of Fields		

Select SAVE and CLOSE for each of these cards.

Select DATABASE\_TRHIST from the DATABASE menu and enter a location of 0.25,0.005, 0.005 in the x, y, z field of the DATABASE\_TRACER card.

Ŧ	Keyword E	ditor			×
	DATABASE_TR	CER			
	TIME	TRACK X, Y, Z	AMMGID NID	*	ID Title ^
	0	1 25.000000 , 5.000000 , 5.000000	0 0 🗸		
	COMMENT	0: PARTICLE FOLLOWS MATERIAL			
		✓ 1: PARTICLE IS FIXED IN SPACE		*	<
					New
				~	Delete
					Save
				-	Close
D	ouble click	to set TRACKING OPTION to default value			

The \*DATABASE\_ELOUT definition, requires a list of the element(s) in which the histories to be output. This is accomplished by using a DATABASE\_HISTORY\_SOLID (or \_SHELL) to list the element numbers that will be included in the ELOUT file data.

To create the DATABASE\_HISTORY\_option card, navigate to the Keyword entry on the Model Explorer and right-click on the DATABASE entry and select the Create option, and then the DATABASE\_HISTORY option on the submenu.

-3	DATABASE •	DATABASE_ADAMS
- 10	DATABASE(ASCII)	DATABASE_CPM_SENSOR
⊞ B.C.S	DATABASE(BINARY)	DATABASE_CROSS_SECTION
Ė- Keywc	DEFINE	DATABASE_EXTENT
Ē. 🔁	DEFORMABLE	DATABASE_FORMAT
	EF 🕨	DATABASE_FREQUENCY_BINARY
	ELEMENT •	DATABASE_FSI
	EOS +	DATABASE_FSI_SENSOR
	FREQUENCY_DOMAIN >	DATABASE_HISTORY
	HOURGLASS	DATABASE_MASSOUT

A Keyword Editor for the DATABASE\_HISTORY card will be displayed. To create a listing of SOLID elements, the user must *select the option SOLID from the drop down listing in the Options field.* 

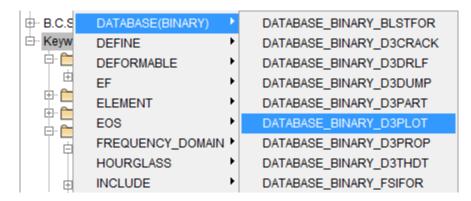
	SHELL	SET					
	🗸 SOLII	1					
	SOLII	_ID					
	SOLII	_SET					
	SPH						
🖗 Keyword	TSHEL	T					
*DATABASE_	TSHEL	T_ID					
OPTIONS	TSHEL	L_SET					
CARD1		1			CARD2	1	
CARD3		1					
ID1	ID2	ID3	ID4	ID5	ID6	ID7	
COMMENT							

To select the individual elements in the model to populate the list, the user must *select the selection icon* ( ) *next to the ID1, ID2, etc. fields.* The user may then select the elements from the Display Area.

Note: another option for output for many elements would be to use the \_SOLID\_SET option, which allows the user to define an SET\_ELEMENT and refer to it on DATABASE\_HISTORY card.

Binary data is defined using the DATABASE\_BINARY\_option card definitions. We will create just a single DATABASE\_BINARY card for this exercise, D3PLOT. This controls the output of binary data that is used to visualize the results in a post processor. The content of the D3PLOT file will vary, depending on the entities in the model. PreSys reads these D3PLOT files easily and allows the user to create animations of any data contained in that file.

Select DATABASE\_BINARY\_D3PLOT from the Keyword entry on the Model Explorer. This will open a keyword editor that allows the user to define the DT parameter of output interval for the D3PLOT file.



Enter a value of 1E-5 in the DT/CYCL field. This will output the complete model state at that interval.

🖗 Keyword Editor		<b></b>
*DATABASE_BINARY_D3PLOT		
DT/CYCL LCDT BEAM NPLTC	*	ID Title ^
1e-005 V V		1
1e-005 v v		· ·
		<
COMMENT		New
		Delete
L		Save
	-	Close
Description of Fields		.H.

This completes all of the input required to set up the simulation. The model may now be exported and submitted to LS-DYNA for calculation.

Select FILE, and EXPORT from the drop down menu at the top of the PreSys window. The user will be provided with several options on what to export and how it should be formatted.

