LS-DYNA®
KEYWORD USER'S MANUAL

VOLUME III

Multi-Physics Solvers

February 19, 2013 (revision: 2754)
R7

LIVERMORE SOFTWARE TECHNOLOGY CORPORATION (LSTC)

Corporate Address
Livermore Software Technology Corporation
P. O. Box 712
Livermore, California 94551-0712

Support Addresses
Livermore Software Technology Corporation
7374 Las Positas Road
Livermore, California 94551
Tel: 925-449-2500 Fax: 925-449-2507
Email: sales@lstc.com
Website: www.lstc.com

Livermore Software Technology Corporation
1740 West Big Beaver Road
Suite 100
Troy, Michigan 48084
Tel: 248-649-4728 Fax: 248-649-6328

Disclaimer

TABLE OF CONTENTS

*TABLE OF CONTENTS

LS-DYNA MULTIPHYSICS USER'S MANUAL

INTRODUCTION

In this manual, there are three main solvers: a compressible flow solver, an incompressible flow solver, and an electromagnetism solver. Each of them implements coupling with the structural solver in LS-DYNA.

The keywords covered in this manual fit into one of three categories. In the first category are the keyword cards that provide input to each of the multiphysics solvers that in turn couple with the structural solver. In the second category are keyword cards involving extensions to the basic solvers. Presently, the chemistry and stochastic particle solvers are the two solvers in this category, and they are used in conjunction with the compressible flow solver discussed below. In the third category are keyword cards for support facilities. A volume mesher that creates volume tetrahedral element meshes from bounding surface meshes is one of these tools. Another is a new data output mechanism for a limited set of variables from the solvers in this manual. This mechanism is accessed through *LSO keyword cards.

The CESE solver is a compressible flow solver based upon the Conservation Element/Solution Element (CE/SE) method, originally proposed by Chang of the NASA Glenn Research Center. This method is a novel numerical framework for conservation laws. It has many non-traditional features, including a unified treatment of space and time, the introduction of separate conservation elements (CE) and solution elements (SE), and a novel shock capturing strategy without using a Riemann solver. This method has been used to solve many types of flow problems, such as detonation waves, shock/acoustic wave interaction, cavitating flows, supersonic liquid jets, and chemically reacting flows. In LS-DYNA, it has been extended to also solve fluid-structure interaction (FSI) problems. It does this with two approaches. The first approach solves the compressible flow equations on an Eulerian mesh while the structural mechanics is solved on a moving mesh that moves through the fixed CE/SE mesh. In the second approach (new with this version), the CE/SE mesh moves in a fashion such that its FSI boundary surface matches the corresponding FSI boundary surface of the moving structural mechanics mesh. This second approach is more accurate for FSI problems, especially with boundary layers flows. Another new feature with the CESE moving mesh solver is conjugate heat transfer coupling with the solid thermal solver. The chemistry and stochastic particle solvers are two addon solvers that extend the CESE solver.

The second solver is the incompressible flow solver (ICFD) that is fully coupled with the solid mechanics solver. This coupling permits robust FSI analysis via either an explicit technique when the FSI is weak, or using an implicit coupling when the FSI coupling is strong. In addition to being able to handle free surface flows, there is also a bi-phasic flow capability that involves modeling using a conservative Lagrangian interface tracking technique. Basic turbulence models are also supported. This solver is the first in LS-DYNA to make use of a new volume mesher that takes surface meshes bounding the fluid domain as input (*MESH keywords). In addition, during the time advancement of the incompressible flow, the solution is adaptively re-meshed as an automatic feature of the solver. Another important feature of the mesher is the ability to create boundary layer meshes. These aniso-

tropic meshes become a crucial part of the model when shear stresses are to be calculated near fluid walls. The ICFD solver is also coupled to the solid thermal solver using a monolithic approach for conjugate heat transfer problems.

The third solver is an electromagnetics (EM) solver. This module solves the Maxwell equations in the Eddy current (induction-diffusion) approximation. This is suitable for cases where the propagation of electromagnetic waves in air (or vacuum) can be considered as instantaneous. Therefore, the wave propagation is not solved. The main applications are Magnetic Metal Forming, bending or welding, induced heating, ring expansions and so forth. The EM module allows the introduction of a source of electrical current into solid conductors and the computation of the associated magnetic field, electric field, as well as induced currents. The EM solver is coupled with the structural mechanics solver (the Lorentz forces are added to the mechanics equations of motion), and with the structural thermal solver (the ohmic heating is added to the thermal solver as an extra source of heat). The EM fields are solved using a Finite Element Method (FEM) for the conductors and a Boundary Element Method (BEM) for the surrounding air/insulators. Thus no air mesh is necessary.

As stated above, the *CHEMISTRY and *STOCHASTIC cards are only used in the CESE solver at this time.

*CESE

The keyword *CESE provides input data for the Conservation Element/Solution Element (CESE) compressible fluid solver:

***CESE_BOUNDARY_AXISYMMETRIC**_{*OPTION*}

***CESE_BOUNDARY_FSI**_{*OPTION*}

***CESE_BOUNDARY_NON_REFLECTIVE**_{*OPTION*}

***CESE_BOUNDARY_PRESCRIBED**_{*OPTION*}

***CESE_BOUNDARY_REFLECTIVE**_{*OPTION*}

***CESE_BOUNDARY_SOLID_WALL**_{*OPTION1*}_{*OPTION2*}

***CESE_CONTROL_LIMITER**

***CESE_CONTROL_MESH_MOV**

***CESE_CONTROL_SOLVER**

***CESE_CONTROL_TIMESTEP**

***CESE_EOS_CAV_HOMOG_EQUILIB**

***CESE_EOS_IDEAL_GAS**

***CESE_INITIAL**

***CESE_INITIAL**_{*OPTION*}

***CESE_INITIAL_CHEMISTRY**

***CESE_INITIAL_CHEMISTRY_ELEMENT**

***CESE_INITIAL_CHEMISTRY_PART**

***CESE_INITIAL_CHEMISTRY_SET**

***CESE_MAT_GAS**

***CESE_PART**

Note that when performing a chemistry calculation with the CESE solver, initialization should only be done with the **CESE_INITIAL_CHEMISTRY** cards, not the **CESE_INITIAL** cards.

An additional option "**_TITLE"** may be appended to all ***CESE** keywords. If this option is used, then an 80 character string is read as a title from the first card of that keyword's input. At present, LS-DYNA does not make use of the title. Inclusion of titles gives greater clarity to input decks.

***CESE_BOUNDARY_AXISYMMETRIC**_*OPTION*

Available options are

PART

SET

SEGMENT

Purpose: Define an axisymmetric boundary condition on the axisymmetric axis for the special 2D axisymmetric CESE compressible flow solver.

For the PART option define the following card:

Card 1	1	2	3	4	5	6	7	8
Variable	SURFPRT							
Type	I							
Default	none							

For the SET option define the following card:

Card 1	1	2	3	4	5	6	7	8
Variable	SSID							
Type	I							
Default	none							

For the SEGMENT option define the following card:

Card 1	1	2	3	4	5	6	7	8
Variable	N1	N2	N3	N4				
Type	I	I	I	I				
Default	none	none	none	none				

VARIABLE	DESCRIPTION
SURFPRT	Surface part ID referenced in ***MESH_SURFACE_ELEMENT** cards.
SSID	Segment set ID
N1,N2...	Node ID's defining a segment

Remarks:

1. This boundary condition can only be used on the axisymmetric axis for the 2D axisymmetric CESE fluid solver.

***CESE_BOUNDARY_FSI**_*OPTION*

Available options are:

PART

SET

SEGMENT

Purpose: Define an FSI boundary condition for the CESE compressible flow solver. This boundary condition must be applied on a surface of the CESE computational domain that is shared with surfaces of the outside boundary of the structural mechanics solver. The nodes of the two meshes will generally not be shared.

For the PART option define the following card:

Card 1	1	2	3	4	5	6	7	8
Variable	SURFPRT							
Type	I							
Default	none							

For the SET option define the following card:

Card 1	1	2	3	4	5	6	7	8
Variable	SSID							
Type	I							
Default	none							

For the SEGMENT option define the following card:

Card 1	1	2	3	4	5	6	7	8
Variable	N1	N2	N3	N4				
Type	I	I	I	I				
Default	none	none	none	none				

VARIABLE	DESCRIPTION
SURFPRT	Surface part ID referenced in ***MESH_SURFACE_ELEMENT** cards.
SSID	Segment set ID.
N1, N2 ...	Node ID's defining a segment

Remarks:

1. This boundary condition card is also needed for conjugate heat transfer problems with the CESE solver.

CESE_BOUNDARY_NON_REFLECTIVE_*OPTION

Available options are:

PART

SET

SEGMENT

Purpose: Define a passive boundary condition for CESE compressible flows. This non-reflective boundary condition (NBC) provides an artificial computational boundary for an open boundary that is passive.

For the PART option define the following card:

Card 1	1	2	3	4	5	6	7	8
Variable	SURFPRT							
Type	I							
Default	none							

For the SET option define the following card:

Card 1	1	2	3	4	5	6	7	8
Variable	SSID							
Type	I							
Default	none							

For the SEGMENT option define the following card:

Card 1	1	2	3	4	5	6	7	8
Variable	N1	N2	N3	N4				
Type	I	I	I	I				
Default	none	none	none	none				

VARIABLE	DESCRIPTION
SURFPRT	Surface part ID referenced in ***MESH_SURFACE_ELEMENT** cards.
SSID	Segment set ID
N1, N2, …	Node ID's defining a segment

Remarks:

1. This boundary condition is usually imposed on an open surface that is far from the main disturbed flow (the farther away, the better), i.e., the flow on that boundary surface should be almost uniform.

2. If any boundary segment has not been assigned a boundary condition by any of the ***CESE _BOUNDARY_**… cards, then it will automatically be assigned this non-reflective boundary condition.

***CESE_BOUNDARY_PRESCRIBED**_*OPTION*

Available options include:

PART

SET

SEGMENT

Purpose: For the CESE compressible flow solver, impose flow variable(s) (velocity, density, pressure, temperature, etc.) at the centroid of the element connected with this boundary segment (***OPTION*=SEGMENT**), or at the centroid of the elements connected with each segment in a set of boundary segments (***OPTION*=SET**) or at the centroid of the elements (created by ***MESH_VOLUME** cards) that are connected with each face in a surface part (PID) defined with ***MESH_SURFACE_ELEMENT** cards (***OPTION*=PART**) .

For the PART option define the following card:

Card 1	1	2	3	4	5	6	7	8
Variable	SURFPRT	IDCOMP						
Type	I	I						
Default	none	none						

For the SET option define the following card:

Card 1	1	2	3	4	5	6	7	8
Variable	SSID	IDCOMP						
Type	I	I						
Default	none	none						

For the SEGMENT option define the following card:

Card 1	1	2	3	4	5	6	7	8
Variable	N1	N1	N3	N4	IDCOMP			
Type	I	I	I	I	I			
Default	none	none	none	none	none			

Card 2	1	2	3	4	5	6	7	8
Variable	LC_U	LC_V	LC_W	LC_RHO	LC_P	LC_T		
Type	I	I	I	I	I	I		
Remarks	1,2	1,2	1,2	1,2	1,2	1,2		

Card 3	1	2	3	4	5	6	7	8
Variable	SF_U	SF_V	SF_W	SF_RHO	SF_P	SF_T		
Type	F	F	F	F	F	F		
Default	1.0	1.0	1.0	1.0	1.0	1.0		
Remarks	1	1	1	1	1	1		

VARIABLE	DESCRIPTION
SURFPRT	A surface part ID referenced in ***MESH_SURFACE_ELEMENT** cards
SSID	Segment set ID
N1, N2, …	Node ID's defining a segment

VARIABLE	DESCRIPTION
IDCOMP	For inflow boundaries in problems involving chemical reacting flows, the chemical mixture of the fluid entering the domain as defined with a ***CHEMISTRY_COMPOSITION** card.
LC_U	Load curve ID to describe the x-component of the velocity versus time; see ***DEFINE_CURVE**.
LC_V	Load curve ID to describe the y-component of the velocity versus time.
LC_W	Load curve ID to describe the z-component of the velocity versus time.
LC_RHO	Load curve ID to describe the density versus time.
LC_P	Load curve ID to describe the pressure versus time.
LC_T	Load curve ID to describe the temperature versus time.
SF_U	Scale factor for LC_U (default = 1.0).
SF_V	Scale factor for LC_V (default = 1.0).
SF_W	Scale factor for LC_W (default = 1.0).
SF_RHO	Scale factor for LC_RHO (default = 1.0).
SF_P	Scale factor for LC_P (default = 1.0).
SF_T	Scale factor for LC_T (default = 1.0).

Remarks:

1. On each centroid or set of centroids, the variables (x-vel, y-vel, z-vel, ρ, P, T) that are given values must be consistent and make the model well-posed (i.e., be such that the solution of the model exists, is unique and physical).

2. If any of the load curves are 0, the corresponding variable will take the constant value of the corresponding scale factor. For instance, if LC_RHO=0, then the constant value of the density for this boundary condition will be SF_RHO.

***CESE_BOUNDARY_REFLECTIVE**_*OPTION*

Available options are:

PART

SET

SEGMENT

Purpose: Define a reflective boundary condition (RBC) for the CESE compressible flow solver. This boundary condition can be applied on a symmetrical surface or a solid wall of the computational domain.

For the PART option define the following card:

Card 1	1	2	3	4	5	6	7	8
Variable	SURFPRT							
Type	I							
Default	none							

For the SET option define the following card:

Card 1	1	2	3	4	5	6	7	8
Variable	SSID							
Type	I							
Default	none							

For the SEGMENT option define the following card:

Card 1	1	2	3	4	5	6	7	8
Variable	N1	N2	N3	N4				
Type	I	I	I	I				
Default	none	none	none	none				

VARIABLE	DESCRIPTION
SURFPRT	Surface part ID referenced in ***MESH_SURFACE_ELEMENT** cards.
SSID	Segment set ID.
N1, N2 ...	Node ID's defining a segment

Remarks:

2. This boundary condition has the same effect as a solid-wall boundary condition for inviscid flows.

***CESE_BOUNDARY_SOLID_WALL**_*OPTION1_OPTION2*

For *OPTION1* the choices are:

PART

SET

SEGMENT

For *OPTION2* the choices are:

<BLANK>

ROTAT

Purpose: Define a solid wall boundary condition (SBC) for this CESE compressible flow solver. This boundary condition can be applied at a solid boundary that is the physical boundary for the flow field. For inviscid flow, this will be a slip boundary condition; while for viscous flows, it is a no-slip boundary condition.

