*DUALCESE

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

DUALCESE_BOUNDARY_AXISYMMETRIC_*{OPTION}

DUALCESE_BOUNDARY_CYCLIC_*{OPTION}

DUALCESE_BOUNDARY_FSI_*{OPTION}

DUALCESE_BOUNDARY_NON_REFLECTIVE_*{OPTION}

DUALCESE_BOUNDARY_PRESCRIBED_*{OPTION}

DUALCESE_BOUNDARY_PRESCRIBED_HYBRID_*{OPTION}

DUALCESE_BOUNDARY_PRESCRIBED_INLET_MACHNUM_*{OPTION}

DUALCESE_BOUNDARY_PRESCRIBED_INLET_MFRATE_*{OPTION}

DUALCESE_BOUNDARY_PRESCRIBED_INLET_PRESSURE_*{OPTION}

DUALCESE_BOUNDARY_PRESCRIBED_OUTLET_EXMFRATE_*{OPTION}

DUALCESE_BOUNDARY_PRESCRIBED_OUTLET_FARFIELD_*{OPTION}

DUALCESE_BOUNDARY_PRESCRIBED_OUTLET_PRESSURE_*{OPTION}

DUALCESE_BOUNDARY_PRESCRIBED_PHASE_CHANGE_*{OPTION}

DUALCESE_BOUNDARY_PRESCRIBED_TWO-PHASE_*{OPTION}

DUALCESE_BOUNDARY_REFLECTIVE_*{OPTION}

DUALCESE_BOUNDARY_SLIDING_*{OPTION}

*DUALCESE_BOUNDARY_SOLID_WALL_*{OPTION1}*_*{OPTION2}*

*DUALCESE_CONTROL_LIMITER

*DUALCESE_CONTROL_MESH_MOV

*DUALCESE_CONTROL_SOLVER (deprecated as of R14)

*DUALCESE_CONTROL_TIMESTEP

*DUALCESE_DARCY-FORCHHEIMER_EQ

*DUALCESE_DATABASE_HISTORY_DRAG_LIFT

*DUALCESE_DATABASE_HISTORY_ELEMENT_SET

*DUALCESE_DATABASE_HISTORY_GLOBALS

*DUALCESE_DATABASE_HISTORY_NODE_SET

*DUALCESE_DATABASE_HISTORY_POINT_SET

*DUALCESE_DATABASE_HISTORY_SEGMENT_SET

*DUALCESE_D3PLOT

*DUALCESE_D3PLOT_FLUID_SSID

*DUALCESE_ELE2D

*DUALCESE_ELE3D

*DUALCESE_ELEMENTSET

*DUALCESE_EOS_CAV_HOMOG_EQUILIB

*DUALCESE_EOS_COCHRAN_CHAN

*DUALCESE_EOS_COOLPROP

*DUALCESE_EOS_IDEAL_GAS

*DUALCESE_EOS_INFLATOR1

*DUALCESE_EOS_INFLATOR2

*DUALCESE_EOS_JWL

*DUALCESE_EOS_NASG

*DUALCESE_EOS_REFPROP

*DUALCESE_EOS_REFPROP_PATH

*DUALCESE_EOS_SET (deprecated as of R14)

*DUALCESE_EOS_STIFFENED_GAS

*DUALCESE_EOS_VAN_DER_WAALS_GENERALIZED

*DUALCESE_FSI_EXCLUDE

*DUALCESE_FSI_EXCLUDE_PARTSET

*DUALCESE_GRAVITY

*DUALCESE_INCLUDE_MODEL

*DUALCESE_INITIAL

DUALCESE_INITIAL_*{OPTION}

*DUALCESE_INITIAL_HYBRID

*DUALCESE_INITIAL_HYBRID_SET

*DUALCESE_INITIAL_PHASE_CHANGE

*DUALCESE_INITIAL_PHASE_CHANGE_SET

*DUALCESE_INITIAL_TWO-PHASE

*DUALCESE_INITIAL_TWO-PHASE_SET

*DUALCESE_MAT_GAS

*DUALCESE_MAT_GAS_0

*DUALCESE_MAT_GAS_2

*DUALCESE_MAT_LIQUID

*DUALCESE_MESH_GEOMETRY

*DUALCESE_MESH_PART

*DUALCESE_MODEL

*DUALCESE_NODE2D

*DUALCESE_NODE3D

*DUALCESE_NODESET

*DUALCESE_PART (deprecated as of R14)

*DUALCESE_PART_MULTIPHASE (deprecated as of R14)

*DUALCESE_POINT_SOURCE

*DUALCESE_POINT_SOURCE_STRUCTNODE

*DUALCESE_POINT_SOURCE_TWO-PHASE

*DUALCESE_POINTSET

*DUALCESE_POROUS_MODEL

*DUALCESE_POROUS_SPECIFY_JUMP

*DUALCESE_REACTION_RATE_IG

*DUALCESE_REACTION_RATE_IG_REDUCED

*DUALCESE_REACTION_RATE_P_DEPEND

*DUALCESE_SEGMENTSET

*DUALCESE_SOLVER_SELECTION

*DUALCESE_SOLVER_CAV_EQNS

*DUALCESE_SOLVER_EULER_EQNS

*DUALCESE_SOLVER_HYBRID_MULTIPHASE

*DUALCESE_SOLVER_NAVIER_STOKES

*DUALCESE_SOLVER_PHASE_CHANGE

*DUALCESE_SOLVER_TWO-PHASE_MULTIPHASE

An additional keyword option TITLE may be appended to the *DUALCESE keywords. If this option is used, then an addition line is read for the DUALCESE card in 80a format which can be used to describe that particular DUALCESE card. At present, the title serves no purpose other than to perhaps lend clarity to input decks.

