LS-DYNA®
KEYWORD USER'S MANUAL

VOLUME III

Multi-Physics Solvers

LS-DYNA R11
10/16/18 (r:10577)

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

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 anisotropic 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_BLAST_LOAD}*

CESE_BOUNDARY_CONJ_HEAT_*{OPTION}

CESE_BOUNDARY_CYCLIC_*{OPTION}

CESE_BOUNDARY_FSI_*{OPTION}

CESE_BOUNDARY_NON_REFLECTIVE_*{OPTION}

CESE_BOUNDARY_PRESCRIBED_*{OPTION}

CESE_BOUNDARY_REFLECTIVE_*{OPTION}

CESE_BOUNDARY_SLIDING_*{OPTION}

CESE_BOUNDARY_SOLID_WALL_*{OPTION1}_{OPTION2}

*CESE_CHEMISTRY_D3PLOT

*CESE_CONTROL_LIMITER

*CESE_CONTROL_MESH_MOV

*CESE_CONTROL_SOLVER

*CESE_CONTROL_TIMESTEP

*CESE_DATABASE_ELOUT

*CESE_DATABASE_FLUXAVG

*CESE_DATABASE_FSIDRAG

*CESE_DATABASE_POINTOUT

*CESE_DATABASE_SSETDRAG

*CESE_DEFINE_NONINERTIAL

*CESE_DEFINE_POINT

*CESE_DRAG

*CESE_EOS_CAV_HOMOG_EQUILIB_

*CESE_EOS_IDEAL_GAS

*CESE_EOS_INFLATOR1

*CESE_EOS_INFLATOR2

*CESE_FSI_EXCLUDE

*CESE_INITIAL

CESE_INITIAL_*{OPTION}

*CESE_INITIAL_CHEMISTRY

*CESE_INITIAL_CHEMISTRY_ELEMENT

*CESE_INITIAL_CHEMISTRY_PART

*CESE_INITIAL_CHEMISTRY_SET

*CESE_MAT_000

*CESE_MAT_001 (*CESE_MAT_GAS)

*CESE_MAT_002

*CESE_PART

*CESE_SURFACE_MECHSSID_D3PLOT

*CESE_SURFACE_MECHVARS_D3PLOT

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.

CESE_BOUNDARY_AXISYMMETRIC_*OPTION

Available options are

MSURF

MSURF_SET

SET

SEGMENT

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

The MSURF and MSURF_SET options are used when the CESE mesh has been created using *MESH cards. The SET and SEGMENT cards are used when *ELEMENT_SOLID cards are used to define the CESE mesh.

Surface Part Card. Card 1 format used when the MSURF keyword option is active. Provide as many cards as necessary. This input ends at the next keyword ("*") card.

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

Surface Part Set Card. Card 1 format used when the MSURF_SET keyword option is active. Provide as many cards as necessary. This input ends at the next keyword ("*") card.

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

Set Card. Card 1 format used when the SET keyword option is active. Provide as many cards as necessary. This input ends at the next keyword ("*") card.

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

Segment Cards. Card 1 format used when SEGMENT keyword option is active. Include an additional card for each corresponding pair of segments. This input ends at the next keyword ("*") card.

Card 1d	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
MSURFID	Mesh surface part ID referenced in *MESH_SURFACE_ELEMENT cards.
MSURF_S	Identifier of a set of mesh surface part IDs created with a *LSO_ID_SET card, where each mesh surface part ID in the set is referenced in *MESH_SURFACE_ELEMENT cards.
SSID	Segment set ID
N1, N2, …	Node IDs 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_BLAST_LOAD_*OPTION

Available options include:

MSURF

MSURF_SET

SET

SEGMENT

Purpose: For the CESE compressible flow solver, set boundary values for velocity, density, and pressure from a blast wave defined by a *LOAD_BLAST_ENHANCED card. Boundary values are applied at the centroid of elements connected with this boundary. OPTION = SET and OPTION = SEGMENT are for user defined meshes whereas OPTION = MSURF or MSURF_SET are associated with the automatic volume mesher (See *MESH keywords).

That is, the MSURF and MSURF_SET options are used when the CESE mesh has been created using *MESH cards. The SET and SEGMENT cards are used when *ELEMENT_SOLID cards are used to define the CESE mesh.

Surface Part Card. Card 1 format used when the MSURF keyword option is active.

Card 1a	1	2	3	4	5	6	7	8
Variable	BID	MSURFID						
Type	I	I						
Default	none	none						

Surface Part Set Card. Card 1 format used when the MSURF_SET keyword option is active.

Card 1b	1	2	3	4	5	6	7	8
Variable	BID	MSURF_S						
Type	I	I						
Default	none	none						

Set Card. Card 1 format used when the SET keyword option is active.

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

Segment Card. Card 1 for SEGMENT keyword option is active.

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

VARIABLE	DESCRIPTION
BID	Blast source ID (see *LOAD_BLAST_ENHANCED).
MSURFID	A mesh surface part ID referenced in *MESH_SURFACE_ELEMENT cards
MSURF_S	Identifier of a set of mesh surface part IDs created with a *LSO_ID_SET card, where each mesh surface part ID in the set is referenced in *MESH_SURFACE_ELEMENT cards.
SSID	Segment set ID
N1, N2, …	Node ID's defining a segment

CESE_BOUNDARY_CONJ_HEAT_*OPTION

Available options are:

MSURF

MSURF_SET

SET

SEGMENT

Purpose: Define a conjugate heat transfer interface condition for CESE compressible flows. This condition identifies those boundary faces of the CESE mesh that are in contact with non-moving structural parts, and through which heat flows. This is only possible when the structural thermal solver is also in being used in the structural parts.

The MSURF and MSURF_SET options are used when the CESE mesh has been created using *MESH cards. The SET and SEGMENT cards are used when *ELEMENT_SOLID cards are used to define the CESE mesh.

Surface Part Card. Card 1 used when the MSURF keyword option is active. Include as many cards as necessary. This input ends at the next keyword ("*") card.

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

Surface Part Set Card. Card 1 used when the MSURF_SET keyword option is active. Include as many cards as necessary. This input ends at the next keyword ("*") card.

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

Set Card. Card 1 used when the SET keyword option is active. Include as many cards as necessary. This input ends at the next keyword ("*") card.

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

Segment Cards. Card 1 used when SEGMENT keyword option is active. Include an additional card for each corresponding pair of segments. This input ends at the next keyword ("*") card.

Card 1d	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
MSURFID	Mesh surface part ID referenced in *MESH_SURFACE_ELEMENT cards.
MSURF_S	Identifier of a set of mesh surface part IDs created with an *LSO_ID_SET card, where each mesh surface part ID in the set is referenced in *MESH_SURFACE_ELEMENT cards.
SSID	Segment set ID
N1, N2, …	Node IDs defining a segment

Remarks:

1. This boundary condition should only be imposed on a CESE mesh boundary that is in contact with non-moving structural parts. An Eulerian CESE solver is required, as is use of the structural thermal solver.

CESE_BOUNDARY_CYCLIC_*OPTION

Available options are:

MSURF

MSURF_SET

SET

SEGMENT

Purpose: Define a cyclic (periodic) boundary condition for CESE compressible flows. This cyclic boundary condition (CBC) can be used on periodic boundary surfaces.

The MSURF and MSURF_SET options are used when the CESE mesh has been created using *MESH cards. The SET and SEGMENT cards are used when *ELEMENT_SOLID cards are used to define the CESE mesh.

Card Sets. The following sequence of cards comprises a *single set*. LS-DYNA will continue reading *CESE_BOUNDARY_SOLID_WALL card sets until the next keyword ("*") card is encountered.

Surface Part Card. Card 1 format used when the MSURF keyword option is active.

Card 1a	1	2	3	4	5	6	7	8
Variable	MSURFID1	MSURFID2	CYCTYP					
Type	I	I	I					
Default	none	none	0					
Remarks			1, 2					

Surface Part Set Card. Card 1 format used when the MSURF_SET keyword option is active.

Card 1b	1	2	3	4	5	6	7	8
Variable	MSRF_S1	MSRF_S2	CYCTYP					
Type	I	I	I					
Default	none	none	0					
Remarks			1, 3					

Set Card. Card 1 format used when the SET keyword option is active.

Card 1c	1	2	3	4	5	6	7	8
Variable	SSID1	SSID2	CYCTYP					
Type	I	I	I					
Default	none	none	0					
Remarks			1, 4					

Segment Card. Card 1 format used when SEGMENT keyword option is active. Include an additional card for each corresponding pair of segments. This input ends at the next keyword ("*") card.

Card 1d	1	2	3	4	5	6	7	8
Variable	ND1	ND2	ND3	ND4	NP1	NP2	NP3	NP4
Type	I	I	I	I	I	I	I	I
Default	none	none	none	none	none	none	none	none

Rotation Case Card. Additional card for the MSURF, MSURF_SET, and SET options when CYCTYP = 1.

Card 2a	1	2	3	4	5	6	7	8
Variable	AXISX1	AXISY1	AXISZ1	DIRX	DIRY	DIRZ	ROTANG	
Type	F	F	F	F	F	F	F	
Default	0.0	0.0	0.0	none	none	none	none	

Translation Case Card. Additional card for the MSURF, MSURF_SET, and SET options when CYCTYP = 2.

Card 2b	1	2	3	4	5	6	7	8
Variable	TRANSX	TRANSY	TRANSZ					
Type	F	F	F					
Default	none	none	none					

VARIABLE	DESCRIPTION
MSURFID1, MSURFID2	Mesh surface part numbers referenced in *MESH_SURFACE_-ELEMENT cards.
MSRF_S1, MSRF_S2	Identifiers of two sets of mesh surface part IDs, each created with a *LSO_ID_SET card, where each mesh surface part ID in each set is referenced in *MESH_SURFACE_ELEMENT cards.
CYCTYP	Relationship between the two cyclic boundary condition surfaces: EQ.0: none assumed (default) EQ.1: The first surface is rotated about an axis to match the second surface. EQ.2: The faces of the first surface are translated in a given direction to obtain the corresponding faces on the second surface.
SSID1 & SSID2	A pair of segment set IDs

ND*i*, NP*i*	Node IDs defining a pair of segments: ND1, ND2, ND3, ND4 define the first segment, while NP1, NP2, NP3, NP4 define the second segment. This pair of segments must match either through a geometric translation or rotation.
AXIS[Z,Y,Z]1	A point on the axis of rotation for CYCTYP.EQ.1.
DIR[X,Y,Z]	The direction which together with AXIS[X,Y,Z]1 defines the axis of rotation for CYCTYP.EQ.1.
ROTANG	The angle of rotation (in degrees) that transforms the centroid of each face on the first surface to the centroid of the corresponding face on the second surface (for CYCTYP.EQ.1).
TRANS[X,Y,Z]	The translation direction that enables the identification of the segment in the second surface that matches a segment in the first surface (for CYCTYP.EQ.2).

Remarks:

1. For the MSURF, MSURF_SET, or SET options with CYCTYP.EQ.0, the code examines the geometry of two faces of the two surfaces in order to determine if the surfaces are approximately parallel (CYCTYP.EQ.2), or related through a rotation (CYCTYP.EQ.1). The geometric parameters required are then computed.

2. For the MSURF option, there must be the same number of mesh surface elements in each mesh surface part, and the mesh surface elements in each mesh surface part are then internally ordered in order to match pairwise between the two mesh surface parts.

3. For the MSURF_SET option, there must be the same number of mesh surface elements in each mesh surface part set, and the mesh surface elements in each mesh surface part set are then internally ordered in order to match pairwise between the two mesh surface part sets.

4. For the SET option, there must be the same number of segments in each set, and the segments in each set are then internally ordered in order to match pairwise between the two sets.

CESE_BOUNDARY_FSI_*OPTION

Available options are:

MSURF

MSURF_SET

SET

SEGMENT

Purpose: Define an FSI boundary condition for the moving mesh CESE compressible flow solver. This card must not be combined with the immersed-boundary method CESE solver, and doing so will result in an error termination condition.

This boundary condition must be applied on a surface of the CESE computational domain that is co-located with surfaces of the outside boundary of the structural mechanics mesh. The nodes of the two meshes will generally not be shared.

The MSURF and MSURF_SET options are used when the CESE mesh has been created using *MESH cards. The SET and SEGMENT cards are used when *ELEMENT_SOLID cards are used to define the CESE mesh.

Surface Part Card. Card 1 format used when the MSURF keyword option is active. Provide as many cards as necessary. This input ends at the next keyword ("*") card.

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

Surface Part Set Card. Card 1 format used when the MSURF_SET keyword option is active. Provide as many cards as necessary. This input ends at the next keyword ("*") card.

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

Set Card. Card 1 format used when the SET keyword option is active. Provide as many cards as necessary. This input ends at the next keyword ("*") card.

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

Segment Cards. Card 1 format used when SEGMENT keyword option is active. Include an additional card for each corresponding pair of segments. This input ends at the next keyword ("*") card.

Card 1d	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
MSURFID	Mesh surface part ID referenced in *MESH_SURFACE_ELEMENT cards.

VARIABLE	DESCRIPTION
MSURF_S	Identifier of a set of mesh surface part IDs created with a *LSO_-ID_SET card, where each mesh surface part ID in the set is referenced in *MESH_SURFACE_ELEMENT cards.
SSID	Segment set ID.
N1, …	Node IDs defining a segment

Remarks:

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

CESE_BOUNDARY_NON_REFLECTIVE_*OPTION

Available options are:

MSURF

MSURF_SET

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.

The MSURF and MSURF_SET options are used when the CESE mesh has been created using *MESH cards. The SET and SEGMENT cards are used when *ELEMENT_SOLID cards are used to define the CESE mesh.

Surface Part Card. Card 1 used when the MSURF keyword option is active. Include as many cards as necessary. This input ends at the next keyword ("*") card.

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

Surface Part Set Card. Card 1 used when the MSURF_SET keyword option is active. Include as many cards as necessary. This input ends at the next keyword ("*") card.

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

Set Card. Card 1 used when the SET keyword option is active. Include as many cards as necessary. This input ends at the next keyword ("*") card.

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

Segment Cards. Card 1 used when SEGMENT keyword option is active. Include an additional card for each corresponding pair of segments. This input ends at the next keyword ("*") card.

Card 1d	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
MSURFID	Mesh surface part ID referenced in *MESH_SURFACE_ELEMENT cards.
MSURF_S	Identifier of a set of mesh surface part IDs created with an *LSO_ID_SET card, where each mesh surface part ID in the set is referenced in *MESH_SURFACE_ELEMENT cards.
SSID	Segment set ID
N1, N2, …	Node IDs defining a segment

Remarks:

1. This boundary condition is usually imposed on an open surface that is far from the main disturbed flow (the further 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:

MSURF

MSURF_SET

SET

SEGMENT

Purpose: For the CESE compressible flow solver, set boundary values for velocity, density, pressure and temperature. Boundary values are applied at the centroid of elements connected with this boundary. OPTION = SET and OPTION = SEGMENT are for user defined meshes whereas OPTION = MSURF or MSURF_SET are associated with the automatic volume mesher (See *MESH keywords).

That is, the MSURF and MSURF_SET options are used when the CESE mesh has been created using *MESH cards. The SET and SEGMENT cards are used when *ELEMENT_SOLID cards are used to define the CESE mesh.

Card Sets:

A set of data cards for this keyword consists of 3 of the following cards:

1. Card 1 specifies the object to which the boundary condition is applied. Its format depends on the keyword option.

2. Card 2 reads in load curve IDs.

3. Card 3 reads in scale factors.

For each boundary condition to be specified include one set of cards. This input ends at the next keyword ("*") card.

Surface Part Card. Card 1 format used when the MSURF keyword option is active.

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

Surface Part Set Card. Card 1 format used when the MSURF_SET keyword option is active.

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

Set Card. Card 1 format used when the SET keyword option is active.

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

Segment Card. Card 1 for SEGMENT keyword option is active.

Card 1d	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			

Load Curve Card.

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	1,2,3	1,2,3	1,2,3	1,2,3	1,2,3	1,2,3		

Scale Factor Card.

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	2	2	2	2	2	2		

VARIABLE	DESCRIPTION
MSURFID	A mesh surface part ID referenced in *MESH_SURFACE_ELEMENT cards
MSURF_S	Identifier of a set of mesh surface part IDs created with a *LSO_ID_SET card, where each mesh surface part ID in the set is referenced in *MESH_SURFACE_ELEMENT cards.
SSID	Segment set ID
N1, N2, …	Node ID's defining a segment
IDCOMP	For inflow boundaries in problems involving chemical reacting flows, the chemical mixture of the fluid entering the domain is defined with a *CHEMISTRY_COMPOSITION card with this ID.
LC_U	Load curve ID to describe the x-component of the velocity versus time; see *DEFINE_CURVE.

VARIABLE	DESCRIPTION
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 (v_x, v_y, v_z, 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.

3. If a load ID is -1 for a given variable, then the boundary value for that variable is computed by the solver, and not specified by the user.

CESE_BOUNDARY_PRESCRIBED_VN_*OPTION

Available options include:

MSURF

MSURF_SET

SET

SEGMENT

Purpose: For the CESE compressible flow solver, set boundary values for velocity, density, pressure and temperature. Boundary values are applied at the centroid of elements connected with this boundary. OPTION = SET and OPTION = SEGMENT are for user defined meshes whereas OPTION = MSURF or MSURF_SET are associated with the automatic volume mesher (See *MESH keywords).

That is, the MSURF and MSURF_SET options are used when the CESE mesh has been created using *MESH cards. The SET and SEGMENT cards are used when *ELEMENT_SOLID cards are used to define the CESE mesh.

Card Sets:

A set of data cards for this keyword consists of 3 of the following cards:

1. Card 1 specifies the object to which the boundary condition is applied. Its format depends on the keyword option.

2. Card 2 reads in load curve IDs.

3. Card 3 reads in scale factors.

For each boundary condition to be specified include one set of cards. This input ends at the next keyword ("*") card.

Surface Part Card. Card 1 format used when the MSURF keyword option is active.

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

Surface Part Set Card. Card 1 format used when the MSURF_SET keyword option is active.

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

Set Card. Card 1 format used when the SET keyword option is active.

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

Segment Card. Card 1 for SEGMENT keyword option is active.

Card 1d	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			

Load Curve Card.

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

Scale Factor Card.

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

VARIABLE	DESCRIPTION
MSURFID	A mesh surface part ID referenced in *MESH_SURFACE_ELEMENT cards
MSURF_S	Identifier of a set of mesh surface part IDs created with a *LSO_ID_SET card, where each mesh surface part ID in the set is referenced in *MESH_SURFACE_ELEMENT cards.
SSID	Segment set ID
N1, N2, …	Node ID's defining a segment
IDCOMP	For inflow boundaries in problems involving chemical reacting flows, the chemical mixture of the fluid entering the domain is defined with a *CHEMISTRY_COMPOSITION card with this ID.
LC_VN	Load curve ID to describe the normal velocity versus time; see *DEFINE_CURVE.

VARIABLE	DESCRIPTION
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_VN	Scale factor for LC_VN (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 (V_N, 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.

3. If a load ID is -1 for a given variable, then the boundary value for that variable is computed by the solver, and not specified by the user.

CESE_BOUNDARY_REFLECTIVE_*OPTION

Available options are:

MSURF

MSURF_SET

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.

The MSURF and MSURF_SET options are used when the CESE mesh has been created using *MESH cards. The SET and SEGMENT cards are used when *ELEMENT_SOLID cards are used to define the CESE mesh.

Surface Part Card. Card 1 format used when the MSURF keyword option is active. Provide as many cards as necessary. This input ends at the next keyword ("*") card.

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

Surface Part Set Card. Card 1 format used when the MSURF_SET keyword option is active. Provide as many cards as necessary. This input ends at the next keyword ("*") card.

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

Set Card. Card 1 format used when the SET keyword option is active. Provide as many cards as necessary. This input ends at the next keyword ("*") card.

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

Segment Cards. Card 1 format used when SEGMENT keyword option is active. Include an additional card for each corresponding pair of segments. This input ends at the next keyword ("*") card.

Card 1d	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
MSURFID	Mesh surface part ID referenced in *MESH_SURFACE_ELEMENT cards.
MSURF_S	Identifier of a set of mesh surface part IDs created with a *LSO_ID_SET card, where each mesh surface part ID in the set is referenced in *MESH_SURFACE_ELEMENT cards.
SSID	Segment set ID.
N1, N2, …	Node IDs defining a segment

Remarks:

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

CESE_BOUNDARY_SLIDING_*OPTION

Available options are:

MSURF

MSURF_SET

SET

SEGMENT

Purpose: Allows nodes of a fluid surface to translate in the main direction of mesh movement. This is useful in piston type applications.

The MSURF and MSURF_SET options are used when the CESE mesh has been created using *MESH cards. The SET and SEGMENT cards are used when *ELEMENT_SOLID cards are used to define the CESE mesh.

Surface Part Card. Card 1 format used when the MSURF keyword option is active. Provide as many cards as necessary. This input ends at the next keyword ("*") card.

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

Surface Part Set Card. Card 1 format used when the MSURF_SET keyword option is active. Provide as many cards as necessary. This input ends at the next keyword ("*") card.

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

Set Card. Card 1 format used when the SET keyword option is active. Provide as many cards as necessary. This input ends at the next keyword ("*") card.

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

Segment Cards. Card 1 format used when SEGMENT keyword option is active. Include an additional card for each corresponding pair of segments. This input ends at the next keyword ("*") card.

Card 1d	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
MSURFID	Mesh surface part ID referenced in *MESH_SURFACE_ELEMENT cards.
MSURF_S	Identifier of a set of mesh surface part IDs created with a *LSO_ID_SET card, where each mesh surface part ID in the set is referenced in *MESH_SURFACE_ELEMENT cards.
SSID	Segment set ID
N1, N2, …	Node IDs defining a segment

CESE_BOUNDARY_SOLID_WALL_*OPTION1_OPTION2

For *OPTION1* the choices are:

MSURF

MSURF_SET

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.

The MSURF and MSURF_SET options are used when the CESE mesh has been created using *MESH cards. The SET and SEGMENT cards are used when *ELEMENT_SOLID cards are used to define the CESE mesh.

Card Sets. The following sequence of cards comprises a *single set*. LS-DYNA will continue reading *CESE_BOUNDARY_SOLID_WALL card sets until the next keyword ("*") card is encountered.

Surface Part Card. Card 1 format used when the MSURF keyword option is active.

Card 1a	1	2	3	4	5	6	7	8
Variable	MSURFID	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

Surface Part Set Card. Card 1 format used when the MSURF_SET keyword option is active.

Card 1b	1	2	3	4	5	6	7	8
Variable	MSURF_S	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

Set Card. Card 1 format used when the SET keyword option is active.

Card 1c	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

Segment Card. Card 1 format used when SEGMENT keyword option is active.

Card 1d	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

Rotating Axis Card. Additional card for the "Segment Card" case that is read when the ROTAT keyword option is used.

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
MSURFID	Mesh surface part ID referenced in *MESH_SURFACE_ELEMENT cards.
MSURF_S	Identifier of a set of mesh surface part IDs created with a *LSO_ID_SET card, where each mesh surface part ID in the set is referenced in *MESH_SURFACE_ELEMENT cards.
SSID	Segment set ID
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_CHEMISTRY_D3PLOT

Purpose: Cause mass fractions of the listed chemical species to be added to the CESE d3plot output. This is only used when chemistry is being solved with the CESE solver.

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

Species Cards. Include one card for each species to be included in the d3plot database. This input ends at the next keyword ("*") card.

Card 2	1	2	3	4	5	6	7	8
Variable	SPECIES							
Type	A							

VARIABLE	DESCRIPTION
MODELID	Identifier of a Chemkin-compatible chemistry model.
SPECIES	Name of a chemical species that is defined in the chemistry model identified by MODELID (see *CHEMISTRY_MODEL).

*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. $\alpha \geq 0$; larger values give more stability, but less accuracy. Usually $\alpha = 2.0$ or 4.0 will be enough for normal shock problems.

2. $0 \leq \beta \leq 1$; larger values give more stability. For problems with shock waves, $\beta = 1.0$ is recommended.

3. $\varepsilon \geq 0$; 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					
Type	I	I	F					
Default	1	100	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).

.

*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	ICESE	IFLOW	IGEOM	IFRAME	MIXID	IDC	ISNAN	
Type	I	I	I	I	I	F	I	
Default	0	0	none	0	none	0.25	0	
Remarks			1, 2			3		

VARIABLE	DESCRIPTION
ICESE	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.2: Two-dimensional (2D) problem EQ.3: Three-dimensional (3D) problem EQ.101: 2D axisymmetric
IFRAME	Choose the frame of reference: EQ.0: Usual non-moving reference frame (default). EQ.1000: Non-inertial rotating reference frame.
MIXID	Chemistry model ID that defines the chemical species to include in the mixing model (see *CHEMISTRY_MODEL). The species information is given through the model's card specifying the Chemkin-compatible input.

VARIABLE	DESCRIPTION
IDC	Contact interaction detection coefficient (for FSI and conjugate heat transfer problems).
ISNAN	Flag to check for a NaN in the CESE solver solution arrays at the completion of each time step. This option can be useful for debugging purposes. There is a cost overhead when this option is active. EQ.0: No checking, EQ.1: Checking is active.

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 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. 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 radial and axial directions respectively.

3. IDC is the same type of variable that is input on the *ICFD_CONTROL_FSI card. For an explanation, see Remark 1 for the *ICFD_CONTROL_FSI card.

*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_DATABASE_ELOUT

Purpose: This keyword enables the output of CESE data on elements. If more than one element set is defined, then several output files will be generated.

Output Options Card.

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

Include as many cards as needed. This input ends at the next keyword ("*") card.

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

VARIABLE	DESCRIPTION
OUTLV	Determines if the output file should be dumped. EQ.0: No output file is generated. EQ.1: The output file is generated.
DTOUT	Time interval to print the output. If DTOUT is equal to 0.0, then the CESE timestep will be used.
ELSID	Solid Elements Set ID.

Remarks:

1. The file name for this database is cese_elout.dat.

*CESE_DATABASE_FLUXAVG

Purpose: This keyword enables the output of CESE data on segment sets. If more than one segment set is defined, then several output files will be generated.

Output Options Card.

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

Include as many cards as needed. This input ends at the next keyword ("*") card.

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

VARIABLE	DESCRIPTION
OUTLV	Determines if the output file should be dumped. EQ.0: No output file is generated. EQ.1: The output file giving the average fluxes is generated.
DTOUT	Time interval to print the output. If DTOUT is equal to 0.0, then the CESE timestep will be used.
SSID	Segment Set ID.

Remarks:

1. The file names for this database is cese_fluxavg.dat.

*CESE_DATABASE_FSIDRAG

Purpose: This keyword enables the output of the total fluid pressure force applied on solid parts in FSI problems at every time step.

Output Options Card.

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

VARIABLE	DESCRIPTION
OUTLV	Determines if the output file should be dumped. EQ.0: No output file is generated. EQ.1: The output file giving the pressure forces is generated.

Remarks:

1. The file names for this database are cese_dragsol.dat, cese_dragshell.dat, cese_dragsol2D.dat and cese_dragbeam.dat .depending on what kind of solid is used.

*CESE_DATABASE_POINTOUT

Purpose: This keyword enables the output of CESE data on points.

Output Options Card.

Card 1	1	2	3	4	5	6	7	8
Variable	PSID	DTOUT	PSTYPE	VX	VY	VZ		
Type	I	F	I	F	F	F		
Default	0	0.	0	0.	0.	0.		

Include as many cards as needed. This input ends at the next keyword ("*") card.

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

VARIABLE	DESCRIPTION
PSID	Point Set ID.
DTOUT	Time interval to print the output. If DTOUT is equal to 0.0, then the CESE timestep will be used.
PSTYPE	Point Set type : EQ.0: Fixed points. EQ.1: Tracer points using prescribed velocity. EQ.2: Tracer points using fluid velocity.
VX, VY, VZ	Constant velocities to be used when PSTYPE = 1
PID	Point ID
X, Y, Z	Point initial coordinates

Remarks:

1. The file name for this database is cese_pointout.dat.

*CESE_DATABASE_SSETDRAG

Purpose: This keyword enables the output of CESE drag forces on segment sets. If more than one segment set is defined, then several output files will be generated.

Output Options Card.

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

Include as many cards as needed. This input ends at the next keyword ("*") card.

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

VARIABLE	DESCRIPTION
OUTLV	Determines if the output file should be dumped. EQ.0: No output file is generated. EQ.1: The output file giving the average fluxes is generated.
DTOUT	Time interval to print the output. If DTOUT is equal to 0.0, then the CESE timestep will be used.
SSID	Segment Set ID.