For the PART option define the following card:

Card 1	1	2	3	4	5	6	7	8
Variable	SURFPRT	LCID	Vx	Vy	Vz	Nx	Ny	Nz
Type	I	I	F	F	F	F	F	F
Default	none	0	0.0	0.0	0.0	0.0	0.0	0.0
Remarks		2, 3	2	2	2	3	3	3

For the SET option define the following card:

Card 1	1	2	3	4	5	6	7	8
Variable	SSID	LCID	Vx	Vy	Vz	Nx	Ny	Nz
Type	I	I	F	F	F	F	F	F
Default	none	0	0.0	0.0	0.0	0.0	0.0	0.0
Remarks		2, 3	2	2	2	3	3	3

For the SEGMENT option define the following two cards:

Card 1	1	2	3	4	5	6	7	8
Variable	N1	N2	N3	N4	LCID	Vx	Vy	Vz
Type	I	I	I	I	I	F	F	F
Default	none	none	none	none	0	0.0	0.0	0.0
Remarks					2, 3	2	2	2

Card 2	1	2	3	4	5	6	7	8
Variable	Nx	Ny	Nz					
Type	F	F	F					
Default	0.0	0.0	0.0					
Remarks	3	3	3					

VARIABLE	DESCRIPTION
SURFPRT	Surface part ID referenced in ***MESH_SURFACE_ELEMENT** cards.
SSID	Segment set ID

VARIABLE	DESCRIPTION
N1, N2 …	Node ID's defining a segment
LCID	Load curve ID to define this solid wall boundary movement

If *OPTION2* = **<BLANK>**:

Vx, Vy, Vz	velocity vector of the solid wall: LCID.EQ.0: it is defined by (Vx,Vy,Vz) itself; LCID.NE.0: it will be defined by both of the load curve and (Vx,Vy,Vz). Nx, Ny, Nz are not used in this case.

If *OPTION2* = **ROTAT**:

Vx, Vy, Vz	x-,y- & z-coordinates of a point on the rotating axis
Nx, Ny, Nz	Unit vector of the rotating axis (for the 2D case, this is not used). The rotating frequency (Hz) is given by the load curve.

Remarks:

1. In this solid-wall condition (SBC), the boundary movement can only be in the tangential direction of the wall and should not affect the fluid domain size and mesh during the calculation, otherwise an FSI or moving mesh solver should be used. Also, this moving SBC only affects viscous flows (no-slip BC).

2. If LCID=0 and Vx=Vy=Vz=0.0 (default), this will be a regular solid wall BC.

3. For rotating SBC, LCID>0 must be used to define the rotating speed (frequency (Hz)). Also, in the 2D case, (Nx, Ny, Nz) does not need to be defined because it is not needed.

*CESE_CONTROL_LIMITER

Purpose: Sets some stability parameters used in the CESE scheme for this CESE compressible flow solver.

Card 1	1	2	3	4	5	6	7	8
Variable	IDLMT	ALFA	BETA	EPSR				
Type	I	F	F	F				
Default	0	0.0	0.0	0.0				
Remarks		1	2	3				

VARIABLE	DESCRIPTION
IDLMT	Set the stability limiter option (See CESE theory manual): EQ.0: limiter format 1 : Re-weighting EQ.1: limiter format 2 : Relaxing
ALFA	Re-weighting coefficient (See CESE theory manual)
BETA	Numerical viscosity control coefficient (See CESE theory manual)
EPSR	Stability control coefficient (See CESE theory manual)

Remarks:

1. $\alpha \geq 0.0$; larger values give more stability, but less accuracy. Usually $\alpha=2.0$ or 4.0 will be enough for normal shock problems.

2. $0.0 \leq \beta \leq 1.0$; larger values give more stability. For problems with shock waves, $\beta=1.0$ is recommended.

3. $\varepsilon \geq 0.0$; larger values give more stability, but less accuracy.

*CESE_CONTROL_MESH_MOV

Purpose: For moving mesh CESE, this keyword is used to choose the type of algorithm to be used for calculating mesh movement.

Card 1	1	2	3	4	5	6	7	8
Variable	MMSH	LIM_ITER	RELTOL	ABSTOL				
Type	I	I	F	F				
Default	1	100	1.0e-3	1.0e-3				

VARIABLE	DESCRIPTION
MMSH	Mesh motion selector: EQ.1: mesh moves using an implicit ball-vertex spring method. EQ.9: the IDW scheme is used to move the mesh.
LIM_ITER	Maximum number of linear solver iterations for the ball-vertex linear system.
RELTOL	Relative tolerance to use as a stopping criterion for the iterative linear solver (conjugate gradient solver with diagonal scaling preconditioner).
ABSTOL	Absolute tolerance measure for the size of mesh displacement changes to use as a stopping criterion for the iterative linear solver.

.

***CESE_CONTROL_SOLVER**

Purpose: Set general purpose control variables for the CESE compressible flow solver.

Card 1	1	2	3	4	5	6	7	8
Variable	IFRAME	IFLOW	IGEOM					
Type	I	I	I					
Default	0	0	0					
Remarks			1, 2, 3, 4					

VARIABLE	DESCRIPTION
IFRAME	Sets the framework of the CESE solver. EQ.0: Fixed Eulerian EQ.100: Moving Mesh FSI EQ.200: Immersed boundary FSI
IFLOW	Sets the compressible flow types: EQ.0: Viscous flows (laminar) EQ.1: Invisid flows
IGEOM	Sets the geometric dimension: EQ.0: 2D or 3D will be selected based upon the input mesh and the given boundary conditions. EQ.2: two-dimensional (2D) problem EQ.3: three-dimensional (3D) problem EQ.101 2D axisymmetric

Remarks:

1. If the user wants to use the 2D (IGEOM=2) or 2D axisymmetric (IGEOM=101) solver, the mesh should only be distributed in the x-y plane (one layer) with the boundary conditions given only at the x-y domain boundaries. Otherwise, a warning message will be given and the 3D solver will be triggered instead.

2. If IGEOM=0 (default), the code will automatically check the mesh and the given boundary conditions to decide the problem geometry type (2D or 3D).

3. The 2D axisymmetric case will work only if the 2D mesh and corresponding boundary conditions are properly defined, with the x and y coordinates corresponding to the axial and radial directions respectively.

4. Currently, the FSI solver only runs in 3D cases.

*CESE_CONTROL_TIMESTEP

Purpose: Sets the time-step control parameters for the CESE compressible flow solver.

Card 1	1	2	3	4	5	6	7	8
Variable	IDDT	CFL	DTINT					
Type	I	F	F					
Default	0	0.9	1.0E-3					

VARIABLE	DESCRIPTION
IDDT	Sets the time step option: EQ.0: Fixed time step size (DTINT, i.e., given initial time step size) NE.0: the time step size will be calculated based on the given CFL-number and the flow solution at the previous time step.
CFL	CFL number (Courant–Friedrichs–Lewy condition) ($0.0 < CFL \leq 1.0$)
DTINT	Initial time step size

*CESE_EOS_CAV_HOMOG_EQUILIB

Purpose: Define the coefficients in the equation of state (EOS) for the homogeneous equilibrium cavitation model.

Card 1	1	2	3	4	5	6	7	8
Variable	EOSID	ρ_{vap}	ρ_{liq}	a_{vap}	a_{liq}	μ_{vap}	μ_{liq}	P_{SatVap}
Type	I	F	F	F	F	F	F	F
Default	none	0.8	880.0	334.0	1386.0	1.435e-5	1.586e-4	1.2e+4

VARIABLE	DESCRIPTION
EOSID	Equation of state identifier
ρ_{vap}	density of the saturated vapor
ρ_{liq}	density of the saturated liquid
a_{vap}	sound speed of the saturated vapor
a_{liq}	sound speed of the saturated liquid
μ_{vap}	dynamic viscosity of the vapor
μ_{liq}	dynamic viscosity of the liquid
P_{SatVap}	pressure of the saturated vapor

Remarks:

1. Once a cavitation EOS is used, the cavitation flow solver will be triggered.

2. In this homogeneous equilibrium cavitation model, a barotropic equation of state is used. This model can be used in small scale & high speed cavitation flows, and it is not good for large-scale, low-speed cavitation calculations.

***CESE_EOS_IDEAL_GAS**

Purpose: Define the coefficients Cv and Cp in the equation of state for an ideal gas in the CESE fluid solver.

Card 1	1	2	3	4	5	6	7	8
Variable	EOSID	Cv	Cp					
Type	I	F	F					
Default	none	717.5	1004.5					

VARIABLE	DESCRIPTION
EOSID	Equation of state identifier
Cv	Specific heat at constant volume
Cp	Specific heat at constant pressure

Remarks:

1. As with other solvers in LS-DYNA, the user is responsible for unit consistency. For example, if a user wants to use dimensionless variables, Cv & Cp above also should be replaced by the corresponding dimensionless ones.

*CESE_INITIAL

Purpose: Specify constant initial conditions (ICs) for flow variables at the centroid of each fluid element.

Card 1	1	2	3	4	5	6	7	8
Variable	U	V	W	RH	P	T		
Type	F	F	F	F	F	F		
Default	0	0.0	0.0	1.225	0.0	0.0		

VARIABLE	DESCRIPTION
U, V, W	x-, y-, z-velocity components respectively
RHO	density ρ
P	pressure P
T	temperature T

Remarks:

1. Usually, only two of ρ, P & T are needed to be specified (besides the velocity). If all three are given, only ρ and P will be used.

2. These initial condition will be applied in those elements that have not been assigned a value by ***CESE_INITIAL_OPTION** cards for individual elements or sets of elements.

***CESE_INITIAL**_*OPTION*

Available options include:

SET

ELEMENT

Purpose: Specify initial conditions for the flow variables at the centroid of each element in a set of elements or at the centroid of a single element.

Card 1	1	2	3	4	5	6	7	8
Variable	EID/ESID	U	V	W	RHO	P	T	
Type	I	F	F	F	F	F	F	
Default	none	0.0	0.0	0.0	1.225	0.0	0.0	
Remarks					1	1	1	

VARIABLE	DESCRIPTION
EID/ESID	Solid element ID (EID) or solid element set ID (ESID)
U,V,W	x-, y-, z-velocity components respectively
RHO	density
P	pressure
T	temperature

Remarks:

1. Usually, only two of ρ, P & T are needed to be specified (besides the velocity). If all three are given, only ρ and P will be used.

2. The priority of this card is higher than ***CESE_INITIAL**, i.e., if an element is assigned a initial value by this card, ***CESE_INITIAL** will not apply to it anymore.

*CESE_INITIAL_CHEMISTRY

Purpose: Initializes the chemistry and fluid state in every element of the CESE mesh that has not already been initialized by one of the other ***CESE_INITIAL_CHEMISTRY** cards. This is only used when chemistry is being solved with the CESE solver.

Card 1	1	2	3	4	5	6	7	8
Variable	CHEMID	COMPID						
Type	I	I						
Default	none	none						

Card 2	1	2	3	4	5	6	7	8
Variable	UIC	VIC	WIC	RHOIC	PIC	TIC	HIC	
Type	F	F	F	F	F	F	F	
Default	none	none	none	none	none	none	none	

VARIABLE	DESCRIPTION
CHEMID	Identifier of chemistry control card to use.
COMPID	Identifier of chemical composition to use.
UIC	X-component of the fluid velocity.
VIC	Y-component of the fluid velocity.
WIC	Z-component of the fluid velocity.
RHOIC	Initial fluid density.
PIC	Initial fluid pressure.
TIC	Initial fluid temperature.
HIC	Initial fluid enthalpy. However, when CHEMID refers to a ZND 1-step reaction card, this is the progressive variable (degree of combustion).

*CESE_INITIAL_CHEMISTRY_ELEMENT

Purpose: Initializes the chemistry and fluid state in every element of the list of CESE elements. This is only used when chemistry is being solved with the CESE solver.

Card 1	1	2	3	4	5	6	7	8
Variable	CHEMID	COMPID						
Type	I	I						
Default	none	none						

Card 2	1	2	3	4	5	6	7	8
Variable	UIC	VIC	WIC	RHOIC	PIC	TIC	HIC	
Type	F	F	F	F	F	F	F	
Default	none	none	none	none	none	none	none	

Card 3…	1	2	3	4	5	6	7	8
Variable	ELE1	ELE2	ELE3	ELE4	ELE5	ELE6	ELE7	ELE8
Type	I	I	I	I	I	I	I	I
Default	none	none	none	none	none	none	none	none

VARIABLE	DESCRIPTION
CHEMID	Identifier of chemistry control card to use.
COMPID	Identifier of chemical composition to use.

VARIABLE	DESCRIPTION
UIC	X-component of the fluid velocity.
VIC	Y-component of the fluid velocity.
WIC	Z-component of the fluid velocity.
RHOIC	Initial fluid density.
PIC	Initial fluid pressure.
TIC	Initial fluid temperature.
HIC	Initial fluid enthalpy. However, when CHEMID refers to a ZND 1-step reaction card, this is the progressive variable (degree of combustion).
ELE1 …	User element numbers to initialize.

***CESE_INITIAL_CHEMISTRY_PART**

Purpose: Initializes the chemistry and fluid state in every element of the specified CESE part that has not already been initialized by ***CESE_INITIAL_CHEMISTRY_ELEMENT** or ***CESE_INITIAL_CHEMISTRY_SET cards**. This is only used when chemistry is being solved with the CESE solver.

Card 1	1	2	3	4	5	6	7	8
Variable	PARTID	CHEMID	COMPID					
Type	I	I	I					
Default	none	none	none					

Card 2	1	2	3	4	5	6	7	8
Variable	UIC	VIC	WIC	RHOIC	PIC	TIC	HIC	
Type	F	F	F	F	F	F	F	
Default	none	none	none	none	none	none	none	

VARIABLE	DESCRIPTION
PARTID	Identifier of the CESE part on which to initialize.
CHEMID	Identifier of chemistry control card to use.
COMPID	Identifier of chemical composition to use.
UIC	X-component of the fluid velocity.
VIC	Y-component of the fluid velocity.
WIC	Z-component of the fluid velocity.
RHOIC	Initial fluid density.
PIC	Initial fluid pressure.
TIC	Initial fluid temperature.

VARIABLE	DESCRIPTION
HIC	Initial fluid enthalpy. However, when CHEMID refers to a ZND 1-step reaction card, this is the progressive variable (degree of combustion).

*CESE_INITIAL_CHEMISTRY_SET

Purpose: Initializes the chemistry and fluid state in every element of the specified element set in the CESE mesh that has not already been initialized by ***CESE_INITIAL_CHEMISTRY_ELEMENT** cards. This is only used when chemistry is being solved with the CESE solver.

Card 1	1	2	3	4	5	6	7	8
Variable	SETID	CHEMID	COMPID					
Type	I	I	I					
Default	none	none	none					

Card 2	1	2	3	4	5	6	7	8
Variable	UIC	VIC	WIC	RHOIC	PIC	TIC	HIC	
Type	F	F	F	F	F	F	F	
Default	none	none	none	none	none	none	none	

VARIABLE	DESCRIPTION
SETID	Identifier of the CESE element set to initialize.
CHEMID	Identifier of chemistry control card to use.
COMPID	Identifier of chemical composition to use.
UIC	X-component of the fluid velocity.
VIC	Y-component of the fluid velocity.
WIC	Z-component of the fluid velocity.
RHOIC	Initial fluid density.
PIC	Initial fluid pressure.
TIC	Initial fluid temperature.