Dual CESE Keyword Deck Structure:

The structure of the keyword setup for the *DUALCESE solvers is different from the way most keyword input is handled in the LS-DYNA input phase. This stems from the fact that there can be several *DUALCESE models in the same problem. In order to deal with this, each such model is restricted to be specified with one file hierarchy that starts with the keyword file designated with the *DUALCESE_MODEL card.

That keyword file can include any number of other keyword files with the *DUALCESE_-INCLUDE_MODEL card, and each of those files can in turn include other keyword files, again with the *DUALCESE_INCLUDE_MODEL card. Standard *INCLUDE cards are not allowed. In fact, in each file in the file hierarchy of a *DUALCESE_MODEL card, only *DUALCESE cards may be used. The only exception to this is when the fluid mesh is defined using *MESH cards; a fatal error will be encountered when using other non-*DU-ALCESE keywords. Any required non-*DUALCESE keyword cards should be defined in some other place in the keyword input (outside the scope of the *DUALCESE_MODEL keyword card).

The mesh for each dual CESE model must be defined within the keyword input file hierarchy for that model. If a mesh created with *MESH cards is used, the *MESH cards for that mesh must all be defined within the scope of the *DUALCESE_MODEL card.

With R14, we introduce an updated, preferred way of specifying the dual CESE problem. The required or recommended keywords to use are:

*DUALCESE_MESH_GEOMETRY

*DUALCESE_MESH_PART

*DUALCESE_SOLVER_SELECTION

along with one of the following specific solver cards:

*DUALCESE_SOLVER_CAV_EQNS

*DUALCESE_SOLVER_EULER_EQNS

*DUALCESE_SOLVER_HYBRID_MULTIPHASE

*DUALCESE_SOLVER_NAVIER_STOKES

*DUALCESE_SOLVER_PHASE_CHANGE (added in R15)

*DUALCESE_SOLVER_TWO-PHASE_MULTIPHASE

The following keywords are deprecated as of R14:

*DUALCESE_CONTROL_SOLVER

*DUALCESE_EOS_SET

*DUALCESE_PART

*DUALCESE_PART_MULTIPHASE

We also added time history cards with R14 that cause certain output to the binout file:

*DUALCESE_DATABASE_HISTORY_ELEMENT_SET

*DUALCESE_DATABASE_HISTORY_GLOBALS

*DUALCESE_DATABASE_HISTORY_NODE_SET

*DUALCESE_DATABASE_HISTORY_POINT_SET

*DUALCESE_DATABASE_HISTORY_SEGMENT_SET

Another time history-type card is new with R15; it causes drag and lift-related quantities to be output to the binout file from the immersed boundary method (IBM) FSI-type dual CESE solvers.

*DUALCESE_DATABASE_HISTORY_DRAG_LIFT

We support two external equation-of-state (EOS) libraries, REFPROP and COOLPROP, for dual CESE. Including these libraries is complex, so we will clarify how to include them here. Each of them is accessed through a shared library that has to be loaded into LS-DYNA at runtime with *MODULE_LOAD like in the follwing:

```
*MODULE_LOAD
UserA                   DUALCESE REFPROP
<path to the installed REFPROP shared library>
```

Note that this *MODULE_LOAD card must not be given inside a keyword file in the file hierarchy of a *DUALCESE_MODEL card. As discussed above, this is the case for all non-*DUALCESE keyword cards. Note also that since *MODULE is not available in the Windows version of LS-DYNA, this capability cannot be used in that version.

REFPROP and COOLPROP Libraries:

While the REFPROP v10.0 version library and its directory of data sets is provided by ANSYS, COOLPROP libraries are not provided by ANSYS.

For the COOLPROP shared library, you can find the current production version here:

https://sourceforge.net/projects/coolprop/files/CoolProp/6.3.0/shared_library/Linux/64bit/

Multiphase Capabilities:

The multiphase capabilities added for R13 with modifications in R14 all have FSI capabilities, including with structural element failure. This includes the 'hybrid' multiphase, 'two-phase' multiphase, and the cavitation solvers. New with R15 is a 'phase-change' multiphase solver. These keywords are involved with these capabilities:

*DUALCESE_BOUNDARY_PRESCRIBED_HYBRID

*DUALCESE_BOUNDARY_PRESCRIBED_PHASE_CHANGE

*DUALCESE_BOUNDARY_PRESCRIBED_TWO-PHASE

*DUALCESE_EOS_CAV_HOMOG_EQUILIB

*DUALCESE_EOS_COCHRAN_CHAN

*DUALCESE_EOS_JWL

*DUALCESE_EOS_NASG

*DUALCESE_EOS_SET (deprecated as of R14; use one of the *DUALCESE_-SOLVER_... cards instead)

*DUALCESE_EOS_STIFFENED_GAS

*DUALCESE_EOS_VAN_DER_WAALS_GENERALIZED

*DUALCESE_INITIAL_HYBRID

*DUALCESE_INITIAL_HYBRID_SET

*DUALCESE_INITIAL_PHASE_CHANGE

*DUALCESE_INITIAL_PHASE_CHANGE_SET

*DUALCESE_INITIAL_TWO-PHASE

*DUALCESE_INITIAL_TWO-PHASE_SET

*DUALCESE_PART_MULTIPHASE (deprecated as of R14; use *DUALCESE_-MESH_PART instead)

