



Using the Workbench LS-DYNA Extension



ANSYS, Inc.
Southpointe
2600 ANSYS Drive
Canonsburg, PA 15317
ansysinfo@ansys.com
<http://www.ansys.com>
(T) 724-746-3304
(F) 724-514-9494

Release 18.1
April 2017

ANSYS, Inc. and
ANSYS Europe,
Ltd. are UL
registered ISO
9001:2008
companies.

Copyright and Trademark Information

© 2017 ANSYS, Inc. Unauthorized use, distribution or duplication is prohibited.

ANSYS, ANSYS Workbench, AUTODYN, CFX, FLUENT and any and all ANSYS, Inc. brand, product, service and feature names, logos and slogans are registered trademarks or trademarks of ANSYS, Inc. or its subsidiaries located in the United States or other countries. ICEM CFD is a trademark used by ANSYS, Inc. under license. CFX is a trademark of Sony Corporation in Japan. All other brand, product, service and feature names or trademarks are the property of their respective owners. FLEMI and FLEXnet are trademarks of Flexera Software LLC.

Disclaimer Notice

THIS ANSYS SOFTWARE PRODUCT AND PROGRAM DOCUMENTATION INCLUDE TRADE SECRETS AND ARE CONFIDENTIAL AND PROPRIETARY PRODUCTS OF ANSYS, INC., ITS SUBSIDIARIES, OR LICENSORS. The software products and documentation are furnished by ANSYS, Inc., its subsidiaries, or affiliates under a software license agreement that contains provisions concerning non-disclosure, copying, length and nature of use, compliance with exporting laws, warranties, disclaimers, limitations of liability, and remedies, and other provisions. The software products and documentation may be used, disclosed, transferred, or copied only in accordance with the terms and conditions of that software license agreement.

ANSYS, Inc. and ANSYS Europe, Ltd. are UL registered ISO 9001: 2008 companies.

U.S. Government Rights

For U.S. Government users, except as specifically granted by the ANSYS, Inc. software license agreement, the use, duplication, or disclosure by the United States Government is subject to restrictions stated in the ANSYS, Inc. software license agreement and FAR 12.212 (for non-DOD licenses).

Third-Party Software

See the [legal information](#) in the product help files for the complete Legal Notice for ANSYS proprietary software and third-party software. If you are unable to access the Legal Notice, contact ANSYS, Inc.

Published in the U.S.A.

Table of Contents

1. Using the Workbench LS-DYNA Extension	1
1.1. Licensing Requirements	1
1.2. Running a Distributed Solution	1
1.3. Setting up a Project	1
1.3.1. Defining Materials	2
1.3.2. Attaching Geometry	2
1.3.3. Defining Part Behavior	2
1.3.3.1. Airbag or Simple Pressure Volume	2
1.3.3.2. Adaptive Region	4
1.3.4. Defining Connections	5
1.3.5. Defining Mesh Settings	5
1.3.6. Defining Analysis Settings	6
1.4. Defining Initial Conditions	6
1.5. Defining Boundary Conditions	6
1.5.1. Using the Input File Include Constraint	7
1.5.2. Using the Keyword Snippet (LS-DYNA) Constraint	7
1.6. Accessing Results	7
1.7. Additional LS-DYNA Analysis Tools	9
Index	11

Chapter 1: Using the Workbench LS-DYNA Extension

The ANSYS LS-DYNA program is able to run the complete keyword set published in the LS-DYNA® KEYWORD USER'S MANUAL (R8.1). In the current implementation, a limited subset of the extensive list of keyword inputs can be generated by Workbench LS-DYNA. The keywords generated through the standard user interface are supported by ANSYS technical support. The following areas of use are not supported with Workbench LS-DYNA:

- User-defined material definition
- LS-DYNA user subroutines
- Mesh-free methods: SPH & EFG
- Some special elements such as Seat belt and others
- Fluid Structure Interaction (FSI)

Advanced, knowledgeable users may modify the input file generated and take advantage of the full keyword set, however ANSYS, Inc. does not provide technical support for modified input files. Note that post-processing may not work correctly with modified .k files. For those cases use LS-Prepost, available in the standard installation.