VARIABLE	DESCRIPTION
HIC	Initial fluid enthalpy. However, when CHEMID refers to a ZND 1-step reaction card, this is the progressive variable (degree of combustion).

***CESE_MAT_GAS**

Purpose: Define the fluid (gas) properties in a viscous flow for the CESE solver.

Card 1	1	2	3	4	5	6	7	8
Variable	MID	C1V	C2	Pmd				
Type	I	F	F	F				
Default	none	1.458E-6	110.4	0.72				

VARIABLE	DESCRIPTION
MID	Material identifier
C1, C2	Two coefficients in the Sutherland's formula for viscosity, i.e., $\mu=C_1T^{3/2}/(T+C_2)$ where C_1 and C_2 are constants for a given gas. For example, for air at moderate temperatures, $C_1=1.458\times10^{-6}$ kg/(m s (^{o}K)$^{1/2}$); $C_2=110.4\,^o$K.
Prnd	The Prandtl Number (used to determine the coefficient of thermal conductivity). It is approximately constant for most gases. For air at standard conditions Prnd=0.72.

Remarks:

1. These material coefficients are only used to calculate the viscosity in viscous flows, so for invisid flows, this material card is not needed.

2. As with other solvers in LS-DYNA, the user is responsible for unit consistency. For example, if dimensionless variables are used, C_1 and C_2 should be replaced by the corresponding dimensionless ones.

*CESE_PART

Purpose: Define CESE solver parts, i.e., connect CESE material and EOS information.

Card 1	1	2	3	4	5	6	7	8
Variable	PID	MID	EOSID					
Type	I	I	I					
Default	none	none	none					

VARIABLE	DESCRIPTION
PID	Part identifier
MID	Material identifier defined by a ***CESE_MAT**_card
EOSID	Equation of state identifier defined by a ***CESE_EOS**_card

Remarks:

1. Since material coefficients are only used in viscous flows, the MID can be left blank for invisid flows.

*CHEMISTRY

The keyword ***CHEMISTRY** is used to access chemistry databases that include Chemkin-based descriptions of a chemical model, as well as to select a method of solving the model. The keyword cards in this section are defined in alphabetical order:

***CHEMISTRY_COMPOSITION**

***CHEMISTRY_CONTROL_0D**

***CHEMISTRY_CONTROL_1D**

***CHEMISTRY_CONTROL_FULL**

***CHEMISTRY_CONTROL_TBX**

***CHEMISTRY_CONTROL_ZND**

***CHEMISTRY_DET_INITIATION**

***CHEMISTRY_PATH**

Currently, the following cards may be used only once in a given problem: ***CHEMISTRY_CONTROL_1D,** ***CHEMISTRY_CONTROL_ZND**, and ***CHEMISTRY_DET_INITIATION**. Also, ***CHEMISTRY_CONTROL_0D** is only used in a standalone fashion. That is, it does not involve any other solvers.

An additional option "**_TITLE"** may be appended to all ***CHEMISTRY** keywords. If this option is used, then an 80 character string is read as a title from the first card of that keyword's input. At present, LS-DYNA does not make use of the title. Inclusion of titles gives greater clarity to input decks.

*CHEMISTRY_COMPOSITION

Purpose: Provides a general way to specify a chemical composition via a list of species mole numbers in the context of a chemkin database model.

REQ 1	1	2	3	4	5	6	7	8
Variable	ID							
Type	I							
Default	none							

REQ 2…	1	2
Variable	MOLFR	SPECIES
Type	F	A
Default	none	none

VARIABLE	DESCRIPTION
ID	A unique identifier among all chemistry compositions.
MOLFR	The mole number corresponding to the species named in the SPECIES field.
SPECIES	The chemkin-compatible name of a chemical species.

*CHEMISTRY_CONTROL_0D

Purpose: Performs an isotropic chemistry calculation that operates standalone (does not call the CESE solver). This is for ISOBARIC or ISOCHORIC cases.

Card 1	1	2	3	4	5	6	7	8
Variable	ID	COMPID	SOLTYP	ERRLIM				
Type	I	I	I	F				
Default	none	none	none	1.0e-3				

Card 2	1	2	3	4	5	6	7	8
Variable	DT	TLIMIT	TIC	PIC	RIC	EIC		
Type	F	F	F	F	F	F		
Default	none	none	none	none	none	none		

Card 3	1
Variable	FILE1
Type	A

Card 4	1
Variable	FILE2
Type	A

Card 5	1
Variable	FILE3
Type	A

VARIABLE	DESCRIPTION
ID	Identifier for this 0D computation.
COMPID	Chemical composition identifier of composition to use.
SOLTYP	Type of 0D calculation: EQ.1: Isochoric EQ.2: Isobaric
ERRLIM	Error tolerance for the calculation
DT	Initial time step
TLIMIT	Time limit for the simulation
TIC	Initial temperature
PIC	Initial pressure
RIC	Initial density
EIC	Initial internal energy
FILE1	Name of the file containing the Chemkin-compatible input.
FILE2	Name of the file containing the chemistry thermodynamics database.
FILE3	Name of the file containing the chemistry transport properties database.

*CHEMISTRY_CONTROL_1D

Purpose: Loads a previously-computed one-dimensional detonation. It is then available for use in the CESE solver for initializing a computation. In the product regions, this card overrides the initialization of the ***CESE_INITIAL_CHEMISTRY** cards.

Card 1	1	2	3	4	5	6	7	8
Variable	ID	XYZD	DETDIR					
Type	I	F	I					
Default	none	none	none					

Card 2	1
Variable	FILE1
Type	A

Card 3	1
Variable	FILE2
Type	A

Card 4	1
Variable	FILE3
Type	A

Card 5	1
Variable	FILE4
Type	A

VARIABLE	DESCRIPTION
ID	Identifier for this one-dimensional detonation solution.
XYZD	Position of the detonation front in the DETDIR direction.
DETDIR	Detonation propagation direction (1 => X; 2 => Y; 3 => Z)
FILE1	Name of the LSDA file containing the one-dimensional solution.
FILE2	Name of the file containing the Chemkin-compatible input.
FILE3	Name of the file containing the chemistry thermodynamics database.
FILE4	Name of the file containing the chemistry transport properties database.

*CHEMISTRY_CONTROL_FULL

Purpose: Computes the full chemistry specified by a chemkin chemistry model. This card can be used for general-purpose chemical reaction calculations.

Card 1	1	2	3	4	5	6	7	8
Variable	ID	ERRLIM						
Type	I	F						
Default	none	none						

Card 2	1
Variable	FILE1
Type	A

Card 3	1
Variable	FILE2
Type	A

Card 4	1
Variable	FILE3
Type	A

VARIABLE	DESCRIPTION
ID	Identifier for this full chemistry calculation.
ERRLIM	Error tolerance for the full chemistry calculation..
FILE1	Name of the file containing the Chemkin-compatible input.

VARIABLE	DESCRIPTION
FILE2	Name of the file containing the chemistry thermodynamics database.
FILE3	Name of the file containing the chemistry transport properties database.

*CHEMISTRY_CONTROL_ZND

Purpose: Computes the one-dimensional reduced chemistry of a ZND model. It is then used in the initialization of the chemistry part of the CESE solver. When this card is used, the ***CESE_INITIAL_CHEMISTRY** cards must specify the progressive variable (degree of combustion) in the HIC field.

Card 1	1	2	3	4	5	6	7	8
Variable	ID							
Type	I							
Default	none							

Card 2	1	2	3	4	5	6	7	8
Variable	F	EPLUS	Q0	GAM	XYZD	DETDIR		
Type	F	F	F	F	F	I		
Default	none	none	none	none	none	none		

VARIABLE	DESCRIPTION
ID	Identifier for this full chemistry calculation.
F	Overdriven factor
EPLUS	EPLUS parameter of the ZND model.
Q0	Q0 parameter of the ZND model.
GAM	GAM parameter of the ZND model.
XYZD	Position of the detonation front in the DETDIR direction.
DETDIR	Detonation propagation direction (1 => X; 2 => Y; 3 => Z)

*CHEMISTRY_DET_INITIATION

Purpose: Performs a one-dimensional detonation calculation based upon a chemical composition and initial conditions. It is then available for use immediately in the CESE solver for initializing a computation, or it can be subsequently used by the ***CHEMISTRY_CONTROL_1D** card in a later run. In the product regions, this card overrides the initialization of the ***CESE_INITIAL_CHEMISTRY** cards.

Card 1	1	2	3	4	5	6	7	8
Variable	ID	COMPID	NMESH	DLEN	CFL	TLIMIT	XYZD	DETDIR
Type	I	I	I	F	F	F	F	I
Default	none	none	none	none	none	none	none	none

Card 2	1
Variable	FILE1
Type	A

Card 3	1
Variable	FILE2
Type	A

Card 4	1
Variable	FILE3
Type	A

Card 5	1
Variable	FILE4
Type	A

VARIABLE	DESCRIPTION
ID	Identifier for this one-dimensional detonation computation.
COMPID	Chemical composition identifier of composition to use.
NMESH	Number of equal-width elements in the one-dimensional domain.
DLEN	Length of the one-dimensional domain.
CFL	Time-step limiting factor.
TLIMIT	Time limit for the simulation
XYZD	Position of the detonation front in the DETDIR direction.
DETDIR	Detonation propagation direction (1 => X; 2 => Y; 3 => Z)
FILE1	Name of the LSDA file in which to write the one-dimensional solution.
FILE2	Name of the file containing the Chemkin-compatible input.
FILE3	Name of the file containing the chemistry thermodynamics database.
FILE4	Name of the file containing the chemistry transport properties database.

*CHEMISTRY_PATH

Purpose: To specify one or more search paths to look for chemistry database files.

Card 1	1
Variable	DIR
Type	A

VARIABLE	DESCRIPTION
DIR	Directory path to add to the search set.

*EM

The *EM keyword cards provide input for a new electromagnetism module for solving 3D eddy-current, inductive heating or resistive heating problems, coupled with mechanical and thermal solvers. Typical applications include magnetic metal forming and welding. A boundary element method in the air is coupled to finite elements in the conductor in order to avoid meshing the air.

***EM_2DAXI**

***EM_BOUNDARY**

***EM_CIRCUIT**

***EM_CIRCUIT_ROGO**

***EM_CONTACT**

***EM_CONTACT_RESISTANCE**

***EM_CONTROL**

***EM_CONTROL_CONTACT**

***EM_CONTROL_TIMESTEP**

***EM_EOS_BURGESS**

***EM_EOS_MEADON**

***EM_EOS_PERMEABILITY**

***EM_EOS_TABULATED1**

***EM_EXTERNAL_FIELD**

***EM_MAT_001**

***EM_MAT_002**

***EM_OUTPUT**

***EM_ROTATION_AXIS**

***EM_SOLVER_BEM**

***EM_SOLVER_BEMMAT**

***EM_SOLVER_FEM**

***EM_SOLVER_FEMBEM**

An additional option "**_TITLE"** may be appended to all ***EM** keywords. If this option is used, then an 80 character string is read as a title from the first card of that keyword's input. At present, LS-DYNA does not make use of the title. Inclusion of titles gives greater clarity to input decks.

*EM_2DAXI

Purpose: Sets up the electromagnetism solver as 2D axisymmetric instead of 3D, on a given part, in order to save computational time as well as memory.

The electromagnetism is solved in 2D on a given cross section of the part (defined by a segment set), with a symmetry axis defined by its direction (at this time, it can be the x,y or z axis). The EM forces and Joule heating are then computed over the full 3D part by rotations. The part needs to be compatible with the symmetry, i.e each node in the part needs to be the child of a parent node on the segment set, by a rotation around the axis. Only the conductor parts (with a ***EM_MAT** of type 2 or 4) should be defined as 2D axisymmetric.

Card 1	1	2	3	4	5	6	7	8
Variable	PID	SSID	DIR	2DOPT				
Type	I	I	I	I				
Default	none	none	none	none				

VARIABLE	DESCRIPTION
PID	Part ID of the part to be solved using 2D axisymmetry
SSID	Segment Set ID : Segment that will define the 2D cross section of the part where the EM field is solved
DIR	Defines the symmetry axis. At this time, only the x,y or z axis can be used :: EQ.1: X axis EQ.2: Y axis EQ.3: Z axis

Remarks:

At this time, either all or none of the conductor parts should be 2D axisymmetric. In the future, a mix between 2D axisymmetric and 3D parts will be allowed.

*EM_BOUNDARY

Purpose: Define some boundary conditions for the electromagnetism problems.

Card 1	1	2	3	4	5	6	7	8
Variable	SSID	BTYPE	LTYPE					
Type	I	I	I					
Default	none	none	none			e		

VARIABLE **DESCRIPTION**

SSID — Segment Set Id

BTYPE
- EQ.9: The faces of this segment set are eliminated from the BEM calculations (used for example for the rear or side faces of a workpiece)
- EQ.10: used by the 2D axisymmetric solver to make the connection between two corresponding boundaries on each side of a slice when the model is a slice of the full 360 circle (See *EM _ROTATION_AXIS card). A boundary of type 10 and LTYPE n is connected to the boundary of type 11 and LTYPE n.
- EQ.11: See EQ.10

LTYPE — Loadtype : used to identify two boundaries of BTYPE 10 and 11. The same value of loadtype identifies two boundaries connected together.

***EM_CIRCUIT**_*{OPTION}*

Available options include

SOURCE

Purpose: Define an electrical circuit.

For the **SOURCE** option, the current will be considered uniform in the circuit. This can be useful in order to save computational time in cases with a low frequency current and where the diffusion of the EM fields is a very fast process. This option is in contrast with the general case where the current density in a circuit is completed in accordance with the solver type defined in EMSOL of *EM _CONTROL. For example, if an eddy current solver is selected, the diffusion of the current in the circuit is taken into account.