Remarks:

1. The file name for this database is *cese_ssetdrag.dat*.

2. In order for the friction drag to give consistent results, special care must be given to the mesh close to the solid wall boundary (Good capturing of the boundary layer behavior). A very fine structured mesh is recommended.

*CESE_DEFINE_NONINERTIAL

Purpose: Define the CESE problem domain as a non-inertial rotating frame that rotates at a constant rate. This is used in rotating problems such as spinning cylinders, wind turbines and turbo machinery.

Card 1	1	2	3	4	5	6	7	8
Variable	FREQ	LCID	PID	Nx	Ny	Nz		
Type	F	I	I	F	F	F		
Default	none	0	none	none	none	none		

Card 2	1	2	3	4	5	6	7	8
Variable	L	R	RELV					
Type	F	F	I					
Default	none	none	0					

VARIABLE	DESCRIPTION
FREQ	Frequency of rotation.
LCID	Load curve ID for scaling factor of FREQ.
PID	Starting point ID for the reference frame (See *CESE_DEFINE_-POINT).
Nx, Ny, Nz	Rotating axis direction.
L	Length of rotating frame.
R	Radius of rotating frame.

VARIABLE	DESCRIPTION
RELV	Velocity display mode: EQ.0: Relative velocity, only the non-rotating components of the velocity are output. EQ.1: Absolute velocity is output.

*CESE_DEFINE_POINT

Purpose: Define points to be used by the CESE solver.

Point Cards. Include one card for each point. This input ends at the next keyword ("*") card.

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

VARIABLE	DESCRIPTION
NID	Identifier for this point.
X, Y, Z	Coordinates of the point.

*CESE_DRAG

Purpose: Provide the far-field (or free-stream) fluid pressure.

Card 1	1	2	3	4	5	6	7	8
Variable	PRESS							
Type	F							

VARIABLE	DESCRIPTION
PRESS	Value of the free-stream fluid pressure (in units used by the current problem).

*CESE_EOS_CAV_HOMOG_EQUILIB

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

Include as many cards as needed. This input ends at the next keyword ("*") card.

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_EOS_INFLATOR1

Purpose: To define an EOS using Cp and Cv thermodynamic expansions for an inflator gas mixture with a single temperature range.

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

Card 2	1	2	3	4	5	6	7	8
Variable	Cp0	Cp1	Cp2	Cp3	Cp4			
Type	F	F	F	F	F			
Default	0.	0.	0.	0.	0.			

Card 3	1	2	3	4	5	6	7	8
Variable	Cv0	Cv1	Cv2	Cv3	Cv4			
Type	F	F	F	F	F			
Default	0.	0.	0.	0.	0.			

VARIABLE	DESCRIPTION
EOSID	Equation of state identifier for the CESE solver.
Cp0, …, Cp4	Coefficients of temperature-dependent specific heat at constant pressure $C_p(T) = C_{p0} + C_{p1}\,T + C_{p2}\,T^2 + C_{p3}\,T^3 + C_{p4}\,T^4$

VARIABLE	DESCRIPTION
Cv0, …, Cv4	Coefficients of temperature-dependent specific heat at constant volume $C_v(T) = C_{v0} + C_{v1}\,T + C_{v2}\,T^2 + C_{v3}\,T^3 + C_{v4}\,T^4$

Remark:

1. These coefficient expansions for the specific heats over the entire temperature range are generated by the 0-D inflator model solver. See *CHEMISTRY_CONTROL_INFLATOR and *CHEMISTRY_INFLATOR_PROPERTIES for details related to running that solver.

*CESE_EOS_INFLATOR2

Purpose: To define an EOS using Cp and Cv thermodynamic expansions for an inflator gas mixture with two temperature ranges, one below 1000 degrees Kelvin, and the other above 1000 degrees Kelvin.

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

Card for the expansion of Specific Heat at Constant Pressure. Valid for T < 1000 ^{0}K

Card 2	1	2	3	4	5	6	7	8
Variable	Cp1_0	Cp1_1	Cp1_2	Cp1_3	Cp1_4			
Type	F	F	F	F	F			
Default	0.	0.	0.	0.	0.			

Card for the expansion of Specific Heat at Constant Pressure. Valid for T > 1000 ^{0}K.

Card 3	1	2	3	4	5	6	7	8
Variable	Cp2_0	Cp2_1	Cp2_2	Cp2_3	Cp2_4			
Type	F	F	F	F	F			
Default	0.	0.	0.	0.	0.			

Card for the expansion of Specific Heat at Constant Volume. Valid for T < 1000 ^{0}K

Card 4	1	2	3	4	5	6	7	8
Variable	Cv1_0	Cv1_1	Cv1_2	Cv1_3	Cv1_4			
Type	F	F	F	F	F			
Default	0.	0.	0.	0.	0.			

Card for the expansion of Specific Heat at Constant Volume. Valid for T > 1000 ^{0}K.

Card 5	1	2	3	4	5	6	7	8
Variable	Cv2_0	Cv2_1	Cv2_2	Cv2_3	Cv2_4			
Type	F	F	F	F	F			
Default	0.	0.	0.	0.	0.			

VARIABLE	DESCRIPTION
EOSID	Equation of state identifier for the CESE solver.
Cp1_0, …, Cp1_4	Coefficients of temperature-dependent specific heat at constant pressure valid for T < 1000 ^{0}K. $C_{p1}(T) = C_{p1\_0} + C_{p1\_1}\,T + C_{p1\_2}\,T^2 + C_{p1\_3}\,T^3 + C_{p1\_4}\,T^4$
Cp2_0, …, Cp2_4	Coefficients of temperature-dependent specific heat at constant pressure valid for T > 1000 ^{0}K. $C_{p2}(T) = C_{p2\_0} + C_{p2\_1}\,T + C_{p2\_2}\,T^2 + C_{p2\_3}\,T^3 + C_{p2\_4}\,T^4$
Cv1_0, …, Cv1_4	Coefficients of temperature-dependent specific heat at constant volume valid for T < 1000 ^{0}K. $C_{v1}(T) = C_{v1\_0} + C_{v1\_1}\,T + C_{v1\_2}\,T^2 + C_{v1\_3}\,T^3 + C_{v1\_4}\,T^4$
Cv2_0, …, Cv2_4	Coefficients of temperature-dependent specific heat at constant volume valid for T > 1000 ^{0}K. $C_{v2}(T) = C_{v2\_0} + C_{v2\_1}\,T + C_{v2\_2}\,T^2 + C_{v2\_3}\,T^3 + C_{v2\_4}\,T^4$

Remark:

2. These coefficient expansions for the specific heats over two temperature ranges are generated by the 0-D inflator model solver. See *CHEMISTRY_CONTROL_INFLATOR and *CHEMISTRY_INFLATOR_PROPERTIES for details related to running that solver.

*CESE_FSI_EXCLUDE

Purpose: Provide a list of mechanics solver parts that are not involve in the CESE FSI calculation. This is intended to be used as an efficiency measure for parts that will not involve significant FSI interactions with the CESE compressible fluid solver..

Include as many cards as needed. This input ends at the next keyword ("*") card.

Card 1	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
PIDn	IDs of mechanics parts that will be excluded from the FSI interaction calculation with the CESE solver.

*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.

Include as many cards as needed. This input ends at the next keyword ("*") card.

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 an initial value by this card, *CESE_INITIAL will no longer apply to that element.

*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.

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_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	

Element List Card. Provide as many cards as necessary. This input ends at the next keyword ("*") card.

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_000

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

Material Definition Cards. Include one card for each instance of this material type. This input ends at the next keyword ("*") card.

Card 1	1	2	3	4	5	6	7	8
Variable	MID	MU	K					
Type	I	F	F					
Default	none	none	none					

VARIABLE	DESCRIPTION
MID	Material identifier
MU	Fluid dynamic viscosity. For Air at 15 °C, MU = 1.81×10^{-5} $^{kg}/_{ms}$
K	Thermal conductivity of the fluid

Remarks:

1. The viscosity is only used viscous flows, so for inviscid flows, it is not necessary to define it. The thermal conductivity is only used to calculate the heat transfer between the structure and the thermal solver when coupling is activated.

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

*CESE_MAT_001(_GAS)

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

Include as many cards as needed. This input ends at the next keyword ("*") card.

Card 1	1	2	3	4	5	6	7	8
Variable	MID	C1	C2	PRND				
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., $$\mu = \frac{C_1 T^{\frac{3}{2}}}{T + C_2}$$ where C_1 and C_2 are constants for a given gas. For example, for air at moderate temperatures, $$C_1 = 1.458 \times 10^{-6}\ \mathrm{kg/msK^{1/2}}, \quad C_2 = 110.4\ \mathrm{K}$$
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. C1 and C2 are only used to calculate the viscosity in viscous flows, so for inviscid flows, this material card is not needed. The Prandtl number is used to extract the thermal conductivity, which is used when thermal coupling with the structure is activated.

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_MAT_002

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

Material Definition Cards. Include one card for each instance of this material type. This input ends at the next keyword ("*") card.

Card 1	1	2	3	4	5	6	7	8
Variable	MID	MU0	SMU	K0	SK	T0		
Type	I	F	F	F	F	F		
Default	none	1.716E-5	111.	0.0241	194.0	273.0		

VARIABLE	DESCRIPTION
MID	Material identifier
MU0 / SMU	Two coefficients appearing in the equation derived by combining Sutherland's formula with the Power law for dilute gases: $$\frac{\mu}{\mu_0} = \left(\frac{T}{T_0}\right)^{3/2} \frac{T_0 + S_\mu}{T + S_\mu}.$$ where μ_0 is a reference value, and S_μ is an effective temperature called the Sutherland constant, which is characteristic of the gas. For air at moderate temperatures, $$\mu_0 = 1.716 \times 10^{-5}\,\mathrm{Ns/m^2}, \qquad S_\mu = 111\ \mathrm{K}$$
K0/SK	Two coefficients appearing in the equation derived by combining Sutherland's formula with the Power law for dilute gases: $$\frac{k}{k_0} = \left(\frac{T}{T_0}\right)^{3/2} \frac{T_0 + S_k}{T + S_k}$$ where k is the thermal conductivity, k_0 is a reference value, and S_k is the Sutherland constant, which is characteristic of the gas. For air at moderate temperatures, $$k_0 = 0.0241\ \mathrm{W/m}, \qquad S_k = 194\ \mathrm{K}$$
T_0	Reference temperature. The default value is for air, K.

Remarks:

1. The viscosity is only used viscous flows, so for inviscid flows, it is not necessary to define it. The thermal conductivity is only used to calculate the heat transfer between the structure and the thermal solver when coupling is activated.

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

*CESE_PART

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

Part Cards. Include one card for each CESE part. This input ends at the next keyword ("*") card.

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 (must be different from any PID on a *PART card)
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 inviscid flows.

*CESE_SURFACE_MECHSSID_D3PLOT

Purpose: Identify the surfaces to be used in generating surface D3PLOT output for the CESE solver. These surfaces must be on the outside of volume element parts that are in contact with the CESE fluid mesh. The variables in question are part of the CESE FSI solution process or of the CESE conjugate heat transfer solver.

Include as many cards as needed. This input ends at the next keyword ("*") card.

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

VARIABLE	DESCRIPTION
SSID	Mechanics solver segment set ID that is in contact with the fluid CESE mesh.
SurfaceLabel	Name to use in d3plot output to identify the SSID for the LSPP user.

*CESE_SURFACE_MECHVARS_D3PLOT

Purpose: List of variables to output on the surfaces designated by the segment set IDs given in the *CESE_SURFACE_MECHSSID_D3PLOT cards. Most of the allowed variables are defined only on the fluid-structure interface, and so the segment set IDs defining a portion of the fluid-structure interface must involve only segments (element faces) that are on the outside of volume element parts that are in contact with the CESE fluid mesh.

Include as many cards as needed. This input ends at the next keyword ("*") card.

Card 1	1	2	3	4	5	6	7	8
Variable	Output Quantity							
Type	A							
Default	none							

VARIABLE	DESCRIPTION

VARIABLE	DESCRIPTION
Output Quantity	Descriptive phrase for the mechanics surface variable to output for the LSPP user. Output will be done on all SSIDs selected by the *CESE_SURFACE_MECHSSID_D3PLOT cards in the problem. Supported variables include: FLUID FSI FORCE FLUID FSI PRESSURE INTERFACE TEMPERATURE SOLID INTERFACE HEAT FLUX FLUID INTERFACE HEAT FLUX INTERFACE HEAT FLUX RATE SOLID INTERFACE DISPLACEMENT SOLID INTERFACE VELOCITY SOLID INTERFACE ACCELERATION Force, displacement, velocity, and acceleration are output as vector quantities. The rest of the variables are scalar quantities. The fluxes are in the normal direction to the fluid/structure interface, with the heat fluxes relative to the normal pointing into the structure.

*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_CSP

*CHEMISTRY_CONTROL_FULL

*CHEMISTRY_CONTROL_INFLATOR†

*CHEMISTRY_CONTROL_TBX

*CHEMISTRY_CONTROL_ZND†

*CHEMISTRY_DET_INITIATION†

*CHEMISTRY_INFLATOR_PROPERTIES†

*CHEMISTRY_MODEL

*CHEMISTRY_PATH

†: Card may be used only once in a given model

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.

In order to use one of the chemistry solvers, the input must include at least one *CHEMISTRY_MODEL card. For each spatial region containing a different chemical composition, at least one *CHEMISTRY_COMPOSITION card is required.

The *CHEMISTRY_CONTROL_0D card is intended to be used in a standalone fashion to verify the validity of a given chemistry model. This model includes the total number

of species and all elementary reactions with their Arrhenius rate parameters. For instance, this solver could be used to check the induction time of the model.

The *CHEMISTRY_CONTROL_1D, *CHEMISTRY_DET_INITIATION, and *CHEMISTRY_CONTROL_ZND cards are intended to provide a one-dimensional initialization to a 2D or 3D chemically-reacting flow.

In order to perform a full, general purpose chemistry calculation in 2D or 3D, the *CHEMISTRY_CONTROL_FULL card should be used.

The *CHEMISTRY_CONTROL_CSP card is an option for reducing the number of species and reactions that are used in a general purpose chemistry calculation. Other reduction mechanisms are planned for the future.

An airbag inflator model is available with *CHEMISTRY_CONTROL_INFLATOR along with *CHEMISTRY_INFLATOR_PROPERTIES and a chemistry model that is referenced via three chemical compositions. This involves zero-dimensional modeling, with pyrotechnic inflator, and cold and hot flow hybrid inflator options.

The *CHEMISTRY_CONTROL_TBX card is intended for use only in a stochastic particle model, where the *STOCHASTIC_TBX_PARTICLES card is used.

*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.

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

Species List Card. Provide as many cards as necessary. This input ends at the next keyword ("*") card.

Card 2	1	2	3	4	5	6	7	8
Variable	MOLFR	SPECIES						
Type	F	A						
Default	none	none						

VARIABLE	DESCRIPTION
ID	A unique identifier among all chemistry compositions.
MODELID	Identifier of a Chemkin-compatible chemistry model.
MOLFR	The number of moles corresponding to the species named in the SPECIES field. But if used with a *STOCHASTIC_TBX_PARTICLES card, it is the molar concentration of the species (in units of moles/[length]^3, where "[length]" is the user's length unit).
SPECIES	The Chemkin-compatible name of a chemical species that is defined in the chemistry model identified by MODELID (see *CHEMISTRY_MODEL).

*CHEMISTRY_CONTROL_0D

Purpose: Performs a zero-dimensional 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	PLOTDT	CSP_SEL			
Type	I	I	I	F	I			
Default	none	none	none	1.0e-6	0			
Remarks					1			

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		

CSP Parameters Card. Include cards for each chemical species in the following format when CSP_SEL.GT.0. This input ends at the next keyword ("*") card.

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

VARIABLE	DESCRIPTION
ID	Identifier for this 0D computation.

VARIABLE	DESCRIPTION
COMPID	Chemical composition identifier of composition to use.
SOLTYP	Type of 0D calculation: EQ.1: Isochoric EQ.2: Isobaric
PLOTDT	Simulation time interval for output both to the screen and to the isocom.csv file. This file can be loaded into LS-PREPOST for curve plotting using the x-y plot facility.
CSP_SEL	CSP solver option: EQ.0: Do not use the CSP solver, and ignore the AMPL and YCUT parameters (default). GT.0: Use the CSP solver, with the AMPL and YCUT parameters.
DT	Initial time step
TLIMIT	Time limit for the simulation
TIC	Initial temperature
PIC	Initial pressure
RIC	Initial density
EIC	Initial internal energy
AMPL	Relative accuracy for the mass fraction of a chemical species in the Chemkin input file.
YCUT	Absolute accuracy for the mass fraction of a chemical species in the Chemkin input file.

Remarks:

1. If CSP_SEL.GT.0, then instead of using the full chemistry solver, the computational singular perturbation (CSP) method solver is used.

*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	CSP_SEL				
Type	I	F	I	I				
Default	none	none	none	0				
Remarks				1				

One-Dimensional Solution LSDA Input File Card.

Card 2	1	2	3	4	5	6	7	8
Variable	FILE							
Type	A							

CSP Parameters Card Include cards for each chemical species in the following format when CSP_SEL > 0. This input ends at the next keyword ("*") card.

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

VARIABLE	DESCRIPTION
ID	Identifier for this one-dimensional detonation solution.

VARIABLE	DESCRIPTION
XYZD	Position of the detonation front in the DETDIR direction.
DETDIR	Detonation propagation direction EQ.1: x EQ.2: y EQ.3: z
CSP_SEL	CSP solver option: EQ.0: Do not use the CSP solver, and ignore the AMPL and YCUT parameters (default). GT.0: Use the CSP solver, with the AMPL and YCUT parameters.
FILE	Name of the LSDA file containing the one-dimensional solution.
AMPL	Relative accuracy for the mass fraction of a chemical species in the chemkin input file.
YCUT	Absolute accuracy for the mass fraction of a chemical species in the chemkin input file.

Remarks:

1. If CSP_SEL > 0, then instead of using the full chemistry solver, the computational singular perturbation (CSP) method solver is used.

*CHEMISTRY_CONTROL_CSP

Purpose: Computes reduced chemistry for a specified Chemkin chemistry model using the Computational Singular Perturbation (CSP) method. This card can be used for general-purpose chemical reaction calculations.

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

CSP Parameters Card. Include cards for each chemical species in the following format as indicated by the value of IERROPT. This input ends at the next keyword ("*") card.

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

VARIABLE	DESCRIPTION
ID	Identifier for this computational singular perturbation solver.
IERROPT	Selector: EQ.0: AMPL and YCUT values for all chemical species are required. EQ.1: One CSP Parameter Card should be provided, and it will be used for all species.
AMPL	Relative accuracy for the mass fraction of a chemical species in the Chemkin input file.
YCUT	Absolute accuracy for the mass fraction of a chemical species in the Chemkin input file.

*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	RHOMIN	TMIN				
Type	I	F	F	F				
Default	none	none	0.0	0.0				

VARIABLE	DESCRIPTION
ID	Identifier for this full chemistry calculation.
ERRLIM	Error tolerance for the full chemistry calculation.
RHOMIN	Minimum fluid density above which chemical reactions are computed.
TMIN	Minimum temperature above which chemical reactions are computed.

*CHEMISTRY_CONTROL_INFLATOR

Purpose: Provide the required properties of an inflator model for airbag inflation.

Card 1	1	2	3	4	5	6	7	8
Variable	MODEL	OUT_TYPE	TRUNTIM	DELT	PTIME			
Type	I	I	F	F	F			
Remarks	1	2,4						

Inflator Output Database File (an ASCII file) Card.

Card 2	1	2	3	4	5	6	7	8
Variable	FILE							
Type	A							

Densities for Condensed Species. Include as many cards as needed. This input ends at the next keyword ("*") card.

Card 3	1	2	3	4	5	6	7	8
Variable	DENSITY	Species Name						
Type	F	A						
Default	none	none						
Remark		3						

VARIABLE	DESCRIPTION

VARIABLE	DESCRIPTION
MODEL	Type of inflator model to compute. EQ.1: Pyrotechnic model EQ.2: Hybrid model with cold flow option in the gas chamber EQ.3: Hybrid model with heat flow in the gas chamber
OUT_TYPE	Selects the output file format that will be used in an airbag simulation. EQ.0: Screen output (see Remark 4) EQ.1: CESE compressible flow solver (default) EQ.2: ALE solver EQ.3: CPM solver (with 2nd-order expansion of C_p) EQ.4: CPM solver (with 4th-order expansion of C_p)
TRUNTIM	Total run time.
DELT	Delta(t) to use in the model calculation.
PTIME	Time interval for output of time history data to FILE.
FILE	Name of the ASCII file in which to write the time history data and other data output by the inflator simulation.
DENSITY	Density of a condensed-phase species present in the inflator.
Species Name	Chemkin-compatible name of a condensed-phase species.

Remarks:

1. If MODEL = 3, the solution of an elementary reaction system is required for the finite-rate chemistry in the gas chamber.

2. Output file includes all of the necessary thermodynamics variables and load curves for the species mass flow rate, temperature, and density curve. This will make it possible to generate the velocity curve which is required by each solver that carries out an airbag simulation.

3. At least one of these cards will be input if condensed-phase species are present during the propellant combustion. In this case, the user must specify each condensed-phase density. This density is then used to compute the volume fractions in both the combustion and gas chamber, where the energy equations are needed.

4. If OUT_TYPE = 0, the propellant information will be displayed on the screen, including total mass, remaining mass percentage, and mass burning rate versus time. With this option, the user can quickly see the effect of changing the parameters on the first three *CHEMISTRY_INFLATOR_PROPERTIES cards.

*CHEMISTRY_CONTROL_TBX

Purpose: Specify a chemistry solver for use in conjunction with stochastic TBX particles. This is intended only for modeling the second phase of an explosion where the explosive has embedded metal (aluminum) particles that are too large to have burned in the first phase of the explosion.

This chemistry card points to a *CHEMISTRY_MODEL card (via IDCHEM) with its associated *CHEMISTRY_COMPOSITION cards to set up the initial conditions. That is, it establishes the spatial distribution of the species in the model.

It is assumed that there is no chemical reaction rate information in the chemistry model files. This is done since a special chemical reaction mechanism is implemented for TBX modeling. If particles other than solid aluminum particles are embedded in the explosive, then another burn model has to be implemented.

Surface Part Card. Card 1 format used when the PART keyword option is active**.**

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

VARIABLE	DESCRIPTION
IDCHEM	Identifier for this chemistry solver.
USEPAR	Coupling flag indicating if a *STOCHASTIC_TBX_PARTICLES card is provided for this model: EQ.1: uses a *STOCHASTIC_TBX_PARTICLES card (default). EQ.0: does not use such a card.

*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

LSDA Output File Card.

Card 2	1	2	3	4	5	6	7	8
Variable	FILE							
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)

VARIABLE	DESCRIPTION
FILE	Name of the LSDA file in which to write the one-dimensional solution.

*CHEMISTRY_INFLATOR_PROPERTIES

Purpose: Provide the required properties of an inflator model.

Card 1	1	2	3	4	5	6	7	8
Variable	COMP_ID	PDIA	PHEIGHT	PMASS	TOTMASS			
Type	I	F	F	F	F			
Remarks	1							

Card 2	1	2	3	4	5	6	7	8
Variable	TFLAME	PINDEX	A0	TDELAY	RISETIME			
Type	F	F	F	F	F			
Default	none	none	none	none	None			

Combustion Chamber Parameter Card.

Card 3	1	2	3	4	5	6	7	8
Variable	COMP1ID	VOL1	AREA1	CD1	P1	T1	DELP1	
Type	I	F	F	F	F	F	F	
Default	none	none	none	none	none	none	none	

Gas Plenum Parameter Card.

Card 4	1	2	3	4	5	6	7	8
Variable	COMP2ID	VOL2	AREA2	CD2	P2	T2	DELP2	
Type	I	F	F	F	F	F	F	
Default	none	none	none	none	none	none	none	

Tank Parameter Card.

Card 5	1	2	3	4	5	6	7	8
Variable	COMP3ID	VOL3	P3	T3				
Type	I	F	F	F				
Default	none	none	none	none				

VARIABLE	DESCRIPTION
COMP_ID	Chemical composition identifier of the composition for the steady-state propellant combustion (see Remark 1).
PDIA	Propellant diameter.
PHEIGHT	Propellant height.
PMASS	Individual cylinder propellant mass.
TOTMASS	Total propellant mass.
TFLAME	Adiabatic flame temperature.
PINDEX	Power of the pressure in rate of burn model.
A0	Steady-state constant.
TDELAY	Ignition time delay.

VARIABLE	DESCRIPTION
RISETIME	Rise time.
COMP1ID	Chemical composition identifier of composition to use in the combustion chamber.
VOL1	Volume of the combustion chamber.
AREA1	Area of the combustion chamber.
CD1	Discharge coefficient of the combustion chamber.
P1	Pressure in the combustion chamber.
T1	Temperature in the combustion chamber.
DELP1	Rupture pressure in the combustion chamber.
COMP2ID	Chemical composition identifier of composition to use in the gas plenum.
VOL2	Volume of the gas plenum.
AREA2	Area of the gas plenum.
CD2	Discharge coefficient of the gas plenum.
P2	Pressure in the gas plenum.
T2	Temperature in the gas plenum.
DELP2	Rupture pressure in the gas plenum.
COMP3ID	Chemical composition identifier of composition to use in the tank.
VOL3	Volume of the tank.
P3	Pressure in the tank.
T3	Temperature in the tank.

Remarks:

1. The propellant composition can be obtained by running a chemical equilibrium program such as NASA CEA, the CHEETAH code, or the PEP code. LSTC

provides a modified version of the PEP code along with documentation for users; it is available upon request.

*CHEMISTRY_MODEL

Purpose: Identifies the files that define a Chemkin chemistry model.

Card 1	1	2	3	4	5	6	7	8
Variable	MODELID	JACSEL	ERRLIM					
Type	I	I	F					
Default	none	1	1.0e-3					

Chemkin Input File Card.

Card 2	1	2	3	4	5	6	7	8
Variable	FILE1							
Type	A							

Thermodynamics Database File Card.

Card 3	1	2	3	4	5	6	7	8
Variable	FILE2							
Type	A							

Transport Properties Database File Card.

Card 4	1	2	3	4	5	6	7	8
Variable	FILE3							
Type	A							

VARIABLE	DESCRIPTION
MODELID	Identifier for this Chemkin-based chemistry model..

VARIABLE	DESCRIPTION
JACSEL	Selects the form of the Jacobian matrix for use in the source term. EQ.1: Fully implicit (default) EQ.2: Simplified implicit
ERRLIM	Allowed error in element balance in a chemical reaction.
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_PATH

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

Include as many cards as needed. This input ends at the next keyword ("*") card.

Card 1	1	2	3	4	5	6	7	8
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_BATTERY_RANDLES

*EM_BOUNDARY

*EM_CIRCUIT

*EM_CIRCUIT_CONNECT

*EM_CIRCUIT_RANDLE

*EM_CIRCUIT_ROGO

*EM_CONTACT

*EM_CONTACT_RESISTANCE

*EM_CONTROL

*EM_CONTROL_CONTACT

*EM_CONTROL_SWITCH

*EM_CONTROL_SWITCH_CONTACT

*EM_CONTROL_TIMESTEP

*EM_DATABASE_CIRCUIT

*EM_DATABASE_CIRCUIT0D

*EM_DATABASE_ELOUT

*EM_DATABASE_FIELDLINE

*EM_DATABASE_GLOBALENERGY

*EM_DATABASE_NODOUT

*EM_VOLTAGE_DROP

*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			STARSSID	ENDSSID	NUMSEC	
Type	I	I			I	I	I	
Default	none	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
STARSSID, ENDSSID	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.
NUMSEC	Number of Sectors. This field gives the ratio of the full circle to the angular extension of the mesh. This has to be a power of two. For example, NUMSEC = 4 means that the mesh of the part represents one fourth of the total circle. If this value is set to 0, then the value from *EM_ROTATION_AXIS is used instead.

Remarks:

1. 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_BATTERY_RANDLES

Purpose: define the distributed Randles circuit parameters for a Randles cell when using a solid mechanical model.