1.1. Licensing Requirements

The Workbench LS-DYNA environment is available to all customers with an ANSYS LS-DYNA license and requires that license to run problems.

SMP and MPP Parallel processing are both available. They do, however, require the use of ANSYS LS-DYNA HPC licenses. The standard ANSYS HPC packs and HPC Workgroup licenses do not work with ANSYS LS-DYNA. Use ANSYS LS-DYNA HPC licenses for running in parallel.

1.2. Running a Distributed Solution

If you are planning to use the LS-DYNA distributed solution, LS-DYNA is using Intel MPI, which needs additional configuration and password caching. To cache your Windows password, open a Command Prompt window and run:

```
%AWP_ROOT180%\commonfiles\MPI\Intel\5.1.3.180\Windows\setimpipassword.bat
```

1.3. Setting up a Project

The general outline of setting up an explicit dynamics project can be found in the Explicit Dynamics Workflow Chapter of the Explicit Dynamics Analysis Guide. LS-DYNA related information is included in that chapter, but that material was originally written to explain LS-DYNA behavior with regard to the LS-DYNA Export system. Workbench LS-DYNA has improved the use of LS-DYNA from within Workbench and those differences are noted in this section.

1.3.1. Defining Materials

You can find information on Materials in the Define Engineering Data section of the Explicit Dynamics Analysis Guide.

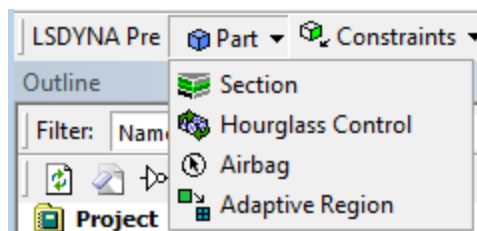
1.3.2. Attaching Geometry

You can find information about Geometry in the Attach Geometry section of the Explicit Dynamics Analysis Guide.

1.3.3. Defining Part Behavior

You can find information about Part Behavior in the Define Part Behavior section of the Explicit Dynamics Analysis Guide.

Add the objects on the LSDYNA Pre toolbar under **Part** to specify **Section** properties such as element formulations, and specify different **Hourglass Control** objects for each part.



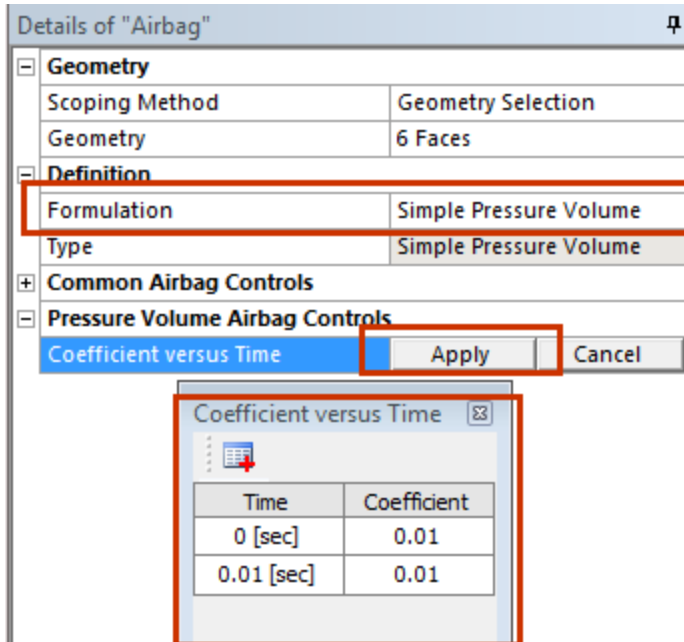
You can also add an airbag/pressure volume object or an adaptive region to your model.

1.3.3.1. Airbag or Simple Pressure Volume

The Airbag object provides a way of defining thermodynamic behavior of the gas flow into the airbag as well as a reference configuration for the fully inflated bag.