Card 1	1	2	3	4	5	6	7	8
Variable	CIRCID	CIRCTYP	LCID	R/F	L/A	C/to	V0	
Type	I	I	I	F	F	F	F	
Default	none	none	none	none	none	none	none	

Card 2	1	2	3	4	5	6	7	8
Variable	SIDCURR	SIDVIN	SIDVOUT	PARTID				
Type	I	I	I	I				
Default	none	none	none	none				

VARIABLE	DESCRIPTION
CIRCID	Circuit ID

VARIABLE	DESCRIPTION
CIRCTYP	Circuit type: EQ.1: Imposed current vs time defined by a load curve. EQ.2: Imposed voltage vs time defined by a load curve. EQ.3: R,L,C,V0 circuit. EQ.4: Induced circuit.(Obsolete) EQ.11: Imposed current defined by an amplitude A, frequency F and initial time t_0: $I = A\sin[2\pi F(t - t_0)]$ EQ.12: Imposed voltage defined by an amplitude A, frequency F and initial time t_0: $V = A\sin[2\pi F(t - t_0)]$
LCID	Load curve ID for CIRCTYP=1 or 2
R/F	Value of the circuit resistance for CIRCTYP.EQ.3 Value of the Frequency for CIRCTYP .EQ. 11 or 12
L/A	Value of the circuit inductance for CIRCTYP.EQ.3 Value of the Amplitude for CIRCTYP .EQ. 11 or 12
C/t0	Value of the circuit capacity for CIRCTYP.EQ.3 Value of the initial time t0 for CIRCTYP .EQ. 11 or 12
V0	Value of the circuit initial voltage for CIRCTYP.EQ.3.
SIDCURR	Segment set ID for the current. It uses the orientation given by the normal of the segments. To use the opposite orientation, use a '–' (minus) sign in front of the segment set id. CIRCTYP.EQ.1/11: The current is imposed through this segment set CIRCTYP.EQ.3: The current needed by the circuit equations is measured through this segment set.
SIDVIN	Segment set ID for input voltage when CIRCTYP.EQ.2/12 or CIRCTYP.EQ.3. It is considered to be oriented as going into the structural mesh, irrespective of the orientation of the segment.
SIDVOUT	Segment set ID for output voltage when CIRCTYP.EQ.2/12 or CIRCTYP.EQ.3. It is considered to be oriented as going out of the structural mesh, irrespective of the orientation of the segment.
PARTID	Part ID associated to the Source Circuit. Only needed if Option=**SOURCE**

Correspondence between circuit type and card entries:

CIRCTYP	1 (imp.curr.)	2 (imp. volt.)	3 (R,L,C)	11 (F/A/t0)	12 (F/A/t0)
LCID	**M**	**M**	**-**	**-**	**-**
R/L/C/V0	**-**	**-**	**M**	**-**	**-**
I/A/t0	**-**	**-**	**-**	**M**	**M**
SIDCURR	**M**	**O**	**M**	**M**	**O**
SIDVIN	**M***	**M**	**M**	**M***	**M**
SIDVOUT	**M***	**M**	**M**	**M***	**M**
PARTID	**-**	**-**	**-**	**-**	**-**

M: mandatory
M*: mandatory with certain exceptions (See Remark 1)
O: optional (See Remark 2)
-: not taken into account

Remarks:

1. When defining a circuit with an imposed current (Type 1 or 11) and in cases of a closed loop geometry (torus), SIDVIN and SIDVOUT cannot be defined and thus, only SIDCURR is necessary.

2. When defining a circuit with an imposed tension (Type 2 or 12), it is possible to also define SIDCURR. This can be useful in circuits where various flow paths are possible for the current in order to force the entire current to go through SIDCURR.

*EM_CIRCUIT_ROGO

Purpose: Define Rogowsky coils to measure a global current vs time through a segment set or a node set.

Card Format

Card 1	1	2	3	4	5	6	7	8
Variable	ROGOID	SETID	SETTYPE	CURTYP				
Type	I	I	I	I				
Default	0	0	0	0				

VARIABLE	DESCRIPTION
ROGOID	Rogowsky coil ID
SETID	Segment or node set ID
SETTYPE	Type of set: EQ.1: Segment set EQ.2: Node set (not available yet}
CURTYP	Type of current measured: EQ.1: Volume current EQ.2: Surface current (not available yet}

Remarks:

1. A node set allows computing only a surface current. An ASCII file “rogo_xxx” , with xxx representing the rogoId, is generated for each ***EM_CIRCUIT_ROGO** card giving the value of the current vs time.

*EM_CONTACT

Purpose: Define an electromagnetic contact between two sets of parts. Used with the ***EM_CONTACT_RESISTANCE** card.

Card 1	1	2	3	4	5	6	7	8
Variable	CONTID	COTYPE	PSIDM	PSIDS				
Type	I	I	I	I				
Default	none	none	none	none				

VARIABLE	DESCRIPTION
CONTID	Electromagnetic contact ID
COTYPE	Type of EM contact EQ.1: Face to face
PSIDM	Master part set ID
PSIDS	Slave part set ID

***EM_CONTACT_RESISTANCE**

Purpose: Calculate the contact resistance of a previously defined EM contact in ***EM_CONTACT.**

Card 1	1	2	3	4	5	6	7	8
Variable	CRID	CONTID	ρ_{probe}	ρ_{sub}	ρ_{oxi}	FACEACTE	FACEAFILM	CIRCID
Type	I	I	F		F	F	F	I
Default	none	none	none	none	none	none	none	none

VARIABLE	DESCRIPTION
CRID	Resistive contact ID
CONTID	EM contact ID defined in ***EM_CONTACT**
ρ_{probe}	Probe resistivity, ρ_{prob}
ρ_{sub}	Substrate resistivity, ρ_{sub}
ρ_{oxi}	Film resistivity ρ_{oxy}
FACEACTE	Scale factor on the constriction area when calculating the constriction resistance. If negative, the factor is time dependent defined by the load curve absolute value (FaceActe).
FACEAFILM	Scale factor on the constriction area when calculating the film resistance. If negative, the factor is time dependent defined by the load curve absolute value (FaceAfilm).
CIRCID	When defined, the contact resistance will be added to the corresponding circuit ID total resistance and taken into account in the circuit equations

Remarks:

1. The contact resistance calculation is based on the book by *Ragmar Holm's "Electric Contacts"*:

$$R_{constriction} = \frac{\rho_{prob} + \rho_{sub}}{\sqrt{faceActe \times ContactArea}}$$

$$R_{film} = \frac{\rho_{oxy}}{\sqrt{faceAfilm \times ContactArea}}$$

$$R_{contact} = R_{constriction} + R_{film}$$

***EM_CONTROL**

Purpose: Enable the EM solver and set its options.

Card 1	1	2	3	4	5	6	7	8
Variable	EMSOL	NUMLS	DTINIT	DTMAX	T_INIT	T_END	NCYLFE	NCYLBE
Type	I	I	F	F	F	F	I	I
Default	0	25	none	none	0.0	ENDTIM	1	1

VARIABLE	DESCRIPTION
EMSOL	Electromagnetism solver selector: EQ.1: eddy current solver EQ.2: induced heating solver EQ.3: resistive heating solver
NUMLS	Number of local EM steps in ½ period for EMSOL=2 Not used for EMSOL=1
DTINIT	Initial electromagnetism time step.
DTMAX	Maximum electromagnetism time step.
T_INIT	Time when electromagnetism solver is turned on (default is at the beginning of the run).
T_END	Time when electromagnetism solver is turned off (default is at the end of the run).
NCYCLFEM	Number of electromagnetism cycles between the recomputation of EM-FEM matrices.
NCYCLBEM	Number of electromagnetism cycles between the recomputation of EM-BEM matrices.

*EM_CONTROL_CONTACT

Purpose: Turns on the electromagnetism contact algorithms to check for contact between conductors and allow the electromagnetic fields to flow from one conductor to another when detected as in contact.

Card 1	1	2	3	4	5	6	7	8
Variable	EMCT							
Type	I							
Default	0							

VARIABLE	DESCRIPTION
EMCT	EQ.0: no contact detection EQ.1: contact detection

***EM_CONTROL_TIMESTEP**

Purpose: Allows to control the EM time step and its evolution

Card 1	1	2	3	4	5	6	7	8
Variable	TSTYPE	DTCONS	LCID	FACTOR				
Type	I	F	I	F				
Default	none	none	none	1.0				

VARIABLE	DESCRIPTION
TSTYPE	Time Step type EQ.1 : constant time step given in DTCONST EQ.2 : time step vs time given by a load curve specified in LCID EQ.3 : automatic time step computation, depending on the solver type. This time step is then multiplied by FACTOR
DTCONST	Constant value for the time step for TSTYPE=1
LCID	Load curve ID giving the time step vs time for TSTYPE=2
FACTOR	Multiplicative factor applied to the time step for TSTYPE=3

Remarks:

1. For an eddy current solver, the time step is based on the diffusion equation for the magnetic field.

$$\sigma\frac{\partial\vec{A}}{\partial t}+\vec{\nabla}\times\frac{1}{\mu}\vec{\nabla}\times\vec{A}+\sigma\vec{\nabla}\varphi=\vec{J_S}$$

It is computed as the minimal elemental diffusion time step over the elements. For a given element, the elemental diffusion time step is given as:

$dt_e=\frac{l_e^2}{2D}$, where:

- D is the diffusion coefficient $D=\frac{1}{\mu_0\sigma_e}$,
- σ_e is the element electrical conductivity,
- μ_0 is the permeability of free space,
- l_e is the minimal edge length of the element (minimal size of the element).

*EM_EOS_BURGESS

Purpose: Define the parameters for a Burgess model giving the electrical conductivity as as a function of the temperature and the density, see:

T.J. Burgess, "Electrical resistivity model of metals", 4th International Conference on Megagauss Magnetic-Field Generation and Related Topics, Santa Fe, NM, USA, 1986

Card 1	1	2	3	4	5	6	7	8
Variable	EOSID	V0	GAMMA	THETA	LF	C1	C2	C3
Type	I	F	F	F	F	F	F	F
Default	none	none	none	none	none	none	none	none

Card 2	1	2	3	4	5	6	7	8
Variable	C4	K	EXPON	LGTUNIT	TIMUNIT	TEMUNI	ADJUST	
Type	F	F	I	F	F	I	I	
Default	none	none	none	none	none	none	none	

In the following, UUS stands for User Units System and BUS for Burgess Units

VARIABLE	DESCRIPTION
EOSID	ID of the EM_EOS (specified by an ***EM_MAT** card)
V0	Reference specific volume V_0 (UUS).
GAMMA0	Reference Gruneisen value γ_0.(no units).
THETA	Reference melting temperature $\theta_{m,0}$ in eV (BUS).
LF	Latent heat of fusion L_F in kJoule/mol (BUS).
C1	C1 constant (BUS)
C2	C2 constant (no units)
C3	C3 constant (no units)

VARIABLE	DESCRIPTION
C4	C4 constant (no units)
K	Parameter k (no units).
EXPON	Exponent in equations (2) (see remarks)
LGTUNIT	Length units for UUS (relative to meter, i.e. =1.e-3 if UUS in mm).
TIMUNIT	Time units for UUS (relative to seconds).
TEMUNIT	Temperature units EQ.1: temperature in Celsius EQ.2: temperature in Kelvins
ADJUST	Conductivity modification EQ.0: (default) The conductivity is given by the Burgess formula. EQ.1: The conductivity is adjusted so that it is equal to the conductivity defined in *EM_MAT card σ_{mat} at room temperature: $$\sigma(\theta) = \sigma_{\text{Burgess}}(\theta)\frac{\sigma_{\text{mat}}}{\sigma_{\text{Burgess}}(\theta_{\text{room}})}$$

Remarks:

1. The Burgess model gives the electrical resistivity vs temperature and density for the solid phase, liquid phase and vapor phase. At this time, only the solid and liquid phases are implemented.

To check which elements are in the solid and in the liquid phase, a melting temperature is first computed by:

$$\theta_m = \theta_{m,0}\left(\frac{V}{V_0}\right)^{-\frac{1}{3}} e^{(2\gamma_0-1)(1-\frac{V}{V_0})}$$

- If $T < \theta_m$: solid phase model

The solid phase electrical resistivity corresponds to the Meadon model:

$$\eta_S = (C_1 + C_2\theta^{C_3}) f_c(\tfrac{V}{V_0}) \quad (1)$$

where θ is the temperature, V is the specific volume, and V_0 is the reference specific volume (zero pressure, solid phase).

In (1), the volume dependence is given by:

$f_c(\frac{V}{V_0}) = (\frac{V}{V_0})^{2\gamma-1}$ if EXPON = -1 (most materials) (2a)

$f_c(\frac{V}{V_0}) = (\frac{V}{V_0})^{2\gamma+1}$ if EXPON = +1 (tungsten) (2b)

$f_c(\frac{V}{V_0}) = (\frac{V}{V_0})^{2\gamma}$ if EXPON = 0 (stainless steel) (2c)

with

$$\gamma = \gamma_0 - (\gamma_0 - \tfrac{1}{2})(1 - \tfrac{V}{V_0}) \quad (3)$$

- If $T > \theta_m$: liquid phase model:

$$\eta_L = (\eta_L)_{\theta_m}\left(\frac{\theta}{\theta_m}\right)^{C_4} \quad (4)$$

with

$$(\eta_L)_{\theta_m} = \Delta\eta(\eta_S)_{\theta_m}$$

where

$\Delta\eta = ke^{0.69L_F/\theta_m}$ if k>0 (5a)

$\Delta\eta = 1 + 0.0772(2 - \theta_m)$ if k = -1 (tungsten) (5b)

$\Delta\eta = 1 + 0.106(0.846 - \theta_m)$ if k = -2 (stainless steel SS-304) (5c)

The following table reports some sets of parameters given by Burgess in his paper:

Parameter	**Cu**	**Ag**	**Au**	**W**	**Al(2024)**	**SS(304)**
V_0(cm³/gm)	0.112	0.0953	0.0518	0.0518	0.370	0.1265
γ_0	2.00	2.55	3.29	1.55	2.13	2.00
$\theta_{m,0}$ (BUS)	0.117	0.106	0.115	0.315	0.0804	0.156
L_F (BUS)	0.130	0.113	0.127	0.337	0.107	0.153
C_1 (BUS)	-4.12e-5	-3.37e-5	-4.95e-5	-9.73e-5	-5.35e-5	0
C_2	0.113	0.131	0.170	0.465	0.233	0.330
C_3	1.145	1.191	1.178	1.226	1.210	0.4133
EXPON	-1	-1	-1	+1	-1	0
C_4	0.700	0.672	0.673	0.670	0.638	0.089
k	0.964	0.910	1.08	-1.	0.878	-2.

***EM_EOS_MEADON**

Purpose: Define the parameters for a Meadon model, giving the electrical conductivity as as a function of the temperature and the density; see:

T.J. Burgess, "Electrical resistivity model of metals", 4th International Conference on Megagauss Magnetic-Field Generation and Related Topics, Santa Fe, NM, USA, 1986

Card 1	1	2	3	4	5	6	7	8
Variable	EOSID	C1	C2	C3	TEMUNI	V0	GAMMA	EXPON
Type	I	F	F	F	I	F	F	I
Default	none	none	none	none	none	none	none	none

Card 2	1	2	3	4	5	6	7	8
Variable	LGTUNIT	TIMUNIT	ADJUST					
Type	F	F	I					
Default	none	none	none					

In the following, UUS stands for User Units System and BUS for Burgess Units.

VARIABLE	DESCRIPTION
EOSID	ID of the EM_EOS
C1	C1 constant (BUS)
C2	C2 constant (no units)
C3	C3 constant (no units)
TEMUNIT	Temperature units EQ.1: temperature in Celsius EQ.2: temperature in Kelvins
V0	Reference specific volume V0 (UUS).