Card 1	1	2	3	4	5	6	7	8
Variable	RDLID	RDLTYPE	RDLAREA	CCPPART	CCNPART	SEPPART	PELPART	NELPART
Type	I	I	I	I	I	I	I	I
Default	none	none	none	none	none	none	none	none

Card 2	1	2	3	4	5	6	7	8
Variable	Q	CQ	SOCINIT	SOCTOU				
Type	F	F	F	F				
Default	none	none	none	none				

Card 3	1	2	3	4	5	6	7	8
Variable	R0CHA	R0DIS	R10CHA	R10DIS	C10CHA	C10DIS		
Type	F	F	F	F	F	F		
Default	none	none	none	none	none	none		

Card 4	1	2	3	4	5	6	7	8
Variable	TEMP	FRTHERM	ROTOTH	DUDT	TEMPU			
Type	F	I	I	F	I			
Default	none	none	none	none	none			

Card 5	1	2	3	4	5	6	7	8
Variable	USESOCS	TAUSOCS	SICSLCID					
Type	I	F	I					
Default	none	none	none					

VARIABLE	DESCRIPTION
RDLID	Id of the Randles Cell
RDLTYPE	Type of Randles Cell EQ.1: Only option available for now.
RDLAREA	Randle Area: EQ.1: The parameters are per unit area and will be scaled in each Randles circuit by a factor depending on the local area of the circuit. EQ.2: Default. The parameters are defined for the whole cell and will be scaled in each Randles circuit by a factor depending on the local area of the circuit and the global area of the cell. EQ.3: The parameters are not scaled by area factors.
CCPPART	Current Collector Positive Part ID
CCNPART	Current Collector Negative Part ID
SEPPART	Separator Part ID

VARIABLE	DESCRIPTION
PELPART	Positive Electrode Part ID
NELPART	Negative Electrode Part ID
Q	Cell capacity.
CQ	SOC conversion factor (%/s), known to be equal to 1/36 in S.I units.
SOCINIT	Initial state of charge of the cell.
SOCTOU	Constant value if positive or load curve ID if negative integer defining the equilibrium voltage (OCV) as a function of the state of charge (SOC).
R0CHA/ R10CHA/ C10CHA	Constant if positive value or load curve or table id (if negative integer) defining r0/r10/c10 when the current flows in the charge direction as a function of: -SOC if load curve -SOC and Temperature if table.
R0DIS/ R10DIS/ C10DIS	Constant if positive value or load curve or table id (if negative integer) defining r0/r10/c10 when the current flows in the discharge direction as a function of: -SOC if load curve -SOC and Temperature if table.
TEMP	Constant temperature value used for the Randles circuit parameters in case there is no coupling with the thermal solver (FRTHERM = 0)
FRTHERM	From Thermal : EQ.0: The temperature used in the Randles circuit parameters is TEMP EQ.1: The temperature used in the Randle circuit parameter is the temperature from the thermal solver.

VARIABLE	DESCRIPTION
R0TOTH	R0 to Thermal : EQ.0: The joule heating in the resistance r0 is not added to the thermal solver EQ.1: The joule heating in the resistance r0 is added to the thermal solver
DUDT	If negative integer, load curve ID of the reversible heat as a function of SOC.
TEMPU	Temperature Unit : EQ.0: The temperature is in Celsius EQ.1: The Temperature is in Kelvin
USESOCS	Use SOC shift (See Remark 1) : EQ.0: Don't use the added SOCshift EQ.1: Use the added SOCshift
TAUSOCS	Damping time in the SOCshift equation (See Remark 1)
SOCSLCID	Load curve giving f(i) where I is the total current in the unit cell

Remarks:

1. Sometimes, an extra term called SOCshift (or SocS) can be added at high rate discharges to account for diffusion limitations. The SOCshift is added to SOC for the calculation of the OCV u(SOC+SOCshift) and r0(Soc+SOCshift). SOC-shift satisfies the following equation:

$$d(SOCshift)/dt + SOCshift/tau = f(i(t))/tau$$

with SOCshift(t = 0)=0

*EM_BOUNDARY

Purpose: Define some boundary conditions for the electromagnetism problems.

Include as many cards as needed. This input ends at the next keyword ("*") card.

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

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).

***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	T0
Type	I	I	I	F	F	F	F	F
Default	none	none	none	none	none	none	none	0.

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. If a negative value is entered for LCID, its absolute value will refer to a DEFINE FUNCTION for a user defined circuit equation. If a DEFINE_FUNCTION is used, the following parameters are accepted: $f(time, emdt, curr, curr1, curr2, pot1, pot2)$. *emdt* is the current timestep, *curr*, *curr*1 and , *curr*2 refer to the current value at *t*, *t-1* and *t-2*, respectfully and *pot*1, *pot*2 refer to the scalar potential at *t-1* and *t-2* respectfully. EQ.3: R, L, C, V0 circuit. 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: $V = A\sin[2\pi F(t - t_0)]$ EQ.21: Imposed current defined by a load curve over one period and a frequency F EQ.22: Imposed voltage defined by a load curve over one period and a frequency F
LCID	Load curve ID for CIRCTYP = 1, 2, 21 or 22
R/F	Value of the circuit resistance for CIRCTYP = 3 Value of the Frequency for CIRCTYP = 11, 12, 21 or 22. . For CIRCTYP = 11 or 12, to have the frequency defined by a load curve function of time, a negative value can be entered, corresponding to the load curve ID.
L/A	Value of the circuit inductance for CIRCTYP = 3 Value of the Amplitude for CIRCTYP = 11 or 12. To have the amplitude defined by a load curve, a negative value can be entered and the solver will look for the corresponding Load Curve ID.
C/t0	Value of the circuit capacity for CIRCTYP = 3 Value of the initial time t0 for CIRCTYP = 11 or 12
V0	Value of the circuit initial voltage for CIRCTYP = 3.

VARIABLE	DESCRIPTION
T0	Starting time for CIRCTYPE = 3. Default is at the beginning of the run.
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.1/11/21: 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 or input current when CIRCTYP.EQ.2/3/12/22 and CIRCTYP.EQ 1/11/21 respectively. 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 or output current when CIRCTYP = 2/3/12/22 and CIRCTYP = 1/11/21 respectively. 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 Circuit. It can be any part ID associated to the circuit.

Variable	Circuit Type (CIRCTYP)				
	Imposed 1: Current	Imposed 2: Voltage	3: R, L, C	11: F, A, t0	12: F, A, t0
LCID	M	M	-	-	-
R/L/C/V0	-	-	M	-	-
F	-	-	-	M	M
A/t0	-	-	-	M	M
SIDCURR	M	O	M	M	O
SIDVIN	M*	M	M	M*	M
SIDVOUT	M*	M	M	M*	M
PARTID	M	M	M	M	M
Variable	21: LCID, F	22 : LCID, F			
LCID	M	M	-	-	-
R/L/C/V0	-	-	-	-	-
F	M	M	-	-	-
A/t0	-	-	-	-	-
SIDCURR	M	O	-	-	-
SIDVIN	M*	M	-	-	-
SIDVOUT	M*	M	-	-	-
PARTID	M	M	-	-	-

Table 4-1. Correspondence between circuit type and card entries. "M" indicates mandatory, "M*" mandatory with exceptions (see Remark 1), "O" indicates optional, and "-" indicates ignored.

Remarks:

1. When defining a circuit with an imposed current (type 1, 11 or 21) 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, 12, 22), 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.

3. Circuit types 21 and 22 are for cases where the periodic current/tension does not exactly follow a perfect sinusoidal. The user has to provide the shape of the current/tension over one period through a LCID as well as the frequency.

*EM_CIRCUIT_CONNECT

Purpose: This keyword connects several circuits together by imposing a linear constraint on the global currents of circuit pairs

$$c_1 i_1 + c_2 i_2 = 0.$$

This is especially useful for 2D axisymmetric models involving spiral or helical coils.

Card 1	1	2	3	4	5	6	7	8
Variable	CONID	CONTYPE	CIRC1	CIRC2	C1	C2		
Type	I	I	I	I	F	F		
Default	none	none	none	none	none	none		

VARIABLE	DESCRIPTION
CONID	Id of the Circuit Connect
CONTYPE	Type of connection between circuits. For the moment, it is only possible to combine circuits by imposing a linear constraint on the global current (=1).
C1/C2	Values of the linear constraints if CONTYPE = 1.

*EM_CIRCUIT_ROGO

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

Include as many cards as needed. This input ends at the next keyword ("*") card.

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

VARIABLE	DESCRIPTION
ROGID	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} EQ.3: Magnetic field flow (B field times Area)

Remarks:

1. An ASCII file "em_rogo_*xxx*" , with *xxx* representing the rogoId, is generated for each *EM_CIRCUIT_ROGO card giving the value of the current or the magnetic field vs time.

*EM_CONTACT

Purpose: Optional card used for defining and specifying options on electromagnetic contacts between two sets of parts. Generally used with the *EM_CONTACT_RESISTANCE card. Fields left empty on this card default to the value of the equivalent field for the *EM_CONTROL_CONTACT keyword.

Contact Definition Cards. Include one card for each contact definition. This input ends at the next keyword ("*") card.

Card 1	1	2	3	4	5	6	7	8
Variable	CONTID	COTYPE	PSIDM	PSIDS	EPS1	EPS2	EPS3	D0
Type	I	I	I	I	F	F	F	F
Default	none	0	none	none	0.3	0.3	0.3	None

VARIABLE	DESCRIPTION
CONTID	Electromagnetic contact ID
COTYPE	Type of EM contact (See Remark 2) EQ.0: Contact type 0 (Default). EQ.1: Contact type 1.
PSIDM	Master part set ID
PSIDS	Slave part set ID
EPS*i*	Contact Coefficients for contact detection conditions. See discussion below.
D0	Contact condition 3 when COTYPE = 1.

Remarks:

Contact is detected when *all of the following three condition are satisfied*:

1. Contact condition 1:

$$\boldsymbol{n}_1 . \boldsymbol{n}_2 \leq -1 + \varepsilon_1$$

2. Contact condition 2:

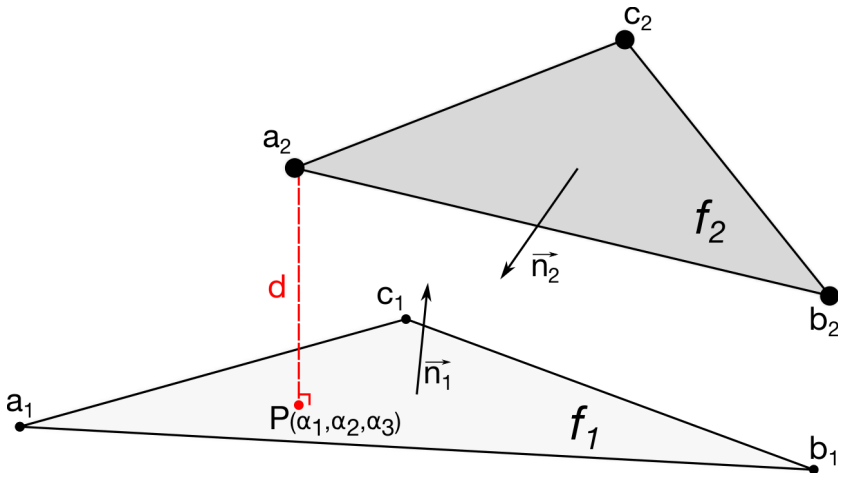


Figure 0-1. Contact detection conditions between two faces.

$$-\varepsilon_2 \le \alpha_1 \le 1+\varepsilon_2$$
$$-\varepsilon_2 \le \alpha_2 \le 1+\varepsilon_2$$
$$-\varepsilon_2 \le \alpha_3 \le 1+\varepsilon_2$$

With n_1 and n_2 the normal vectors of faces f_1 and f_2 respectfully and P the projection of point a_2 on face f_1 with $(\alpha_1, \alpha_2, \alpha_3)$ its local coordinates (See Figure 0-1).

3. Contact condition 3 depends on the contact type.

   a) For contact type 0:

   $$d \le \varepsilon_3 S_1$$

   where d is the distance between P and a_2 and where S_1 the minimum side length:

   $$S_1 = \min[d(a_1, b_1), d(b_1, c_1), d(c_1, a_1)]$$

   b) For contact type 1 :

   $$d \le D_0$$

*EM_CONTACT_RESISTANCE

Purpose: Calculate the contact resistance of a previously defined EM contact in *EM_-CONTACT. Most contact resistance calculations are based on *Ragmar Holm's "Electric Contacts"*.

Card 1	1	2	3	4	5	6	7	8
Variable	CRID	CONTID	CTYPE	CIRCID	JHRTYPE			
Type	I	I	I	I	I			
Default	none	none	none	none	none			

Card 2 if CTYPE = 1.

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

Card 2 if CTYPE = 2.

Cards 2	1	2	3	4	5	6	7	8
Variable	RHO	RAD						
Type	F	F						
Default	0.	0.						

Card 2 if CTYPE = 3.

Cards 2	1	2	3	4	5	6	7	8
Variable	RHO	RAD	D	CURLCID	EPS	HB		
Type	F	F	F	I	F	F		
Default	0.	0.	0.	0	0.	0.		

Card 2 if CTYPE = 4.

Cards 2	1	2	3	4	5	6	7	8
Variable	RHO	RAD	D	CURLCID	E	CURV		
Type	F	F	F	I	F	F		
Default	0.	0.	0.	0	0.	0.		

Card 2 if CTYPE = 5.

Cards 2	1	2	3	4	5	6	7	8
Variable	RHOPROB	RHOSUB	RHOOXY	FACTE	FACFILM			
Type	F	F	F	F	F			
Default	none	none	none	none	none			

VARIABLE	DESCRIPTION
CRID	Resistive contact ID
CONTID	EM contact ID defined in *EM_CONTACT

VARIABLE	DESCRIPTION
CTYPE	Contact Resistance type : EQ.1: Contact resistance defined by user defined load curve. EQ.2: Classic Holm's formula for contact resistances (See Remark 1). EQ.3: Modified contact resistance for cases with plastic deformation in the contact area (See Remarks 2 and 3). EQ.4: Modified contact resistance for cases with elastic deformation in the contact area (See Remarks 2 and 3). EQ.5: Basic contact resistance definition (See Remark 4).
CIRCID	Circuit ID: When defined, the contact resistance will be added to the corresponding circuit total resistance and taken into account in the circuit equations.
JHRTYPE	Indicates how the Joule heating calculated by the contact resistance shall be taken into account: EQ.0: No addition: The Joule heating calculated by the contact resistance is not taken into account. EQ.1: The Joule heating coming from the contact resistance is divided and distributed evenly among all elements neighboring the contact surface.
LCID	Load Curve ID defining the contact resistance versus time.
RHO	Material resistivity ρ_{mat}. If not defined or EQ. 0.0, the solver will automatically calculate an average resistivity based on the conductivity of the elements that are in contact.
RAD	Radius of the contact sphere a. If not defined or EQ. 0.0, the solver will automatically calculate an equivalent radius based on the contact area: $a = \sqrt{\text{Area}}/\pi$.
D	Diameter of the Electrode.
CURLCID	Load Curve ID defining the current intensity of the electrode. If not defined or EQ. 0, the solver will automatically look for the circuit's current intensity using the circuit defined in CIRID.
EPS	Constant ε with values typically between 0.35 and 1.
HB	Brinell hardness, H_b.

VARIABLE	DESCRIPTION
E	Material Young's modulus.
CURV	Radius of curvature of the contact surface, r.
RHOPROB	Probe resistivity, ρ_{prob}
RHOSUB	Substrate resistivity, ρ_{sub}
RHOOXY	Film resistivity, ρ_{oxi}
FACTE	Scale factor on the constriction area when calculating the constriction resistance. If negative, the factor is time-dependent and defined by the load curve absolute value (FACTE).
FACFILM	Scale factor on the constriction area when calculating the film resistance. If negative, the factor is time-dependent and defined by the load curve absolute value (FACFILM).

Remarks:

1. **Holm's formula for Contact Resistance.** A very good approximation of the electric contact resistance is given by Holm's formula :

$$R_{\text{contact}} = \frac{\rho_{\text{mat}}}{2a}$$

where ρ_{mat} is the material's resistivity and a is the radius of the contact surface assuming the contact surface area is close to that of a circle : Area $= \pi a^2$.

It is recommended to use this method (CTYPE = 2) in a first approach since most other contact resistance definitions are extensions of this formula.

2. **Contact Area formulations.** For certain types of applications such as resistance spot welding (RSW) it is advantageous to better approximate the area by taking into account the deformation and the heterogeneities of the materials that come into contact at a microscopic level. For a plastic deformation of the contact zone, the contact area, assumed to be circular, can be defined approximated as:

$$\text{Area} = \frac{F_c}{\varepsilon\, H_b}$$

where F_c is the contact force, ε a constant with values between 0.35 and 1, and H_b the Brinell hardness of the material.

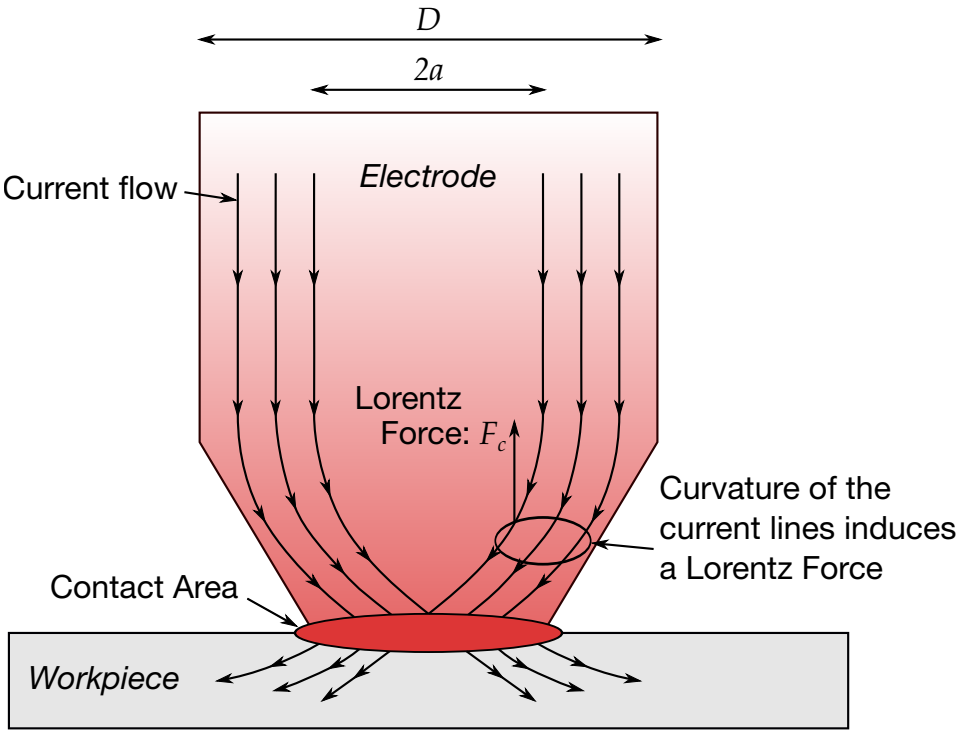


Figure 4-2. Electrode coming into contact with workpiece (RSW application).

For an elastic deformation in the contact area, the radius of the contact surface is now given by:

$$a = \frac{rF_c}{E}^{1/3}$$

where r is the radius of curvature of the contact surface and E is Young's modulus.
The Holm formula can then be modified in order to give:

$$R_{\text{contact}} = \frac{\rho_{\text{mat}}}{2} \times \sqrt{\frac{\pi \varepsilon H_b}{F_c}}$$

and

$$R_{\text{contact}} = \frac{\rho_{\text{mat}}}{2} \times \left(\frac{E}{rF_c}\right)^{1/3}$$

in the cases of plastic (CTYPE = 3) and elastic (CTYPE = 4) deformations respectfully.

3. **Lorentz Force from a Spherical Electrode.** When a spherical electrode comes into contact with a work piece, the curvature of the current flowing from the electrode to the work piece induces a Lorentz force parallel to the

normal of the contact surface thus forcing the electrode and the work piece away from each other. Its intensity can be written as:

$$F_c = \frac{\mu_0}{4\pi} I^2 \ln\left(\frac{D}{2a}\right)$$

where I is the current intensity and D the diameter of the electrode. See Figure 4-2.

4. **Basic resistive contact formulation (CTYPE = 5).** In the case of a clean metal contact with no film the resistance calculation involves only the constriction term. If a film is present and both sides have different metals, the contact resistance, R_{contact}, is the sum of the constriction resistance $R_{\text{constriction}}$ and the film resistance R_{film}. In the basic resistive model, the following expressions determine the resistance:

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

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

$$R_{\text{contact}} = R_{\text{constriction}} + R_{\text{film}}.$$

*EM_CONTROL

Purpose: Enable the EM solver and set its options.

Card 1	1	2	3	4	5	6	7	8
Variable	EMSOL	NUMLS	MACRODT					
Type	I	I	F					
Default	0	100	none					

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 a whole period for EMSOL = 2. Not used for EMSOL = 1. If a negative value is entered, it will give NUMLS function of the macro time.
MACRODT	Macro time step when EMSOL = 2. Can be used as constant EM time step when EMSOL = 1. Obsolete: use *EM_CONTROL_TIMESTEP.

*EM_CONTROL_CONTACT

Purpose: This keyword activates the electromagnetism contact algorithms, which detects contact between conductors. 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	CCONLY		COTYPE	EPS1	EPS2	EPS3	D0
Type	I	I		I	F	F	F	F
Default	0	0		0	0.3	0.3	0.3	none

VARIABLE	DESCRIPTION
EMCT	EM contact activation flag: EQ.0: No contact detection EQ.1: Contact detection
CCONLY	Determines on which parts of the model the EM contact should be activated. EQ.0: Contact detection between all active parts associated with a conducting material. (Default) EQ.1: Only look for EM contact between parts associated through the EM_CONTACT card. In some cases this option can reduce the calculation time.
COTYPE	Type of EM contact. If *EM_CONTACT is not defined, the solver will look for global contact options in *EM_CONTROL_CONTACT. EQ.0: Contact type 0 (Default). EQ.1: Contact type 1.
EPS*i*	Global contact coefficients used if the equivalent fields in *EM_-CONTACT are empty.
D0	Global contact condition 3 value when COTYPE = 1

*EM_CONTROL_SWITCH

Purpose: It is possible to active a control "switch" that will shut down the solver based on a load curve information. LS-DYNA incorporates complex types of curves (See *DEFINE_CURVE_FUNCTION) that allow the setting up of complex On/Off switches, for instance, by using a nodal temperature value.

Card 1	1	2	3	4	5	6	7	8
Variable	LCID	FEMCOMP	BEMCOMP					
Type	I	I	I					
Default	0	0	0					

VARIABLE	DESCRIPTION
LCID	Load Curve ID. Negative values switch the solver off, positive values switch it back on.
FEMCOMP	Determines if FEM matrices are recomputed each time the EM solver is turned back on : EQ.0 : FEM matrices are recomputed EQ.1 : FEM matrices are not recomputed
BEMCOMP	Determines if BEM matrices are recomputed each time the EM solver is turned back on : EQ.0 : BEM matrices are recomputed EQ.1 : BEM matrices are not recomputed

*EM_CONTROL_SWITCH_CONTACT

Purpose: It is possible to active a control "switch" that will shut down the electromagnetic contact detection. This can be useful in order to save some calculation time in cases where the user knows when contact between conductors will occur or stop occurring.

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

VARIABLE	DESCRIPTION
LCID	Load Curve ID. Negative values switch the contact detection off, positive values switch it back on.
NCYLFEM	Determines the number of cycles before FEM matrix recomputation. If defined this will overwrite the previous NCYCLFEM as long as the contact detection is turned on.
NCYLBEM	Determines the number of cycles before BEM matrix recomputation. If defined this will overwrite the previous NCYCLBEM as long as the contact detection is turned on.

*EM_CONTROL_TIMESTEP

Purpose: Controls 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 $dt_e = {l_e^2}/{2D}$, where:

- D is the diffusion coefficient $D = {1}/{\mu_0 \sigma_e}$,

- 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_DATABASE_CIRCUIT

Purpose: This keyword enables the output of EM data for every circuit defined.

Output options card

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

VARIABLE	DESCRIPTION
OUTLV	Determines if the output file should be dumped. EQ.0: No output file is generated. EQ.1: The output file is generated.
DTOUT	Time interval to print the output. If DTOUT is equal to 0.0, then the EM timestep will be used.

Remarks:

1. The file name for this database is em_circuit_*XXX*.dat with *XXX* the circuit ID.

2. *ResistanceD* is calculated in the following way:

   a) A scalar potential difference of 1 is imposed at the circuit's boundaries SIDVIN and SIDVOUT.

   b) The system to be solved at SIDCURR is then $\nabla^2\varphi = 0$ with $\varphi_{\text{SIDVIN}} = 1$ and $\varphi_{\text{SIDVOUT}} = 0$. No diffusive effects are taken into account meaning that the current density can be written as $\mathbf{j} = \nabla\varphi$ and the total current as $I = \mathbf{j} \cdot \mathbf{n}dA$.

   c) The resistance can then be estimated using $R_D = U/I$. The calculation of this R_D resistance is solely based on the circuit's geometry and conductivity. It is therefore equivalent to the resistance as commonly defined in the circuit equations:

$$R_D = L/\sigma S$$

where L is the length of the circuit and S its surface area.

3. *ResistanceJ* is calculated by using the data provided during the EM solve : $R_J = J/I^2$ where J and I are, respectively, the joule heating and the current. Compared with *ResistanceD*, *ResistanceJ* is not so much a resistance calculation since it accounts for the resistive effects (when using the Eddy current solver). Rather, it corresponds to the resistance that the circuit would need in order to get the same Joule heating in the context of a circuit equation. If all EM fields are diffused or the RH solver is being used, *ResistanceJ* should be close to *ResistanceD*.

4. Only the mutual inductances between the first three circuits defined are output.

*EM_DATABASE_CIRCUIT0D

Purpose: This keyword enables the output of EM data for every circuit defined.

Output options card

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

VARIABLE	DESCRIPTION
OUTLV	Determines if the output file should be dumped. EQ.0: No output file is generated. EQ.1: The output file is generated.
DTOUT	Time interval to print the output. If DTOUT is equal to 0.0, then the EM timestep will be used.

Remarks:

1. The file name for this database is em_circuit0D_*XXX*.dat with *XXX* the circuit ID.

2. At the start of the run, based on the initial values of the meshes resistances and inductances, the solver will calculate the results for a so-called "0D" solution which does not take into account the current's diffusion, the part's displacements or the EM material property changes. It is therefore a crude approximation. This can be useful in some cases especially in R,L,C circuits if the users wishes to have an first idea of how the source current will behave.

3. Since the calculation of this 0D circuit can take time depending on the problems size, it should only be used in cases where the output results are useful to the comprehension of the analysis.

4. This card has no influence on the results of the EM run itself.

*EM_DATABASE_ELOUT

Purpose: This keyword enables the output of EM data on elements.

Output Options Card.

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

Include as many cards as needed. This input ends at the next keyword ("*") card.

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

VARIABLE	DESCRIPTION
OUTLV	Determines if the output file should be dumped. EQ.0: No output file is generated. EQ.1: The output file is generated.
DTOUT	Time interval to print the output. If DTOUT is equal to 0.0, then the EM timestep will be used.
ELSID	Solid Elements Set ID.

Remarks:

1. The file name for this database is em_elout.dat.

*EM_DATABASE_FIELDLINE

Purpose: The EM solver uses a BEM method to calculate the EM fields between conductors. With this method, the magnetic field in the air or vacuum between conductors is therefore not explicitly computed. However, in some cases, it may be interesting to visualize some magnetic field lines for a better analysis. This keyword allows the output of field line data. It has no influence on the results of the EM solve.

Output Options Card.

Card 1	1	2	3	4	5	6	7	8
Variable	FLID	PSID	DTOUT	NPOINT				
Type	I	I	F	I				
Default	none	none	0.	100				

Remaining cards are optional.†

Card 2	1	2	3	4	5	6	7	8
Variable	INTEG	H	HMIN	HMAX	TOLABS	TOLREL		
Type	I	F	F	F	F	F		
Default	2	0.	0.	1E10	1E-3	1E-5		

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

VARIABLE	DESCRIPTION
FLID	Field line set ID
PSID	Point Set ID associated to the field line set (See *EM_POINT_SET). The coordinates given by the different points will be the starting points of the field lines.
DTOUT	Time interval to print the output. If DTOUT is equal to 0.0, then the EM time step will be used.
NPOINT	Number of points per field line. The points are regularly spaced.
INTEG	Type of numerical integrator used to compute the field lines : EQ.1: RK4, Runge Kutta 4. See Remark 2. EQ.2: DOP853, Dormand Prince 8(5,3). See Remark 2.
H	Value of the step size. In case of an integrator with adaptive step size, it is the initial value of the step size.
HMIN	Minimal step size value. Only used in the case of an integrator with adaptive step size.
HMAX	Maximal step size value. Only used in the case of an integrator with adaptive step size.
TOLABS	Absolute tolerance of the integrator. Only used in the case of an integrator with adaptive step size.
TOLREL	Relative tolerance of the integrator. Only used in the case of an integrator with adaptive step size.
BTYPE	Method to compute the magnetic field : EQ.1: Direct method (every contribution is computed by the Biot Savart Law and summed up : very slow). EQ.2: Multipole method (approximation of the direct method using the multipole expansion). EQ.3: Multicenter method (approximation of the direct method using a weighted subset of points only in order to compute the magnetic field).