Details of "Airbag"	
[-] Geometry	
Scoping Method	Geometry Selection
Geometry	6 Faces
[-] Definition	
Formulation	Simple Pressure Volume
Type	Simple Pressure Volume
[-] Common Airbag Controls	
Volume Scale Factor	1
Pressure Scale Factor	1
Initial filled Volume	0 [m m m]
Mass Weighted Damping Factor	500
Stagnation Pressure Scale Factor	0
[-] Pressure Volume Airbag Controls	
Coefficient versus Time	Tabular Data

Simple Pressure Volume



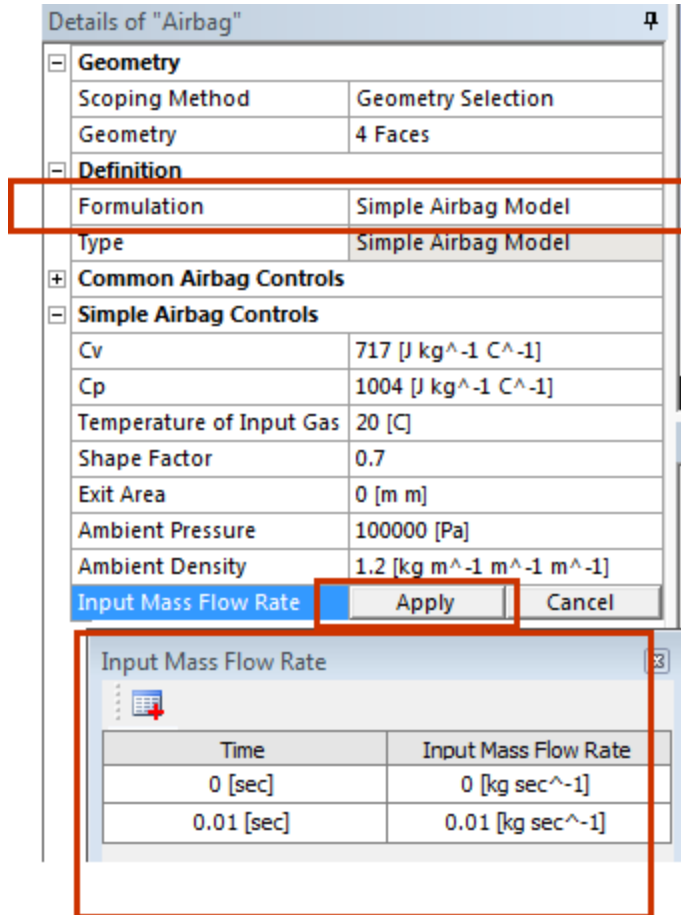
When **Formulation** is set to Simple Pressure Volume:

- The pressure is a function of the ratio of current volume to the initial volume.
- This simple model can be used when an initial pressure is given.
- No leakage, no temperature change, and no input mass flow are assumed.
- A typical application is the modeling of air in automobile tires.
- Pressure Volume Airbag controls: Coefficient versus Time (CN) can be defined in a table.

$$Pressure = \beta \frac{CN}{Relative\ Volume}$$

$$Relative\ Volume = \frac{Current\ Volume}{Initial\ Volume}$$

Simple Airbag Model



The volume pressure relationships is defined by the Simple Airbag Model for control volumes.

- The gamma law equation of state used to determine the pressure in the airbag:

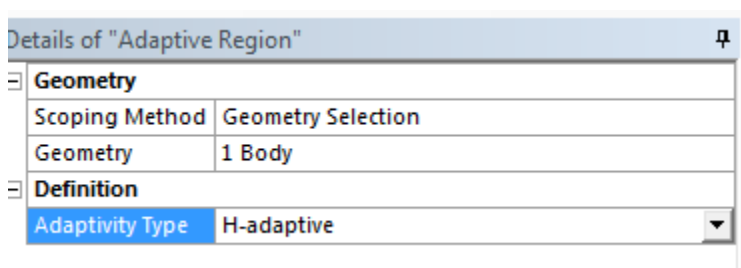
$$p = (\gamma - 1)\rho e$$

where p is the pressure, ρ is the density, e is the specific internal energy of the gas, and γ is the ratio of the specific heats:

$$\gamma = c_p/c_v$$

- Input Mass Flow Rate can be defined in a table.