*DUALCESE_REACTION_RATE_IG

*DUALCESE_REACTION_RATE_IG_REDUCED

*DUALCESE_REACTION_RATE_P_DEPEND

*DUALCESE_SOLVER_HYBRID_MULTIPHASE (added in R14)

*DUALCESE_SOLVER_PHASE_CHANGE (added in R15)

*DUALCESE_SOLVER_TWO-PHASE_MULTIPHASE (added in R14)

Flow Inlet and Exit Boundary Conditions:

The following prescribed inlet and prescribed outlet boundary condition cards are new with R14:

DUALCESE_BOUNDARY_PRESCRIBED_INLET_MACHNUM_*{OPTION}

DUALCESE_BOUNDARY_PRESCRIBED_INLET_MFRATE_*{OPTION}

DUALCESE_BOUNDARY_PRESCRIBED_INLET_PRESSURE_*{OPTION}

DUALCESE_BOUNDARY_PRESCRIBED_OUTLET_EXMFRATE_*{OPTION}

DUALCESE_BOUNDARY_PRESCRIBED_OUTLET_FARFIELD_*{OPTION}

DUALCESE_BOUNDARY_PRESCRIBED_OUTLET_PRESSURE_*{OPTION}

Comparison to CESE:

The capabilities implemented in the dual CESE solvers are only a part of what is available in the *CESE solvers, especially those that involve couplings with the *CHEMISTRY and *STOCHASTIC_PARTICLE solvers. Another *CESE capability that is missing in the dual CESE solver is conjugate heat transfer. Some capabilities are only in the newer dual CESE solvers, in particular, the multiphase solvers, the many added EOSes, the binout-based time history capabilities, and the point-source-type fluid injection methods implemented in the single phase CFD solvers and the two-phase multiphase solvers. Also, drag, lift, and related variables are only computed for the Dual CESE IBM FSI solvers (starting with R15).

DUALCESE_BOUNDARY_AXISYMMETRIC_*OPTION

Available options are:

MSURF

SEGMENT_SET

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

The MSURF option should be used when the dual CESE mesh has been created using *MESH cards. The SEGMENT_SET option should be used when *DUALCESE_ELE2D or *DUALCESE_ELE3D cards specify the dual CESE mesh.

Card Summary:

Card 1a. This card is included for the MSURF keyword option. Provide as many cards as necessary. This input ends at the next keyword ("*") card.

MSPID							

Card 1b. This card is included for the SEGMENT_SET keyword option. Provide as many cards as necessary. This input ends at the next keyword ("*") card.

SSID							

Data Card Definitions:

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	MSPID							
Type	I							
Default	none							

VARIABLE	DESCRIPTION
MSPID	Mesh surface part ID that is referenced by *MESH_SURFACE_ELEMENT cards

Set Card. Card 1 used when the SEGMENT_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	SSID							
Type	I							
Default	none							

VARIABLE	DESCRIPTION
SSID	Segment set ID for the segment set created with *DUALCESE_SEGMENTSET

Remarks:

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

DUALCESE_BOUNDARY_CYCLIC_*OPTION

Available options are:

MSURF

SEGMENT_SET

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

The MSURF option should be used when the dual CESE mesh has been created using *MESH cards. The SEGMENT_SET option should be used when *DUALCESE_ELE2D or *DUALCESE_ELE3D cards specify the dual CESE mesh.

Card Summary:

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

Card 1a. This card is included if the MSURF keyword option is used.

MSPID1	MSPID2	CYCTYP					

Card 1b. This card is included if the SEGMENT_SET keyword option is used.

SSID1	SSID2	CYCTYP					

Card 2a. This card is included when CYCTYP = 1.

AXISX1	AXISY1	AXISZ1	DIRX	DIRY	DIRZ	ROTANG	

Card 2b. This card is included when CYCTYP = 2.

TRANSX	TRANSY	TRANSZ					

Data Card Definitions:

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	MSPID1	MSPID2	CYCTYP					
Type	I	I	I					
Default	none	none	0					
Remarks			1, 2					

VARIABLE	DESCRIPTION
MSPID1, MSPID2	Mesh surface part IDs that are referenced by *MESH_SURFACE_ELEMENT cards
CYCTYP	Relationship between the two cyclic boundary condition surfaces: EQ.0: Relationship determined by LS-DYNA (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.

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

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

VARIABLE	DESCRIPTION
SSID1, SSID2	Segment set IDs for the segment sets created with *DUALCESE_-SEGMENTSET
CYCTYP	Relationship between the two cyclic boundary condition surfaces: EQ.0: Relationship determined by LS-DYNA (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.

Rotation Case Card. Additional card 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	

VARIABLE	DESCRIPTION
AXIS[X,Y,Z]1	A point on the axis of rotation for the transformation between the surfaces
DIR[X,Y,Z]	The direction which together with AXIS[X,Y,Z]1 defines the axis of rotation for the transformation between the surfaces
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

Translation Case Card. Additional card 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
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

Remarks:

1. **Unspecified Relationship between Surfaces.** For the case CYCTYP = 0, LS-DYNA examines the geometry of two faces of the two surfaces in order to determine if the surfaces are approximately parallel (CYCTYP = 2) or related through a rotation (CYCTYP = 1). The geometric parameters required are then computed.

2. **MSURF.** For the MSURF option, each mesh surface part must contain the same number of mesh surface elements. The mesh surface elements in each mesh surface part are internally ordered for pairwise matching between the two mesh surface parts.