Remarks:

1. **File Names.** The file name for this database is em_fieldLine_*XX*_*YYY*.dat where *XX* is the field line ID and *YYY* is the point set ID defined in *EM_POINT_SET.

2. **Integrators.** The Runge Kutta 4 integrator is an explicit iterative method for solving ODEs. It is a fourth order method with a constant step size. The Dormand Prince 8(5,3) integrator is an explicit iterative method for solving IDEs. Particularly, this integrator is an embedded Runge Kutta integrator of order 8 with an adaptive step size. This integrator allows a step size control which is done though an error estimate at each step. The Dormand Prince 8(5,3) is a Dormand Prince 8(6) for which the 6^{th} order error estimator has been replaced by a 5^{th} order estimator with 3^{rd} order correction in order to make the integrator more robust.

*EM_DATABASE_GLOBALENERGY

Purpose: This keyword enables the output of global EM.

Output Options Card.

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

VARIABLE	DESCRIPTION
OUTLV	Determines if the output file should be dumped. EQ.0: No output file is generated. EQ.1: The output file is generated.
DTOUT	Time interval to print the output. If DTOUT is equal to 0.0, then the EM timestep will be used.

Remarks:

1. The file name for this database is em_globEnergy.dat.

2. Outputs the global EM energies of the mesh, the air and the source circuit. Also outputs the global kinetic energy and the global plastic work energy.

*EM_DATABASE_NODOUT

Purpose: This keyword enables the output of EM data on nodes.

Output Options Card.

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

Include as many cards as needed. This input ends at the next keyword ("*") card.

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

VARIABLE	DESCRIPTION
OUTLV	Determines if the output file should be dumped. EQ.0: No output file is generated. EQ.1: The output file is generated.
DTOUT	Time interval to print the output. If DTOUT is equal to 0.0, then the EM timestep will be used.
NSID	Node Set ID.

Remarks:

1. The file name for this database is em_nodout.dat.

*EM_DATABASE_PARTDATA

Purpose: This keyword enables the output of EM data for every part defined. .

Output Options Card.

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

VARIABLE	DESCRIPTION
OUTLV	Determines if the output file should be dumped. EQ.0: No output file is generated. EQ.1: The output file is generated.
DTOUT	Time interval to print the output. If DTOUT is equal to 0.0, then the EM timestep will be used.

Remarks:

1. The file name for this database is em_partData_*XXX*.dat with *XXX* the part ID.
2. Outputs the part EM energies of the part as well as the Lorentz force. Also outputs the part kinetic energy and the part plastic work energy.

*EM_DATABASE_POINTOUT

Purpose: This keyword enables the output of EM data on points sets.

Output Options Card.

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

Include as many cards as needed. This input ends at the next keyword ("*") card.

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

VARIABLE	DESCRIPTION
OUTLV	Determines if the output file should be dumped. EQ.0: No output file is generated. EQ.1: The output file is generated.
DTOUT	Time interval to print the output. If DTOUT is equal to 0.0, then the ICFD timestep will be used.
PSID	Point Set ID (See *EM_POINT_SET card).

Remarks:

1. The file name for this database is em_pointout.dat.

*EM_DATABASE_ROGO

Purpose: This keyword enables the output of EM data for every circuit defined. .

Output Options Card.

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

VARIABLE	DESCRIPTION
OUTLV	Determines if the output file should be dumped. EQ.0: No output file is generated. EQ.1: The output file is generated.
DTOUT	Time interval to print the output. If DTOUT is equal to 0.0, then the EM timestep will be used.

Remarks:

1. The file name for this database is em_rogoCoil_*XXX*.dat where *XXX* is the rogo Coil ID.

*EM_DATABASE_TIMESTEP

Purpose: This keyword enables the output of EM data regarding the EM timestep.

Output options card.

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

VARIABLE	DESCRIPTION
OUTLV	Determines if the output file should be dumped. EQ.0: No output file is generated. EQ.1: The output file is generated.

Remarks:

1. The file name for this database is em_timestep.dat.

2. Outputs the run's EM tim estep versus the time step calculated using the EM CFL condition as criteria (autotimestep). This can be useful in cases with big deformations and/or material property changes and a fixed time step is being used in case that time step becomes to big compared to the stability time step.

*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)

VARIABLE	DESCRIPTION
C2	C2 constant (no units)
C3	C3 constant (no units)
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:

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

   a) If T < θ_m: solid phase model applies.

      The solid phase electrical resistivity corresponds to the Meadon model:

$$\eta_S = (C_1 + C_2\theta^{C_3}) f_c\left(\frac{V}{V_0}\right), \tag{1}$$

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:

$$f_c\left(\frac{V}{V_0}\right) = \begin{cases} \left(\frac{V}{V_0}\right)^{2\gamma-1} & \text{EXPON.EQ.} -1 \quad \text{(most materials)} \\ \left(\frac{V}{V_0}\right)^{2\gamma+1} & \text{EXPON.EQ.} +1 \quad \text{(tungsten)} \\ \left(\frac{V}{V_0}\right)^{2\gamma} & \text{EXPON.EQ. 0} \quad \text{(stainless steel)} \end{cases} \tag{2}$$

with

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

b) If $T > \theta_m$: liquid phase model:

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

with

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

where

$$\Delta\eta = \begin{cases} ke^{0.69L_F/\theta_m} & k > 0 \\ 1 + 0.0772(2 - \theta_m) & k = -1 \quad \text{(tungsten)} \\ 1 + 0.106(0.846 - \theta_m) & k = -2 \quad \text{(stainless steel SS-304)} \end{cases} \tag{5}$$

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

Parameter	Cu	Ag	Au	W	Al(2024)	SS(304)
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 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

VARIABLE	DESCRIPTION
V0	Reference specific volume V0 (UUS).
GAMMA0	Reference Gruneisen value γ_0.(no units).
EXPON	Exponent in equations (7)
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: $$\sigma(\theta) = \sigma_{\text{Burgess}}(\theta)\frac{\sigma_{\text{mat}}}{\sigma_{\text{Burgess}}(\theta_{\text{room}})}$$

Remarks:

1. The Meadon model is a simplified Burgess model with the solid phase equations only.

   The electrical resistivity is given by:

   $$\eta_S = (C_1 + C_2\theta^{C_3})f_c\left(\frac{V}{V_0}\right) \tag{6}$$

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

   In (6), the volume dependence is given by:

   $$f_c\left(\frac{V}{V_0}\right) = \begin{cases} \left(\frac{V}{V_0}\right)^{2\gamma-1} & \text{EXPON.EQ.}-1 \quad \text{(most materials)} \\ \left(\frac{V}{V_0}\right)^{2\gamma+1} & \text{EXPON.EQ.}+1 \quad \text{(tungsten)} \\ \left(\frac{V}{V_0}\right)^{2\gamma} & \text{EXPON.EQ.0} \quad \text{(stainless steel)} \\ 1 & \text{VO.EQ. 0} \quad \text{(default value for } V_0 \text{ is zero)} \end{cases} \tag{7}$$

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

with,

$$\gamma = \gamma_0 - \left(\gamma_0 - \frac{1}{2}\right)\left(1 - \frac{V}{V_0}\right) \quad (8)$$

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(cm^3/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 B function of H curve ($B = \mu H$) EQ.2: Permeability defined by a H function of B curve ($H = B/\mu$)
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_EOS_TABULATED2

Purpose: Define the electrical conductivity as a function of time 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 time (abscissa). The user needs to make sure the time 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 t = 0 at after a long time) to avoid bad extrapolations of the conductivity if the run time gets out of the load curve bounds.

2. LCID can also refer to a DEFINE FUNCTION. If a DEFINE_FUNCTION is used, the following parameters are allowed: $f(vx, vy, vz, temp, pres, vol, mass, Ex, Ey, Ez, Bx, By, Bz, Fx, Fy, Fz, JHrate, time)$. Fx, Fy, Fz refers to the Lorentz force vector.

*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_ISOPOTENTIAL

Purpose: Defining an isopotential, i.e. constrain nodes so that they have the same scalar potential value. This card is to be used with the EM solver of type 3 and the distributed Randles circuits only at this time.

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

VARIABLE	DESCRIPTION
ISOID	ID of the Isopotential
SETTYPE	Set type: EQ.2: Node Set.
SETID	Set ID
RDLTYPE	Used for the application: composite Tshell battery, with *EM_RANDLES_LAYERED. Selects which layers of the underlying EM mesh is included in the isopotential: EQ.1: Current Collector Positive EQ.2: Positive Electrode EQ.3: Separator EQ.4: Negative Electrode EQ.5: Current Collector Negative The layers functions are defined in *EM_MAT.

*EM_ISOPOTENTIAL_CONNECT

Purpose: Define a connection between two isopotentials.

Card 1	1	2	3	4	5	6	7	8
Variable	CONID	CONTYPE	ISOID1	ISOID2	VAL	LCID/RDLID	PSID	
Type	I	I	I	I	F	I	I	
Default	none	none	none	none	none	none	none	

VARIABLE	DESCRIPTION
CONID	Connection ID
CONTYPE	Connection type : EQ.1: Short Circuit. EQ.2: Resistance. EQ.3: Voltage Source. EQ.4: Current Source. EQ.5: Meshless Randles circuit (used to represent a cell by one lumped Randles circuit)
ISOID1	ID of the first isopotential to be connected
ISOID2	ID of the second isopotential to be connected
VAL	Value of the resistance, voltage or current depending on CONTYPE Ignored if LCID defined.
LCID /RDLID	Load curve ID defining the value of the resistance, voltage or current function of time and depending on CONTYPE. If not defined, VAL will be used. Or ID of the Randles circuit defined by *EM_RANDLES_MESHLESS.

VARIABLE	DESCRIPTION
PSID	Used for the application: meshless Randles circuit (CONTYPE = 5) if the variable R0TOTH of *EM_RANDLES_MESHLESS is equal to 1. Part Set ID where the joule heating corresponding to the resistance r0 in *EM_RANDLES_MESHLESS is added, averaged over the part set.

*EM_MAT_001

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

Include as many cards as needed. This input ends at the next keyword ("*") card.

Card 1	1	2	3	4	5	6	7	8
Variable	MID	MTYPE	SIGMA	EOSID				RDLTYPE
Type	I	I	F	I				I
Default	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 cards).

VARIABLE	DESCRIPTION
RDLTYPE	Used for the application: composite Tshell battery, with *EM_-RANDLES_LAYERED . Defines the function of the layer associated to MID: EQ.1: Current Collector Positive EQ.2: Positive Electrode EQ.3: Separator EQ.4: Negative Electrode EQ.5: Current Collector Negative

*EM_MAT_002

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

Include as many cards as needed. This input ends at the next keyword ("*") card.

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 cards).
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_MAT_003

Purpose: Define an electromagnetic material type whose electromagnetic conductivity is defined by a (3*3) tensor matrix. Applications include composite materials.

Orthotropic Card 1.

Card 1	1	2	3	4	5	6	7	8
Variable	MID	MTYPE	SIGMA11	SIGMA22	SIGMA33			
Type	I	I	F	F	F			

Orthotropic Card 2.

Card 2	1	2	3	4	5	6	7	8
Variable	SIGMA12	SIGMA13	SIGMA21	SIGMA23	SIGMA31	SIGMA32	AOPT	
Type	F	F	F	F	F	F	I	

Orthotropic Card 3.

Card 1	1	2	3	4	5	6	7	8
Variable	XP	YP	ZP	A1	A2	A3	MACF	
Type	F	F	F	F	F	F	I	

Orthotropic Card 4.

Card 2	1	2	3	4	5	6	7	8
Variable	V1	V2	V3	D1	D2	D3		
Type	F	F	F	F	F	F		

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.

SIGMA11 The 1, 1 term in the 3 × 3 electromagnetic conductivity tensor matrix. Note that 1 corresponds to the *a* material direction

SIGMA12 The 1, 2 term in the 3 × 3 electromagnetic conductivity tensor matrix. Note that 2 corresponds to the *b* material direction

⋮ ⋮

SIGMA33 The 3, 3 term in the 3 × 3 electromagnetic conductivity tensor matrix.

Define AOPT for both options:

AOPT Material axes option, see the figure in *MAT_002.

EQ.0.0: locally orthotropic with material axes determined by element nodes as shown in part (a) the figure in *MAT_002. The a-direction is from node 1 to node 2 of the element. The b-direction is orthogonal to the a-direction and is in the plane formed by nodes 1, 2, and 4.

EQ.1.0: locally orthotropic with material axes determined by a point in space and the global location of the element center; this is the a-direction.

EQ.2.0: globally orthotropic with material axes determined by vectors defined below, as with *DEFINE_COORDINATE_VECTOR.

EQ.3.0: locally orthotropic material axes determined by rotating the material axes about the element normal by an angle, BETA, from a line in the plane of the element defined by the cross product of the vector v with the

element normal. The plane of a solid element is the midsurface between the inner surface and outer surface defined by the first four nodes and the last four nodes of the connectivity of the element, respectively.

EQ.4.0: locally orthotropic in cylindrical coordinate system with the material axes determined by a vector v, and an originating point, P, which define the centerline axis. This option is for solid elements only.

EQ.5.0: globally defined reference frame with (a,b,c)=(X0,Y0,Z0).

XP, YP, ZP	Define coordinates of point **p** for AOPT = 1 and 4.
A1, A2, A3	Define components of vector **a** for AOPT = 2.
MACF	Material axes change flag for solid elements: EQ.1: No change, default,
V1, V2, V3	Define components of vector **v** for AOPT = 3 and 4.
D1, D2, D3	Define components of vector **d** for AOPT = 2.

Remarks:

This card works in a similar way to *MAT_002.

The procedure for describing the principle material directions is explained for solid elements for this material model. We will call the material direction the **a**-**b**-**c** coordinate system. The AOPT options illustrated in the AOPT figure of *MAT_002 can define the **a**-**b**-**c** system for all elements of the parts that use the material.

*EM_MAT_004

Purpose: Define the electromagnetic material type and properties for conducting shells in a 3D problem.

Include as many cards as needed. This input ends at the next keyword ("*") card.

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

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 cards).
NELE	Number of elements in the thickness of the shell. It is up to the user to make sure his mesh is fine enough to correctly capture the inductive-diffusive effects (see skin depth definition).

*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

VARIABLE	DESCRIPTION
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
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_POINT_SET

Purpose: This keyword creates a set of points which can be used by the *EM_DATABASE_POINTOUT keyword.

Output Options Card.

Card 1	1	2	3	4	5	6	7	8
Variable	PSID	PSTYPE	VX	VY	VZ			
Type	I	I	F	F	F			
Default	0	0	0.	0.	0.			

Include as many cards as needed. This input ends at the next keyword ("*") card.

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

VARIABLE	DESCRIPTION
PSID	Point Set ID.
PSTYPE	Point Set type : EQ.0: Fixed points. EQ.1: Tracer points using prescribed velocity.
VX, VY, VZ	Constant velocities to be used when PSTYPE = 1
PID	Point ID
X, Y, Z	Point initial coordinates

VARIABLE	DESCRIPTION
POS	Position flag (for 2D see Remark 1) : EQ.0 (default) : The solver determines if the point is inside or outside of the conductors. EQ.1: Point outside of the conductors during the entire simulation. The solver does not check; hence a gain in computation time.

Remarks:

1. If using *EM_2DAXI notice that the conductors represents the corresponding 3D conductors.

*EM_RANDLES_LAYERED

Purpose: define the distributed Randles circuit parameters for a Randles cell when using a composite Tshell mechanical model.

Card 1	1	2	3	4	5	6	7	8
Variable	RDLID	RDLTYPE	PSID	RDLAREA				
Type	I	I	I	I				
Default	none	none	none	none				

Card 2	1	2	3	4	5	6	7	8
Variable	Q	CQ	SOCINIT	SOCTOU				
Type	F	F	F	F				
Default	none	none	none	none				

Card 3	1	2	3	4	5	6	7	8
Variable	R0CHA	R0DIS	R10CHA	R10DIS	C10CHA	C10DIS		
Type	F	F	F	F	F	F		
Default	none	none	none	none	none	none		

Card 4	1	2	3	4	5	6	7	8
Variable	TEMP	FRTHERM	ROTOTH	DUDT	TEMPU			
Type	F	I	I	F	I			
Default	none	none	none	none	none			

Card 5	1	2	3	4	5	6	7	8
Variable	USESOCS	TAUSOCS	SICSLCID					
Type	I	F	I					
Default	none	none	none					

VARIABLE	DESCRIPTION
RDLID	Id of the Randles Cell
RDLTYPE	Type of Randles Cell EQ.1: Only option available for now.
PSID	Part Set ID of all the parts composing the cell
RDLAREA	Randle Area: EQ.1: The parameters are per unit area and will be scaled in each Randle circuit by a factor depending on the local area of the circuit. EQ.2: Default. The parameters are defined for the whole cell and will be scaled in each Randle circuit by a factor depending on the local area of the circuit and the global area of the cell. EQ.3: The parameters are not scaled by area factors.
Q	Cell capacity.

VARIABLE	DESCRIPTION
CQ	SOC conversion factor (%/s), known to be equal to 1/36 in S.I units.
SOCINIT	Initial state of charge of the cell.
SOCTOU	Constant value if positive or load curve ID if negative integer defining the equilibrium voltage (OCV) as a function of the state of charge (SOC).
R0CHA/ R10CHA/ C10CHA	Constant if positive value or load curve or table id (if negative integer) defining r0/r10/c10 when the current flows in the charge direction as a function of: -SOC if load curve -SOC and Temperature if table.
R0DIS/ R10DIS/ C10DIS	Constant if positive value or load curve or table id (if negative integer) defining r0/r10/c10 when the current flows in the discharge direction as a function of: -SOC if load curve -SOC and Temperature if table.
TEMP	Constant temperature value used for the Randles circuit parameters in case there is no coupling with the thermal solver (FRTHERM = 0)
FRTHERM	From Thermal : EQ.0: The temperature used in the Randles circuit parameters is TEMP EQ.1: The temperature used in the Randles circuit parameter is the temperature from the thermal solver.
R0TOTH	R0 to Thermal : EQ.0: The joule heating in the resistance r0 is not added to the thermal solver EQ.1: The joule heating in the resistance r0 is added to the thermal solver
DUDT	If negative integer, load curve ID of the reversible heat as a function of SOC.

VARIABLE	DESCRIPTION
TEMPU	Temperature Unit : EQ.0: The temperature is in Celsius EQ.1: The Temperature is in Kelvin
USESOCS	Use SOC shift (See Remark 1) : EQ.0: Don't use the added SOCshift EQ.1: Use the added SOCshift
TAUSOCS	Damping time in the SOCshift equation (See Remark 1)
SOCSLCID	Load curve giving f(i) where I is the total current in the unit cell

Remarks:

2. Each part of PSID is defined by *PART_COMPOSITE_TSHELL. The function (CCP, CCN, Sep, PosEl, NegEl) of each part's layer is defined through its associated material, in *EM_MAT (see in particular field RANDTYPE there).

3. Sometimes, an extra term called SOCshift (or SocS) can be added at high rate discharges to account for diffusion limitations. The SOCshift is added to SOC for the calculation of the OCV u(SOC+SOCshift) and r0(Soc+SOCshift). SOCshift satisfies the following equation:

$$d(SOCshift)/dt + SOCshift/tau = f(i(t))/tau$$

with SOCshift(t = 0)=0

*EM_RANDLES_MESHLESS

Purpose: define the distributed Randles circuit parameters for a Randles cell which is not associated with a mesh (lumped Randles circuit).

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

Card 2	1	2	3	4	5	6	7	8
Variable	Q	CQ	SOCINIT	SOCTOU				
Type	F	F	F	F				
Default	none	none	none	none				

Card 3	1	2	3	4	5	6	7	8
Variable	R0CHA	R0DIS	R10CHA	R10DIS	C10CHA	C10DIS		
Type	F	F	F	F	F	F		
Default	none	none	none	none	none	none		

Card 4	1	2	3	4	5	6	7	8
Variable	TEMP			DUDT	TEMPU			
Type	F			F	I			
Default	none			none	none			

Card 5	1	2	3	4	5	6	7	8
Variable	USESOCS	TAUSOCS	SICSLCID					
Type	I	F	I					
Default	none	none	none					

VARIABLE	DESCRIPTION
RDLID	Id of the Randles Cell
RDLTYPE	Type of Randles Cell EQ.1: Only option available for now.
Q	Cell capacity.
CQ	SOC conversion factor (%/s), known to be equal to 1/36 in S.I units.
SOCINIT	Initial state of charge of the cell.
SOCTOU	Constant value if positive or load curve ID if negative integer defining the equilibrium voltage (OCV) as a function of the state of charge (SOC).
R0CHA/ R10CHA/ C10CHA	Constant if positive value, or load curve (if negative integer) defining r0/r10/c10 when the current flows in the charge direction as a function of SOC.

VARIABLE	DESCRIPTION
R0DIS/ R10DIS/ C10DIS	Constant if positive value, or load curve (if negative integer) defining r0/r10/c10 when the current flows in the discharge direction as a function of SOC.
TEMP	Constant temperature value used for the Randles circuit parameters in case there is no coupling with the thermal solver.
DUDT	If negative integer, load curve ID of the reversible heat as a function of SOC.
TEMPU	Temperature Unit : EQ.0: The temperature is in Celsius EQ.1: The Temperature is in Kelvin
USESOCS	Use SOC shift (See Remark 1) : EQ.0: Don't use the added SOCshift EQ.1: Use the added SOCshift
TAUSOCS	Damping time in the SOCshift equation (See Remark 1)
SOCSLCID	Load curve giving f(i) where I is the total current in the unit cell

Remarks:

1. Sometimes, an extra term called SOCshift (or SocS) can be added at high rate discharges to account for diffusion limitations. The SOCshift is added to SOC for the calculation of the OCV u(SOC+SOCshift) and r0(Soc+SOCshift). SOC-shift satisfies the following equation:

$$d(SOCshift)/dt + SOCshift/tau = f(i(t))/tau$$

with SOCshift(t = 0)=0

*EM_RANDLES_SHORT

Purpose: For battery cell internal short, define conditions to turn on a Randles short (replace one or several Randles circuits by resistances), and to define the value of the short resistance.

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

VARIABLE	DESCRIPTION
AREATYPE	Works the same way as RDLAREA in *EM_BATTERY_RANDLES or in *EM_RANDLES_LAYERED : EQ.1: The resistance in FUNCTID is per unit area. EQ.2: Default. The resistance in FUNCTID is for the whole cell (the whole cell is shorted), and then a factor based on areaLocal/areaGlobal is applied. EQ.3: The resistance in FUNCTID is taken as is in each Randles circuit.
FUNCTID	DEFINE_FUNCTION ID giving the local resistance function of local parameters for the local Randles circuit. Accepted values are: $f(x_{ccp}, y_{ccp}, z_{ccp}, x_{sep}, y_{sep}, z_{sep}, x_{sem}, y_{sem}, z_{sem}, x_{ccm}, y_{ccm}, z_{ccm}, time)$.

Remarks:

1. If the return value of the function is negative, there is no short, the Randles circuit is maintained. If it is negative, the function gives the value of the resistance.

2. The parameter description is :

   a) x_ccp: x of boundary between positive current collector and positive electrode

b) x_sep: x of boundary between positive electrode and separator

c) x_sem: x of boundary between separator and negative electrode

d) x_ccm: x of boundary between negative electrode and negative current collector

3. An example of a function :

```
*DEFINE_FUNCTION

FID (Function Id)

Float resistance_short_randle(

float time,

 float x_ccp,float y_ccp,float z_ccp,

float x_sep,float y_sep,float z_sep,

float x_sem,float y_sem,float z_sem,

float x_ccm,float y_ccm,float z_ccm)

{ float seThick0;

seThick0 = 1.e-5;

  seThick=(sqrt(x_sep-x_sem)^2+(y_sep-y_sem)^2+(z_sep-z_
sem)^2);

if (seThick >= seThick0) then

return -1;

else

return 1.e-2;

endif
```

*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
XP, YP, ZP	x, y, and z coordinates of the point
XD, YD, ZD	x, y, and z components of direction of the axis
NUMSEC	Number of Sectors. This field gives the ratio of the full circle to the angular extension of the mesh. This has to be a power of two. For example, NUMSEC = 4 means that the mesh of the part represents one fourth of the total circle. If NUMSEC = 0 for *EM_2DAXI, the solver will replace it with this value.

*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	NCYCLBEM		
Type	I	I	I	I	I	I		
Default	1E-6	1000	2	2	1	5000		

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.

VARIABLE	DESCRIPTION
USELAST	This is used only for iterative solvers (PCG or GMRES). EQ.-1: Start from 0 as initial guess for solution of the linear system. EQ.1: Starts from the previous solution normalized by the RHS change.
NCYLBEM	Number of electromagnetism cycles between the recalculation of BEM matrices.

Remarks:

1. Using USELAST = 1 can save many iterations in the subsequent 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.

2. Since the BEM matrices depend on (and only on) the surface node coordinates of the conductors, it is important to recalculate them when the conductors are moving. The frequency with which they are updated is controlled by NCYLBEM. Note that very small values, for example NCYLBEM = 1, should, generally, be avoided since this calculation involves a high computational cost. However, when two conductors are moving and in contact with each other it is recommended to recalculate the matrices at *every* time step.

*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-6

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 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	USELAST	NCYCLFEM		
Type	I	I	I	I	I	I		
Default	10^{-3}	1000	1	1	1	5000		

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 right-hand-side change change.
NCYCLFEM	Number of electromagnetism cycles between the recalculation of FEM matrices.

Remarks:

1. Using USELAST = 1 can save many iterations in the subsequent solves if the vector solution of the present solve is assumed to be nearly parallel to the vec-

tor solution of the previous solve, as usually happens in time-domain eddy-current problems.

2. The default values are only valid when the PCG resolution method (STYPE = 2). For the default direct solve (STYPE = 1) those values are ignored.

3. When the conductor parts are deforming or undergoing changes in their EM material properties (conductivity for example), it is important to change the default value of NCYLFEM to recalculate the FEM matrices more often.

*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..

*EM_VOLTAGE_DROP

Purpose: Impose a voltage drop between two segment sets.

Card 1	1	2	3	4	5	6	7	8
Variable	VDID	VDTYPE	SSID1	SSID2	VOLT			
Type	I	I	I	I	F			
Default	none	none	none	none	none			

VARIABLE	DESCRIPTION
VDID	Voltage Drop ID
VDTYPE	Voltage Drop Type: EQ.1: Voltage drop between the two corresponding nodes of the two segment sets SSID1 and SSID2.
SSID1	Segment Set ID 1
SSID2	Segment Set ID 2
VOLT	Value of the voltage drop

*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_FLUX_TEMP

*ICFD_BOUNDARY_FREESLIP

*ICFD_BOUNDARY_FSI

*ICFD_BOUNDARY_FSWAVE

*ICFD_BOUNDARY_GROUND

*ICFD_BOUNDARY_NONSLIP

*ICFD_BOUNDARY_PRESCRIBED_MOVEMESH

*ICFD_BOUNDARY_PRESCRIBED_PRE

*ICFD_BOUNDARY_PRESCRIBED_TEMP

*ICFD_BOUNDARY_PRESCRIBED_TURBULENCE

*ICFD_BOUNDARY_PRESCRIBED_VEL

*ICFD_BOUNDARY_WINDKESSEL

*ICFD_CONTROL_ADAPT

*ICFD_CONTROL_ADAPT_SIZE

*ICFD_CONTROL_CONJ

*ICFD_CONTROL_DEM_COUPLING

*ICFD_CONTROL_EMBEDSHELL

*ICFD_CONTROL_FSI

*ICFD_CONTROL_GENERAL

*ICFD_CONTROL_IMPOSED_MOVE

*ICFD_CONTROL_LOAD

*ICFD_CONTROL_MESH

*ICFD_CONTROL_MESH_MOV

*ICFD_CONTROL_MONOLITHIC

*ICFD_CONTROL_OUTPUT

*ICFD_CONTROL_OUTPUT_SUBDOM

*ICFD_CONTROL_PARTITION

*ICFD_CONTROL_POROUS

*ICFD_CONTROL_STEADY

*ICFD_CONTROL_SURFMESH

*ICFD_CONTROL_TAVERAGE

*ICFD_CONTROL_TIME

*ICFD_CONTROL_TRANSIENT

*ICFD_CONTROL_TURB_SYNTHESIS

*ICFD_CONTROL_TURBULENCE

*ICFD_DATABASE_AVERAGE

*ICFD_DATABASE_DRAG

*ICFD_DATABASE_FLUX

*ICFD_DATABASE_HTC

*ICFD_DATABASE_NODEAVG

*ICFD_DATABASE_NODOUT

*ICFD_DATABASE_POINTAVG

*ICFD_DATABASE_POINTOUT

*ICFD_DATABASE_RESIDUALS

*ICFD_DATABASE_TEMP

*ICFD_DATABASE_TIMESTEP

*ICFD_DATABASE_UINDEX

*ICFD_DEFINE_HEATSOURCE

*ICFD_DEFINE_NONINERTIAL

*ICFD_DEFINE_POINT

*ICFD_DEFINE_WAVE_DAMPING

*ICFD_INITIAL

*ICFD_INITIAL_TURBULENCE

*ICFD_MAT

*ICFD_MODEL_NONNEWT

*ICFD_MODEL_POROUS

*ICFD_PART

*ICFD_PART_VOL

*ICFD_SECTION

*ICFD_SET_NODE

*ICFD_SOLVER_SPLIT

*ICFD_SOLVER_TOL_LSET

*ICFD_SOLVER_TOL_MMOV

*ICFD_SOLVER_TOL_MOM

*ICFD_SOLVER_TOL_MONOLITHIC

*ICFD_SOLVER_TOL_PRE

*ICFD_SOLVER_TOL_TEMP

*ICFD_BOUNDARY_CONJ_HEAT

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

Include as many cards as needed. This input ends at the next keyword ("*") card.