VARIABLE	DESCRIPTION
GAMMA0	Reference Gruneisen value γ_0.(no units).
EXPON	Exponent in equations (2)
LGTUNIT	Length units for UUS (relative to meter, i.e. =1.e-3 if UUS in mm).
TIMUNIT	Time units for UUS (relative to seconds).
ADJUST:	EQ.0: (default) the conductivity is given by the Burgess formula. EQ.1: The conductivity is adjusted so that it is equal to the conductivity defined in the *EM_MAT card σ_{mat} at room temperature: $$\sigma(\theta) = \sigma_{Burgess}(\theta)\frac{\sigma_{mat}}{\sigma_{Burgess}(\theta_{room})}$$

Remarks:

1. The Meadon model is a simplified Burgess model with the solid phase equations only.
The electrical resistivity is given by:

$$\eta_S = (C_1 + C_2\theta^{C_3}) f_c(\frac{V}{V_0}) \quad (1)$$

where θ is the temperature, V is the specific volume, and V_0 is the reference specific volume (zero pressure, solid phase).
In (1), the volume dependence is given by:

$f_c(\frac{V}{V_0}) = (\frac{V}{V_0})^{2\gamma-1}$ if EXPON = -1 (most materials) (2a)

$f_c(\frac{V}{V_0}) = (\frac{V}{V_0})^{2\gamma+1}$ if EXPON = +1 (tungsten) (2b)

$f_c(\frac{V}{V_0}) = (\frac{V}{V_0})^{2\gamma}$ if EXPON = 0 (stainless steel) (2c)

$f_c(\frac{V}{V_0}) = 1.$ if V_0 is not defined or equal to 0 (2d)

(In this last case, only EOSID, C1, C2, C3, TEMUNIT, TIMUNIT and LGTUNIT need to be defined)

with

$$\gamma = \gamma_0 - (\gamma_0 - \frac{1}{2})(1 - \frac{V}{V_0}) \quad (3)$$

The following table reports some sets of parameters given by Burgess in his paper:

Parameter	Cu	Ag	Au	W	Al(2024)	SS(304)
V_0(cm³/gm)	0.112	0.0953	0.0518	0.0518	0.370	0.1265
γ_0	2.00	2.55	3.29	1.55	2.13	2.00
C_1 (BUS)	-4.12e-5	-3.37e-5	-4.95e-5	-9.73e-5	-5.35e-5	0
C_2	0.113	0.131	0.170	0.465	0.233	0.330
C_3	1.145	1.191	1.178	1.226	1.210	0.4133
EXPON	-1	-1	-1	+1	-1	0

*EM_EOS_PERMEABILITY

Purpose: Define the parameters for the behavior of a material's permeability

Card 1	1	2	3	4	5	6	7	8
Variable	EOSID	EOSTYPE	LCID					
Type	I	I	I					
Default	none	none	none					

VARIABLE	DESCRIPTION
EOSID	ID of the EM_EOS
EOSTYPE	Define the type of EOS: EQ.1: Permeability defined by a H function of B curve (H=B/μ) EQ.2: Permeability defined by a B function of H curve (B=μH)
LCID	Load curve ID

*EM_EOS_TABULATED1

Purpose: Define the electrical conductivity as a function of temperature by using a load curve.

Card 1	1	2	3	4	5	6	7	8
Variable	EOSID	LCID						
Type	I	I						
Default	none	none						

VARIABLE	DESCRIPTION
EOSID	ID of the EM_EOS
LCID	Load curve ID.

Remarks:

1. The load curve describes the electrical conductivity (ordinate) vs the temperature (abscissa). The user needs to make sure the temperature and the electrical conductivity given by the load curve are in the correct units. Also, it is advised to give some bounds to the load curve (conductivities at very low and very high temperatures) to avoid bad extrapolations of the conductivity if the temperature gets out of the load curve bounds.

*EM_EXTERNAL_FIELD

Purpose: Define the components of a time dependent exterior field uniform in space applied on the conducting parts.

Card 1	1	2	3	4	5	6	7	8
Variable	FIELDID	FTYPE	FDEF	LCIDX	LCIDY	LCIDZ		
Type	I	I	F	I	I	I		
Default	0	0	0	0	0	0		

VARIABLE	DESCRIPTION
FIELDID	External Field ID
FTYPE	Field type: EQ.1: Magnetic field EQ.2: Electric field (not available yet)
FDEF	Field defined by : EQ.1: Load Curves
LCID(X,Y,Z)	Load curve ID defining the (X,Y,Z) component of the field function of time

***EM_MAT_001**

Purpose: Define the electromagnetic material type and properties for a material whose permeability equals the free space permeability.

Card 1	1	2	3	4	5	6	7	8
Variable	MID	MTYPE	SIGMA	EOSID				
Type	I	I	F	I				
Default	none	none	none	none				

VARIABLE	DESCRIPTION
MID	Material ID: refers to MID in the ***PART** card.
MTYPE	Defines the electromagnetism type of the material: EQ.0: Air or vacuum EQ.1: Insulator material: these materials have the same electromagnetism behavior as EQ.0 EQ.2: Conductor carrying a source. In these conductors, the eddy current problem is solved, which gives the actual current density. Typically, this would correspond to the coil. EQ.4: Conductor not connected to any current or voltage source, where the Eddy current problem is solved. Typically, this would correspond to the workpiece
SIGMA	Initial electrical conductivity of the material
EOSID	ID of the EOS to be used for the electrical conductivity (see ***EM_EOS** card).

*EM_MAT_002

Purpose: Define an electromagnetic material type and properties whose permeability is different than the free space permeability.

Card 1	1	2	3	4	5	6	7	8
Variable	MID	MTYPE	SIGMA	EOSID	MUREL	EOSMU		
Type	I	I	F	I	F	I		
Default	none	none	none	none	none	none		

VARIABLE	DESCRIPTION
MID	Material ID: refers to MID in the ***PART** card.
MTYPE	Defines the electromagnetism type of the material: EQ.0: Air or vacuum EQ.1: Insulator material: these materials have the same electromagnetism behavior as EQ.0 EQ.2: Conductor carrying a source. In these conductors, the eddy current problem is solved, which gives the actual current density. Typically, this would correspond to the coil. EQ.4: Conductor not connected to any current or voltage source, where the Eddy current problem is solved. Typically, this would correspond to the workpiece
SIGMA	Initial electrical conductivity of the material
EOSID	ID of the EOS to be used for the electrical conductivity (see ***EM_EOS** card).
MUREL	Relative permeability : Is the ratio of the permeability of a specific medium to the permeability of free space ($\mu_r = \mu/\mu_0$)
EOSMU	ID of the EOS to be used to define the behavior of μ by an equation of state (Note : if EOSMU is defined, MUREL will be used for the initial value only).

***EM_OUTPUT**

Purpose: Define the level of EM related output on the screen and in the messag file.

Card 1	1	2	3	4	5	6	7	8
Variable	MATS	MATF	SOLS	SOLF	MESH	MEM	TIMING	
Type	I	I	I	I	I	I	I	
Default	0	0	0	0	0	0	0	

VARIABLE	DESCRIPTION
MATS	Level of matrix assembly output to the screen: EQ.0: No output EQ.1: Basic assembly steps EQ.2: Basic assembly steps+percentage completed+final statistics EQ.3: Basic assembly steps+percentage completed+statistics at each percentage of completion
MATF	Level of matrix assembly output to the messag file: EQ.0: No output EQ.1: Basic assembly steps EQ.2: Basic assembly steps+percentage completed+final statistics EQ.3: Basic assembly steps+percentage completed+statistics at each percentage of completion
SOLS	Level of solver output on the screen: EQ.0: No output EQ.1: Global information at each FEM iteration EQ.2: Detailed information at each FEM iteration
SOLF	Level of solver output to the messag file: EQ.0: No output EQ.1: Global information at each FEM iteration EQ.2: Detailed information at each FEM iteration

VARIABLE	DESCRIPTION
MESH	Controls the output of the mesh data to the d3hsp file EQ.0: No mesh output EQ.1: Mesh info is written to the d3hsp file
MEMORY	Controls the output of information about the memory used by the EM solve to the messag file: EQ.0: no memory information written. EQ.1: memory information written.
TIMING	Controls the output of information about the time spent in the different parts of the EM solver to the messag file EQ.0: no timing information written. EQ.1: timing information written.

*EM_ROTATION_AXIS

Purpose: Define a rotation axis for the EM solver. This is used with the 2D axisymmetric solver. The axis is defined by a point and a direction.

Card 1	1	2	3	4	5	6	7	8
Variable	XP	YP	ZP	XD	YD	ZD	NUMSEC	
Type	F	F	F	F	F	F	I	
Default	none	none	none	none	none	none	none	

VARIABLE	DESCRIPTION
NP	X,Y,Z coordinate of the point
ND	X,Y,Z coordinate of direction of the axis
NUMSEC	Number of sectors : ratio of the full circle to the angular extension of the mesh. This has to be a power of 2. For example NUMSEC=4 means that the mesh represents one fourth of the full 360 degrees circle.

*EM_SOLVER_BEM

Purpose: Define the type of linear solver and pre-conditioner as well as tolerance for the EM_BEM solve.

Card 1	1	2	3	4	5	6	7	8
Variable	RELTOL	MAXITE	STYPE	PRECON	USELAS			
Type	I	I	I	I	I			
Default	1E-3	1000	2	2	1			

VARIABLE	DESCRIPTION
RELTOL	Relative tolerance for the iterative solvers (PCG or GMRES). The user should try to decrease this tolerance if the results are not accurate enough. More iterations will then be needed.
MAXITER	Maximal number of iterations.
STYPE	Solver type: EQ.1: Direct solve – the matrices will then be considered as dense. EQ.2: Pre-Conditioned Gradient method (PCG) - this allows to have block matrices with low rank blocks, and thus reduce memory used. EQ.3 : GMRES method - this allows to have block matrices with low rank blocks and thus reduce memory used. The GMRES option only works in Serial for now.
PRECON	Preconditioner type for PCG or GMRES iterative solves: EQ.0: no preconditioner EQ.1: Diagonal line EQ.2: diagonal block EQ.3: broad diagonal including all neighbor faces EQ.4: LLT factorization. The LLT factorization option only works in serial for now.
USELAST	This is used only for iterative solvers (PCG or GMRES). EQ.-1 : starts from 0 as initial solution of the linear system. EQ.1: starts from previous solution normalized by the rhs change.

Remarks:

1. Using USELAST=1 can save many iterations in the further solves if the vector solution of the present solve is assumed to be nearly parallel to the vector solution of the previous solve, as usually happens in time-domain eddy-current problems.

*EM_SOLVER_BEMMAT

Purpose: Define the type of BEM matrices as well as the way they are assembled.

Card 1	1	2	3	4	5	6	7	8
Variable	MATID							RELTOL
Type	I							F
Default	none							1E-3 or -2

VARIABLE	DESCRIPTION
MATID	Defines which BEM matrix the card refers to: EQ.1: P matrix EQ.2: Q matrix
RELTOL	Relative tolerance on the sub-blocks of the matrix when doing low rank approximations. The default values are 1.e-3 for the P matrix and 1.e-2 for the Q matrix. The user should try to decrease these tolerances if the results are not accurate enough. More memory will then be needed.

***EM_SOLVER_FEM**

Purpose: Define some parameters for the EM_FEM solver.

Card 1	1	2	3	4	5	6	7	8
Variable	RELTOL	MAXITE	STYPE	PRECON	USELAS			
Type	I	I	I	I	I			
Default	1E-3	1000	2	2	1			

VARIABLE	DESCRIPTION
RELTOL	Relative tolerance for the iterative solvers (PCG or GMRES). The user should try to decrease this tolerance if the results are not accurate enough. More iterations will then be needed.
MAXITER	Maximal number of iterations.
STYPE	Solver type: EQ.1: Direct solve EQ.2: Conditioned Gradient Method (PCG)
PRECON	Preconditioner type for PCG. EQ.0: no preconditioner EQ.1: Diagonal line
USELAST	This is used only for iterative solvers (PCG). EQ.-1 : starts from 0 as initial solution of the linear system. EQ.1: starts from previous solution normalized by the rhs change.

Remarks:

1. Using USELAST=1 can save many iterations in the further solves if the vector solution of the present solve is assumed to be nearly parallel to the vector solution of the previous solve, as usually happens in time-domain eddy-current problems.

*EM_SOLVER_FEMBEM

Purpose: Define some parameters for the coupling between the EM_FEM and EM_BEM solvers.

Card 1	1	2	3	4	5	6	7	8
Variable	RELTOL	MAXITE	FORCON					
Type	F	I	I					
Default	1E-2	50	0					

VARIABLE	DESCRIPTION
RELTOL	Relative tolerance for the solver. The user should try to decrease this tolerance if the results are not accurate enough. More iterations will then be needed.
MAXITER	Maximal number of iterations.
FORCON	EQ.0: the code stops with an error if no convergence EQ.1: the code continues to the next time step even if the RELTOL convergence criteria has not been reached..

*ICFD

The keyword *ICFD covers all the different options available in the incompressible fluid solver. The keyword cards in this section are defined in alphabetical order:

***ICFD_BOUNDARY_CONJ_HEAT**
***ICFD_BOUNDARY_FREESLIP**
***ICFD_BOUNDARY_FSI**
***ICFD_BOUNDARY_NONSLIP**
***ICFD_BOUNDARY_PRESCRIBED_VEL**
***ICFD_BOUNDARY_PRESCRIBED_PRE**
***ICFD_BOUNDARY_PRESCRIBED_TEMP**
***ICFD_CONTROL_ADAPT**
***ICFD_CONTROL_ADAPT_SZIE**
***ICFD_CONTROL_FSI**
***ICFD_CONTROL_MESH**
***ICFD_CONTROL_MESH_MOV**
***ICFD_CONTROL_OUTPUT**
***ICFD_CONTROL_PARTITION**
***ICFD_CONTROL_SPLIT**
***ICFD_CONTROL_SURFMESH**
***ICFD_CONTROL_TIME**
***ICFD_CONTROL_TURBULENCE**
***ICFD_DATABASE_AVERAGE**
***ICFD_DATABASE_DRAG**
***ICFD_DEFINE_POINT**
***ICFD_INITIAL**
***ICFD_MAT**
***ICFD_PART**
***ICFD_PART_VOL**
***ICFD_SECTION**
***ICFD_SET_NODE_LIST**

An additional option "**_TITLE"** may be appended to all ***ICFD** keywords. If this option is used, then an 80 character string is read as a title from the first card of that keyword's input. At present, LS-DYNA does not make use of the title. Inclusion of titles gives greater clarity to input decks.

*ICFD_BOUNDARY_CONJ_HEAT

Purpose: Specify which boundary of the fluid domain will exchange heat with the solid.