1.3.3.2. Adaptive Region



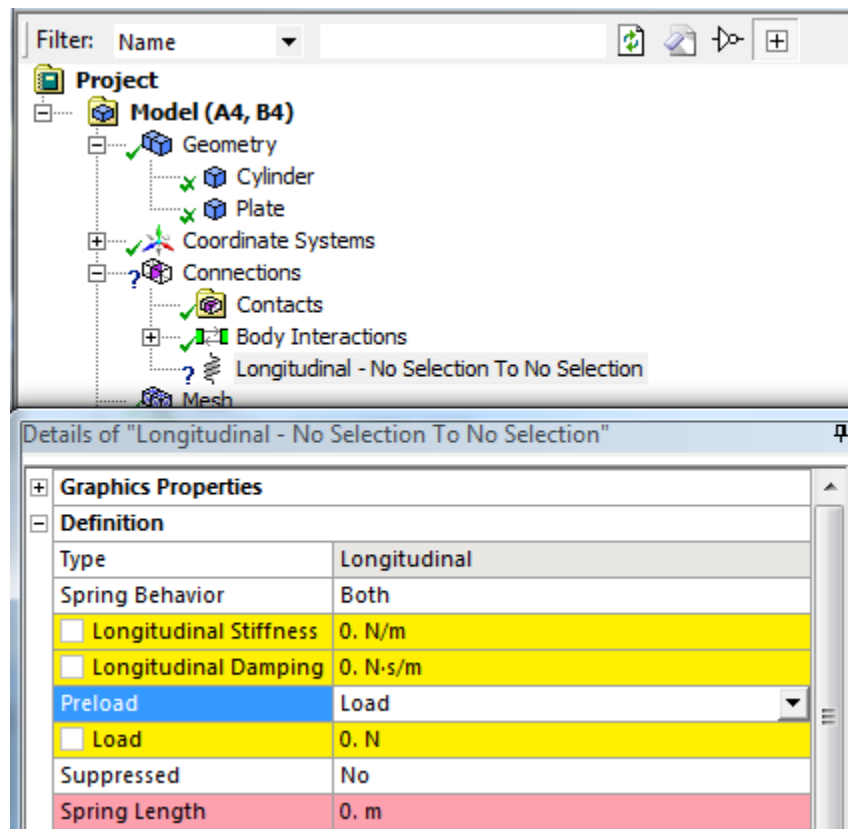
In metal forming and high-speed impact analyses, a body may experience very large amounts of plastic deformation. Single point integration explicit elements, which are usually robust for large deformations, may give inaccurate results in these situations due to inadequate element aspect ratios. To counteract this problem, the ANSYS LS-DYNA program has the ability to automatically remesh a surface during an analysis to improve its integrity. This capability, known as adaptive meshing, is controlled with the adaptive region:

- -adaptive for 3-D shells.
- Passive -adaptive for 3-D shells. The elements in this part will not be split unless their neighboring elements in other parts need to be split more than one level.

At 18.1 when this feature is used, the postprocessing has to be done within LS-Prepost

1.3.4. Defining Connections

You can find information about Connections in the Define Connections section of the Explicit Dynamics Analysis Guide. In Workbench LS-DYNA Springs can be added in Connections. Both Longitudinal Stiffness and Damping are supported. Pre-loads are supported. Supported spring behavior is linear only.



1.3.5. Defining Mesh Settings

You can find information about Mesh Settings in the Apply Mesh Controls/Preview Mesh section of the Explicit Dynamics Analysis Guide.

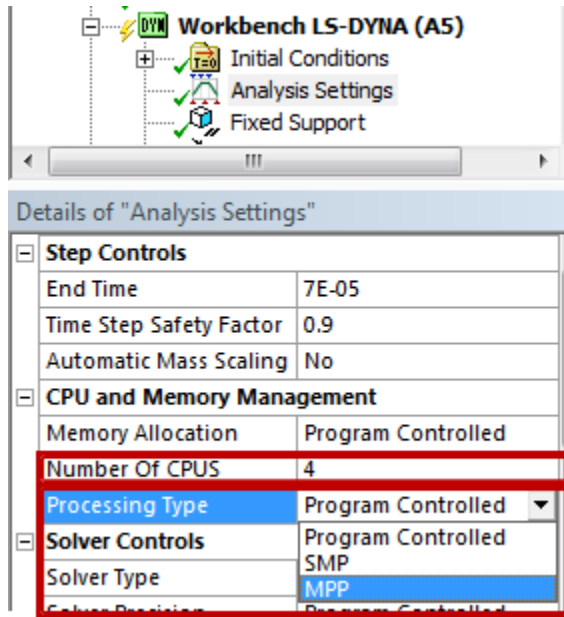
Dimensionally Reduced Rigid Body Behavior Is not available for Workbench LS-DYNA.