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_FLUX_TEMP

Purpose: Impose a heat flux on the boundary expressed as $q = \nabla T$

Include as many cards as needed. This input ends at the next keyword ("*") card.

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, *DEFINE_CURVE_FUNCTION or *DEFINE_FUNCTION. . If a DEFINE_FUNCTION is used, the following parameters are allowed: $f(x, y, z, vx, vy, vz, temp, pres, time)$.
SF	Load curve scale factor. (default = 1.0)
DEATH	Time at which the imposed motion/constraint is removed: EQ.0.0: default set to 10e28
BIRTH	Time at which the imposed pressure is activated starting from the initial abscissa value of the curve

*ICFD_BOUNDARY_FREESLIP

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

Include as many cards as needed. This input ends at the next keyword ("*") card.

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.

Include as many cards as needed. This input ends at the next keyword ("*") card.

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_FSWAVE

Purpose: Impose a wave inflow boundary condition.

Include as many cards as needed. This input ends at the next keyword ("*") card.

Card 1	1	2	3	4	5	6	7	8
Variable	PID	WTYPE	HEIGHT	WAMP	WLENG	WMAX	SFLCID	WANG
Type	I	I	F	F	F	F	I	F
Default	none	none	none	none	none	none	none	none

VARIABLE	DESCRIPTION
PID	PID for a fluid surface.
WTYPE	Wave Type: EQ.1: Stokes wave of first order EQ.2: Stokes wave of second order EQ.4: Solitary wave EQ.5: Irregular waves using JONSWAP spectrum
HEIGHT	Free surface equilibrium level.
WAMP	Wave amplitude for WTYPE = 1 to WTYPE = 4. Significant wave height for WTYPE = 5.
WLENG	Wave Length for WTYPE = 1 and WTYPE = 2. Not used for WTYPE = 4. Minimum wave frequency in spectrum (rad/sec) for WTYPE = 5.
WMAX	Maximum wave frequency in spectrum (rad/sec) for WTYPE = 5.
SFLCID	Scale factor LCID on the wave amplitude for WTYPE = 1 and WTYPE = 2. Number of Wave modes (default = 1024) for WTYPE = 5.
WANG	Angle between incoming wave direction and x-axis for z and y-aligned gravity vector, or angle between incoming wave direction and y-axis for x-aligned gravity vector.

*ICFD_BOUNDARY_GROUND

Purpose: Specify the fluid boundary with a ground boundary condition. The ground boundary condition is similar to the nonslip boundary condition except that it will keep V = 0 in all circumstances, even if the surface nodes are moving. This is useful in cases where the nodes are allowed to move or translate (using ICFD_BOUNDARY_PRESCRIBED_MOVEMESH for example) but those displacements are only to accommodate for mesh movement and do not correspond to a physical motion.

Include as many cards as needed. This input ends at the next keyword ("*") card.

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 ground boundary condition is applied.

*ICFD_BOUNDARY_NONSLIP

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

Include as many cards as needed. This input ends at the next keyword ("*") card.

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.

Include as many cards as needed. This input ends at the next keyword ("*") card.

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_PRE

Purpose: Impose a fluid pressure on the boundary.

Include as many cards as needed. This input ends at the next keyword ("*") card.

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, *DEFINE_CURVE_FUNCTION or *DEFINE_FUNCTION. . If a DEFINE_FUNCTION is used, the following parameters are allowed: $f(x, y, z, vx, vy, vz, temp, pres, time)$.
SF	Load curve scale factor. (default = 1.0)
DEATH	Time at which the imposed motion/constraint is removed: EQ.0.0: default set to 10E28
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.

Include as many cards as needed. This input ends at the next keyword ("*") card.

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, *DEFINE_CURVE_FUNCTION or *DEFINE_FUNCTION. . If a DEFINE_FUNCTION is used, the following parameters are allowed: $f(x, y, z, vx, vy, vz, temp, pres, time)$.
SF	Load curve scale factor. (default = 1.0)
DEATH	Time at which the imposed temperature is removed: EQ.0.0: default set to 10E28
BIRTH	Time at which the imposed temperature is activated starting from the initial abscissa value of the curve

*ICFD_BOUNDARY_PRESCRIBED_TURBULENCE

Purpose: Optional keyword that allows the user to strongly impose the turbulence quantities when a RANS turbulence model is selected. See ICFD_CONTROL_TURBULENCE. Mainly used to modify the default boundary conditions at the inlet.

Include as many cards as needed. This input ends at the next keyword ("*") card.

Card 1	1	2	3	4	5	6	7	8
Variable	PID	VTYPE	IMP	LCID				
Type	I	I	I	I				
Default	none	none	0	none				

VARIABLE	DESCRIPTION
PID	PID for a fluid surface.
VTYPE	Variable type. EQ.1: kinetic turbulent energy EQ.2: turbulent dissipation rate EQ.3: specific dissipation rate EQ.4: modified turbulent viscosity
IMP	Imposition method. EQ.0: Direct imposition through value specified by LCID EQ.1: Using turbulent Intensity specified by LCID if VTYPE = 1. Using turbulence length scale specified by LCID if VTYPE = 2,3 and 4. EQ.2: Using turbulent viscosity ratio specified by LCID. Only available for VTYPE = 2 and VTYPE = 3.
LCID	Load curve ID to describe the variable value versus time, see *DEFINE_CURVE, *DEFINE_CURVE_FUNCTION or *DEFINE_FUNCTION. . If a DEFINE_FUNCTION is used, the following parameters are allowed: $f(x, y, z, vx, vy, vz, temp, pres, time, k, e, mut)$.

Remarks:

1. At the inlet, the relationship between the turbulent kinetic energy k and the turbulence intensity I is given by :

$$k = \frac{3}{2}(U_{avg}{}^2 I^2)$$

By default, the solver uses an inlet intensity of 0.05 (5%).

2. At the inlet, if specifying the turbulent dissipation rate using a length scale, l, the following relationship will be used :

$$\epsilon = C_\mu{}^{3/4} \frac{k^{3/2}}{l}$$

By default, the solver estimates a length scale based on the total height of the channel. Otherwise, if using the turbulent viscosity ratio $r = \frac{\mu_t}{\mu}$ method:

$$\epsilon = \rho C_\mu \frac{k^2}{\mu\, r}$$

3. At the inlet, if specifying the specific dissipation rate using a length scale, l, the following relationship will be used :

$$\omega = C_\mu{}^{-1/4} \frac{k^{1/2}}{l}$$

By default, the solver estimates a length scale based on the total height of the channel. Otherwise, if using the turbulent viscosity ratio $r = \frac{\mu_t}{\mu}$ method:

$$\omega = \rho \frac{k}{\mu\, r}$$

4. At the inlet, the relationship between the modified turbulent viscosity $\tilde{\nu}$ is given and the length scale, l is given by :

$$\tilde{\nu} = 0.05\sqrt{\frac{3}{2}}(U_{avg}\, l)$$

*ICFD_BOUNDARY_PRESCRIBED_VEL

Purpose: Impose the fluid velocity on the boundary.

Include as many cards as needed. This input ends at the next keyword ("*") card.

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, EQ.4: Normal direction degree of freedom,
VAD	Velocity flag: EQ.1: Linear velocity EQ.2: Angular velocity EQ.3: Parabolic velocity profile EQ.4: Activates synthetic turbulent field on part. See *ICFD_CONTROL_TURB_SYNTHESIS.
LCID	Load curve ID used to describe motion value versus time, see *DEFINE_CURVE, *DEFINE_CURVE_FUNCTION, or *DEFINE_FUNCTION. If a DEFINE_FUNCTION is used, the following parameters are allowed: $f(x,y,z,vx,vy,vz,temp,pres,time)$.
SF	Load curve scale factor. (default = 1.0)
VID	Point ID for angular velocity application point, see *ICFD_DEFINE_POINT.

VARIABLE	DESCRIPTION
DEATH	Time at which the imposed motion/constraint is removed: EQ.0.0: default set to 10^{28}
BIRTH	Time at which the imposed motion/constraint is activated starting from the initial abscissa value of the curve

*ICFD_BOUNDARY_WINDKESSEL

Purpose: This boundary condition imposes the pressure function of circuit parameters where an analogy is made between the pressure and scalar potential as well as between the flux and the current intensity. Such conditions are frequently encountered in hemodynamics.

Card 1	1	2	3	4	5	6	7	8
Variable	PID	WTYPE	R1	C1	R2	L1		
Type	I	I	F	F	F	F		
Default	none	none	0.	0.	0.	0.		

Optional card if WTYPE = 3 or 4.

Card 2	1	2	3	4	5	6	7	8
Variable	P2LCID	C2	R3					
Type	I	F	F					
Default	None	0.	0.					

VARIABLE	DESCRIPTION
PID	PID for a fluid surface
WTYPE	Circuit type (See Remarks) : EQ.1: Windkessel circuit EQ.2: Windkessel circuit with inverted flux EQ.3: CV type circuit EQ.4: CV type circuit with inverted flux
R1/C1/L1/R 2/C2	Parameters (Resistances, inductances, capacities) for the different circuits.
P2LCID	Load curve ID describing behavior of P2(t) function of time for CV type circuit.

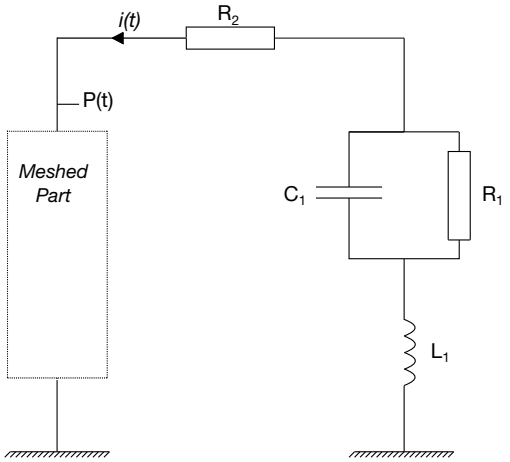


Figure [1]. Windkessel circuit

Remarks:

1. Figure 1 shows a Windkessel circuit and Figure 2 a CV circuit.

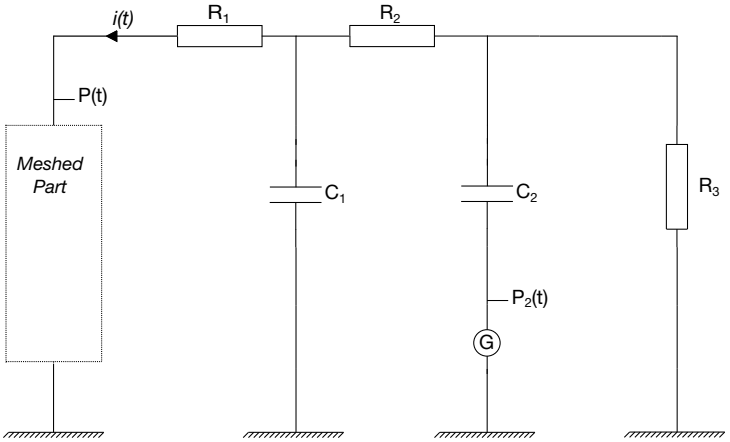


Figure [2]. CV Circuit

*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 in contrast to the default algorithm which only checks for inverted elements.

Card 1	1	2	3	4	5	6	7	8
Variable	ASIZE	NIT						
Type	I	I						
Default	0	none						

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.
NIT	Number of iterations before a re-meshing is forced. If a negative integer is entered, then a load curve function of time will be used to define NIT.

*ICFD_CONTROL_CONJ

Purpose: This keyword allows to pick between the different coupling methods for conjugate heat transfer applications

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

VARIABLE	DESCRIPTION
CTYPE	Indicates the thermal coupling type. EQ.0: Robust and accurate monolithic coupling where the temperature field are solved simultaneously between the fluid and the structure. EQ.1: Weak thermal coupling. The fluid passes the heat flux to the solid at the fluid-structure interface and the solid returns the temperature which is applied as a Dirichlet condition.

Remarks:

1. The keyword ICFD_BOUNDARY_CONJ_HEAT is ignored if CTYPE = 1 but the keyword ICFD_BOUNDARY_FSI is needed in all thermal coupling cases.

*ICFD_CONTROL_DEM_COUPLING

Purpose: This keyword is needed to activate coupling between the ICFD and DEM solvers.

Card 1	1	2	3	4	5	6	7	8
Variable	CTYPE	BT	DT	SF				
Type	I	F	F	F				
Default	0	0.	1E+28	1.				

VARIABLE	DESCRIPTION
CTYPE	Indicates the coupling direction to the solver. EQ.0: two-way coupling between the fluid and the solid particles. EQ.1: one-way coupling: The DEM particles transfer their location to the fluid solver. EQ.2: one-way coupling: The fluid solver transfers forces to the DEM particles
BT	Birth time for the DEM coupling.
DT	Death time for the DEM coupling.
SF	Scale factor applied to the force transmitted by the fluid to the structure.

*ICFD_CONTROL_EMBEDSHELL

Purpose: This keyword allows the user to control specific options related to the use of the keyword MESH_EMBEDSHELL.

Card 1	1	2	3	4	5	6	7	8
Variable	GTYPE	DIST						
Type	I	F						
Default	0	0.1						

VARIABLE	DESCRIPTION
GTYPE	Gap type. Defines the criteria for selecting a distance to build the gap between the embedded nodes and the newly generated : EQ.0: Automatic and based on the surface mesh size multiplied by a scale factor given by DIST. Default method. EQ.1: Specific gap size given by the user and defined by DIST.
DIST	Distance value if GTYPE = 1 or scale factor value if GTYPE = 0.

*ICFD_CONTROL_FSI

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

Card 1	1	2	3	4	5	6	7	8
Variable	OWC	BT	DT	IDC	LDICSF	XPROJ		
Type	I	F	F	F	I	I		
Default	0	0	1E+28	0.25	0	0		

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 mechanics solver transfers displacements to the fluid solver. EQ.2: one-way coupling: The fluid solver transfers stresses to the solid mechanics 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 mechanics solver.
DT	Death time for the FSI coupling. After DT the fluid solver will not transfer any loads to the solid mechanics solver but the fluid will continue to deform with the solid.
IDC	Interaction detection coefficient. See Remark 1.
LCIDSF	Optional load curve ID to apply a scaling factor on the forces transferred to the solid : GT.0: Load curve ID function of iterations LT.0: Load curve ID function of time

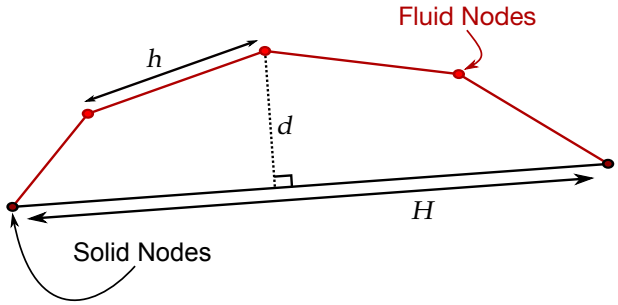


Figure 0-1. Geometry of FSI contact.

VARIABLE	DESCRIPTION
XPROJ	Projection of the nodes of the CFD domain that are at the FSI interface onto the structural mesh. EQ.0: No projection EQ.1: Projection

Remarks:

2. One of the criteria to automatically detect the fluid and solid surfaces that will interact in FSI problems is the distance d between a fluid (solid) node and a solid (fluid) element respectively:

$$d \leq \text{IDC} \times \min(h, H)$$

where h is the size of the fluid mesh, H the size of the solid mechanics mesh, and IDC a detection coefficient criteria with IDC = 0.25 by default. In the majority of cases, this default value is sufficient to ensure FSI interaction. However, it can happen in special cases that the fluid and solid geometries have curvatures that differ too much (example: pipe flows in conjugate heat transfer applications). In such cases, a bigger IDC value may be needed. This flag should be handled with care.

3. XPROJ = 1 is recommended for cases with rotation.

*ICFD_CONTROL_GENERAL

Purpose: This keyword allows choosing between the different types of CFD analyses.

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

VARIABLE	DESCRIPTION
ATYPE	Analysis type : EQ. -1: Turns off the ICFD solver after initial keyword reading. EQ.0: Transient analysis (Default) EQ.1: Steady state analysis
MTYPE	Solving Method type : EQ.0: Fractional Step Method EQ.1: Monolithic solve EQ.2: Potential flow solve

*ICFD_CONTROL_IMPOSED_MOVE

Purpose: This keyword allows the user to impose a velocity on specific ICFD parts or on the whole volume mesh. Global translation, global rotation and local rotation components can be defined and combined. This can be used in order to save calculation time in certain applications such as sloshing where the modeling of the whole fluid box and the solving of the consequent FSI problem is not necessarily needed.

Card 1	1	2	3	4	5	6	7	8
Variable	PID	LCVX	LCVY	LCVZ	VADT			
Type	I	I	I	I	I			
Default	none	none	none	none	0			

Optional Card. Rotational velocity components using Euler angles (See Remark 1).

Card 2	1	2	3	4	5	6	7	8
Variable	ALPHAL	BETAL	GAMMAL	ALPHAG	BETAG	GAMMAG	VADR	
Type	I	I	I	I	I	I	I	
Default	none	none	none	none	none	none	0	

Optional Card. Local reference frame definition if ALPHAL, BETAL or GAMMAL used.

Card 3	1	2	3	4	5	6	7	8
Variable	PTID	X1	Y1	Z1	X2	Y2	Z2	
Type	I	F	F	F	F	F	F	
Default	0	1.	0.	0.	0.	1.	0.	

VARIABLE	DESCRIPTION
PID	PID. This can be any part ID referenced in *ICFD_PART or *ICFD_PART_VOL. If PID = 0, then the whole volume mesh will be used.
LCVX, LCVY, LCVZ	LCID for the velocity/displacements in the three global directions (x, y, z).
VADT	Velocity/Displacements flag for translation components EQ.0: Prescribe Velocity EQ.1: Prescribe Displacements
ALPHAL, BETAL, GAMMAL	LCID for the three Euler angle rotational velocities/displacements in the local reference frame (See Remark 2).
ALPHAG, BETAG, GAMMAG	LCID for the three Euler angle rotational velocities/displacements in the global reference frame (See Remark 2).
VADR	Velocity/Displacements flag for rotation components EQ.0: Prescribe Velocity EQ.1: Prescribe Displacements
PTID	Point ID for the origin of the local reference frame. If not defined, the barycenter of the volume mesh will be used.
X1, Y1, Z1	Three components of the local reference X1 axis. If not defined, the global x axis will be used.
X2, Y2, Z2	Three components of the local reference X2 axis. If not defined, the global y axis will be used.

Remarks:

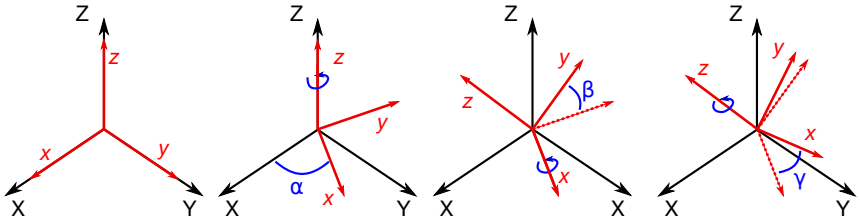


Figure 5-1. A rotation represented by Euler angles (α, β, γ) using $\mathbf{Z}(\alpha)\mathbf{X}(\beta)\mathbf{Z}(\gamma)$ intrinsic rotations.

1. **Rotations.** Any target orientation can be reached starting from a known reference orientation using a specific sequence of intrinsic rotations whose magnitudes are the Euler angles (α, β, γ). Equivalently, any rotation matrix R can be decomposed as a product of three elemental rotation matrices. For instance:

$$\mathbf{R} = \mathbf{X}(\alpha)\mathbf{Y}(\beta)\mathbf{Z}(\gamma)$$

However, different definition of the elemental rotation matrices (x, y, z) and their multiplication order can be adopted. The ICFD solver uses the following approach and rotation matrix:

$$\mathbf{Z}(\alpha)\mathbf{X}(\beta)\mathbf{Z}(\gamma) = \begin{bmatrix} c_\alpha c_\gamma - c_\beta s_\alpha s_\gamma & -c_\beta c_\gamma s_\alpha - c_\alpha s_\gamma & s_\alpha s_\beta \\ c_\gamma s_\alpha + c_\alpha c_\beta s_\gamma & c_\alpha c_\beta c_\gamma - s_\alpha s_\gamma & -c_\alpha s_\beta \\ s_\beta s_\gamma & c_\gamma s_\beta & c_\beta \end{bmatrix}$$

where $\boldsymbol{X}(\alpha)$, $\boldsymbol{Y}(\beta)$, and $\boldsymbol{Z}(\gamma)$ *are* the matrices representing the elemental rotations about the axes (x, y, z), $s_\alpha = \sin(\alpha)$, and $c_\beta = \cos(\beta)$.

2. **Local Coordinate Systems.** It is possible to have the ICFD parts or ICFD_PART_VOLs rotate around the global reference frame but also to define and use a local reference frame by defining its point of origin and two of its vectors $\mathbf{v}_1 = (\text{X1}, \text{Y1}, \text{Z1})$ and $\mathbf{v}_2 = (\text{X2}, \text{Y2}, \text{Z2})$. The third vector is, then, in the direction of $\mathbf{v}_1 \times \mathbf{v}_2$. See Figure 5-1.

*ICFD_CONTROL_LOAD

Purpose: This keyword resets the body load in the ICFD solver to zero, while leaving the body load unchanged for the solid mechanics solver. It is useful in problems where the gravity acceleration may be neglected for the fluid problem, but not for the solid mechanics problem.

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.

Card 1	1	2	3	4	5	6	7	8
Variable	MGSF		MSTRAT	2DSTRUC	NRMSH			
Type	F		I	I	I			
Default	1.41		0	0	0			

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.
MSTRAT	Mesh generation strategy : EQ.0: Mesh generation based on Delaunay criteria EQ.1: Mesh generation based on octree (See Remark 2)
2DSTRUC	Flag to decide between a unstructured mesh generation strategy in 2D or a structured mesh strategy : EQ.0: Structured mesh EQ.1: Unstructured mesh
NRMSH	Flag to turn off any remeshing : EQ.0: Remeshing possible EQ.1: Remeshing impossible

Remarks:

4.For MGSF, 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). MGSF has a fixed value of 1 in 2D.

5. If the user knows in advance that no remeshing will occur during the analysis, then setting NRMSH to 1may be useful as it will free space used to back up the mesh and consequently lower memory consumption.

6. The Default Mesh generation strategy (based on Delaunay criteria) yields a linear interpolation of the mesh size between two surfaces facing each other whereas the octree based generation strategy allows for elements' sizes to remain close to the element surface mesh size over a longer distance. This can be useful in configurations where two surface meshes facing each other have very distinct sizes in order to create a smoother transition.

*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					
Type	I	I	F					
Default	2	100	1.0e-3					

VARIABLE	DESCRIPTION
MMSH	Mesh motion selector: EQ.-1: completely shuts off any mesh movement 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.(default) 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).

*ICFD_CONTROL_MONOLITHIC

Purpose: This keyword allows to choose between the Fractional Step Solver and the Monolithic Solver.

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

VARIABLE	DESCRIPTION
SID	Solver ID : EQ.0: Fractional Step Solver. Default. EQ.1: Monolithic 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	LSPPOUT		ITOUT		
Type	I	I	F	I		I		
Default	0	0	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. EQ.4: full output information is also copied to the messag file.

VARIABLE	DESCRIPTION
OUTL	Output the fluid results in other file formats apart from d3plot. EQ.0: only d3plot output EQ.2: output a file with mesh statistics and the fluid results in OpenDX format. A directory named dx will be created in the work directory where the output files will be written. EQ.6: output a file with mesh statistics and the fluid results in VTK format readable by Paraview. A directory named vtk will be created in the work directory where the output files will be written. EQ.7: output a file with mesh statistic and the fluid results in VTU format readable by Paraview. A directory named vtk will be created in the work directory where the output files will be written. EQ.10: output a file with mesh statistic and the fluid results in Fieldview ASCII format. . A directory named fv will be created in the work directory where the output files will be written. Only available in 3D. EQ.11: output a file with mesh statistic and the fluid results in Fieldview binary format. . A directory named fv will be created in the work directory where the output files will be written. Only available in 3D.
DTOUT	Time interval to print the output when OUTL is different than 0.
LSPPOUT	EQ.1: outputs a file with the automatically created fluid volume mesh in a format compatible for LSPP.
ITOUT	Iteration interval to print the output, including the d3plot files when the steady state solver is selected (See ICFD_CONTROL_GENERAL).

.

*ICFD_CONTROL_OUTPUT_SUBDOM

Purpose: Defines a specific zone that should be output in the format specified by the ICFD_CONTROL_OUTPUT card rather than the whole domain.

Remeshing Control. First card specifies the shape of the output sub domain.

Card 1	1	2	3	4	5	6	7	8
Variable	SNAME							
Type	A							
Default	none							

Box Case. Card 2 for Sname = box

Cards 2	1	2	3	4	5	6	7	8
Variable	PMINX	PMINY	PMINZ	PMAXX	PMAXY	PMAXZ		
Type	F	F	F	F	F	F		
Default	none	none	none	none	none	none		

Sphere Case. Card 2 for Sname = sphere

Cards 3	1	2	3	4	5	6	7	8
Variable	RADIUS	CENTERX	CENTERY	CENTERZ				
Type	F	F	F	F				
Default	none	none	none	none				

Cylinder Case. Card 2 for Sname = cylinder

Cards 4	1	2	3	4	5	6	7	8
Variable	Radius	PMINX	PMINY	PMAXZ	PMAXX	PMAXY	PMAXZ	
Type	F	F	F	F	F	F	F	
Default	none	none	none	none	none	none	none	

VARIABLE	DESCRIPTION
SNAME	Shape name. Possibilities include 'box', 'cylinder' and 'sphere'
PMINX, 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*.

*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_POROUS

Purpose: This keyword modifies the porous media solve.

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

VARIABLE	DESCRIPTION
PMSTYPE	Indicates the porous media solve type. EQ.0: Anisotropic Generalized Navier-Stokes model for porous media (See *ICFD_MODEL_POROUS) using Fractional step method. EQ.1: Anisotropic Darcy-Forcheimer model using a Monolithic approach for the solve. This method is better suited for very low Reynolds flows through porous media (Frequently encountered in Resin Transfer Molding (RTM) applications).

Remarks:

1. When using the Anisotropic Darcy-Forcheimer model, the convective term in the Navier Stokes formulation is neglected.

*ICFD_CONTROL_STEADY

Purpose: This keyword allows to specify convergence options for the steady state solver.

Card 1	1	2	3	4	5	6	7	8
Variable	ITS	TOL1	TOL2	TOL3	REL1	REL2	UREL	ORDER
Type	I	F	F	F	F	F	F	I
Default	1e6	1.e-3	1.e-3	1.e-3	0.3	0.7	1.	0

VARIABLE	DESCRIPTION
ITS	Maximum number of iterations to reach convergence.
TOL1/2/3	Tolerance limits for the momentum pressure and temperature equations respectfully.
REL1/2	Relaxation parameters for the velocity and pressure respectfully. Decreasing those values may add stability but more iterations may be needed to reach convergence.
UREL	Under relaxation parameter. Lowering this value may improve the final accuracy of the solution but more iterations may be needed to achieve convergence.
ORDER	Analysis order : EQ.0: Second order. More accurate but more time consuming. EQ.1: First order: More stable and faster but may be less accurate.