Card 1	1	2	3	4	5	6	7	8
Variable	PID							
Type	I							
Default	none							

VARIABLE	DESCRIPTION
PID	PID of the fluid surface in contact with the solid.

***ICFD_BOUNDARY_FREESLIP**

Purpose: Specify the fluid boundary with free-slip boundary condition.

Card 1	1	2	3	4	5	6	7	8
Variable	PID							
Type	I							
Default	none							

VARIABLE	DESCRIPTION
PID	PID of the fluid surface where a free-slip boundary condition is applied.

*ICFD_BOUNDARY_FSI

Purpose: This keyword defines which fluid surfaces will be considered in contact with the solid surfaces for fluid-structure interaction (FSI) analysis. This keyword should not be defined if ***ICFD_CONTROL_FSI** is not defined.

Card 1	1	2	3	4	5	6	7	8
Variable	PID							
Type	I							
Default	none							

VARIABLE	DESCRIPTION
PID	PID of the fluid surface in contact with the solid domain.

*ICFD_BOUNDARY_NONSLIP

Purpose: Specify the fluid boundary with a non-slip boundary condition.

Card 1	1	2	3	4	5	6	7	8
Variable	PID							
Type	I							
Default	none							

VARIABLE	DESCRIPTION
PID	PID of the fluid surface where a non-slip boundary condition is applied.

*ICFD_BOUNDARY_PRESCRIBED_MOVEMESH

Purpose: Allows the node of a fluid surface to translate in certain directions using an ALE approach. This is useful in piston type applications or can also be used in certain cases to avoid big mesh deformation.

Card 1	1	2	3	4	5	6	7	8
Variable	PID	dofx	dofy	dofz				
Type	I	I	I	I				
Default	none	1	1.	1				

VARIABLE	DESCRIPTION
PID	PID for a fluid surface.
dofx,dofy,dofz	Degrees of freedom in the X,Y and Z directions : EQ.0: degree of freedom left free (Surface nodes can translate in the chosen direction) EQ.1: prescribed degree of freedom (Surface nodes are blocked)

*ICFD_BOUNDARY_PRESCRIBED_VEL

Purpose: Impose the fluid velocity on the boundary.

Card 1	1	2	3	4	5	6	7	8
Variable	PID	DOF	VAD	LCID	SF	VID	DEATH	BIRTH
Type	I	I	I	I	F	I	F	F
Default	none	none	1	none	1.	0	1.E+28	0.0

VARIABLE	DESCRIPTION
PID	PID for a fluid surface.
DOF	Applicable degrees-of-freedom: EQ.1: x- degree-of-freedom, EQ.2: y- degree-of-freedom, EQ.3: z- degree-of-freedom,
VAD	Velocity flag: EQ.1 linear velocity EQ.2 angular velocity EQ.3 parabolic velocity profile
LCID	Load curve ID used to describe motion value versus time, see ***DEFINE_CURVE**, ***DEFINE_CURVE_FUNCTION**, or ***DEFINE_FUNCTION**. See BIRTH below.
SF	Load curve scale factor. (default=1.0)
VID	Point ID for angular velocity application point, see ***ICFD_DEFINE_POINT**.
DEATH	Time at which the imposed motion/constraint is removed: EQ.0.0: default set to 1028
BIRTH	Time at which the imposed motion/constraint is activated starting from the initial abscissa value of the curve

*ICFD_BOUNDARY_PRESCRIBED_PRE

Purpose: Impose a fluid pressure on the boundary.

Card 1	1	2	3	4	5	6	7	8
Variable	PID	LCID	SF	DEATH	BIRTH			
Type	I	I	F	F	F			
Default	none	none	1.	1.E+28	0.0			

VARIABLE	DESCRIPTION
PID	PID for a fluid surface.
LCID	Load curve ID to describe the pressure value versus time, see ***DEFINE _CURVE**.
SF	Load curve scale factor. (default=1.0)
DEATH	Time at which the imposed motion/constraint is removed: EQ.0.0: default set to 1028
BIRTH	Time at which the imposed pressure is activated starting from the initial abscissa value of the curve

***ICFD_BOUNDARY_FLUX_TEMP**

Purpose: Impose a temperature flux on the boundary.

Card 1	1	2	3	4	5	6	7	8
Variable	PID	LCID	SF	DEATH	BIRTH			
Type	I	I	F	F	F			
Default	none	none	1.	1.E+28	0.0			

VARIABLE	DESCRIPTION
PID	PID for a fluid surface.
LCID	Load curve ID to describe the temperature flux value versus time, see ***DEFINE_CURVE**.
SF	Load curve scale factor. (default=1.0)
DEATH	Time at which the imposed motion/constraint is removed: EQ.0.0: default set to 1028
BIRTH	Time at which the imposed pressure is activated starting from the initial abscissa value of the curve

*ICFD_BOUNDARY_PRESCRIBED_TEMP

Purpose: Impose a fluid temperature on the boundary.

Card 1	1	2	3	4	5	6	7	8
Variable	PID	LCID	SF	DEATH	BIRTH			
Type	I	I	F	F	F			
Default	none	none	1.	1.E+28	0.0			

VARIABLE	DESCRIPTION
PID	PID for a fluid surface.
LCID	Load curve ID to describe the temperature value versus time; see ***DEFINE_CURVE**.
SF	Load curve scale factor. (default=1.0)
DEATH	Time at which the imposed temperature is removed: EQ.0.0: default set to 1028
BIRTH	Time at which the imposed temperature is activated starting from the initial abscissa value of the curve

*ICFD_CONTROL_ADAPT

Purpose: This keyword will activate the adaptive mesh refinement feature. The solver will use an a-posteriori error estimator to compute a new mesh size bounded by the user to satisfy a maximum perceptual global error.

Card 1	1	2	3	4	5	6	7	8
Variable	MINH	MAXH	ERR	MTH	NIT			
Type	F	F	F	I	I			
Default	none	none	none	0	0			

VARIABLE	DESCRIPTION
MINH	Minimum mesh size allowed to the mesh generator. The resulting mesh will not have an element smaller than MINH even if the minimum size does not satisfy the maximum error.
MAXH	Maximum mesh size.
ERR	Maximum perceptual error allowed in the whole domain.
MTH	Specify if the mesh size is computed based on function error or gradient error. EQ.0: Function error. EQ.1: Gradient error.
NIT	Number of iterations before a re-meshing is forced. Default forces a re-meshing at every timestep.

*ICFD_CONTROL_ADAPT_SIZE

Purpose: This keyword controls the re-meshing of elements taking into account the element quality and distortion overwriting the default which only checks for inverted elements.

Card 1	1	2	3	4	5	6	7	8
Variable	ASIZE							
Type	I							
Default	0							

VARIABLE	DESCRIPTION
ASIZE	EQ.0: only re-mesh in cases where elements invert. EQ.1: re-mesh if elements invert or if element quality deteriorates.

*ICFD_CONTROL_FSI

Purpose: This keyword modifies default values for the fluid-structure interaction coupling.

Card 1	1	2	3	4	5	6	7	8
Variable	OWC	BT	DT					
Type	I	F	F					
Default	0	0	1e20					

VARIABLE	DESCRIPTION
OWC	Indicates the coupling direction to the solver. EQ.0: two-way coupling. Loads and displacements are transferred across the FSI interface and the full non-linear problem is solved. EQ.1: one-way coupling. The solid solver transfers displacements to the fluid solver. EQ.2: one-way coupling. The fluid solver transfers stresses to the solid solver.
BT	Birth time for the FSI coupling. Before BT the fluid solver will not pass any loads to the structure but it will receive displacements from the solid solver.
DT	Death time for the FSI coupling. After DT the fluid solver will not transfer any loads to the solid solver but it will continue to deform with the solid.

*ICFD_CONTROL_LOAD

Purpose: This keyword can reset the body load in the fluid problem provided by keyword *LOAD _BODY. It is useful in problems where the gravity acceleration may be neglected for the fluid problem but not for the solid.

Card 1	1	2	3	4	5	6	7	8
Variable	ABL							
Type	I							
Default	1							

VARIABLE	DESCRIPTION
ABL	EQ.0: the body load provided in *LOAD_BODY is reset to zero only for the fluid analysis.

***ICFD_CONTROL_MESH**

Purpose: This keyword modifies default values for the automatic volume mesh generation. Only used in 3D cases.

Card 1	1	2	3	4	5	6	7	8
Variable	MGSF							
Type	F							
Default	1.41							

VARIABLE	DESCRIPTION
MGSF	Mesh Growth Scale Factor : Specifies the maximum mesh size that the volume mesher is allowed to use when generating the volume mesh based on the mesh surface element sizes defined in ***MESH_SURFACE_ELEMENT.** Values between 1 and 2 are allowed. Values closer to 1 will result in a finer volume mesh (1 means the volume mesh is not allowed to be coarser than the element size from the closest surface meshes) and values closer to 2 will result in a coarser volume mesh (2 means the volume can use elements as much as twice as coarse as those from the closest surface mesh).

*ICFD_CONTROL_MESH_MOV

Purpose: With this keyword the user can choose the type of algorithm for mesh movement.

Card 1	1	2	3	4	5	6	7	8
Variable	MMSH	LIM _ITER	RELTOL	ABSTOL				
Type	I	I	F	F				
Default	2	100	1.0e-3	5.0e-4				

VARIABLE	DESCRIPTION
MMSH	Mesh motion selector: EQ.1: mesh moves based on the distance to moving walls. EQ.2: mesh moves by solving a linear elasticity problem using the element sizes as stiffness. EQ.3: mesh uses a Laplacian smoothing with stiffness on edges and from node to opposite faces. Very robust, but costly. EQ.4: full Lagrangian: The mesh moves with the velocity of the flow. EQ.11: mesh moves using an implicit ball-vertex spring method
LIM_ITER	Maximum number of linear solver iterations for the ball-vertex linear system.
RELTOL	Relative tolerance to use as a stopping criterion for the ball-vertex method iterative linear solver (conjugate gradient solver with diagonal scaling preconditioner).
ABSTOL	Absolute tolerance measure for the size of mesh displacement changes to use as a stopping criterion for the ball-vertex iterative linear solver.

.

*ICFD_CONTROL_OUTPUT

Purpose: This keyword modifies default values for screen and file outputs related to this fluid solver only.

Card 1	1	2	3	4	5	6	7	8
Variable	MSGL	OUTL	DTOUT					
Type	I	I	F					
Default	0	0	0					

VARIABLE	DESCRIPTION
MSGL	Message level. EQ.0: only time step information is output. EQ.1: first level solver information. EQ.2: full output information with details about linear algebra and convergence steps.
OUTL	Output the fluid results in other file formats apart from d3plot. EQ.0: only d3plot output EQ.1: output a file with mesh statistics and fluid results in GMV format. A directory named output/gmv has to be created one level above the executable. EQ.2: output a file with mesh statistics and the fluid results in OpenDX format. A directory named output/dx has to be created one level above the executable. EQ.3: output a file with mesh statistics and the fluid results in OpenDX, GMV and the mesh visualization tool medit format. Three new directories have to be created: output/dx, output/gmv, and output/medit. each one level above the executable.
DTOUT	Time interval to print the output when OUTL is different than 0.

.

*ICFD_CONTROL_PARTITION

Purpose: This keyword changes the default option for the partition in MPP, thus it is only valid in MPP.

Card 1	1	2	3	4	5	6	7	8
Variable	PTECH							
Type	I							
Default	1							

VARIABLE	DESCRIPTION
PTECH	Indicates the type of partition. EQ.1: the library Metis is used. EQ.2: partition along the axis with higher aspect ratio. EQ.3: partition along X axis. EQ.4: partition along Y axis. EQ.5: partition along Z axis.

*ICFD_CONTROL_SURFMESH

Purpose: This keyword enables automatic surface re-meshing. The objective of the re-meshing is to improve the mesh quality on the boundaries. It should not be used on a regular basis.

Card 1	1	2	3	4	5	6	7	8
Variable	RSRF							
Type	I							
Default	0							

VARIABLE	DESCRIPTION
RSRF	Indicates whether or not to perform a surface re-meshing. EQ.0: no re-meshing is applied. EQ.1: allows the surface mesh to be re-meshed

.

*ICFD_CONTROL_SPLIT

Purpose: This keyword provides an option to trigger an iterative procedure on the fluid system. This procedure aims to bring more precision to the final pressure and velocity values but is often very time consuming. It must therefore be used with caution in specific cases. For stability purposes, this method is automatically used for the first ICFD time step.

Card 1	1	2	3	4	5	6	7	8
Variable	NIT	TOL						
Type	I	F						
Default	none	none						

VARIABLE	DESCRIPTION
NIT	Maximum Number of iterations of the system for each fluid time step. If TOL criteria is not reached after NIT iterations, the run will proceed.
TOL	Tolerance Criteria for the pressure residual during the fluid system solve.

*ICFD_CONTROL_TIME

Purpose: This keyword is used to change the defaults related to time parameters in the fluid problem.

Card 1	1	2	3	4	5	6	7	8
Variable	TTM	DT	CFL					
Type	F	F	F					
Default	1.E28	0	1					

VARIABLE	DESCRIPTION
TTM	Total time of simulation for the fluid problem.
DT	Time step for the fluid problem. If different from zero, the time step will be set constant and equal to this value. If DT = 0, then the time step is automatically computed based on the CFL condition.
CFL	Scale factor that multiplies DT.

*ICFD_CONTROL_TURBULENCE

Purpose: This keyword enables the user to modify the default values for the turbulence model.

Card 1	1	2	3	4	5	6	7	8
Variable	TMOD							
Type	I							
Default	0							

(This card is optional and may be used if TMOD=1).

Card 2	1	2	3	4	5	6	7	8
Variable	Ce1	Ce2	σ_ε	σ_k	C_μ			
Type	F	F	F	F	F			
Default	1.44	1.92	1.3	1.0	0.09			

(This card is optional and may be used if TMOD=2).

Card 2	1	2	3	4	5	6	7	8
Variable	Cs							
Type	F							
Default	0.2							

VARIABLE	DESCRIPTION
TMOD	Indicates what turbulence model will be used. EQ.0: Turbulence model based on a variational multiscale approach is used by default. EQ.1: RANS k-ε approach. EQ.2: LES Smagorinsky sub-grid scale model.
Ce1,Ce2, σ_ε, σ_k C_μ	k-ε model constants

VARIABLE	DESCRIPTION
Cs	Smagorinsky constant

*ICFD_DATABASE_DRAG

Purpose: This keyword enables the computation of drag forces over given parts of the model. If multiple keywords are given, the forces over the PID surfaces are given in separate files and are also added and output in a separate file.

Card 1	1	2	3	4	5	6	7	8
Variable	PID							
Type	I							
Default	none							

VARIABLE	DESCRIPTION
PID	Part ID of the surface where the drag force will be computed.

Remarks:

1. The file name for this database is *icfdragi* for instantaneous drag and *icfdraga* for the drag computed using average values of pressure and velocities.