3. **SEGMENT_SET.** For the SEGMENT_SET option, each segment set must contain the same number segments. The segments in each set are internally ordered for pairwise matching between the two sets.

DUALCESE_BOUNDARY_FSI_*OPTION

Available options are:

MSURF

SEGMENT_SET

Purpose: Define an FSI boundary condition for the moving mesh dual CESE compressible flow solver. This keyword must not be combined with the dual CESE immersed-boundary method FSI solver in the same dual CESE part on the same dual CESE mesh. Doing so will result in an error termination condition.

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

The MSURF option should be used when the dual CESE mesh has been created using *MESH cards. The SEGMENT_SET option should be used when *DUALCESE_ELE2D or *DUALCESE_ELE3D cards specify the dual CESE mesh.

Card Summary:

Card 1a. Include this card for the MSURF keyword option. Provide as many cards as necessary. This input ends at the next keyword ("*") card.

MSPID	REF_P						

Card 1b. Include this card for the SEGMENT_SET keyword option. Provide as many cards as necessary. This input ends at the next keyword ("*") card.

SSID	REF_P						

Data Card Definitions:

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	MSPID	REF_P						
Type	I	F						
Default	none	0.0						

VARIABLE	DESCRIPTION
MSPID	Mesh surface part ID that is referenced by *MESH_SURFACE_ELEMENT cards
REF_P	Ambient/reference pressure of the fluid domain on the side opposite this structural interface to the fluid simulation domain. This ambient pressure only needs to be specified in the case where the FSI structural part(s) connected with this FSI interface are not immersed in the dual CESE mesh. This reference pressure defaults to 0.0 since moving mesh FSI calculations most often involve structures surrounded by the dual CESE mesh, and there is no need for a reference pressure in that case.

Set Card. Card 1 used when the SEGMENT_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	SSID	REF_P						
Type	I	F						
Default	none	0.0						

VARIABLE	DESCRIPTION
SSID	Segment set ID for the segment set created with *DUALCESE_SEGMENTSET

VARIABLE	DESCRIPTION
REF_P	Ambient/reference pressure of the fluid domain on the side opposite this structural interface to the fluid simulation domain. This ambient pressure only needs to be specified in the case where the FSI structural part(s) connected with this FSI interface are not immersed in the dual CESE mesh. This reference pressure defaults to 0.0 since moving mesh FSI calculations most often involve structures surrounded by the dual CESE mesh, and there is no need for a reference pressure in that case..

Remarks:

This boundary condition card is also needed for conjugate heat transfer problems with the moving mesh dual CESE solver. However, the conjugate heat transfer capability is *not* implemented in the dual CESE solver.

DUALCESE_BOUNDARY_NON_REFLECTIVE_*OPTION

Available options are:

MSURF

SEGMENT_SET

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

The MSURF option should be used when the dual CESE mesh has been created using *MESH cards. The SEGMENT_SET option should be used when *DUALCESE_ELE2D or *DUALCESE_ELE3D cards are used to specify the dual CESE mesh.

Card Summary:

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

MSPID	DIRX	DIRY	DIRZ				

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

SSID	DIRX	DIRY	DIRZ				

Data Card Definitions:

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	MSPID	DIRX	DIRY	DIRZ				
Type	I	F	F	F				
Default	none	0.0	0.0	0.0				

VARIABLE	DESCRIPTION
MSPID	Mesh surface part ID that is referenced by *MESH_SURFACE_ELEMENT cards
DIRX, DIRY, DIRZ	If this vector is nonzero, then it is used as the prescribed flow direction.

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 1b	1	2	3	4	5	6	7	8
Variable	SSID	DIRX	DIRY	DIRZ				
Type	I	F	F	F				
Default	none	0.0	0.0	0.0				

VARIABLE	DESCRIPTION
SSID	Segment set ID for the segment set created with *DUALCESE_SEGMENTSET
DIRX, DIRY, DIRZ	If this vector is nonzero, then it is used as the prescribed flow direction.

Remarks:

1. **Boundary Surface Flow.** This boundary condition is usually imposed on an open surface that is far from the main disturbed flow (the further away, the better), meaning the flow on the boundary surface should be almost uniform.

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

DUALCESE_BOUNDARY_PRESCRIBED_*OPTION

Available options include:

MSURF

SEGMENT_SET

Purpose: For the dual 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.

The MSURF option should be used when the dual CESE mesh has been created using *MESH cards. The SEGMENT_SET card should be used when *DUALCESE_ELE2D or *DUALCESE_ELE3D cards specify the dual 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.

Card Summary:

Card 1a. This card is included if the MSURF keyword option is used.

MSPID	IDCOMP	DIRX	DIRY	DIRZ			

Card 1b. This card is included if the SEGMENT_SET keyword option is used.

SSID	IDCOMP	DIRX	DIRY	DIRZ			

Card 2. This card is required.

LC_U	LC_V	LC_W	LC_RHO	LC_P	LC_T		

Card 3. This card is required.

SF_U	SF_V	SF_W	SF_RHO	SF_P	SF_T		

Data Card Definitions:

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	MSPID	IDCOMP	DIRX	DIRY	DIRZ			
Type	I	I	F	F	F			
Default	none	none	0.0	0.0	0.0			

VARIABLE	DESCRIPTION
MSPID	Mesh surface part ID that is referenced by *MESH_SURFACE_ELEMENT cards
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 [Not yet available].
DIRX, DIRY, DIRZ	If this vector is non-zero, then it is used as the prescribed flow direction.