*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	SADAPT						
Type	I	I						
Default	0	0						

VARIABLE	DESCRIPTION
RSRF	Indicates whether or not to perform a surface re-meshing. EQ.0: no re-meshing is applied. EQ.1: Laplacian smoothing surface remeshing EQ.2: Curvature preserving surface remeshing
SADAPT	Indicates whether or not to trigger adaptive surface remeshing. EQ.0: no adaptive surface re-meshing is applied. EQ.1: automatic surface remeshing when quality deteriorates (3D only).

.

*ICFD_CONTROL_TAVERAGE

Purpose: This keyword controls the restarting time for computing the time average values. By default, there is no restarting and the average quantities are given starting from $t = 0$. This keyword can be useful in turbulent problems that admit a steady state.

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

VARIABLE	DESCRIPTION
DT	Over each DT time interval, the average quantities are reset.

*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	LCIDSF	DTMIN	DTMAX	DTINIT	TDEATH
Type	F	F	F	I	F	F	F	F
Default	1E28	0	1	none	none	none	none	1E28

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	CFL number for DT = 0. In general, CFL specifies a scale factor that is applied to the time step. When DT = 0, the time step is set to the maximum value satisfying the CFL condition, in which case this scale factor is equal to the *CFL number*.
LCIDSF	Load Curve ID specifying the CFL number when DT = 0 as a function of time, and more generally LCIDSF specifies the time step scale factor as the function of time.
DTMIN	Minimum time step. When an automatic time step is used and DTMIN is defined, the time step cannot drop below DTMIN.
DTMAX	Maximum time step. When an automatic time step is used and DTMAX is defined, the time step cannot increase beyond DTMAX.
DTINIT	Initial time step. If not defined, the solver will automatically determine an initial timestep based on the flow velocity or dimensions of the problem in cases where there is no inflow.

VARIABLE	DESCRIPTION
TDEATH	Death time for the Navier Stokes solve. After TDEATH, the velocity and pressure will no longer be updated. But the temperature and other similar quantities still can.

*ICFD_CONTROL_TRANSIENT

Purpose: This keyword allows to specify different integration scheme options for the transient solver.

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

VARIABLE	DESCRIPTION
TORD	Time integration order : EQ.0: Second order. EQ.1: First order.
FSORD	Fractional step integration order : EQ.0: Second order. EQ.1: First order.

*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	SUBMOD	WLAW	KS	CS		LCIDS1	LCID2
Type	I	I	I	F	F		I	I
Default	0	1	1	0.	0.		none	none

Optional card if TMOD = 1.

Card 2	1	2	3	4	5	6	7	8
Variable	Ce1	Ce2	σ_e	σ_k	C_μ	C_{cut}		
Type	F	F	F	F	F	F		
Default	1.44	1.92	1.3	1.0	0.09	-1.		

Optional card TMOD = 2 or TMOD = 3.

Card 2	1	2	3	4	5	6	7	8
Variable	Cs							
Type	F							
Default	0.18							

Optional card if TMOD = 4.

Card 2	1	2	3	4	5	6	7	8
Variable	γ	β_{01}	$\sigma_{\omega 1}$	σ_{k1}	β_0^*	C_{cut}		
Type	F	F	F	F	F	F		
Default	1.44	0.072	2	2	0.09	-1.		

Optional card if TMOD = 4.

Card 3	1	2	3	4	5	6	7	8
Variable	$a1$	β_{02}	$\sigma_{\omega 2}$	σ_{k2}	C_l			
Type	F	F	F	F	F			
Default	0.31	0.0828	2	2	0.875			

Optional card if TMOD = 5.

Card 2	1	2	3	4	5	6	7	8
Variable	C_{b1}	C_{b1}	σ_ν	C_{v1}	C_{w1}	C_{w2}		
Type	F	F	F	F	F	F		
Default	0.1355	0.622	0.66	7.2	0.3	2.0		

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. EQ.3: LES Wall adapting local eddy-viscosity (WALE) model. EQ.4: RANS k - w approach. EQ.5: RANS Spalart Allmaras approach.
SUBMOD	Turbulence sub-model. If TMOD = 1 : EQ.1: Standard model EQ.2: Realizable model If TMOD = 4 : EQ.1: Standard Wilcox 98 model EQ.2: Standard Wilcox 06 model EQ.3: SST Menter 2003
WLAW	Law of the wall ID is RANS turbulence model selected : EQ.1: Standard classic law of the wall. EQ.2: Standard Launder and Spalding law of the wall. EQ.4: Non equilibrium Launder and Spalding law of the wall. EQ.5: Automatic classic law of the wall.
KS/CS	Roughness physical height and Roughness constant. Only used if RANS turbulence model selected.
LCIDS1	Load curve describing user defined source term in turbulent kinetic energy equation function of time. See *DEFINE_CURVE, *DEFINE_CURVE_FUNCTION, or *DEFINE_FUNCTION. If a DEFINE_FUNCTION is used, the following parameters are allowed: $f(x,y,z,vx,vy,vz,temp,pres,time,k,e,mut)$.

VARIABLE	DESCRIPTION
LCIDS2	Load curve describing user defined source term in turbulent dissipation equation function of time. See *DEFINE_CURVE, *DEFINE_CURVE_FUNCTION, or *DEFINE_FUNCTION. If a DEFINE_FUNCTION is used, the following parameters are allowed: $f(x, y, z, vx, vy, vz, temp, pres, time, k, e, mut)$.
Ce1, Ce2, σ_e, σ_k, C_μ, C_{cut}	k - ε model constants
Cs	Smagorinsky constant if TMOD = 2 or WALE constant if TMOD = 3
$\gamma, \beta_{01}, \sigma_{\omega 1}, \sigma_{k1}, \beta_0^*, a1, \beta_{02}, \sigma_{\omega 2}, \sigma_{k2}, C_l, C_{cu}$	k -ω model constants
$C_{b1}, C_{b2}, \sigma_\nu, C_{v1}, C_{w1} C_{w2}$	Spalart-Allmaras constants

Remarks:

1. For the Standard k - ε model, the following two equations are solved for the turbulent kinetic energy and the turbulent dissipation respectively k and ε :

$$\frac{\partial k}{\partial t} + \frac{\partial (k u_i)}{\partial x_i} = \frac{\partial}{\partial x_j}\left[(\frac{\mu}{\rho} + \frac{\mu_t}{\rho\, \sigma_k}) \frac{\partial k}{\partial x_j} \right] + P_k + P_b - \epsilon + S_k$$

$$\frac{\partial \epsilon}{\partial t} + \frac{\partial (\epsilon u_i)}{\partial x_i} = \frac{\partial}{\partial x_j}\left[(\frac{\mu}{\rho} + \frac{\mu_t}{\rho\, \sigma_\epsilon}) \frac{\partial \epsilon}{\partial x_j} \right] + C_{1\epsilon} \frac{\varepsilon}{k} P_k - C_{2\varepsilon} \frac{\epsilon^2}{k} + S_e$$

With P_k the k production term, P_b the production term due to buoyancy and S_k, S_e are the user defined source terms. P_k and P_b are expressed as :

$$P_k = \frac{\mu_t}{\rho} S^2$$

$$P_b = \frac{\beta \mu_t}{\rho Pr_t} g_i \frac{\partial T}{\partial x_i}$$

With S the modulus of the mean rate of strain tensor ($S^2 = 2 S_{ij} S_{ij}$), β the coefficient of thermal expansion, and Pr_t the turbulent Prandtl number. The turbulent viscosity is then expressed as:

$$\mu_t = \rho C_\mu \frac{k^2}{\varepsilon}$$

For the realizable k - ε model, the equation for the turbulent kinetic energy does not change, but the equation for the turbulent dissipation is now expressed as:

$$\frac{\partial \epsilon}{\partial t} + \frac{\partial (\epsilon u_i)}{\partial x_i} = \frac{\partial}{\partial x_j}\left[(\frac{\mu}{\rho} + \frac{\mu_t}{\rho\, \sigma_\epsilon}) \frac{\partial \epsilon}{\partial x_j} \right] + C_1 S \epsilon - C_{2\varepsilon} \frac{\epsilon^2}{k + \sqrt{\frac{\mu}{\rho}\epsilon}} - \epsilon + S_e$$

With $C_1 = max\left[0.43, \frac{\eta}{\eta+5}\right]$, $\eta = S\frac{k}{\epsilon}$.

Furthermore, while the turbulent viscosity is still expressed the same way, C_μ is no longer a constant:

$$C_\mu = \frac{1}{A_0 + A_s k \frac{U^*}{\epsilon}}$$

$$U^* = \sqrt{\Omega_{ij}\Omega_{ij} + S_{ij}S_{ij}}$$

$$A_0 = 4.04$$

$$A_s = \sqrt{6} cos\left(\frac{1}{3} cos^{-1}\left(\sqrt{6} \frac{S_{ij}S_{jk}S_{ki}}{(S_{ij}S_{ij})^{3/2}} \right) \right)$$

It can be noted that in this case, the constant value C_μ that can be input by the user serves as the limit values that C_μ can take. By default $C_\mu = 0.09$ so:

$$0.0009 < C_\mu < 0.09$$

2. For the Standard Wilcox 06 k -ω model, the following two equations are solved for the turbulent kinetic energy and the specific turbulent dissipation rate respectively k and ω :

$$\frac{\partial k}{\partial t} + \frac{\partial (k u_i)}{\partial x_i} = \frac{\partial}{\partial x_j}\left[(\frac{\mu}{\rho} + \frac{\mu_t}{\rho\, \sigma_{k1}}) \frac{\partial k}{\partial x_j} \right] + P_k - \beta^* k\omega + S_k$$

$$\frac{\partial w}{\partial t} + \frac{\partial (w u_i)}{\partial x_i} = \frac{\partial}{\partial x_j}\left[(\frac{\mu}{\rho} + \frac{\mu_t}{\rho\, \sigma_{w1}}) \frac{\partial \epsilon}{\partial x_j} \right] + \gamma \frac{\omega}{k} P_k - \beta\omega^2 + \sigma_d X_k \omega^2 + S_\omega$$

With P_k the k production term and S_k, S_ω are the user defined source terms. P_k, β^* and β are expressed as:

$$P_k = \frac{\mu_t}{\rho} S^2$$

$$\beta^* = \beta_0^* f_{\beta*} \quad \beta = \beta_{01} f_\beta$$

$$f_\beta = \frac{1 + 85X_\omega}{1 + 100X_\omega} \quad f_{\beta*} = 1. \quad \sigma_d = \begin{cases} 0. & X_k \le 0. \\ 1/8 & X_k > 0. \end{cases}$$

$$X_k = \frac{1}{\omega^3} \frac{\partial k}{\partial x_j} \frac{\partial \omega}{\partial x_j} \quad X_\omega = \left| \frac{\Omega_{ij}\Omega_{jk}S_{ki}}{(\beta_0^*\omega)^3} \right|$$

The turbulent viscosity is then:

$$\mu_t = \rho \frac{k}{max\left[\omega, C_l \sqrt{\frac{2S_{ij}S_{ij}}{\beta_0^*}}\right]}$$

For the Standard Wilcox 98 model, the following terms are modified:

$$f_\beta = \frac{1 + 70X_\omega}{1 + 80X_\omega} \quad f_{\beta*} = \begin{cases} 1 & X_k \le 0. \\ \frac{1 + 680\ {X_k}^2}{1 + 400\ {X_k}^2} & X_k > 0. \end{cases} \quad \sigma_d = 0.$$

The turbulent viscosity is then:

$$\mu_t = \rho \frac{k}{\omega}$$

For the Menter SST 2003 model, the following equations are solved:

$$\frac{\partial k}{\partial t} + \frac{\partial (k u_i)}{\partial x_i} = \frac{\partial}{\partial x_j}\left[\left(\frac{\mu}{\rho} + \frac{\mu_t}{\rho\ \sigma_k}\right) \frac{\partial k}{\partial x_j} \right] + P_k - \beta_0^* k\omega + S_k$$

$$\frac{\partial w}{\partial t} + \frac{\partial (w u_i)}{\partial x_i} = \frac{\partial}{\partial x_j}\left[\left(\frac{\mu}{\rho} + \frac{\mu_t}{\rho\ \sigma_w}\right) \frac{\partial \epsilon}{\partial x_j} \right] + \frac{\gamma}{\mu_t} P_k - \beta\omega^2 + 2(1 - F_1)\ \sigma_{w2} X_k \omega^2 + S_\omega$$

Each of the constants, $\gamma, \beta, \sigma_k, \sigma_w$ are now computed by a blend via:

$$\alpha = \alpha_1 F_1 + \alpha_2 (1 - F_1)$$

Where the blending function F_1 is defined by:

$$F_1 = \tanh\left\langle\left[min\left(max\left(\frac{\sqrt{k}}{\beta_0^*\omega y}, \frac{500\nu}{y^2\omega}\right), \frac{4\rho\sigma_{w2}k}{CD\,y^2}\right)\right]^4\right\rangle$$

With y the distance to the nearest wall and:

$$CD = max(2\rho\sigma_{\omega 2}X_k\omega^2, 10^{-10})$$

The turbulent viscosity is then:

$$\mu_t = \rho\frac{a_1 k}{max(a_1\omega, S\,F_2)}$$

With:

$$F_2 = tanh\left[\left(max\left(\frac{2\sqrt{k}}{\beta_0^*\omega y}, \frac{500\nu}{y^2\omega}\right)\right)^2\right]$$

3. It is possible to activate a limiter on the production term P_k. If $C_{cut} \geq 0.$, then :

   $P_k = min(P_k, C_{cut}\varepsilon)$ if TMOD = 1, $P_k = min(P_k, C_{cut}\beta_0^* k\omega)$ if TMOD = 4. This is especially common when using the Menter SST 2003 model.

4. For RANS models, the following laws of the wall are available :

   a) STANDARD CLASSIC :

$$U^+ = \frac{1}{\kappa}\ln(E\,Y^+)$$

If $Y^+ > 11.225$, $U^+ = Y^+$ otherwise

$$Y^+ = \frac{\rho y U_\tau}{\mu}$$

$$U^+ = \frac{U}{U_\tau}$$

$$U_\tau = \sqrt{\frac{\tau_w}{\rho}}$$

   This is the default for TMOD = 1

   b) STANDARD LAUNDER and SPALDING :

$$U^* = \frac{1}{\kappa}\ln(E\,Y^*)$$

If $Y^* > 11.225$, $U^* = Y^*$ otherwise

$$Y^* = \frac{\rho {C_\mu}^{1/4} k^{1/2} y}{\mu}$$

$$U^* = \frac{U {C_\mu}^{1/4} k^{1/2}}{{U_\tau}^2}$$

$$U_\tau = \sqrt{\frac{\tau_w}{\rho}}$$

c) The NON EQUILIBRUM laws of the wall modify the expression of the velocity at the wall making it sensitive to the pressure gradient :

$$U = U - \frac{1}{2}\frac{dP}{dx}\left[\frac{y_v}{\rho\kappa\sqrt{k}} ln\left(\frac{y}{y_v}\right) + \frac{y - y_v}{\rho\kappa\sqrt{k}} + \frac{{y_v}^2}{\mu}\right]$$

With:

$$y_v = \frac{11.225}{y^*} y$$

This law is recommended with TMOD = 1 and in cases of complex flows involving separation, reattachment and recirculation.

d) The automatic wall law attempts to blend the viscous and log layers to better account for the transition zone. In the buffer region, we have :

$$U^+ = \frac{U}{U_\tau}$$

$$U_\tau = \sqrt[4]{(\frac{U}{y^+})^4 + (\frac{U}{\frac{1}{\kappa}\ln(Ey^+)})^4}$$

This is the recommended approach for TMOD = 4.

5. The LES Smagorinsky turbulence model uses the Van Driest damping function close to the wall :

$$f_v = 1 - e^{-\frac{y^+}{A^+}}$$

6. When a RANS turbulence model is selected, it is possible to define extra parameters to account for the rugosity effects. In such cases, an extra term will be added to the logarithmic part of the different laws of the wall :

$$U^+ = \frac{1}{\kappa}\ln(E\ Y^+) - \Delta B$$

If we introduce the non-dimensional roughness height $K^+ = \frac{\rho K_s {C_\mu}^{1/4} k^{1/2}}{\mu}$, we have:

$$\Delta B = 0 \, for \ K^+ \leq 2.25$$

$$\Delta B = \frac{1}{\kappa} ln\left[\frac{K^+ - 2.25}{87.75} + C_s K^+\right] \times sin(0.4258(\ln K^+ - 0.811)) \, for \ 2.25 < K^+ \leq 90.0$$

$$\Delta B = \frac{1}{\kappa} ln(1 + C_s K^+) \, for \ 90. < K^+$$

*ICFD_CONTROL_TURB_SYNTHESIS

Purpose: This keyword enables the user impose a divergence-free turbulent field on inlets.

Card must be used jointly with VAD = 4 of keyword *ICFD_BOUNDARY_PRESCRIBED_VEL.

Card 1	1	2	3	4	5	6	7	8
Variable	PID	IU	IV	IW	LS			
Type	I	F	F	F	F			
Default	0	10^{-3}	10^{-3}	10^{-3}	h_{min}			

VARIABLE	DESCRIPTION
PID	Part ID of the surface with the turbulent velocity inlet condition.
IU, IV, IW	Intensity of field fluctuations over x, y, and z directions, $\text{IU} = \frac{u'}{u_{\text{avg}}}$.
LS	Integral length scale of turbulence

Remarks:

1. If this card is not defined but a turbulent field inlet has been activated. See VAD = 4 of *ICFD_BOUNDARY_PRESCRIBED_VEL, the default parameters will be used.

*ICFD_DATABASE_AVERAGE

Purpose: This keyword enables the computation of time average variables 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_DATABASE_DRAG_{OPTION}

Available options include

VOL

Purpose: This keyword enables the computation of drag forces over given surface 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.

For the VOL option, drag calculation can also be applied on a volume defined by ICFD_PART_VOL. This is mostly useful in porous media applications to output the pressure drag of the porous media domain.

Surface Drag Cards. Include one card for each surface on which drag is applied. This input ends at the next keyword ("*") card.

Card 1	1	2	3	4	5	6	7	8
Variable	PID	CPID	DTOUT	PEROUT	DIVI	ELOUT	SSOUT	
Type	I	I	F	I	I	I	I	
Default	none	none	0.	0	10	0	0	

VARIABLE	DESCRIPTION
PID	Part ID of the surface where the drag force will be computed.
CPID	Center point ID used for the calculation of the force's moment. By default the reference frame center is used is $\mathbf{0} = (0, 0, 0)$.
DTOUT	Time interval to print the output. If DTOUT is equal to 0.0, then the ICFD timestep will be used.
PEROUT	Outputs the contribution of the different elements on the total drag in fractions of the total drag in the d3plots.
DIVI	Number of drag divisions for PEROUT. Default is 10 which means the contributions will be grouped in 10 deciles.
ELOUT	Outputs the drag value of each element in the d3plots.
SSOUT	Outputs the pressure loads caused by the fluid on each solid segment set in keyword format. FSI needs to be activated.

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.

2. The output contains:

   a) "Fpx" , "Fpy", and "Fpz" refer to the three components of the pressure drag force

   $$\mathbf{F}_p = \int P d\boldsymbol{A},$$

   where P is the pressure and A the surface area.

   b) "Fvx", "Fxy", and "Fvz" refer to the three components of the viscous drag force

   $$\mathbf{F}_v = \int \mu \frac{\partial \mathrm{u}}{\partial \mathrm{y}} \mathrm{d}\boldsymbol{A}.$$

   where $\frac{\partial \mathrm{u}}{\partial \mathrm{y}}$ is the shear velocity at the wall, μ is the viscosity and A is the surface area.

   c) "Mpx", "Mpy", "Mpz", "Mvx", "Mvy", and "Mvz" refer to the three components of the pressure and viscous force moments respectively.

*ICFD_DATABASE_FLUX

Purpose: This keyword enables the computation of the flow rate and average pressure over given parts of the model. If multiple keywords are given, separate files are output.

Include as many cards as needed. This input ends at the next keyword ("*") card.

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

VARIABLE	DESCRIPTION
PID	Part ID of the surface where the flow rates will be computed.
DTOUT	Output frequency. Default is at every fluid timestep.

Remarks:

1. The file name for this database is icfd_flux.dat.

2. The flux database contains the flow rate through a section, called "output flux",

$$\Phi = \sum_i (\mathbf{V}_i \cdot \mathbf{n}_i) A_i,$$

the average pressure, called "Pre-avg",

$$P_{\text{avg}} = \frac{\sum_i P_i A_i}{\sum_i A_i},$$

and the total area, called "Areatot".

*ICFD_DATABASE_HTC

Purpose: This keyword allows the user to trigger the calculation of the Heat transfer coefficient using different methods and to control the output options.

Card 1	1	2	3	4	5	6	7	8
Variable	OUT	HTC	TB					OUTDT
Type	I	I	F					F
Default	0	0.	0.					0.

VARIABLE	DESCRIPTION
OUT	Determines if the solver should calculate the heat transfer coefficient and how to output it : EQ.0: No HTC calculation EQ.1: HTC calculated and output in LSPP as a surface variable. EQ.2: The solver will also look for FSI boundaries and output the HTC value at the solid nodes in an ASCII file called icfdhtci.dat. EQ.3: The solver will also look for FSI boundaries that are part of SEGMENT_SETS and output the HTC for those segments in an ASCII file called icfd_convseg.****.key in a format that can directly read by LS-DYNA for a subsequent pure structural thermal analysis.
HTC	Determines how the HTC is calculated. EQ.0: Automatically calculated by the solver based on the average temperature flowing through the pipe section (See Remark 1). EQ.1: User imposed value (See Remark 2).
TB	Value of the bulk temperature if HTC = 1.
OUTDT	Output frequency of the HTC in the various ASCII files. If left to 0., the solver will output the HTC at every timestep.

Remarks:

1. The heat transfer coefficient is frequently used in thermal applications to estimate the effect of the fluid cooling and it derived from a CFD calculation.

2. The heat transfer coefficient is defined as follows:

$$h = \frac{q}{T_s - T_b}$$

with q the heat flux, T_s the surface temperature and T_b the so called "bulk" temperature. For external aerodynamic applications, this bulk temperature is often defined as a constant (ambient or far field conditions, HTC = 1). However, for internal aerodynamic application, this temperature is often defined as an average temperature flowing through the pipe section with the flow velocity being used as a weighting factor (HTC = 0).

*ICFD_DATABASE_NODEAVG

Purpose: This keyword enables the computation of the average quantities on surface nodes defined in *ICFD_DATABASE_NODOUT.

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

VARIABLE	DESCRIPTION
ON	If equal to 1, the average quantities will be computed.

Remarks:

1. The file name for this database is icfd_nodeavg.dat.

*ICFD_DATABASE_NODOUT

Purpose: This keyword enables the output of ICFD data on surface nodes. For data in the fluid volume, it is advised to use points or tracers (See *ICFD_DATABASE_POINTOUT).

Output Options Card.

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

Include as many cards as needed. This input ends at the next keyword ("*") card.

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
OUTLV	Determines if the output file should be dumped. EQ.0: No output file is generated. EQ.1: The output file is generated.
DTOUT	Time interval to print the output. If DTOUT is equal to 0.0, then the ICFD timestep will be used.
NID..	Node IDs.

Remarks:

1. The file name for this database is icfd_nodout.dat.

*ICFD_DATABASE_POINTAVG

Purpose: This keyword enables the computation of the average quantities on point sets using the parameters defined in *ICFD_DATABASE_POINTOUT.

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

VARIABLE	DESCRIPTION
ON	If equal to 1, the average quantities will be computed.

Remarks:

1. The file name for this database is icfd_psavg.dat.

*ICFD_DATABASE_POINTOUT

Purpose: This keyword enables the output of ICFD data on points.

Output Options Card.

Card 1	1	2	3	4	5	6	7	8
Variable	PSID	DTOUT	PSTYPE	VX	VY	VZ		
Type	I	F	I	F	F	F		
Default	0	0.	0	0.	0.	0.		

Include as many cards as needed. This input ends at the next keyword ("*") card.

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

VARIABLE	DESCRIPTION
PSID	Point Set ID.
DTOUT	Time interval to print the output. If DTOUT is equal to 0.0, then the ICFD timestep will be used.
PSTYPE	Point Set type : EQ.0: Fixed points. EQ.1: Tracer points using prescribed velocity. EQ.2: Tracer points using fluid velocity. EQ.3: Tracer points using mesh velocity..
VX, VY, VZ	Constant velocities to be used when PSTYPE = 1
PID	Point ID

VARIABLE	DESCRIPTION
X, Y, Z	Point initial coordinates

Remarks:

1. The file name for this database is icfd_pointout.dat.

*ICFD_DATABASE_RESIDUALS

Purpose: This keyword allows the user to output the residuals of the various systems.

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

VARIABLE	DESCRIPTION
RLVL	Residual output level :

EQ.0: No output.

EQ.1: Only outputs the number of iterations needed for solving the pressure Poisson equation.

EQ.2: Outputs the number of iterations for the momentum, pressure, mesh movement and temperature equations.

EQ.3: Also gives the residual for each iteration during the solve of the momentum, pressure, mesh movement and temperature equations.

Remarks:

1. The file names for the momentum, pressure, mesh movement and temperature equations are called icfd_residuals.moms.dat, icfd_residuals.pres.dat, icfd_residuals.mmov.dat, and icfd_residuals.temp.dat respectively.

*ICFD_DATABASE_TEMP

Purpose: This keyword enables the computation of the average temperature and the heat flux over given parts of the model. If multiple keywords are given, separate files are output.

Include as many cards as needed. This input ends at the next keyword ("*") card.

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

VARIABLE	DESCRIPTION
PID	Part ID of the surface where the average temperature and heat flux will be computed.
DTOUT	Output frequency. Default is at every fluid timestep.

Remarks:

1. The file name for this database is icfd_thermal.dat.

2. Two average temperature are given in the icfd_thermal.dat file: "Temp-avg" and "Temp-sum". The average temperature is calculated using the local node area as weighting factor,

$$T_{\text{avg}} = \frac{\sum_i^N T_i A_i}{\sum_i^N A_i},$$

whereas, the sum is not weighted by area

$$T_{\text{sum}} = \frac{\sum_i^N T_i}{N}$$

If the mesh is regular, the two values will be of similar value. The icfd_thermal.dat output file also includes the average heat flux, the total surface area, and the average heat transfer coefficients (See *ICFD_DATABASE_HTC).

*ICFD_DATABASE_TIMESTEP

Purpose: This keyword enables the output of ICFD data regarding the ICFD timestep.

Output Options Card.

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

VARIABLE	DESCRIPTION
OUTLV	Determines if the output file should be dumped. EQ.0: No output file is generated. EQ.1: The output file is generated.

Remarks:

1. The file name for this database is icfd_tsout.dat.

2. Outputs the run's ICFD timestep versus the timestep calculated using the ICFD CFL condition as criteria (autotimestep). This can be useful in cases using a fixed timestep where big mesh deformations and/or big fluid velocity changes occur in order to track how that fixed timestep value compares to the reference autotimestep.

*ICFD_DATABASE_UINDEX

Purpose: This keyword allows the user to have the solver calculate the uniformity index (See Remark 1).

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

VARIABLE	DESCRIPTION
OUT	Determines if the solver should calculate the uniformity index. EQ.0: Off. EQ.1: On.

Remarks:

1. **Uniformity Index.** The uniformity index is a post treatment quantity which measures how uniform the flow is through a given section. It is especially useful in internal aerodynamics cases. It is expressed as :

$$\gamma = 1 - \frac{1}{2nA}\sum_{i=1}^{n}\left[\frac{\sqrt{(u_i - \bar{u})^2}}{\bar{u}}A_i\right]$$

with A_i, the local cell area, A the total section area, u_i the local velocity, $\bar{u}$ the average velocity through the section, and n the number of cells.

Values close to 0 means that the flow is very unevenly distributed. This can be used to identify bends, corners or turbulent effects. Values close to 1 imply smooth or equally distributed flow through the surface.

*ICFD_DEFINE_HEATSOURCE

Purpose: This keyword defines a volumetric heat source for the heat equation solve.