1.3.6. Defining Analysis Settings

You can find information about Analysis Settings in the Establish Analysis Settings section of the Explicit Dynamics Analysis Guide.

You can set **Solver Precision** to Single or Double under **Solver Controls**, or leave it as Program Controlled.

In Workbench LS-DYNA, the **End Time** is the only required input. SMP or MPP parallel processing can be activated by changing the Number of CPU's. Note that ANSYS LS-DYNA HPC licenses are required for using more than 1 core.

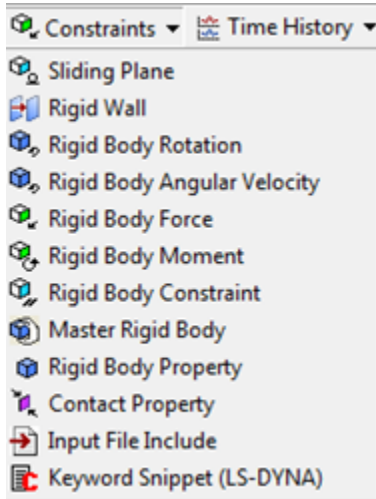


1.4. Defining Initial Conditions

You can find information about Initial Conditions in the Define Initial Conditions section of the Explicit Dynamics Analysis Guide.

1.5. Defining Boundary Conditions

You can find information about Boundary Conditions in the Apply Loads and Supports section of the Explicit Dynamics Analysis Guide. In Workbench LS-DYNA. In addition to the boundary conditions discussed there, Remote Displacement is available, and also the Rigid Body and Rigid Wall constraints from the drop-down on the LSDYNA Pre toolbar.



1.5.1. Using the Input File Include Constraint

Advanced users can insert additional LS-DYNA keywords by using the **Input File Include** constraint to specify the name of a file that contains LS-DYNA keywords. An include file `*INCLUDE Filename1` keyword card will be generated pointing to the file you specify.

Included files can contain any valid LS-DYNA keyword cards. Using include files eliminates the need for you to edit the `.k` input file each time the file is created by Workbench LS-DYNA if you have other keywords you want to use. Note that the included file can contain other include file statements, providing a fairly general capability for easily adding predefined inputs.

Note

ANSYS Support does not provide technical assistance for keywords not generated by Workbench LS-DYNA.

1.5.2. Using the Keyword Snippet (LS-DYNA) Constraint

You can also insert additional LS-DYNA keywords by using the **Keyword Snippet (LS-DYNA)** constraint to insert individual LS-DYNA keywords. Note that this can only be inserted under Connections to use contact types not supported by Workbench LS-DYNA.

Note

ANSYS Support does not provide technical assistance for keywords not generated by Workbench LS-DYNA.

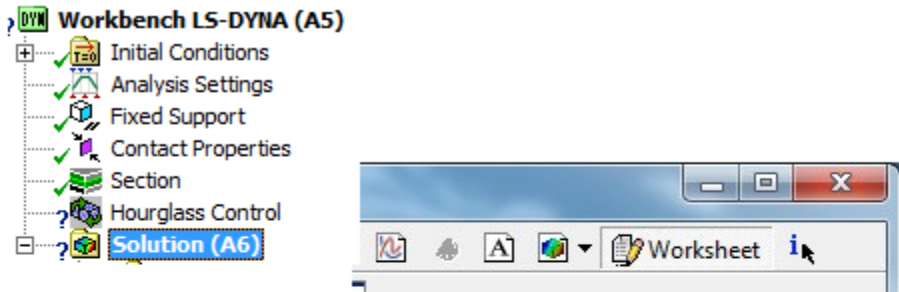
1.6. Accessing Results

You can find information about Results Processing in the Postprocessing section of the Explicit Dynamics Analysis Guide.