Export							
Range							
All	🔘 Displaye	ed 🔘 🤇	Customize				
Include File							
Preserve							
Export Keyword's Default Value							
Yes		No					
Unit Settings							
Unit	MM, S, TONNE						
Length	MM 💌	Angle	RAD 💌				
Time	S 🔻	TEMP	К 💌				
Mass		Mole	MOL 💌				
ОК		0	ancel				

The user may select All or a portion of the model to export. Select the default options for each of the sections of the Export Task Panel, and select OK.

The model will be exported to the file name that you supplied

## 9 LS-DYNA Analysis

The user may submit the model created in the preceding pages to LS-DYNA for calculation. When the simulation run is completed, there are certain files created, depending upon the output defined for the model.

Output files generated by LS-DYNA:

d3hsp (This is a log file in ascii format) message (This is a log file in ascii format)

The user should check both files for "Warning" and "Error" messages. The description of all nodes, elements, sections, material, loads, curves and all other data defined in the input file would be written to d3hsp file.

BINARY output files generated by LS-DYNA, as a result of the DATABASE\_BINARY\_xxx cards defined by the user. You should find one or more files with the D3PLOT designation. These are the binary plot files for animation and graphics. It contains all the results corresponding to the number of the frequency output defined.

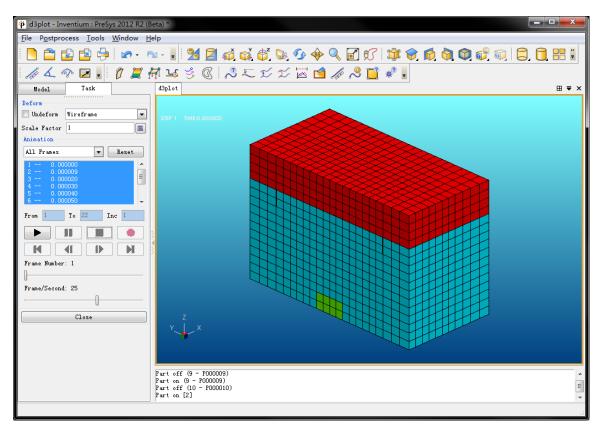
ASCII output files generated by LS-DYNA as requested through the definition of DATABASE\_ASCII\_xxx cards. For our exercise we requested:

GLSTAT: Provides details of various energy for graphical variation ELOUT: Provides details of forces, moments, stress, strain for graphical purposes TRTHIST: Provide details of position, velocities and stress of the selected particle

## 10 Post Processing

The results of the simulation can be viewed using PreSys. To read the binary D3PLOT result files, we Open the files using the FILE/ OPEN command. Navigate to the folder where your results files are located, and select the D3PLOT file. All D3PLOTS located in that folder will be read.

When the complete files are read, the model will be displayed in the Display Area, and the Model Explorer will be populated with Parts for each of the model parts.

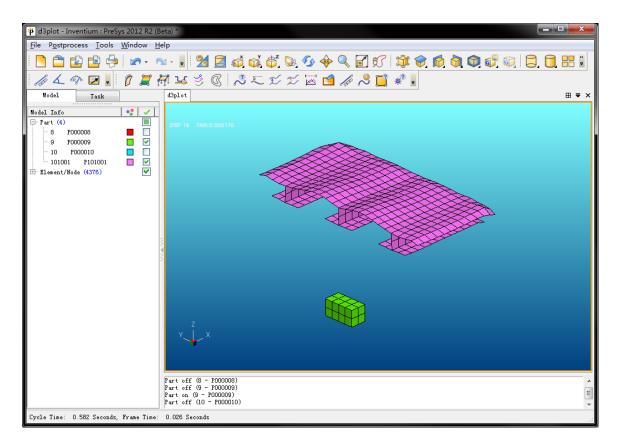


The model should look like the image above. Note that the SHADE and ELEMENT EDGES options are shown, and the model is rotated such that the explosive component is visible.

By default, D3PLOT files are opened with the DEFORM Task Panel active. This panel allows the user to select the desire simulation steps and will construct an animated image of the steps. To begin an animation, *select the PLAY button in the middle of the DEFORM Task Panel.* 

Initially, the user will see no deformation, since the ALE mesh will not deform. However, the user may turn off the ALE mesh parts (part IDs 2 and 3). This will display the Lagrangian shell elements, and the user will see the deformation of that component.