Card 1	1	2	3	4	5	6	7	8
Variable	HSID	LCID	SHAPE	R	PTID1	PTID2		
Type	I	I	I	F	I	I		
Default	none	none	none	none	none	none		

VARIABLE	DESCRIPTION
HSID	Heat source ID.
LCID	Load curve ID specifying the evolution of the heat source function of time.
SHAPE	Shape of the volumetric heat source: EQ.1 : Box shape EQ.2 : Cylinder shape EQ.3 : Sphere shape
R	Radius of the sphere is SHAPE = 3
PTID1	ID of point (See ICFD_DEFINE_POINT) of minimum coordinates if SHAPE = 1, tail point if SHAPE = 2, origin if SHAPE = 3.
PTID2	ID of point of maximum coordinates if SHAPE = 2, head point if SHAPE = 2.

.

*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	CONSTPID			
Type	I	F	F	F	I			
Default	none	none	none	none	none			

Optional Card 2. Load curve IDS specifying velocity components of translating point

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

Optional Card 3. Load curve IDS and rotation axis of rotating point

Card 2	1	2	3	4	5	6	7	8
Variable	LCIDW	XT	YT	ZT	XH	YH	ZH	
Type	I	F	F	F	F	F	F	
Default	0	none	none	none	none	none	none	

VARIABLE	DESCRIPTION
POID	Point ID.
X/Y/Z	x, y ,z coordinates for the point.

VARIABLE	DESCRIPTION
CONSTPID	Surface Part ID to which the point is constrained. This means that if the selected surface moves, then the localization of the point will update as well.
LCIDX/LCIDY/LCIDZ	The point can be made to translate. Those are the three load curve IDs for the three translation components.
LCIDW	The point can also be made to rotate. This load curve specifies the angular velocity.
XT/YT/ZT	Rotation axis tail point coordinates.
XH/YH/ZH	Rotation axis head point coordinates.

.

*ICFD_DEFINE_NONINERTIAL

Purpose: This keyword defines a non-inertial reference frame in order to avoid heavy mesh distortions and attendant remeshing associated with large-scale rotations. This is used to model, for example, spinning cylinders, wind turbines, and turbo machinery.

Include as many cards as needed. This input ends at the next keyword ("*") card.

Card 1	1	2	3	4	5	6	7	8
Variable	W1	W2	W3	R	PTID	L	LCID	RELV
Type	F	F	F	F	I	F	I	I
Default	none	none	none	none	none	none	none	0

VARIABLE	DESCRIPTION
W1, W2, W3	Rotational Velocity along the X,Y,Z axes
R	Radius of the rotating reference frame. If a negative value if given, then the absolute value will refer to a *DEFINE_FUNCTION ID. If a DEFINE_FUNCTION is used, the following parameters are allowed: $f(x, y, z, vx, vy, vz, temp, pres, time)$.
PTID	Starting point ID for the reference frame (See *ICFD_DEFINE_-POINT)
L	Length of the rotating reference frame

VARIABLE	DESCRIPTION
LCID	Load curve for scaling factor of w. If a negative value is entered, then the absolute value will refer to a *DEFINE_FUNCTION ID. If a DEFINE_FUNCTION is used, the following parameters are allowed: $f(x,y,z,vx,vy,vz,temp,pres,time)$.
RELV	Velocities computed and displayed: EQ.0: Relative velocity, only the non-rotating components of the velocity are used and displayed. EQ.1: Absolute velocity . All the components of the velocity are used. Useful in cases where several or at least one non-inertial reference frame is combined with an inertial "classical" reference frame.

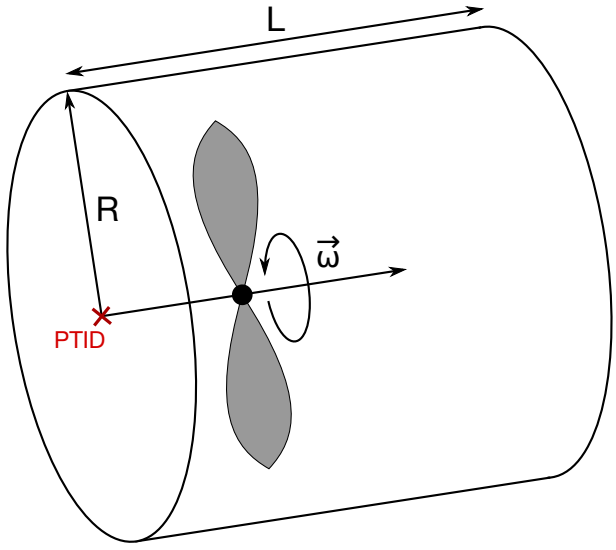


Figure 5-2. Non Inertial Reference Frame Example

*ICFD_DEFINE_WAVE_DAMPING

Purpose: This keyword defines a damping zone for free surface waves.

Card 1	1	2	3	4	5	6	7	8
Variable	PID	NID	L	F1	F2	N	LCID	
Type	I	F	F	F	F	I	I	
Default	none	none		10	10	1	none	

VARIABLE	DESCRIPTION
PID	Point ID defining the start of the damping layer.
NID	Normal ID defined using ICFD_DEFINE_POINT and pointing to the outgoing direction of the damping layer.
L	Length of damping layer. If no is value specified, the damping layer will have a length corresponding to five element lengths.
F1/F2	Linear and quadratic damping factor terms.
N	Damping term factor.
LCID	Load curve ID acting as temporal scale factor on damping term.

Remarks:

1. The damping is achieved by adding a source term to the momentum equations :

$$s^d = w\ (f_1 + f_2|u|)\ u$$

with w the weight function :

$$w = \frac{e^{\gamma} - 1}{e - 1}$$

and γ the blending function which allows a smooth insertion of the source term in the damping layer :

$$\gamma = \left(\frac{x - x_{sd}}{x_{ed} - x_{sd}}\right)^n$$

x_{sd} and x_{ed} representing the start and end coordinates of the damping zone.

*ICFD_INITIAL

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

Include as many cards as needed. This input ends at the next keyword ("*") card.

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

VARIABLE	DESCRIPTION
PID	Part ID for the volume elements or the surface elements where the values are initialized (see *ICFD_PART_VOL and *ICFD_PART). PID = 0 to assign the initial condition to all nodes at once.
Vx	x coordinate for the velocity.
Vy	y coordinate for the velocity.
Vz	z coordinate for the velocity.
T	Initial temperature.
P	Initial Pressure.

*ICFD_INITIAL_TURBULENCE

Purpose: When a RANS turbulent model is selected, it is possible to modify the default initial values of the turbulent quantities using this keyword.

Include as many cards as needed. This input ends at the next keyword ("*") card.

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

VARIABLE	DESCRIPTION
PID	Part ID for the volume elements or the surface elements where the values are initialized (see *ICFD_PART_VOL and *ICFD_PART). PID = 0 to assign the initial condition to all nodes at once.
I	Initial turbulent intensity.
R	Initial turbulent viscosity to laminar viscosity ratio ($r = \frac{\mu_{turb}}{\mu}$).

Remarks:

1. If no initial conditions have been assigned to a specific PID, the solver will automatically pick I = 0.05 (5%) and R = 10000.

***ICFD_MAT**_*{OPTION}*

Available options include

TITLE

Purpose: Specify physical properties for the fluid material.

Fluid Material Card Sets:

The Material Fluid Parameters Card is required. If a second card is given, it must be a Thermal Fluid Parameters Card. If the fluid thermal properties are not needed, the second card can be a blank card. In the third card, it is possible to associate the fluid material to a Non-Newtonian model and/or to a Porous media model (See *ICFD_MODEL_NONNEWT and *ICFD_MODEL_POROUS).

Material Fluid Parameters Card.

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

Thermal Fluid Parameters Card. Only to be defined if the thermal problem is solved.

Card 2	1	2	3	4	5	6	7	8
Variable	HC	TC	BETA	PRT	HCSFLCID	TCSFLCID		
Type	F	F	F	F	I	I		
Default	0	0	0	0.85	none	none		

Additional fluid models. Only to be defined if the fluid is non-newtonian and/or is a porous media.

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

VARIABLE	DESCRIPTION
MID	Material ID.
FLG	Flag to choose between fully incompressible, slightly compressible, or barotropic flows. EQ.0 : Vacuum (free surface problems only) EQ.1 : Fully incompressible fluid.
RO	Flow density.
VIS	Dynamic viscosity.
ST	Surface tension coefficient.
STSFLCID	Load curve ID for scale factor applied on ST function of time. See *DEFINE_CURVE, *DEFINE_CURVE_FUNCTION, or *DEFINE_FUNCTION. If a DEFINE_FUNCTION is used, the following parameters are allowed: $f(x,y,z,vx,vy,vz,temp,pres,time)$.
HC	Heat capacity.
TC	Thermal conductivity.
BETA	Thermal expansion coefficient used in the Boussinesq approximation for buoyancy.
PRT	Turbulent Prandlt number. Only used if K-Epsilon turbulence model selected.

VARIABLE	DESCRIPTION
HCSFLCID	Load curve ID for scale factor applied on HC function of time. See *DEFINE_CURVE, *DEFINE_CURVE_FUNCTION, or *DEFINE_FUNCTION. If a DEFINE_FUNCTION is used, the following parameters are allowed: $f(x, y, z, vx, vy, vz, temp, pres, time)$.
TCSFLCID	Load curve ID for scale factor applied on TC function of time. See *DEFINE_CURVE, *DEFINE_CURVE_FUNCTION, or *DEFINE_FUNCTION. If a DEFINE_FUNCTION is used, the following parameters are allowed: $f(x, y, z, vx, vy, vz, temp, pres, time)$.
NNMOID	Non-Newtonian model ID. This refers to a Non-Newtonian fluid model defined using *ICFD_MODEL_NONNEWT.
PMMOID	Porous media model ID. This refers to a porous media model defined using *ICFD_MODEL_POROUS.

Remarks:

1. If a K-Epsilon turbulence model is used and the heat transfer equation is solved, then the effective thermal conductivity will be determined by :

$$k_{eff} = k + \frac{Cp\mu_{turb}}{Pr_{turb}}$$

*ICFD_MODEL_NONNEWT

Purpose: Specify a non-newtonian model or a viscosity law that can associated to a fluid material.

Non-Newtonian Model ID and type.

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

Non-Newtonian Fluid Parameters Card.

Card 2	1	2	3	4	5	6	7	8
Variable	K	N	MUMIN	LAMBDA	ALPHA	TALPHA		
Type	F	F	F	F	F	F		
Default	0.0	0.0	0.0	1.e30	0.0	0.0		

VARIABLE	DESCRIPTION
NNMOID	Non-Newtonian Model ID.
NNID	Non-Newtonian fluid model type : EQ.1 : Power-Law model EQ.2 : Carreau model EQ.3 : Cross model EQ.4 : Herschel-Bulkley model EQ.5 : Cross II model EQ.6 : Sutherland formula for temperature dependent viscosity EQ.7 : Power-Law for temperature dependent viscosity EQ.8 : Viscosity defined by Load Curve ID or Function ID

VARIABLE	DESCRIPTION
K	Consistency index if NNID = 1 and 4. Zero shear Viscosity if NNID = 2,3 and 5.Reference viscosity if NNID = 6 and NNID = 7. Load curve ID or function ID if NNID = 8.
N	Measure of the deviation of the fluid from Newtonian (Power Law index) for NNID = 1,2,3,4,5,7. Not used for NNID = 6 and 8.
MUMIN	Minimum acceptable viscosity value if NNID = 1. Infinite Shear Viscosity if NNID = 2,5.Yielding viscosity if NNID = 4.Not used if NNID = 3,6,7,8.
LAMBDA	Maximum acceptable viscosity value if NNID = 1. Time constant if NNID = 2, 3, 5. Yield Stress Threshold if NNID = 4.Sutherland constant if NNID = 6. Not used if NNID = 7,8.
ALPHA	Activation energy if NNID = 1, 2. Not used if NNID = 3,4,5,6,7,8.
TALPHA	Reference temperature if NNID = 2. Not used if NNID = 1,3,4,5,6,7,8

Remarks:

1. For the Non-Newtonian models, the viscosity is expressed as :

   a) POWER-LAW :

$$\mu = k\dot{\gamma}^{n-1}e^{\alpha T_0/T}$$

$$\mu_{min} < \mu < \mu_{max}$$

   With k the consistency index, n the power law index, α the activation energy, T_0 the initial temperature, T the temperature at any given time t, μ_{min} the minimum acceptable viscosity and μ_{max} the maximum acceptable viscosity.

   b) CARREAU :

$$\mu = \mu_\infty + (\mu_0 - \mu_\infty)[1 + (H(T)\dot{\gamma}\lambda)^2]^{(n-1)/2}$$

$$H(T) = exp\left[\alpha(\frac{1}{T - T_0} - \frac{1}{T_\alpha - T_0})\right]$$

   With μ_∞ the infinite shear viscosity, μ_0 the zero shear viscosity, n the power law index, λ a time constant, α the activation energy, T_0 the initial temperature, T

the temperature at any given time t and T_α the reference temperature at which $H(T) = 1$.

c) CROSS :

$$\mu = \frac{\mu_0}{1 + (\lambda\dot{\gamma})^{1-n}}$$

With μ_0 the zero shear viscosity, n the power law index and λ a time constant.

d) HERSCHEL-BULKLEY :

$$\mu = \mu_0 \; if \; (\dot{\gamma} < \tau_0/\mu_0)$$

$$\mu = \frac{\tau_0 + k[\dot{\gamma}^n - (\tau_0/\mu_0)^n]}{\dot{\gamma}}$$

With k the consistency index, τ_0 the Yield stress threshold, μ_0 the yielding viscosity and n the power law index.

e) CROSS II :

$$\mu = \mu_\infty + \frac{\mu_0 - \mu_\infty}{1 + (\lambda\dot{\gamma})^n}$$

With μ_0 the zero shear viscosity, μ_∞ the infinite shear viscosity, n the power law index and λ a time constant.

2. For the temperature dependent viscosity models, the viscosity is expressed as :

a) SUTHERLAND's LAW :

$$\mu = \mu_0 (\frac{T}{T_0})^{3/2} \frac{T_0 + S}{T + S}$$

With μ_0 a reference viscosity, T_0 the initial temperature (which therefore must not be 0.), T the temperature at any given time t and S Sutherland's constant.

b) POWER LAW :

$$\mu = \mu_0 (\frac{T}{T_0})^n$$

With μ_0 a reference viscosity, T_0 the initial temperature (which therefore must not be 0.), T the temperature at any given time t and n the power law index.

3. For NNID = 8, a load curve function of time, a curve function or a function can be used. If it references a DEFINE_FUNCTION, the following arguments are allowed $f(x, y, z, vx, vy, vz, temp, pres, shear, time)$.

*ICFD_MODEL_POROUS

Purpose: Specify a porous media model.

Porous Media Model ID and type.

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

Porous Media Parameters Card.

Card 2	1	2	3	4	5	6	7	8
Variable	POR	PER/THX	FF/THY	THZ	PVLCIDX	PVLCIDY	PVLCIDZ	
Type	F	F	F	F	I	I	I	
Default	0.	0.	0.	0.	none	none	none	

Permeability Vector Card in local reference frame. Only to be defined if the porous media is anisotropic.

Card 3	1	2	3	4	5	6	7	8
Variable	KX'	KY'	KZ'					
Type	F	F	F					
Default	0.	0.	0.					

Projection of local Vectors in global reference frame. Only to be defined if the porous media is anisotropic.

Card 4	1	2	3	4	5	6	7	8
Variable	1-X/1-PID	1-Y/2-PID	1-Z	2-X	2-Y	2-Z		
Type	F/I	F/I	F/I	F/I	F/I	F/I		
Default	0	0.	0.	0.	0.	0.		

VARIABLE	DESCRIPTION
PMMOID	Porous media model ID.
PMID	Porous media model type : EQ.1 : Isotropic porous media - Ergun Correlation. EQ.2 : Isotropic porous media - Darcy-Forchheimer model. EQ.3 : Isotropic porous media - Permeability defined through Pressure-Velocity Data. EQ.4 : Anisotropic porous media - Fixed local reference frame (See Figure 5-3). EQ.5 : Anisotropic porous media model - Moving local reference frame and permeability vector in local reference frame (x', y', z') defined by three Pressure-Velocity curves. EQ.6 : Anisotropic porous media model - Moving local reference frame and permeability vector constant. EQ.7: Anisotropic porous media model - Moving local reference frame and permeability vector constant. This model differs from PMID = 6 in the way the local reference frame is moved.
POR	Porosity ε.
PER/THX	Permeability κ if PMID = 1 or 2. Probe Thickness Δx if PMID = 3 or PMID = 5.
FF/THY	Forchheimer factor. To Be defined if PMID = 2. Probe Thickness Δy if PMID = 5.

VARIABLE	DESCRIPTION
THZ	Probe Thickness Δz if PMID = 5.
PVLCIDX	Pressure function of Velocity Load Curve ID. To be defined if PMID = 3 and PMID = 5. If PMID = 5, this refers to P-V curve in global X direction. For PMID = 1 and PMID = 2, this flags acts as an optional load curve ID, define curve function ID or define function ID. If a DEFINE_FUNCTION is used, the following parameters are allowed: $f(x, y, z, vx, vy, vz, temp, pres, time)$.
PVLCIDY	Pressure function of Velocity Load Curve ID. To be defined if PMID = 5. This refers to P-V curve in global Y direction.
PVLCIDZ	Pressure function of Velocity Load Curve ID. To be defined if PMID = 5. This refers to P-V curve in global Z direction.
KX′/KY′/KZ′	Permeability vector in local reference frame (x', y', z'). To be defined in PMID = 4, 5, 6 or 7. Those values become scale factors if PMID = 5.
1-X/1-Y/1-Z	Projection of local permeability vector **x**′ in global reference frame (x, y, z). To be defined if PMID = 4. If PMID = 6, those become load curve IDs so the coordinates of the local **x**′ vector can be made to move through time.
2-X/2-Y/2-Z	Projection of local permeability vector **y**′ in global reference frame (x, y, z). To be defined if PMID = 4. If PMID = 6, those become load curve IDs so the coordinates of the local **y**′ vector can be made to move through time.
1-PID/2-PID	If PMID = 5 or PMID = 7, the two local reference frame vectors are defined by the coordinates of the two point IDs defined by 1-PID and 2-PID. (See ICFD_DEFINE_POINT). Since those points can be made to move, it is therefore possible to define a moving reference frame for the anisotropic porous media domain.

Remarks:

1. Being ε the porosity and κ the permeability of the porous media respectively, one can define:

$$\varepsilon = \frac{void\ volume}{total\ volume}$$

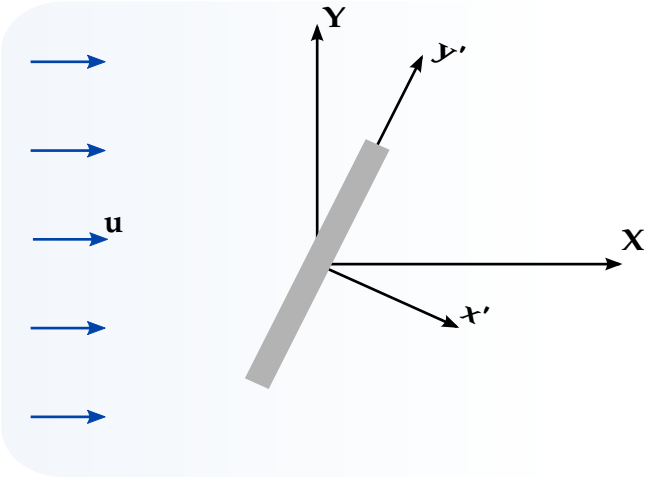


Figure 5-3. Anisotropic porous media vectors definition (PMID = 4,5,6,7). The vectors **X** and **Y** are the global axes; **x**′ and **y**′ define system for the primed coordinate(x', y', z').

And being u_i the volume averaged velocity field defined in terms of the fluid velocity field u_{if} as:

$$u_i = \varepsilon u_{if}$$

The generalized flow equations of momentum and mass conservation can be expressed as :

$$\frac{\partial u_i}{\partial x_i} = 0$$

$$\frac{\rho}{\varepsilon}\left[\frac{\partial u_i}{\partial t} + \frac{\partial}{\partial x_j}\left(\frac{\partial u_i u_j}{\varepsilon}\right)\right] = -\frac{1}{\varepsilon}\frac{\partial (P\varepsilon)}{\partial x_i} + \frac{\mu}{\varepsilon}\left(\frac{\partial^2 u_i}{\partial x_j \partial x_j}\right) + \rho g_i - D_i$$

Where D_i are the forces exerted to the fluid by the porous matrix. For the isotropic model, the porous forces are a function of the matrix porosity and its permeability. For the isotropic case, three models are available :

a) Model 1 : Ergun correlation $D_i = \frac{\mu u_i}{\kappa} + \frac{1.75\rho|U|}{\sqrt{150}\sqrt{\kappa}\varepsilon^{3/2}} u_i$

b) Model 2 : Darcy-Forcheimer $D_i = \frac{\mu u_i}{\kappa} + \frac{F\varepsilon\rho|U|}{\sqrt{\kappa}} u_i$

c) Model 3 : Using the $\Delta P - V$ experimental data. In this case, it is assumed that the pressure velocity curve was obtained applying a pressure difference or pressure drop on both ends of a porous slab of thickness Δx with porous properties κ and ε. It then becomes possible for the solver to fit that experimental curve with a quadratic polynomial of the form

$\Delta P(u_x) = \alpha {u_x}^2 + \beta u_x$. Once α and β are known, it is possible to estimate D_i.

2. The anisotropic (See Figure 0-1) version of the Darcy-Forcheimer term can be written as :

$$D_i = \mu B_{ij} \mu_j + F\varepsilon |U| C_{ij} u_j$$

$$B_{ij} = \left(K_{ij}\right)^{-1}$$

$$C_{ij} = \left(K_{ij}\right)^{-1/2}$$

Where K_{ij} is the anisotropic permeability tensor.

***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 optional line with a HEADING in order to associate a name to the part.

Card 1	1	2	3	4	5	6	7	8
Variable	HEADING							
Type	A							
Default	none							

Part Material Card. Include as many cards as needed. This input ends at the next keyword ("*") card.

Card 2	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: This keyword assigns material properties to the nodes enclosed by surface ICFD parts.

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

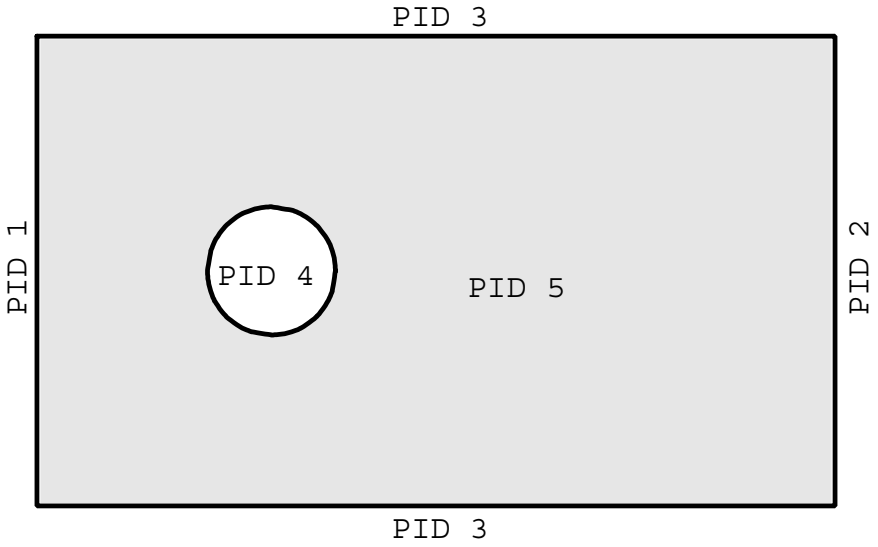
Title	1	2	3	4	5	6	7	8
Variable	HEADING							
Type	A							
Default	none							

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

Provide as many cards as necessary. This input ends at the next keyword ("*") card

Card 2	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.
SPID1, …	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_LIST

Purpose: Only used in cases where the mesh is specified by the user (See *MESH_ VOLUME_ELEMENT). Defines a set of nodes associated with 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						

Node List Card. Provide as many cards as necessary. This input ends at the next keyword ("*") card

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 IDs

Remarks:

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

*ICFD_SOLVER_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. It is intended only for special 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	1	10^{-3}						

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_SOLVER_TOL_LSET

Purpose: This keyword allows the user to change the default tolerance values for the advection equation for levelset. *Care should be taken when deviating from the default values*.

Card 1	1	2	3	4	5	6	7	8
Variable	ATOL	RTOL		MAXIT				
Type	F	F		I				
Default	10^{-8}	10^{-8}		1000				

VARIABLE	DESCRIPTION
ATOL	Absolute convergence criteria. Convergence is achieved when $\text{Residual}_{i+1} - \text{Residual}_i \leq \text{ATOL}$. If a negative integer is entered, then that value will be used as a load curve ID for ATOL.
RTOL	Relative convergence criteria. Convergence is achieved when $(\text{Residual}_{i+1} - \text{Residual}_i)/\text{Residual}_{\text{initial}} \leq RTOL$. If a negative integer is entered, then that value will be used as a load curve ID for RTOL.
MAXIT	Maximum number of iterations allowed to achieve convergence. If a negative integer is entered, then that value will be used as a load curve ID for MAXIT.

*ICFD_SOLVER_TOL_MMOV

Purpose: This keyword allows the user to change the default tolerance values for the mesh movement algorithm. *Care should be taken when deviating from the default values*.

Card 1	1	2	3	4	5	6	7	8
Variable	ATOL	RTOL		MAXIT				
Type	F	F		I				
Default	1e-8	1e-8		1000				

VARIABLE	DESCRIPTION
ATOL	Absolute convergence criteria. Convergence is achieved when $\text{Residual}_{i+1} - \text{Residual}_i \leq \text{ATOL}$. If a negative integer is entered, then that value will be used as a load curve ID for ATOL.
RTOL	Relative convergence criteria. Convergence is achieved when $(\text{Residual}_{i+1} - \text{Residual}_i)/\text{Residual}_{\text{initial}} \leq \text{RTOL}$. If a negative integer is entered, then that value will be used as a load curve ID for RTOL.
MAXIT	Maximum number of iterations allowed to achieve convergence. If a negative integer is entered, then that value will be used as a load curve ID for MAXIT.

*ICFD_SOLVER_TOL_MOM

Purpose: This keyword allows the user to change the default tolerance values for the momentum equation solve. *Care should be taken when deviating from the default values*.

Card 1	1	2	3	4	5	6	7	8
Variable	ATOL	RTOL		MAXIT				
Type	F	F		I				
Default	10^{-8}	10^{-8}		1000				

VARIABLE	DESCRIPTION
ATOL	Absolute convergence criteria. Convergence is achieved when $\text{Residual}_{i+1} - \text{Residual}_i \leq \text{ATOL}$. If a negative integer is entered, then that value will be used as a load curve ID for ATOL.
RTOL	Relative convergence criteria. Convergence is achieved when $(\text{Residual}_{i+1} - \text{Residual}_i)/\text{Residual}_{\text{initial}} \leq \text{RTOL}$. If a negative integer is entered, then that value will be used as a load curve ID for RTOL.
MAXIT	Maximum number of iterations allowed to achieve convergence. If a negative integer is entered, then that value will be used as a load curve ID for MAXIT.

*ICFD_SOLVER_TOL_MONOLITHIC

Purpose: This keyword allows the user to change the default tolerance values for the monolithic solver. *Care should be taken when deviating from the default values*.

Card 1	1	2	3	4	5	6	7	8
Variable	ATOL	RTOL		MAXIT				
Type	F	F		I				
Default	10^{-8}	10^{-8}		1000				

VARIABLE	DESCRIPTION
ATOL	Absolute convergence criteria. Convergence is achieved when $\text{Residual}_{i+1} - \text{Residual}_i \leq \text{ATOL}$. If a negative integer is entered, then that value will be used as a load curve ID for ATOL.
RTOL	Relative convergence criteria. Convergence is achieved when $(\text{Residual}_{i+1} - \text{Residual}_i)/\text{Residual}_{\text{initial}} \leq \text{RTOL}$. If a negative integer is entered, then that value will be used as a load curve ID for RTOL.
MAXIT	Maximum number of iterations allowed to achieve convergence. If a negative integer is entered, then that value will be used as a load curve ID for MAXIT.

*ICFD_SOLVER_TOL_PRE

Purpose: This keyword allows the user to change the default tolerance values for the Poisson equation for pressure. *Care should be taken when deviating from the default values*.