*ICFD_DATABASE_AVERAGE

Purpose: This keyword enables the computation of average variable values at given time intervals.

Card 1	1	2	3	4	5	6	7	8
Variable	DT							
Type	F							
Default	none							

VARIABLE	DESCRIPTION
DT	Over each DT time interval, an average of the different fluid variables will be calculated and then reset when moving to the next DT interval.

Remarks:

1. The file name for this database is *icfdavg.*.dat* with the different averaged variable values copied in a ASCII format.

*ICFD_DEFINE_POINT

Purpose: This keyword defines a point in space that could be used for multiple purposes.

Card 1	1	2	3	4	5	6	7	8
Variable	POID	X	Y	Z				
Type	I	F	F	F				
Default	none	none	none	none				

VARIABLE	DESCRIPTION
POID	Point ID.
X	x coordinate for the point.
Y	y coordinate for the point.
Z	z coordinate for the point.

.

***ICFD_INITIAL**

Purpose: Simple initialization of velocity and temperature within a volume.

Card 1	1	2	3	4	5	6	7	8
Variable	PID	Vx	Vy	Vz	T			
Type	I	F	F	F	F			
Default	none	none	none	none	none			

VARIABLE	DESCRIPTION
PID	Part ID for the volume elements where the values are initialized (see ***ICFD_PART_VOL**).
Vx	x coordinate for the velocity.
Vy	y coordinate for the velocity.
Vz	z coordinate for the velocity.
T	Initial temperature.

*ICFD_MAT

Purpose: Specify physical properties for the fluid material.

Card 1	1	2	3	4	5	6	7	8
Variable	MID	FLG	RO	VIS	ST	THD		
Type	I	I	F	F	F	F		
Default	none	1	none	0	0	0		

Card 2	1	2	3	4	5	6	7	8
Variable	HC	TC	Beta					
Type	F	F	F					
Default	0	0	0					

Card 3	1	2	3	4	5	6	7	8
Variable	NNID	k	n	nmin	nmax			
Type	F	F	F	F	F			
Default	0	0	0	0	1E30			

VARIABLE	DESCRIPTION
MID	Material ID.
FLG	Flag to choose between fully incompressible, slightly compressible, or barotropic flows. So far, only fully incompressible flow is supported.
RO	Flow density.
VIS	Dynamic viscosity.
ST	Surface tension coefficient.

VARIABLE	DESCRIPTION
THD	Thermal diffusion used in the solution of the thermal problem. (Does not need to be defined if HC and TC exist).
HC	Heat capacity.
TC	Thermal conductivity
Beta	Thermal expansion coefficient used in the Boussinesq approximation for buoyancy.
NNID	Non Newtonian flows model ID. EQ.1 : Power Law
k	Power Law input parameter. Measure of the average velocity of the fluid (consistency index).
n	Measure of the deviation of the fluid from Newtonian.
nmin	Lower n value limit of the Power Law.
nmax	Upper n value limit of the Power Law.

***ICFD_PART**_*{OPTION}*

Available options include

TITLE

Purpose: Define parts for this incompressible flow solver.

The **TITLE** option allows the user to define an additional line with a HEADING in order to associate a name to the part.

Optional	1
Variable	HEADING
Type	C
Default	none
Remark	1

Card 1	1	2	3	4	5	6	7	8
Variable	PID	SECID	MID					
Type	I	I	I					
Default	none	none	none					

VARIABLE	DESCRIPTION
PID	Part identifier for fluid surfaces.
SECID	Section identifier defined with the ***ICFD_SECTION** card.
MID	Material identifier defined with the ***ICFD_MAT** card.

***ICFD_PART_VOL**_*{OPTION}*

Available options include

TITLE

Purpose: Define parts for a volume enclosed by surfaces. This keyword is used when no volume mesh is provided by the user. The solver will build a volume mesh and assign to it the PID defined by this card.

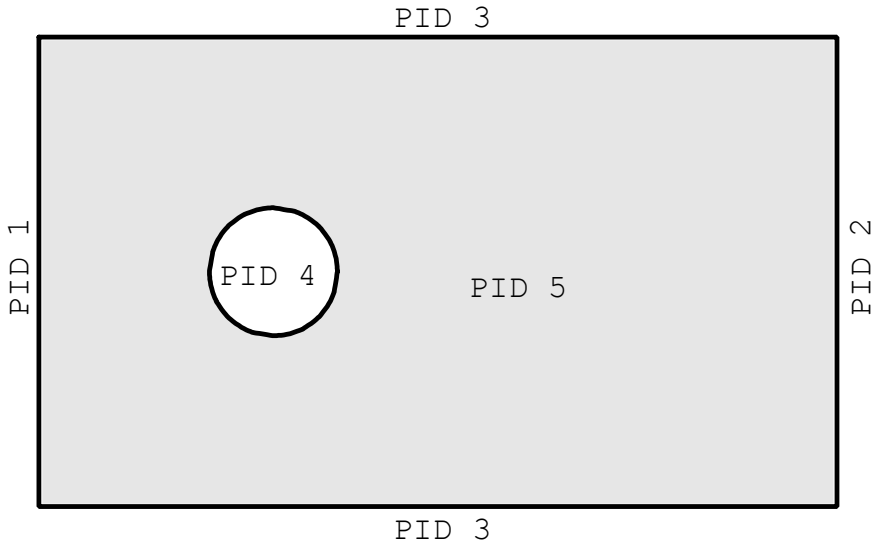
The **TITLE** option allows the user to define an additional line with a HEADING in order to associate a name to the part.

Optional	1
Variable	HEADING
Type	C
Default	none
Remark	1

Card 1	1	2	3	4	5	6	7	8
Variable	PID	SECID	MID					
Type	I	I	I					
Default	none	none	none					

Card 2,3…	1	2	3	4	5	6	7	8
Variable	SPID1	SPID2	SPID3	SPID4	SPID5	SPID6	SPID7	SPID8
Type	I	I	I	I	I	I	I	I
Default	none	none	none	none	none	none	none	none

VARIABLE	DESCRIPTION
PID	Part identifier for fluid volumes.
SECID	Section identifier defined by the ***ICFD_SECTION** card.
MID	Material identifier.
SPIDn	Part IDs for the surface elements that define the volume mesh.




```
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$$$$   *ICFD_PART_VOL
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$ PART ID 5 is defined by the surfaces that enclose it.
$
*ICFD_PART_VOL
$
$...>....1....>....2....>....3....>....4....>....5....>....6....>....7....>....8
$      pid       secid       mid
         5           1         1
$...>....1....>....2....>....3....>....4....>....5....>....6....>....7....>....8
$      pid1      pid2      pid3      pid4      pid5      pid6      pid7      pid8
         1         2         3         4
```

***ICFD_SECTION**

Purpose: Define a section for the incompressible flow solver.

Card 1	1	2	3	4	5	6	7	8
Variable	SID							
Type	I							
Default	none							

VARIABLE	DESCRIPTION
SID	Section identifier.

*ICFD_SET_NODE

Purpose: Only used in cases where the mesh is specified by the user (See ***MESH_VOLUME_ELEMENT**). Defines a set of nodes associated to a part ID on which boundary conditions can be applied.

Card 1	1	2	3	4	5	6	7	8
Variable	SID	PID						
Type	I	I						
Default	none	none						

Card 2	1	2	3	4	5	6	7	8
Variable	NID1	NID2	NID3	NID4	NID5	NID6	NID7	NID8
Type	I	I	I	I	I	I	I	I
Default	none	none	none	none	none	none	none	none

VARIABLE	DESCRIPTION
SID	Set ID
PID	Associated Part ID.
NID1	Node ID n

Remarks:

1. The convention is the similar to the one used by the keyword ***SET_NODE_LIST** and serves a similar purpose.

*MESH

The keyword ***MESH** is used to create a mesh that will be used in the analysis. So far only tetrahedral (or triangular in 2-d) elements can be generated. The keyword cards in this section are defined in alphabetical order:

***MESH_BL**

***MESH_BL_SYM**

***MESH_EMBEDSHELL**

***MESH_INTERF**

***MESH_SIZE**

***MESH_SIZE_SHAPE**

***MESH_SURFACE_ELEMENT**

***MESH_SURFACE_NODE**

***MESH_VOLUME**

***MESH_VOLUME_ELEMENT**

***MESH_VOLUME_NODE**

An additional option "**_TITLE"** may be appended to all ***MESH** keywords. If this option is used, then an 80 character string is read as a title from the first card of that keyword's input. At present, LS-DYNA does not make use of the title. Inclusion of titles gives greater clarity to input decks.

***MESH_BL**

Purpose: Specify the part ID meshed by a boundary layer mesh and how many elements to place in the thickness.

Card 1	1	2	3	4	5	6	7	8
Variable	PID	NELTH						
Type	I	I						
Default	none	none						

VARIABLE	DESCRIPTION
PID	Part identifier for the surface element.
NELTH	Number of elements normal to the surface (in the boundary layer).

*MESH_BL_SYM

Purpose: Specify the part ID that will have symmetry conditions for the boundary layer. On these surfaces, the boundary layer mesh follows the surface tangent.

Card 1	1	2	3	4	5	6	7	8
Variable	PID							
Type	I							
Default	none							

VARIABLE	DESCRIPTION
PID	Part identifier for the surface element. This is the surface with symmetry.

***MESH_EMBEDSHELL**

Purpose: Define surfaces that the mesher will embed inside the volume mesh. These surfaces will have no thickness and will conform to the rest of the volume mesh having matching nodes on the interface.

Card 1	1	2	3	4	5	6	7	8
Variable	VOLID							
Type	I							
Default	none							

Cards 2, 3, 4, … (the next "*" card terminates the input.)

	1	2	3	4	5	6	7	8
Variable	PID1	PID2	PID3	PID4	PID5	PID6	PID7	PID8
Type	I	I	I	I	I	I	I	I
Default	none	none	none	none	none	none	none	none

VARIABLE	DESCRIPTION
VOLID	ID assigned to the new volume in the keyword ***MESH_VOLUME**. The surface mesh size will be applied to this volume.
PIDn	Part IDs for the surface elements that will be embedded in the volume mesh.

*MESH_INTERF

Purpose: Define the surfaces that will be used by the mesher to specify fluid interfaces in multi-fluid simulations.

Card 1	1	2	3	4	5	6	7	8
Variable	VOLID							
Type	I							
Default	none							

Cards 2, 3, 4, … (the next "*" card terminates the input.)

	1	2	3	4	5	6	7	8
Variable	PID1	PID2	PID3	PID4	PID5	PID6	PID7	PID8
Type	I	I	I	I	I	I	I	I
Default	none	none	none	none	none	none	none	none

VARIABLE	DESCRIPTION
VOLID	ID assigned to the new volume in the keyword ***MESH_VOLUME**. The interface meshes will be applied to this volume.
PIDn	Part IDs for the surface elements.

***MESH_SIZE**

Purpose: Define the surfaces that will be used by the mesher to specify a local mesh size inside the volume. If no internal mesh is used to specify the size, the mesher will use a linear interpolation of the surface sizes that define the volume enclosure**.**

Card 1	1	2	3	4	5	6	7	8
Variable	VOLID							
Type	I							
Default	none							

Cards 2, 3, 4, … (the next "*" card terminates the input.)

	1	2	3	4	5	6	7	8
Variable	PID1	PID2	PID3	PID4	PID5	PID6	PID7	PID8
Type	I	I	I	I	I	I	I	I
Default	none	none	none	none	none	none	none	none

VARIABLE	DESCRIPTION
VOLID	ID assigned to the new volume in the keyword ***MESH_VOLUME**. The mesh sizing will be applied to this volume.
PIDn	Part IDs for the surface elements that are used to define the mesh size next to the surface mesh.

*MESH_SIZE_SHAPE

Purpose: Defines a local mesh size in specific zones corresponding to given geometrical shapes (box, sphere, cylinder). The solver will automatically apply the conditions specified during the generation of the volume mesh. This zone does not need to be entirely defined in the volume mesh.

Card 1	1	2	3	4	5	6	7	8
Variable	Sname							
Type	C							
Default	none							

if Sname=box

Cards 2	1	2	3	4	5	6	7	8
Variable	MSIZE	PminX	PminY	PminZ	PmaxX	PmaxY	PmaxZ	
Type	F	F	F	F	F	F	F	
Default	none	none	none	none	none	none	none	

if Sname=sphere

Cards 2	1	2	3	4	5	6	7	8
Variable	MSIZE	Radius	CenterX	CenterY	CenterZ			
Type	F	F	F	F	F			
Default	none	none	none	none	none			

if Sname=cylinder

Cards 2	1	2	3	4	5	6	7	8
Variable	MSIZE	Radius	PminX	PminY	PminZ	PmaxX	PmaxY	PmaxZ
Type	F	F	F	F	F	F	F	F
Default	none	none	none	none	none	none	none	none

VARIABLE	DESCRIPTION
Sname	Shape name. Possibilities include ‘box’, ‘cylinder’ and ‘sphere’
MSIZE	Mesh size that needs to be applied in the zone of the shape defined by Sname
Pmin(X,Y,Z)	X,Y,Z for the point of minimum coordinates
Pmax(X,Y,Z)	X,Y,Z for the point of maximum coordinates
Center(X,Y,Z)	Coordinates of the sphere center in cases where Sname is Sphere
Radius	Radius of the sphere if Sname is Sphere or of the cross section disk if Sname is Cylinder.

*MESH_SURFACE_ELEMENT

Purpose: Specify a set of surface elements (quadrilateral or triangular in 3-d and linear segments in 2-d) that will be used by the mesher to construct a volume mesh. These surface elements may be used to define the enclosed volume to be meshed, or alternatively they could be used to apply different mesh sizes inside the volume (see card ***MESH_SIZE).**

Card 1	1	2	3	4	5	6	7	8
Variable	EID	PID	N1	N2	N3	N4		
Type	I	I	I	I	I	I		
Default	none	none	none	none	none	none		

VARIABLE	DESCRIPTION
EID	Element ID. A unique number with respect to all ***MESH_SURFACE_ELEMENTS** cards.
PID	Part ID. A unique part identification number.
N1	Nodal point 1.
N2	Nodal point 2.
N3	Nodal point 3.
N4	Nodal point 4.

Remarks:

1. The convention is the same used by the keyword ***ELEMENT_SHELL**. In the case of a triangular face N3=N4. In 2-d N2=N3=N4.

***MESH_SURFACE_NODE**

Purpose: Define a node and its coordinates. These nodes will be used in the mesh generation process by the ***MESH_SURFACE_ELEMENT** keyword**.**

Card 1	1	2	3	4	5	6	7	8
Variable	NID	X	Y	Z				
Type	I	F	F	F				
Default	none	0	0	0				

VARIABLE	DESCRIPTION
NID	Node ID. A unique number with respect to the other surface nodes.
X	x coordinate.
Y	y coordinate.
Z	z coordinate.