Go to the MODEL tab on the Model Explorer and expand the PART entry. The user may use the visibility check boxes to turn off PIDs 2 and 4.



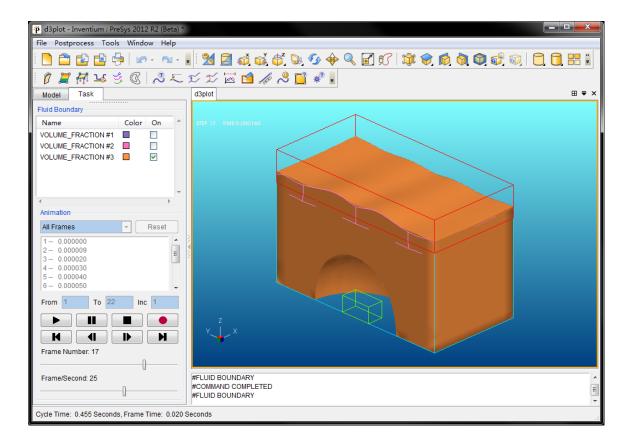
The user can also view the results in the form of Fluid Boundary animations, which display the moving fluid boundary on the model.

Postprocess
Deform
Contour
Vector
Part Interference Check
SPH Control
Isosurface
Fluid Boundary
Change Title
Node Trace

To create an animation of the boundary of the material in fluid meshes, the user can select Fluid Boundary from the drop down Postprocess menu.

The Fluid Boundary command may display the boundary of the material, which help the user to check the result of Fluid-Structure Interaction model. When entering the interface, the model is displayed with the boundaries of each material.

Select VOL\_FRACTION #3 in Part list in Fluid Boundary Task Window. The user may then select the PLAY button, and the animation will be constructed.



# 11 Graphical Data

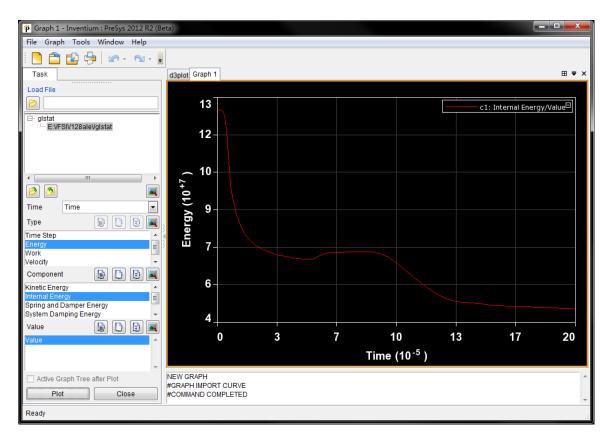
The simulation output that was written to the GLSTAT, ELOUT and TRHIST files may be plotted using X-Y curves.

To create a graph of the energy in the model versus time, we *will navigate to the FILE / NEW GRAPH option using the drop down menu.* A new model tab will be opened where the graph will be displayed.

File	
<u>N</u> ew Model	
New <u>G</u> raph	
<u>O</u> pen	Control-O
<u>C</u> lose	
<u>I</u> mport	FЗ
<u>E</u> xport	Shift-F3
<u>R</u> un Session	

LOAD the GLSTAT result file using the Open File icon. All of the data available for graphing will be displayed. Select ENERGY and then the INTERNAL ENERGY component.

Select PLOT at the bottom of the Task Panel. A graph of the model's internal energy versus time will be displayed.



Note: Additional post processing may be performed on the data generated from the analysis. Please refer to the PreSys Tutorial: Inventium PreSys R2 Post Processing Tutorial for additional information on post processing features.

## 12 Conclusion

This concludes the training overview of the FSI module and post processing using Inventium.

The user should now have basic skills required to generate the Finite element model, set up the analysis and extract the necessary results. The user should also refer to the PreSys and VPG manuals and LS-DYNA User's Manual for more details.



Engineering Technology Associates, Inc. 1133 E. Maple Road, Suite 200, Troy, MI 48083 Phone: +1(248) 729.3010 • Fax: +1(248) 729.3020 sales@eta.com • www.eta.com • www.inventiumsuite.com