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

Card 1b	1	2	3	4	5	6	7	8
Variable	SSID	IDCOMP	DIRX	DIRY	DIRZ			
Type	I	I	F	F	F			
Default	none	none	0.0	0.0	0.0			

VARIABLE	DESCRIPTION
SSID	ID for the segment set created with *DUALCESE_SEGMENTSET
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 [Not yet available].
DIRX, DIRY, DIRZ	If this vector is non-zero, then it is used as the prescribed flow direction.

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	1	1	1	1	1		

VARIABLE	DESCRIPTION
LC_U	Load curve ID (see *DEFINE_CURVE) to describe the x-component of the velocity as a function of time or function ID (see *DEFINE_FUNCTION) to give the x-component of the velocity as a function of position, velocity, temperature, pressure, and time, f(x, y, z, vx, vy, vz, temp, pres, time). EQ.0: x-component of velocity is a constant with value SF_U. EQ.-1: x-component of velocity is computed by the solver.
LC_V	Load curve ID to describe the y-component of the velocity as a function of time or function ID to give the y-component of the velocity as a function of position, velocity, temperature, pressure, and time, f(x, y, z, vx, vy, vz, temp, pres, time). EQ.0: y-component of velocity is a constant with value SF_V. EQ.-1: y-component of velocity is computed by the solver.
LC_W	Load curve ID to describe the z-component of the velocity as a function of time or function ID to give the z-component of the

VARIABLE	DESCRIPTION
	velocity as a function of position, velocity, temperature, pressure, and time, f(x, y, z, vx, vy, vz, temp, pres, time). EQ.0: z-component of velocity is a constant with value SF_W. EQ.-1: z-component of velocity is computed by the solver.
LC_RHO	Load curve ID to describe the density as a function of time or function ID to give the density as a function of position, velocity, temperature, pressure, and time, f(x, y, z, vx, vy, vz, temp, pres, time). EQ.0: Density is a constant with value SF_RHO. EQ.-1: Density is computed by the solver.
LC_P	Load curve ID to describe the pressure as a function of time or function ID to give the pressure as a function of position, velocity, temperature, pressure, and time, f(x, y, z, vx, vy, vz, temp, pres, time). EQ.0: Pressure is a constant with value SF_P. EQ.-1: Pressure is computed by the solver.
LC_T	Load curve ID to describe the temperature as a function of time or function ID to give the temperature as a function of position, velocity, temperature, pressure, and time, f(x, y, z, vx, vy, vz, temp, pres, time). EQ.0: Temperature is a constant with value SF_T. EQ.-1: Temperature is computed by the solver.

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		

VARIABLE	DESCRIPTION
SF_U	Scale factor for LC_U

VARIABLE	DESCRIPTION
SF_V	Scale factor for LC_V
SF_W	Scale factor for LC_W
SF_RHO	Scale factor for LC_RHO
SF_P	Scale factor for LC_P
SF_T	Scale factor for LC_T

Remarks:

1. **Consistent Boundary Values.** 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 (meaning be such that the solution of the model exists, is unique, and is physical).

DUALCESE_BOUNDARY_PRESCRIBED_HYBRID_*OPTION

Available options include:

MSURF

SEGMENT_SET

Purpose: For the dual CESE compressible flow solver, set boundary values for velocity, density, pressure, temperature, and other values in the hybrid multiphase model. Boundary values are applied at the centroid of elements connected with this boundary.

The MSURF option should be used when the dual CESE mesh has been created using *MESH cards. The SEGMENT_SET card should be used when *DUALCESE_ELE2D or *DUALCESE_ELE3D cards specify the dual 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. Cards 2 and 3 provide load curve IDs.

3. Cards 4 and 5 provide scale factors.

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

Card Summary:

Card 1a. This card is included if the keyword option is set to MSURF.

MSPID	IDCOMP	DIRX	DIRY	DIRZ			

Card 1b. This card is included if the keyword option is set to SSID.

SSID	IDCOMP	DIRX	DIRY	DIRZ			

Card 2. This card is required.

LC_Z1	LC_RA	LC_U	LC_V	LC_W	LC_D1	LC_DA	LC_DB

Card 3. This card is required.

LC_P	LC_T						

Card 4. This card is required.

SF_Z1	SF_RA	SF_U	SF_V	SF_W	SF_D1	SF_DA	SF_DB

Card 5. This card is required.

SF_P	SF_T						

Data Card Definitions:

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

Card 1a	1	2	3	4	5	6	7	8
Variable	MSPID	IDCOMP	DIRX	DIRY	DIRZ			
Type	I	I	F	F	F			
Default	none	none	0.0	0.0	0.0			

VARIABLE	DESCRIPTION
MSPID	Mesh surface part ID that is referenced by *MESH_SURFACE_ELEMENT cards
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 [Not yet available].
DIRX, DIRY, DIRZ	If this vector is nonzero, then it is used as the prescribed flow direction.

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

Card 1b	1	2	3	4	5	6	7	8
Variable	SSID	IDCOMP	DIRX	DIRY	DIRZ			
Type	I	I	F	F	F			
Default	none	none	0.0	0.0	0.0			

VARIABLE	DESCRIPTION
SSID	ID for the segment set created with *DUALCESE_SEGMENTSET
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 [Not yet available].
DIRX, DIRY, DIRZ	If this vector is nonzero, then it is used as the prescribed flow direction.

Load Curve Card. See Remark 1.