*MESH_VOLUME

Purpose: This keyword defines the volume space that will be meshed. The boundaries of the volume are the surfaces defined by ***MESH_SURFACE_ELEMENT**. The list of surfaces have to be non-overlapping, and should not leave any gaps or open spaces between the surface boundaries. The nodes on the boundary of two neighbor surfaces have to be uniquely defined by the keyword ***MESH_SURFACE_NODE**, and should match exactly on the interface. This card will be ignored if the volume mesh is specified by the user and not generated automatically.

Card 1	1	2	3	4	5	6	7	8
Variable	VOLID							
Type	I							
Default	none							

Cards 2, 3, 4, … (the next "*" card terminates the input.)

	1	2	3	4	5	6	7	8
Variable	PID1	PID2	PID3	PID4	PID5	PID6	PID7	PID8
Type	I	I	I	I	I	I	I	I
Default	none	none	none	none	none	none	none	none

VARIABLE	DESCRIPTION
VOLID	ID assigned to the new volume.
PIDn	Part IDs for the surface elements that are used to define the volume.

***MESH_VOLUME_ELEMENT**

Purpose: Specify a set of volume elements for the fluid volume mesh in cases where the volume mesh is specified by the user and not generated automatically. The Nodal point are specified in the ***MESH_VOLUME_NODE** keyword. Only Tets are supported (triangles in 2D).

Card 1	1	2	3	4	5	6	7	8
Variable	EID	PID	N1	N2	N3	N4		
Type	I	I	I	I	I	I		
Default	none	none	none	none	none	none		

VARIABLE	DESCRIPTION
EID	Element ID. A unique number with respect to all ***MESH_VOLUME_ELEMENTS** cards.
PID	Part ID. A unique part identification number.
N1	Nodal point 1.
N2	Nodal point 2.
N3	Nodal point 3.
N4	Nodal point 4.

Remarks:

1. The convention is the same used by the keyword ***ELEMENT_SHELL**.

*MESH_VOLUME_NODE

Purpose: Define a node and its coordinates. This keyword is only used in cases where the fluid Volume Mesh is provided by the user and is not automatically generated. It serves the same purpose as the ***NODE** keyword for solid mechanics. Only Tets are supported**.**

Card 1	1	2	3	4	5	6	7	8
Variable	NID	X	Y	Z	H			
Type	I	F	F	F	F			
Default	none	0	0	0	0			

VARIABLE	DESCRIPTION
NID	Node ID. A unique number with respect to the other volume nodes.
X	x coordinate.
Y	y coordinate.
Z	z coordinate.

*MESH_VOLUME_PART

Purpose: Associate a volume part number created by a ***MESH_VOLUME** card with the part number of a part card from a selected solver (designated by the SOLVER field).

REQ 1	1	2	3	4	5	6	7	8
Variable	VOLPRT	SOLPRT	SOLVER					
Type	I	I	A					
Default								

VARIABLE	DESCRIPTION
VOLPRT	Part ID of a volume part created by a ***MESH_VOLUME** card.
SOLPRT	Part ID of a part created using **SOLVER**'s part card.
SOLVER	Name of a solver using a mesh created with ***MESH** cards.

.

*STOCHASTIC

The keyword ***STOCHASTIC** is used to describe the particles and numerical details for solving a set of stochastic PDEs. Currently, there are two types of stochastic PDE models in the code: a model of embedded particles in TBX explosives, and a spray model. The cards for using these models are:

***STOCHASTIC_SPRAY_PARTICLES**

***STOCHASTIC_TBX_PARTICLES**

An additional option "**_TITLE"** may be appended to all ***STOCHASTIC** keywords. If this option is used, then an 80 character string is read as a title from the first card of that keyword's input. At present, LS-DYNA does not make use of the title. Inclusion of titles gives greater clarity to input decks.

*STOCHASTIC_SPRAY_PARTICLES

Purpose: Specify particle and other model details for spray modeling using stochastic PDEs that approximate such processes. A pair of cards is required to specify the characteristics of each nozzle (cards 3 and 4 describe the first nozzle).

Card 1	1	2	3	4	5	6	7	8
Variable	INJDIST	IBRKUP	ICOLLDE	IEVAP	IPULSE	LIMPR	IDFUEL	
Type	I	I	I	I	I	I	I	
Default	1	none	none	none	none	none	1	

Card 2	1	2	3	4	5	6	7	8
Variable	RHOP	TIP	PMASS	PRTRTE	STRINJ	DURINJ		
Type	F	F	F	F	F	F		
Default	none	none	none	none	none	none		

Card 3, 5, …

	1	2	3	4	5	6	7	8
Variable	XORIG	YORIG	ZORIG	SMR	VELINJ	DRNOZ	DTHNOZ	
Type	F	F	F	F	F	F	F	

Card 4, 6, …

Card N	1	2	3	4	5	6	7	8
Variable	TILTXY	TILTXZ	CONE	DCONE	ANOZ	AMP0		
Type	F	F	F	F	F	F		
Default								

VARIABLE	DESCRIPTION
INJDIST	Spray particle size distribution: EQ.1: uniform EQ.2: Rosin-Rammler (default) EQ.3: Chi-squared degree of 2 EQ.4: Chi-squared degree of 6
IBRKUP	Type of particle breakup model: EQ.0: off (no breakup) EQ.1: TAP EQ.2: KHRT
ICOLLDE	Turn collision modeling on or off
IEVAP	Turn evaporation on or off
IPULSE	Type of injection: EQ.0: continuous injection EQ.1: sine wave EQ.2: square wave
LIMPRT	Upper limit on the number of parent particles modeled in this spray. This is not used with the continuous injection case (IPULSE=0).

VARIABLE	DESCRIPTION
IDFUEL	Selected spray liquid fuels: EQ.1: (Default), H20 EQ.2: Benzene, C6H6 EQ.3: Diesel # 2,C12H26 EQ.4: Diesel # 2,C13H13 EQ.5: Ethanol, C2H5OH EQ.6: Gasoline, C8H18 EQ.7: Jet-A, C12H23 EQ.8: Kerosene, C12H23 EQ.9: Methanol, CH3OH EQ.10: N_dodecane, C12H26
RHOP	Particle density
TIP	Initial particle temperature.
PMASS	Total particle mass
PRTRTE	Number of particles injected per second for continuous injection.
STRINJ	Start of injection(s)
DURINJ	Duration of injection(s)
XORIG	X-coordinate of center of a nozzle's exit plane
YORIG	Y-coordinate of center of a nozzle's exit plane
ZORIG	Z-coordinate of center of a nozzle's exit plane
SMR	Sauter mean radius
VELINJ	Injection velocity
DRNOZ	Nozzle radius
DTHNOZ	Azimuthal angle (in degrees measured counterclockwise) of the injector nozzle from the j=1 plane
TILTXY	Rotation angle (in degrees) of the injector in the x-y plane, where 0.0 points towards the 3 o'clock position (j=1 line), and the angle increases counter-clockwise from there.

VARIABLE	DESCRIPTION
TILTXZ	Inclination angle (in degrees) of the injection in the x-z plane, where 0.0 points straight down, x>0.0 points in the positive x direction, and x<0.0 points in the negative x direction.
CONE	Spray mean cone angle (in degrees) for hollow cone spray; spray cone angle (in degrees) for solid cone spray.
DCONE	Injection liquid jet thickness in degrees.
ANOZ	Area of injector
AMP0	Initial amplitude of droplet oscillation at injector

*STOCHASTIC_TBX_PARTICLES

Purpose: Specify particle and other model details for stochastic PDEs that model embedded particles in TBX explosives.

Card 1	1	2	3	4	5	6	7	8
Variable	PCOMB	NPRTCL	MXCNT	PMASS	SMR	RHOP	TICP	T_IGNIT
Type	I	I	I	F	F	F	F	F
Default	0	none	none	none	none	none	none	none

Card 2	1	2	3	4	5	6	7	8
Variable	INITDST	AZIMTH	ALTITD	CPS/CVS	HVAP	EMISS	BOLTZ	
Type	I	F	F	F	F	F	F	
Default	1	none	none	none	none	none	none	
Remarks						1	1	

Card 3	1	2	3	4	5	6	7	8
Variable	XORIG	YORIG	ZORIG	XVEL	YVEL	ZVEL		
Type	F	F	F	F	F	F		
Default	none	none	none	0.0	0.0	0.0		

VARIABLE **DESCRIPTION**

VARIABLE	DESCRIPTION
PCOMB	Particle combustion mode EQ.0: no burning EQ.1: K-model EQ.2: Hybrid
NPRTCL	Initial total number of parent particles (discrete particles for calculation)
MXCNT	Maximum allowed number of parent particles (during the simulation)
PMASS	Total particle mass
SMR	Sort mean particle radius
RHOP	Particle density
TICP	Initial particle temperature
T_IGNIT	Particle ignition temperature
INITDST	Initial particle distribution EQ.1: spatially uniform EQ.2: Rosin-Rammler EQ.3: Chi-squared
AZIMTH	Angle in degrees from X-axis in X-Y plane of reference frame of TBX explosive (0 < AZMITH < 360)
ALTITD	Angle in degrees from Z-axis of reference frame of TBX explosive (0 < ALTITD < 180)
CPS/CVS	Heat coefficient
HVAP	Latent heat of vaporization
EMISS	Particle emissivity
BOLTZ	Boltzmann coefficient
XORIG	X coordinate of the origin of the initial reference frame of the TBX explosive
YORIG	Y coordinate of the origin of the initial reference frame of the TBX explosive
ZORIG	Z coordinate of the origin of the initial reference frame of the TBX explosive
XVEL	X component of the initial particle velocity the TBX explosive
YVEL	Y component of the initial particle velocity the TBX explosive
ZVEL	Z component of the initial particle velocity the TBX explosive

Remarks:

1. If radiation heat transfer is being modeled, then EMISS and BOLTZ are required.

*LSO

These cards provide a general data output mechanism, causing the creation of a sequence of LSDA files. This facility is intended to allow several different time sequences of data to be output in the same simulation. In addition, any number of domains (and any number of variables on those domains) may be specified within each time sequence. The keyword cards in this section are defined in alphabetical order:

***LSO_DOMAIN**

***LSO_POINT_SET**

***LSO_TIME_SEQUENCE**

An additional option "**_TITLE"** may be appended to all ***LSO** keywords. If this option is used, then an 80 character string is read as a title from the first card of that keyword's input. At present, LS-DYNA does not make use of the title. Inclusion of titles gives greater clarity to input decks.

***LSO_DOMAIN**

Purpose: This command provides a way to output data for specific variables at a frequency defined by ***LSO_TIME_SEQUENCE** and for a specific set of data structure (Points, nodes, elements, parts, etc. See ***LSO_POINT_SET**) in a specific binary LSO file.

Card 1	1
Variable	DOMAIN_TYPE
Type	C

Card 2	1
Variable	SOLVER_NAME
Type	C

When DOMAIN_TYPE is THIST_POINT, the following cards apply:

Card 3	1	2	3	4	5	6	7	8
Variable	OUTID	SETID						
Type	I	I						
Default	none	none						

Card 4,…	1
Variable	VARIABLE_NAME
Type	C

VARIABLE	DESCRIPTION
DOMAIN_TYPE	The type of domain for which LSO output may be generated. Accepted entries so far are ‘thist_point’

VARIABLE	DESCRIPTION
SOLVER _NAME	Selects the solver from which data is output on this domain. Accepted entries so far are 'em', 'cese' and 'icfd'.
OUTID	Output frequency ID associated to the domain. See ***LSO_TIME _SEQUENCE**.
SET_ID	Support set ID. See ***LSO_POINT_SET**
VARIABLE _NAME	Either the name of a single output variable or a variable group. See remarks.

Remarks:

1. When **DOMAIN_TYPE** is **THIST_POINT,** the following variable names are accepted:

	Solver Name		
Variable type	**EM**	**ICFD**	**CESE**
Vectors	**magneticField_point**	**velocity_point**	**velocity_point**
	electricField_point		
	vecpotField_point		
	currentDensity2_point		
Scalars	**ScalarPotential_point**	**pressure_point**	**pressure_point**
		temperature_point	**temperature _point**
		density_point	**density_point**
		lset_point	

***LSO_POINT_SET**

Purpose: Define a list of points used to sample variables in time. Of the different sampling methods, the most common one is to specify points for time history output.

Card 1	1	2	3	4	5	6	7	8
Variable	SETID							
Type	I							
Default	none							

Card 2,3,4	1	2	3	4	5	6	7	8
Variable	X	Y	Z					
Type	F	F	F					
Default	none	none	none					

VARIABLE	DESCRIPTION
SETID	Identifier for this point set. Called by ***LSO_DOMAIN**
X, Y, Z	Coordinates of the point. As many points as desired can be specified.

Remarks:

1. For the ICFD and CESE solvers, the points have to remain inside the fluid mesh. For the EM solver, the points can be defined inside the conductors or in the air. In the latter case, the fields will be computed using a Biot-Savart type integration.

*LSO_TIME_SEQUENCE

Purpose: This command provides users with maximum flexibility in specifying exactly the frequency of output of the LSO data points.. Each instance of the *LSO_TIME_SEQUENCE command creates a new time sequence with an independent output frequency.

Card 1	1
Variable	SOLVER_NAME
Type	C

Card 2	1	2	3	4	5	6	7	8
Variable	DT	LCDT	LCOPT	NPLTC	TBEG	TEND		
Type	F	I	I	I	F	F		
Default	0.0	0	1	0	0.0	0.0		

Card 3	1	2	3	4	5	6	7	8
Variable	DOMID							
Type	I							
Default	none							

VARIABLE	DESCRIPTION
SOLVER _NAME	Selects the solver from which data is output in this time sequence. Accepted entries so far are ‘em’, ‘cese’ and ‘icfd’
DT	Time interval between outputs.
LCDT	Optional load curve ID specifying the time interval between dumps.

VARIABLE	DESCRIPTION
LCOPT	Flag to govern behavior of plot frequency load curve: EQ.1: At the time each plot is generated, the load curve value is added to the current time to determine the next plot time (this is the default behavior). EQ.2: At the time each plot is generated, the next plot time T is computed so that T = the current time plus the load curve value at the time T. EQ.3: A plot is generated for each ordinate point in the load curve definition. The actual value of the load curve is ignored.
NPLTC	**DT=ENDTIM/NPLTC** overrides **DT** specified in the first field.
TBEG	The problem time at which to begin writing output to this time sequence
TEND	The problem time at which to terminate writing output to this time sequence
DOMID	Output set ID defining the domain over which variable output is to be performed in this time sequence. Each **DOMID** refers to the domain identified in an *LSO_DOMAIN keyword card.

Remarks:

1. If **LCDT** is nonzero, then it is used and **DT** and **NPLTC** are ignored. If **LCDT** is zero and **NPLTC** is non-zero, then **NPLTC** determines the snapshot time increment. If **LCDT** and **NPLTC** are both zero, then the minimum non-zero time increment specified by **DT** is used to determine the snapshot times.