Card 1	1	2	3	4	5	6	7	8
Variable	ATOL	RTOL		MAXIT				
Type	F	F		I				
Default	10^{-8}	10^{-8}		1000				

VARIABLE	DESCRIPTION
ATOL	Absolute convergence criteria. Convergence is achieved when $\text{Residual}_{i+1} - \text{Residual}_i \leq \text{ATOL}$. If a negative integer is entered, then that value will be used as a load curve ID for ATOL.
RTOL	Relative convergence criteria. Convergence is achieved when $(\text{Residual}_{i+1} - \text{Residual}_i)/\text{Residual}_{\text{initial}} \leq RTOL$. If a negative integer is entered, then that value will be used as a load curve ID for RTOL.
MAXIT	Maximum number of iterations allowed to achieve convergence. If a negative integer is entered, then that value will be used as a load curve ID for MAXIT.

*ICFD_SOLVER_TOL_TEMP

Purpose: This keyword allows the user to change the default tolerance values for the heat equation. To be handled with great care.

Card 1	1	2	3	4	5	6	7	8
Variable	ATOL	RTOL		MAXIT				
Type	F	F		I				
Default	1e-8	1e-8		1000				

VARIABLE	DESCRIPTION
ATOL	Absolute convergence criteria. Convergence is achieved when $Residual_{i+1} - Residual_i \leq ATOL$. If a negative integer is entered, then that value will be used as a load curve ID for ATOL.
RTOL	Relative convergence criteria. Convergence is achieved when $(Residual_{i+1} - Residual_i)/Residual_{initial} \leq RTOL$. If a negative integer is entered, then that value will be used as a load curve ID for RTOL.
MAXIT	Maximum number of iterations allowed to achieve convergence. If a negative integer is entered, then that value will be used as a load curve ID for MAXIT.

*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_NODE

*MESH_SIZE_

*MESH_SIZE_SHAPE

*MESH_SURFACE_ELEMENT

*MESH_SURFACE_NODE

*MESH_VOLUME

*MESH_VOLUME_ELEMENT

*MESH_VOLUME_NODE

*MESH_VOLUME_PART

*MESH_BL

Purpose: This keyword is used to define a boundary-layer mesh as a refinement on volume-mesh. The boundary layer mesh is constructed by subdividing elements near the surface.

Boundary Layer Cards. Define as many cards as are necessary. The next "*" card terminates the input.

Card 1	1	2	3	4	5	6	7	8
Variable	PID	NELTH	BLTH	BLFE	BLST			
Type	I	I	F	F	I			
Default	none	none	0.	0.	0			

VARIABLE	DESCRIPTION
PID	Part identifier for the surface element.
NELTH	Number of elements normal to the surface (in the boundary layer) is NELTH+1.
BLTH	Boundary layer mesh thickness if BLST = 1 or BLST = 2. Growth scale factor if BLST = 3. Ignored if BLST = 0.
BLFE	Distance between the wall and the first volume mesh node if BLST = 3. Scaling coefficient if BLST = 1 or BLST = 2. Ignored if BLST = 0.
BLST	Boundary layer mesh generation strategy : EQ.0: Default. $2^{NELTH+1}$ subdivision based on surface mesh size. EQ.1: Power law using BLTH, and NELTH with BLFE as a scale factor. EQ.2: Geometric series based on BLTH and BLFE. EQ.3: Repartition following a growth scale factor (BLTH).

Remarks:

1. For BLST = 0, for every additional NELTH, the automatic volume mesher will divide the elements closest to the surface by two so that the smallest element in the boundary layer mesh will have an aspect ratio of $2^{\text{NELTH}+1}$. A default boundary layer mesh thickness based on the surface mesh size will be chosen.

2. For a constant repartition of the nodes in the boundary layer, use BLST = 1 with BLFE = 1. For BLST = 1, starting from the wall, the position of node n in the normal direction is given by :

$$X_n = \left(\frac{n}{NELTH+1}\right)^{[5\times(1-BLFE)]} \frac{BLTH}{\sum_{i=1}^{NELTH+1}[i/(NELTH+1)]^{[5\times(1-BLFE)]}}$$

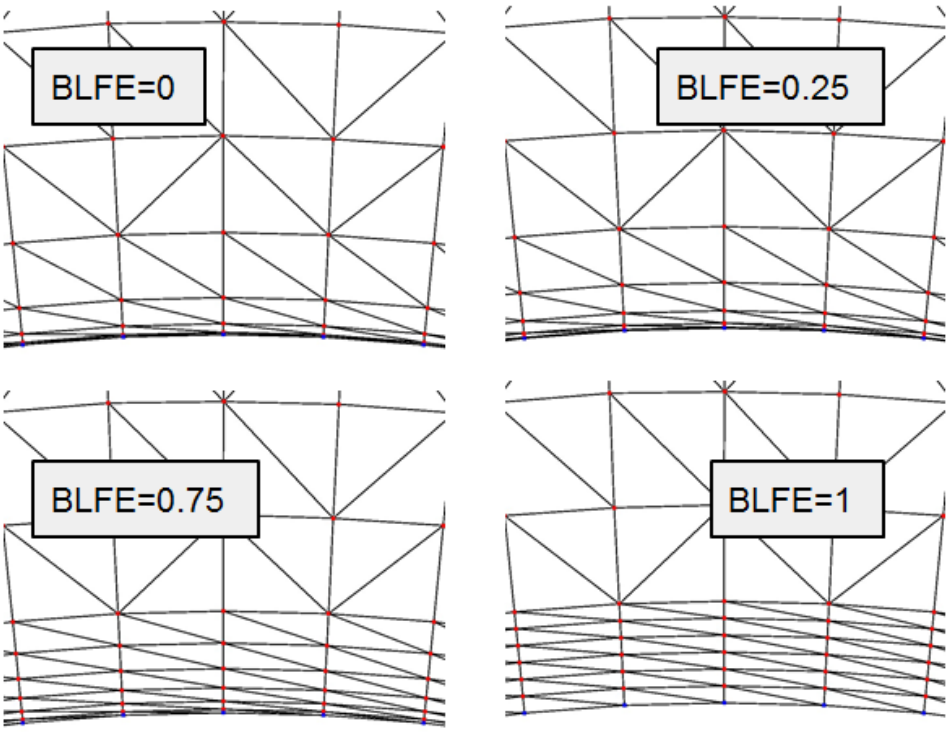


Figure [1]. BLST = 1 example

3. Setting BLFE = 1 makes BLST = 2 equivalent to BLST = 0 except that BLST = 0 allows BLTH to be specified by the user instead of automatically using the local surface mesh size. For BLST = 2, starting from BLTH, each newly inserted node will have its location closer to the wall, following this law :

$$X_n = (0.5 \times BLFE)^n * BLTH * (1 - 0.5 * BLFE)$$

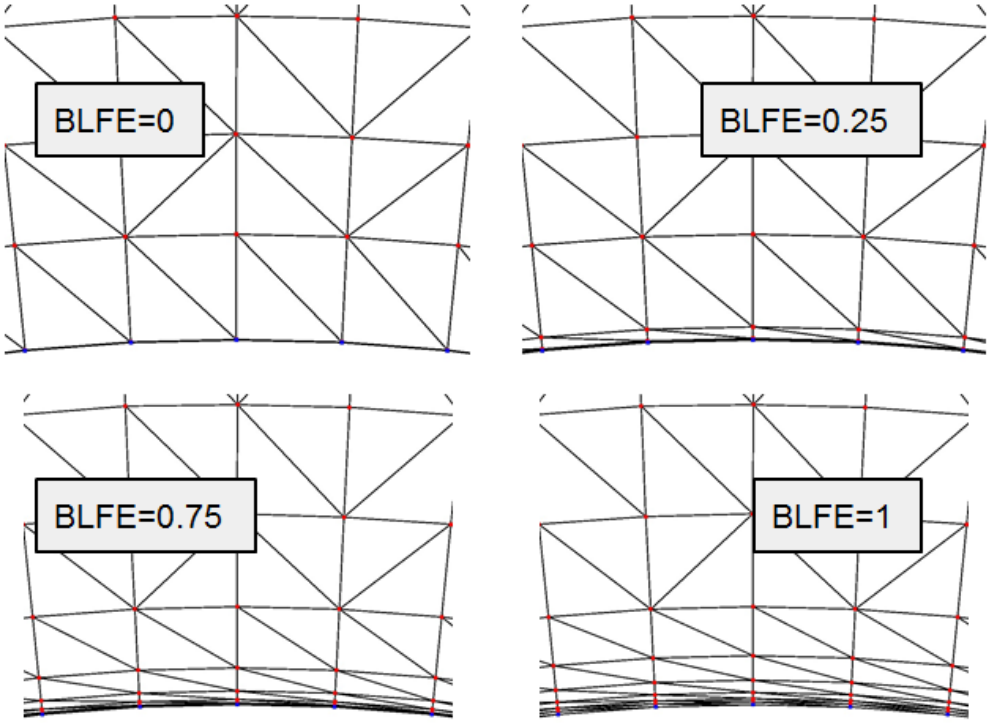


Figure [2]. BLST = 2 example

4. For BLST = 3, starting from the wall, the position of node n in the normal direction is given by :

$$X_n = \sum_{i=0}^{n} BLFE * BLTH^i \ \ with\ 0 \leq n \leq NELTH$$

*MESH_BL_SYM

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

Boundary Layer with Symmetry Condition Cards. Define as many cards as necessary. The next "*" card terminates the input.

Card 1	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
PID1, …	Part identifiers 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							

Define as many cards as are necessary based on the number of PIDs (the next "*" card terminates the input.)

Card 2	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.
PID*n*	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							

Define as many cards as are necessary based on the number of PIDs. This input ends at the next keyword ("*") card.

Card 2	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.
PID*n*	Part IDs for the surface elements.

*MESH_NODE

Purpose: Define a fluid node and its coordinates. These nodes are used in the mesh generation process by the *MESH_SURFACE_ELEMENT keyword, or as user defined volume nodes by the *MESH_VOLUME_ELEMENT keyword.

Node Cards. Include one additional card for each node. This input ends at the next keyword ("*") card.

Card 1	1	2	3	4	5	6	7	8	9	10
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.

Remarks:

1. The data card format for the *MESH_NODE keyword is identical to *NODE.
2. The *MESH_NODE keyword supersedes *MESH_SURFACE_NODE, which was for surfaces nodes as well as *MESH_VOLUME_NODE for, which was for volume nodes in user defined.

*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							

Define as many cards as are necessary based on the number of PIDs (the next "*" card terminates the input.).

Card 2	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.
PID*n*	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 and polynomial). 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. In the polynomial case, it is recommended to define several zones for a better mesh size control.

Remeshing Control Card sets:

Add as many *remeshing control cards* paired with a *case card* as desired. The input of such pairs ends at the next keyword "*" card.

Remeshing Control. First card specifies whether to maintain this mesh sizing criterion through a remesh operation.

Card 1	1	2	3	4	5	6	7	8
Variable	SNAME	FORCE	METHOD	BT	DT			
Type	A	I	I	F	F			
Default	none	0	0	0.	1.E12			

Box Case. Card 2 for SNAME = "box" and METHOD = 0

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	

Sphere Case. Card 2 for SNAME = "sphere" and METHOD = 0

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			

Cylinder Case. Card 2 for SNAME = "cylinder" and METHOD = 0

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

Polynomial Case. Card 2 for SNAME = "pol" and METHOD = 0

Cards 2	1	2	3	4	5	6	7	8
Variable	MSIZE	X	Y	Z	NX	NY	NZ	
Type	F	F	F	F	F	F	F	
Default	none	none	none	none	none	none	none	

Card 2 for METHOD = 1

Cards 2	1	2	3	4	5	6	7	8
Variable	MSIZE	RADIUS	PTID1	PTID2				
Type	F	F	I	I				
Default	none	none	none	none				

VARIABLE	DESCRIPTION
SNAME	Shape name. Possibilities include "box", "cylinder", "pol" and "sphere"
FORCE	Force to keep the mesh size criteria even after a remeshing is done. EQ.0: Off, mesh size shape will be lost if a remeshing occurs EQ.1: On.
METHOD	Specifies which method to use when defining the second card. EQ.0: Default, directly input the coordinates. EQ.1: Define the coordinates via the introduction of ICFD_DEFINE_POINT IDs. The biggest advantage of using this method is that the ICFD_DEFINE_POINTs are allowed to move which allows the user to control how the mesh size area should evolve function of time in cases where there is remeshing.
BT/DT	Birth and death time of the mesh size area in cases where remeshing occurs.
MSIZE	Mesh size that needs to be applied in the zone of the shape defined by SNAME
PMIN[X, Y, Z]	x, y, or z value for the point of minimum coordinates
PMAX[X, Y, Z]	x, y, or z value for the point of maximum coordinates
CEN-TER[X, Y, Z]	Coordinates of the sphere center in cases where SNAME is sphere

VARIABLE	DESCRIPTION
RADIUS	Radius of the sphere if SNAME is Sphere or of the cross section disk if SNAME is Cylinder.
X/Y/Z	Coordinates of starting point in cases where SNAME is pol.
NX/NY/NZ	Direction in which mesh size will be applied in cases where SNAME is pol.
PTID1	Point ID1 referring to ICFD_DEFINE_POINT. Replaces PMIN, X/Y/Z or CENTER for the various SNAME cases.
PTID2	Point ID2. Not needed if SNAME is Sphere. Replaces PMAX or NX/NY/NZ for the various SNAME cases.

*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**).**

Surface Element Card. Define as many cards as necessary. The next "*" card terminates the input.

Card 1	1	2	3	4	5	6	7	8	9	10
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	Mesh surface part ID. A unique identifier for the surface to which this mesh surface element belongs.
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. Note that the accepted card format is 6i8 (not 6i10)

*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.

*MESH_NODE supersedes this card; so please use *MESH_NODE instead of this card.

Surface Node Cards. Include one card for each node. Include as many cards a necessary. This input ends at the next keyword ("*") card.

Card 1	1	2	3	4	5	6	7	8	9	10
Variable	NID	X		Y		Z				
Type	I	F		F		F				
Default	none	0		0		0				

VARIABLE	DESCRIPTION
NID	Node ID. This NID must be unique within the set of 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 surfaces listed have to be non-overlapping, and should not leave any gaps or open spaces between the surface boundaries. On the boundary between two neighbor surfaces, nodes have to be in common (no duplicate nodes) and should match exactly on the interface. They are defined by the keyword *MESH_SURFACE_NODE. 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							

Define as many cards as are necessary based on the number of PIDs (the next "*" card terminates the input.)

Card 2	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.
PID*n*	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 tetrahedral elements are supported (triangles in 2D).

Volume Element Card. Define as many cards as necessary. The next "*" card terminates the input.

Card 1	1	2	3	4	5	6	7	8	9	10
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_SOLID.

*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 tetrahedral elements are supported.

*MESH_NODE supersedes this card; so please use *MESH_NODE instead of this card.

Volume Node Cards. Include as many cards in the following format as desired. This input ends at the next keyword ("*") card.

Card 1	1	2	3	4	5	6	7	8	9	10
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 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).

Mesh Volume Part Card. Include as many cards in the following format as desired. This input ends at the next keyword ("*") card.

Card 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	0	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		

Nozzle card 1: Provide as many pairs of nozzle cards 1 and 2 as necessary. This input ends at the next keyword ("*") card (following a nozzle card 2).

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

Nozzle card 2: Provide as many pairs of nozzle cards 1 and 2 as necessary. This input ends at the next keyword ("*") card.

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

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: TAB EQ.2: KHRT
ICOLLDE	Turn collision modeling on or off
IEVAP	Evaporation flag: EQ.0: off (no evaporation) EQ.1: Turn evaporation on (see Remark 1)
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), H_2O EQ.2: Benzene, C_6H_6 EQ.3: Diesel # 2, $C_{12}H_{26}$ EQ.4: Diesel # 2, $C_{13}H_{13}$ EQ.5: Ethanol, C_2H_5OH EQ.6: Gasoline, C_8H_{18} EQ.7: Jet-A, $C_{12}H_{23}$ EQ.8: Kerosene, $C_{12}H_{23}$ EQ.9: Methanol, CH_3OH EQ.10: N-dodecane, $C_{12}H_{26}$
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.

VARIABLE	DESCRIPTION
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 counterclockwise from there.
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

Remarks:

1. When IEVAP = 1, the keyword input file must be modified in a fashion similar to a chemistry problem. This is illustrated in a portion of an example keyword file below. That is, the following keywords need to be used, along with the inclusion of other chemistry-related files (i.e. evap.inp and the corresponding thermodynamics data file):

   *CHEMISTRY_MODEL

   *CHEMISTRY_COMPOSITION

   *CHEMISTRY_CONTROL_FULL

   *CESE_INITIAL_CHEMISTRY

```
$ Setup stochastic particles
$
*STOCHASTIC_SPRAY_PARTICLES
$  injdist    ibrkup  icollide     ievap    ipulse    limprt    fuelid
         3         1         0         1         0    100000         1
$     rhop       tip pmass[Kg]    prtrte   str_inj   dur_inj
    1000.0      300.      0.01     1.0e7       0.0      10.0
$  the next card is needed for fireball position and max.  particle velocity:
$    XORIG     YORIG     ZORIG       SMR   Velinj      Drnoz    Dthnoz
     0.005     0.005    1.0e-5    5.0e-6    200.0     9.0e-5
$   TILTXY    TILTXZ      CONE     DCONE      ANOZ       AMP0
       0.0       0.0      15.0      15.0   2.5e-8        0.0
$
*CHEMISTRY_MODEL
$ model_id    jacsel    errlim
```

```
        10         1       0.0
  evap.inp
 therm.dat
  tran.dat
$
*CHEMISTRY_COMPOSITION
$  comp_id  model_id
        11        10
$  molefra   Species
       1.0        O2
      3.76        N2
$
*CHEMISTRY_CONTROL_FULL
$   sol_id    errlim
         5
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$ Set global initial conditions for fluid
$
*CESE_INITIAL_CHEMISTRY
$   sol_id   comp_id
         5        11
$INITIAL CONDITIONS
$      uic       vic       wic       ric       pic       tic       hic
       0.0       0.0       0.0       1.2   101325.     300.0       0.0
```

*STOCHASTIC_TBX_PARTICLES

Purpose: Specify particle and other model details for stochastic PDEs that model embedded particles in TBX explosives. Note that the components listed on the corresponding *CHEMISTRY_COMPOSITION card are in terms of molar concentrations of the species (in units of moles/[length]^3, where "[length]" is the user's length unit).

For further information on the theory of the TBX model that has been implemented, a document on this topic can be found at this URL:

http://www.lstc.com/applications/cese_cfd/documentation

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	FRADIUS	
Type	F	F	F	F	F	F	F	
Default	none	none	none	0.0	0.0	0.0	none	

VARIABLE	DESCRIPTION
PCOMB	Particle combustion model EQ.0: no burning EQ.1: K-model
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

VARIABLE	DESCRIPTION
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
FRADIUS	Radius of the explosive area.

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_ID_SET (*not available in the single-precision version of LS-DYNA*)

*LSO_POINT_SET

*LSO_TIME_SEQUENCE

*LSO_VARIABLE_GROUP

Note that only the mechanics solver is available in the single-precision version of LS-DYNA, and therefore, only LSO mechanics variables are available for output from single precision LS-DYNA. These mechanics variables are listed by domain type in a separate document. This document (LSO_VARIABLES.TXT) is created by running the command: LS-DYNA print_lso_doc. Contrary to LSO_VARIABLES.TXT, element quantities such as stress are not available for output from the mechanics solver to the "lso" database.

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 specify variables on a subset of the domain for a given solver. This domain can be a subset of the mesh used by that solver, a set of output points created with *LSO_POINT_SET, or a set of objects created with *LSO_-ID_SET. The frequency and duration of the output for any given domain is determined by each *LSO_TIME_SEQUENCE card that references this *LSO_DOMAIN card. Note that for the single-precision version of LS-DYNA, the only allowed value of SOLVER_-NAME = MECH.

Card 1	1	2	3	4	5	6	7	8
Variable	DOMAIN_TYPE							
Type	A							

Card 2	1	2	3	4	5	6	7	8
Variable	SOLVER_NAME							
Type	A							

Special Domains Card. Card 3 when DOMAIN_TYPE is one of ROGO, CIRCUIT, THIST_POINT or TRACER_POINT.

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

Miscellaneous Domain Card. Card 3 when DOMAIN_TYPE is one of NODE, PART, SEGMENT, SURFACE_NODE, SURFACE_ELEMENT, VOLUME_ELEMENT, SURFACE_PART, VOLUME_PART.

Card 3	1	2	3	4	5	6	7	8
Variable	OUTID	REFID	OVERRIDE	REDUCT				
Type	I	I	I	I				
Default	none	0	0	none				

Variable Name Card. Provide as many cards as necessary. This input ends at the next keyword ("*") card

Card 4	1	2	3	4	5	6	7	8
Variable	VARIABLE_NAME							
Type	A							

VARIABLE	DESCRIPTION
DOMAIN_TYPE	The type of domain for which LSO output may be generated.
SOLVER_NAME	Selects the solver from which data is output on this domain. Accepted entries so far are "MECH", "EM", "CESE", and "ICFD".
OUTID	LSO domain ID associated with this domain, and used by *LSO_TIME_SEQUENCE cards.
REFID	Support set ID. This can be a set defined by a *SET card, a *LSO_ID_SET, card, or a *LSO_POINT_SET card. Unless OVERRIDE is specified, this set must be of the same type as DOMAIN_TYPE.
OVERRIDE	If non-zero, then REFID is interpreted as: EQ.1: a PART set for SOLVER_NAME EQ.2: a PART set of volume parts created with a *LSO_-ID_SET card (volume parts are defined with

	EM	ICFD	CESE
VECTORS	magneticField_point electricField_point vecpotField_point currentDensity2_point	velocity_point	velocity_point
SCALARS	ScalarPotential_point	pressure_point temperature_point density_point lset_point	pressure_point temperature_point density_point

Table 8-1. Selected LSO Varriables

VARIABLE	DESCRIPTION
	*MESH_VOLUME cards). EQ.3: a PART set of surface parts created with a *LSO_-ID_SET card (surface parts are defined with *MESH_SURFACE_ELEMENT cards). EQ.4: a set of segment sets created with a *LSO_ID_SET card.
REDUCT	A function that operates on the entire domain and returns a single value for scalar variables, three values for vector variables, or 6 values for symmetric tensor variables. For REDUCT="range", the number of returned values doubles. The following are the supported functions: EQ.BLANK: no reduction (default) EQ."none": Same as above EQ."avg": the average by component EQ."average": Same as above EQ."min": the minimum by component EQ."minimum": Same as above EQ."max": the maximum by component EQ."maximum": Same as above EQ."sum": the sum by component

VARIABLE	DESCRIPTION
	EQ."range": the minimum by component followed by the maximum by component
VARIABLE_NAME	Either the name of a single output variable or a variable group. See remarks.

Remarks:

1. Supported choices for VARIABLE_NAME are listed by DOMAIN_TYPE for each SOLVER_NAME in a separate document. This document (LSO_VARIABLES.TXT) is created by running the command: LS-DYNA print_lso_doc. The following table shows a sample of the point output variables available when DOMAIN_TYPE = THIST_POINT:

*LSO_ID_SET

Purpose: Provides a way to create a set of existing sets (segment sets), or to define a set that is not available with other set-related keyword cards. These are then used in other *LSO cards to specify LSO output. This card is not available in the single precision version of LS-DYNA.

Card 1	1	2	3	4	5	6	7	8
Variable	SETID	TYPE	SOLVER					
Type	I	A	A					
Default	none	none	MECH					

Referenced IDs. Provide as many cards as necessary. This input ends at the next keyword ("*") card

Card	1	2	3	4	5	6	7	8
Variable	ID1	ID2	ID3	ID4	ID5	ID6	ID7	ID8
Type	I	I	I	I	I	I	I	I
Default	none	none	none	none	none	none	none	none

VARIABLE	DESCRIPTION
SETID	Identifier for this ID set.

VARIABLE	DESCRIPTION
TYPE	The kind of IDs in this set: EQ.'SEG_SETS': Each ID is a segment set connected with SOLVER. EQ.'CIRCUIT': Each ID is a circuit ID (from *EM cards) EQ.'SURF_PARTS': Each ID is a surface part number (See *MESH_SURFACE_ELEMENT) EQ.'VOL_PARTS': Each ID is a volume part number (See *MESH_VOLUME) EQ.'SURF_ELES': Each ID is a surface element number (See *MESH_SURFACE_ELEMENT)
SOLVER	Name of the solver (MECH, ICFD, CESE, EM, …)
ID1, …	IDs of the TYPE kind.

*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	USE						
Type	I	I						
Default	none	1						
Remarks		1						

Point Cards. Provide as many cards as necessary. This input ends at the next keyword ("*") card

Card	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. Used by *LSO_DOMAIN
USE	Points in this set are used as: EQ.1: Fixed time history points (default) EQ.2: Positions of tracer particles
X, Y, Z	Coordinates of a point. As many points as desired can be specified.

Remarks:

1. For USE = 1, with the ICFD and CESE solvers, the fixed points have to remain inside the fluid mesh or a zero result is returned, while 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. For USE = 2, a massless tracer particle is tracked for the ICFD and CESE solvers using their local velocity field to integrate the position of each particle in time.

*LSO_TIME_SEQUENCE

Purpose: This command provides users with maximum flexibility in specifying exactly what they want to have appear in the output LSO binary database. Each instance of the *LSO_TIME_SEQUENCE command creates a new time sequence with an independent output frequency and duration. Furthermore, while the default domain for each output variable will be the entire mesh on which that variable is defined, at all selected snapshot times, the *LSO_DOMAIN keyword commands can be used to specify that output will only occur on a portion of SOLVER_NAME's mesh, and for a limited time interval, or that it will occur at a set of points (see *LSO_POINT_SET), or over a set of object IDs (see *LSO_ID_SET). Note that for the single-precision version of LS-DYNA, the only allowed value of SOLVER_NAME = MECH.

Card 1	1	2	3	4	5	6	7	8
Variable	SOLVER_NAME							
Type	A							

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		
Remarks	1	1	1	1				

Domain IDs. Provide as many cards as necessary. This input ends at the next keyword ("*") card, or when a global variable name card appears

Card 3	1	2	3	4	5	6	7	8
Variable	DOMID1	DOMID2	DOMID3	DOMID4	DOMID5	DOMID6	DOMID7	DOMID8
Type	I	I	I	I	I	I	I	I
Default	none	none	none	none	none	none	none	none

Global variable names. Provide as many cards as necessary. This input ends at the next keyword ("*") card

Card 4	1	2	3	4	5	6	7	8
Variable	GLOBAL_VAR							
Type	A							

VARIABLE	DESCRIPTION
SOLVER_ NAME	Selects the solver from which data is output in this time sequence. Accepted entries so far are 'MECH', 'EM', 'CESE' and 'ICFD'
DT	Time interval between outputs.
LCDT	Optional load curve ID specifying the time interval between dumps.
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.

VARIABLE	DESCRIPTION
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
DOMID1, …	Output set ID defining the domain over which variable output is to be performed in this time sequence. Each DOMID refers to the domain identifier in an *LSO_DOMAIN keyword card.
GLOBAL_VAR	The name of a global output variable computed by SOLVER_-NAME. This variable must have a single value (scalar, vector, or tensor), and therefore does not depend upon any DOMID. Any number of such variables may be specified with a given time sequence. These variables are listed as having "global" domain for SOLVER_NAME in a separate document. This document (LSO_VARIABLES.TXT) is created by running the command: LS-DYNA print_lso_doc.

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.

*LSO_VARIABLE_GROUP

Purpose: To provide a means of defining a shorthand name for a group of variables. That is, wherever the given group name is used, it is replaced by the list of variables given in this command. Note that for the single-precision version of LS-DYNA, the only allowed value of SOLVER_NAME = MECH.

Card 1	1	2	3	4	5	6	7	8
Variable	SOLVER_NAME							
Type	A							

Card 2	1	2	3	4	5	6	7	8
Variable	DOMAIN_TYPE							
Type	A							

Card 3	1	2	3	4	5	6	7	8
Variable	GROUP_NAME							
Type	A							

List Of Variables In Group. Provide as many cards as necessary. This input ends at the next keyword ("*") card

Card 4	1	2	3	4	5	6	7	8
Variable	VAR_NAME							
Type	A							

VARIABLE	DESCRIPTION
SOLVER_NAME	Selects the solver for which data is output in a time sequence.

VARIABLE	DESCRIPTION
DOMAIN_TYPE	Name of the type of domain on which each VAR_NAME is defined.
GROUP_NAME	Name of (or alias for) the group of names given by the listed VAR_NAMEs
VAR_NAME	The name of an output variable computed by SOLVER_-NAME

Remarks:

1. Valid VAR_NAMEs depend both upon the SOLVER_NAME and the DOMAIN_TYPE. These variables are listed by DOMAIN_TYPE for each SOLVER_NAME in a separate document. This document (LSO_VARIABLES.TXT) is created by running the command: LS-DYNA print_lso_doc.