Card 2	1	2	3	4	5	6	7	8
Variable	LC_Z1	LC_RA	LC_U	LC_V	LC_W	LC_D1	LC_DA	LC_DB
Type	I	I	I	I	I	I		

VARIABLE	DESCRIPTION
LC_Z1	Load curve ID or function ID to describe the volume fraction of material 1 as a function of time or a function of position, velocity, temperature, pressure, and time, f(x, y, z, vx, vy, vz, temp, pres, time), respectively. EQ.0: The volume fraction is a constant with value SF_Z1. EQ.-1: The volume fraction is computed by the solver.
LC_RA	Load curve or function ID to describe the mass fraction of reactant (material α) with respect to the explosive mixture (material 2) as a

VARIABLE	DESCRIPTION
	function of time or a function of position, velocity, temperature, pressure, and time, f(x, y, z, vx, vy, vz, temp, pres, time), respectively. EQ.0: The mass fraction is a constant with value SF_RA. EQ.-1: The mass fraction is computed by the solver.
LC_U	Load curve or defined function ID to describe the *x*-component of the velocity as a function of time or a function of position, velocity, temperature, pressure, and time, f(x, y, z, vx, vy, vz, temp, pres, time), respectively. EQ.0: The *x*-component of velocity is a constant with value SF_U. EQ.-1: The *x*-component of velocity is computed by the solver.
LC_V	Load curve or defined function ID to describe the *y*-component of the velocity as a function of time or a function of position, velocity, temperature, pressure, and time, f(x, y, z, vx, vy, vz, temp, pres, time), respectively. EQ.0: The *y*-component of velocity is a constant with value SF_V. EQ.-1: The *y*-component of velocity is computed by the solver.
LC_W	Load curve or defined function ID to describe the *z*-component of the velocity as a function of time or a function of position, velocity, temperature, pressure, and time, f(x, y, z, vx, vy, vz, temp, pres, time), respectively. EQ.0: The *z*-component of velocity is a constant with value SF_W. EQ.-1: The *z*-component of velocity is computed by the solver.
LC_D1	Load curve or defined function ID to describe the density of the first multiphase material as a function of time or a function of position, velocity, temperature, pressure, and time, f(x, y, z, vx, vy, vz, temp, pres, time), respectively. EQ.0: The density of the first multiphase material is a constant with value SF_D1. EQ.-1: The density of the first multiphase material is computed by the solver.

VARIABLE	DESCRIPTION
LC_DA	Load curve or defined function ID to describe the density of the reactant (material α) as a function of time or a function of position, velocity, temperature, pressure, and time, f(x, y, z, vx, vy, vz, temp, pres, time), respectively. EQ.0: The density of the reactant is a constant with value SF_DA. EQ.-1: The density of the reactant is computed by the solver.
LC_DB	Load curve or defined function ID to describe the density of the product (material β) as a function of time or a function of position, velocity, temperature, pressure, and time, f(x, y, z, vx, vy, vz, temp, pres, time), respectively. EQ.0: The density of the product is a constant with value SF_DB. EQ.-1: The density of the product is computed by the solver.

Load Curve Card 2. See Remark 1.

Card 3	1	2	3	4	5	6	7	8
Variable	LC_P	LC_T						
Type	I	I						

VARIABLE	DESCRIPTION
LC_P	Load curve or defined function ID to describe the pressure as a function of time or a function of position, velocity, temperature, pressure, and time, f(x, y, z, vx, vy, vz, temp, pres, time), respectively. EQ.0: The pressure is a constant with value SF_P. EQ.-1: The pressure is computed by the solver.
LC_T	Load curve or defined function ID to describe the temperature as a function of time or a function of position, velocity, temperature, pressure, and time, f(x, y, z, vx, vy, vz, temp, pres, time), respectively. EQ.0: The temperature is a constant with value SF_T.

VARIABLE	DESCRIPTION
	EQ.-1: The temperature is computed by the solver.

Scale Factor Card.

Card 4	1	2	3	4	5	6	7	8
Variable	SF_Z1	SF_RA	SF_U	SF_V	SF_W	SF_D1	SF_DA	SF_DB
Type	F	F	F	F	F	F	F	F
Default	1.0	1.0	1.0	1.0	1.0	1.0	1.0	1.0

VARIABLE	DESCRIPTION
SF_Z	Scale factor for LC_Z1
SF_RA	Scale factor for LC_RA
SF_U	Scale factor for LC_U
SF_V	Scale factor for LC_V
SF_W	Scale factor for LC_W
SF_D1	Scale factor for LC_D1
SF_DA	Scale factor for LC_DA
SF_DB	Scale factor for LC_DB

Scale Factor Card 2.

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

VARIABLE	DESCRIPTION
SF_P	Scale factor for LC_P
SF_T	Scale factor for LC_T

Remark:

1. **Consistent Boundary Values.** On each centroid or set of element centroids, the variables ($v_x, v_y, v_z, \rho, P, T, \ldots$) that are given values must be consistent and make the model well-posed (meaning be such that the solution of the model exists, is unique, and is physical).

DUALCESE_BOUNDARY_PRESCRIBED_INLET_MACHNUM_*OPTION

Available options include:

MSURF

SEGMENT_SET

Purpose: For the dual CESE compressible flow solver, set inlet boundary values using total pressure, total temperature, and Mach number. Specifying an arbitrary inlet flow direction is an option. Boundary values are applied at the centroid of elements connected with this boundary.

The MSURF option should be used when the dual CESE mesh has been created using *MESH cards. The SEGMENT_SET card should be used when *DUALCESE_ELE2D or *DUALCESE_ELE3D cards specify the dual 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.