To collect Nodal Data, insert one or more Result Trackers under Solution prior to running the simulation. Result trackers must be scoped to a node.

Stress and Plastic Strain are written by default (can be suppressed) but Strain is not written by default and you must select it if desired.

LS-DYNA keeps track of Total Strain and Plastic Strain. Elastic Strain will always show as 0. To plot elastic plastic strain or elastic strain, click on Solution, then click on Worksheet and select from the User Defined Result Expressions within the Worksheet. Right click on the expression and select Create User Defined Result.



Worksheet

User Defined Result Expressions

Type	Data Type	Data Style	Component	Expression	Output Unit
EPPL	Element Nodal	Scalar	XY	EPPLXY	Strain
EPPL	Element Nodal	Scalar	YZ	EPPLYZ	Strain
EPPL	Element Nodal	Scalar	XZ	EPPLXZ	Strain
EPPL	Element Nodal	Scalar	1	EPPL1	Strain
EPPL	Element Nodal	Scalar	2	EPPL2	Strain
EPPL	Element Nodal	Scalar	3	EPPL3	Strain
EPPL	Element Nodal	Scalar	INT	EPPLINT	Strain
EPPL	Element Nodal	Tensor	VECTORS	EPPLVECTORS	Strain
EPPL	Element Nodal	Scalar	MAXSHEAR	EPPLMAXSHEAR	Strain
EPS	Element Nodal	Scalar		FPS	Strain
TEMP	Nodal	Scalar			Temperature
BEAM_AXIAL_FX	Element Nodal	Scalar		BEAM_AXIAL_FX	Force

A context menu is open over the 'EPS' row, showing the option 'Create User Defined Result'.

Time History Outputs (ASCII files) can be generated. All the files shown below can be created by ANSYS LS-DYNA from the Workbench LS-DYNA GUI. Note however that some data in the ASCII files currently cannot be viewed.

Details of "Analysis Settings"	
+	Step Controls
+	CPU and Memory Management
+	Solver Controls
+	Damping Controls
+	Hourglass Controls
+	ALE Controls
+	Output Controls
-	Time History Output Controls
Calculate Results At	Equally Spaced Points
--- Value	40
Output	Boundary Condition Forces and Energy Deformed Geometry Data Discrete Elements Data Geometry Contact Entities Joint Force Data Material energies Data Nodal Constraint Reaction Force Data (spotwel Nodal Data Nodal Interface Forces Resultant Interface Force Data Rigid Body Data Sliding Interface Energy Spc Reaction Forces Data Wall Forces

To view Time History Outputs select the ones desired from the ASCII drop down menu.

Using LS-DYNA terminology, the following results can be viewed within Workbench LS-DYNA:

- GLSTAT (Global Data : Kinetic Energy, Hourglass, ...).
- BNDOUT (Boundary Conditions Data).
- RCFORC (Contact Forces Data).
- SPCFORC (Reaction Forces on Boundary Conditions) using *BOUNDARY_SPC (Fixed Support) to view Reaction Force.
- MATSUM (Body Data).
- NODOUT (Nodal Data) Trackers must be defined during pre-processing for the nodal data to be available during Results processing.

1.7. Additional LS-DYNA Analysis Tools

When the Workbench LS-DYNA system is selected in the tree view, you will see these additional analysis tools on the toolbar:



- - use this tool to specify rigid body(ies) by directly scoping to the geometry.



- - use this tool to merge multiple rigid bodies together into a single rigid body.



- - use this tool to add additional nodes from flexible body to the existing rigid body.



- - use this tool to add Dynamic Relaxation to the project.



- - use this tool to define the time step for the analysis of rigid body dynamics.



- - use this tool to calculate CFL time step before the calculation starts.



- - use this tool to write the .k file of the Workbench LS-DYNA model to the specified directory, including all keywords from Keyword Snippets and Include Files.

Index