Card Summary:

Card 1a. Include this card if the MSURF keyword option is used.

MSPID	IWALL	DIR_X	DIR_Y	DIR_Z			

Card 1b. Include this card if the SEGMENT_SET keyword option is used.

SSID	IWALL	DIR_X	DIR_Y	DIR_Z			

Card 2. This card is required.

LC_TOTP	LC_TOTT	LC_MACH					

Card 3. This card is required.

SF_TOTP	SF_TOTT	SF_MACH					

Data Card Definitions:

Surface Part Card. Include this card when the MSURF keyword option is active.

Card 1a	1	2	3	4	5	6	7	8
Variable	MSPID	IWALL	DIRX	DIRY	DIRZ			
Type	I	I	F	F	F			
Default	none	0	0.0	0.0	0.0			

VARIABLE	DESCRIPTION
MSPID	Mesh surface part ID that is referenced by *MESH_SURFACE_ELEMENT cards
IWALL	Artificial wall flag: EQ.0: Do not treat as an artificial wall if there is flow recirculation at this inlet boundary. EQ.1: Do treat this inlet boundary as an artificial wall whenever flow recirculation exists there.
DIRX, DIRY, DIRZ	If this vector is nonzero, it is used as the prescribed flow direction.

Set Card. Include this card when the SEGMENT_SET keyword option is active.

Card 1b	1	2	3	4	5	6	7	8
Variable	SSID	IWALL	DIRX	DIRX	DIRY			
Type	I	I	F	F	F			
Default	none	0	0.0	0.0	0.0			

VARIABLE	DESCRIPTION
SSID	ID for the segment set created with *DUALCESE_SEGMENTSET
IWALL	Artificial wall flag: EQ.0: Do not treat as an artificial wall if there is flow recirculation at this inlet boundary. EQ.1: Do treat this inlet boundary as an artificial wall whenever flow recirculation exists there.
DIRX, DIRY, DIRZ	If this vector is nonzero, it is used as the prescribed flow direction.

Load Curve Card.

Card 2	1	2	3	4	5	6	7	8
Variable	LC_TOTP	LC_TOTT	LC_MACH					
Type	I	I	I					
Remarks	1	1	1					

VARIABLE	DESCRIPTION
LC_TOTP	Load curve ID (see *DEFINE_CURVE) to describe the total pressure as a function of time or function ID (see *DEFINE_FUNCTION) to give the total pressure as a function of position, velocity, temperature, pressure, and time, f(x, y, z, vx, vy, vz, temp, pres, time). EQ.0: Total pressure is a constant with value SF_TOTP.
LC_TOTT	Load curve ID (see *DEFINE_CURVE) to describe the total temperature as a function of time or function ID (see *DEFINE_FUNCTION) to give the total temperature as a function of position, velocity, temperature, pressure, and time, f(x, y, z, vx, vy, vz, temp, pres, time). EQ.0: Total temperature is a constant with value SF_TOTT.
LC_MACH	Load curve ID to describe the Mach number as a function of time or function ID to give the Mach number as a function of position,

VARIABLE	DESCRIPTION
	velocity, temperature, pressure, and time, f(x, y, z, vx, vy, vz, temp, pres, time).
	EQ.0: Mach number is a constant with value SF_MACH.

Scale Factor Card.

Card 3	1	2	3	4	5	6	7	8
Variable	SF_TOTP	SF_TOTT	SF_MACH					
Type	F	F	F					
Default	1.0	1.0	1.0					

VARIABLE	DESCRIPTION
SF_TOTP	Scale factor for LC_TOTP
SF_TOTT	Scale factor for LC_TOTT
SF_MACH	Scale factor for LC_MACH

Remarks:

1. **Consistent boundary values.** On each centroid or set of centroids, the variables (P_{total}, T_{total}, Mach number) that are given values must be consistent and make the model well-posed (meaning be such that the solution of the model exists, is unique, and is physical).

DUALCESE_BOUNDARY_PRESCRIBED_INLET_MFRATE_*OPTION

Available options include:

MSURF

SEGMENT_SET

Purpose: For the dual CESE compressible flow solver, set inlet boundary values using total pressure, total temperature, and mass flow rate. Specifying an arbitrary inlet flow direction is an option. Boundary values are applied at the centroid of elements connected with this boundary.

The MSURF option should be used when the dual CESE mesh has been created using *MESH cards. The SEGMENT_SET card should be used when *DUALCESE_ELE2D or *DUALCESE_ELE3D cards specify the dual 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.

Card Summary:

Card 1a. Include this card if the MSURF keyword option is used.

MSPID	IWALL	DIR_X	DIR_Y	DIR_Z			

Card 1b. Include this card if the SEGMENT_SET keyword option is used.

SSID	IWALL	DIR_X	DIR_Y	DIR_Z			

Card 2. This card is required.

LC_TOTP	LC_TOTT	LC_MFRT					

Card 3. This card is required.

SF_TOTP	SF_TOTT	SF_MFRT					

Data Card Definitions:

Surface Part Card. Include this card when the MSURF keyword option is active.

Card 1a	1	2	3	4	5	6	7	8
Variable	MSPID	IWALL	DIRX	DIRY	DIRZ			
Type	I	I	F	F	F			
Default	none	0	0.0	0.0	0.0			

VARIABLE	DESCRIPTION
MSPID	Mesh surface part ID that is referenced by *MESH_SURFACE_ELEMENT cards
IWALL	Artificial wall flag: EQ.0: Do not treat as an artificial wall if there is flow recirculation at this inlet boundary. EQ.1: Do treat this inlet boundary as an artificial wall whenever flow recirculation exists there.
DIRX, DIRY, DIRZ	If this vector is nonzero, it is used as the prescribed flow direction.

Set Card. Include this card when the SEGMENT_SET keyword option is active.

Card 1b	1	2	3	4	5	6	7	8
Variable	SSID	IWALL	DIRX	DIRX	DIRY			
Type	I	I	F	F	F			
Default	none	0	0.0	0.0	0.0			

VARIABLE	DESCRIPTION
SSID	ID for the segment set created with *DUALCESE_SEGMENTSET
IWALL	Artificial wall flag: EQ.0: Do not treat as an artificial wall if there is flow recirculation at this inlet boundary. EQ.1: Do treat this inlet boundary as an artificial wall whenever flow recirculation exists there.
DIRX, DIRY, DIRZ	If this vector is nonzero, it is used as the prescribed flow direction.

Load Curve Card.

Card 2	1	2	3	4	5	6	7	8
Variable	LC_TOTP	LC_TOTT	LC_MFRT					
Type	I	I	I					
Remarks	1	1	1					

VARIABLE	DESCRIPTION
LC_TOTP	Load curve ID (see *DEFINE_CURVE) to describe the total pressure as a function of time or function ID (see *DEFINE_FUNCTION) to give the total pressure as a function of position, velocity, temperature, pressure, and time, f(x, y, z, vx, vy, vz, temp, pres, time). EQ.0: Total pressure is a constant with value SF_TOTP.
LC_TOTT	Load curve ID (see *DEFINE_CURVE) to describe the total temperature as a function of time or function ID (see *DEFINE_FUNCTION) to give the total temperature as a function of position, velocity, temperature, pressure, and time, f(x, y, z, vx, vy, vz, temp, pres, time). EQ.0: Total temperature is a constant with value SF_TOTT.
LC_MFRT	Load curve ID to describe the mass flow rate as a function of time or function ID to give the mass flow rate as a function of position,

VARIABLE	DESCRIPTION
	velocity, temperature, pressure, and time, f(x, y, z, vx, vy, vz, temp, pres, time). EQ.0: Mass flow rate is a constant with value SF_MFRT.

Scale Factor Card.

Card 3	1	2	3	4	5	6	7	8
Variable	SF_TOTP	SF_TOTT	SF_MFRT					
Type	F	F	F					
Default	1.0	1.0	1.0					

VARIABLE	DESCRIPTION
SF_TOTP	Scale factor for LC_TOTP
SF_TOTT	Scale factor for LC_TOTT
SF_MFRT	Scale factor for LC_MFRT

Remarks:

1. **Consistent boundary values.** On each centroid or set of centroids, the variables (P_{total}, T_{total}, mass flow rate) that are given values must be consistent and make the model well-posed (meaning be such that the solution of the model exists, is unique, and is physical).

DUALCESE_BOUNDARY_PRESCRIBED_INLET_PRESSURE_*OPTION

Available options include:

MSURF

SEGMENT_SET

Purpose: For the dual CESE compressible flow solver, set inlet boundary values using total pressure, total temperature, and static pressure. Specifying an arbitrary inlet flow direction is an option. Boundary values are applied at the centroid of elements connected with this boundary.

The MSURF option should be used when the dual CESE mesh has been created using *MESH cards. The SEGMENT_SET card should be used when *DUALCESE_ELE2D or *DUALCESE_ELE3D cards specify the dual 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.

Card Summary:

Card 1a. Include this card if the MSURF keyword option is used.

MSPID	IWALL	DIR_X	DIR_Y	DIR_Z			

Card 1b. Include this card if the SEGMENT_SET keyword option is used.

SSID	IWALL	DIR_X	DIR_Y	DIR_Z			

Card 2. This card is required.

LC_TOTP	LC_TOTT	LC_PSTAT					

Card 3. This card is required.

SF_TOTP	SF_TOTT	SF_PSTAT					

Data Card Definitions:

Surface Part Card. Include this card when the MSURF keyword option is active.

Card 1a	1	2	3	4	5	6	7	8
Variable	MSPID	IWALL	DIRX	DIRY	DIRZ			
Type	I	I	F	F	F			
Default	none	0	0.0	0.0	0.0			

VARIABLE	DESCRIPTION
MSPID	Mesh surface part ID that is referenced by *MESH_SURFACE_ELEMENT cards
IWALL	Artificial wall flag: EQ.0: Do not treat as an artificial wall if there is flow recirculation at this inlet boundary. EQ.1: Do treat this inlet boundary as an artificial wall whenever flow recirculation exists there.
DIRX, DIRY, DIRZ	If this vector is nonzero, it is used as the prescribed flow direction.

Set Card. Include this card when the SEGMENT_SET keyword option is active.

Card 1b	1	2	3	4	5	6	7	8
Variable	SSID	IWALL	DIRX	DIRX	DIRY			
Type	I	I	F	F	F			
Default	none	0	0.0	0.0	0.0			

VARIABLE	DESCRIPTION
SSID	ID for the segment set created with *DUALCESE_SEGMENTSET
IWALL	Artificial wall flag: EQ.0: Do not treat as an artificial wall if there is flow recirculation at this inlet boundary. EQ.1: Do treat this inlet boundary as an artificial wall whenever there is flow recirculation there.
DIRX, DIRY, DIRZ	If this vector is nonzero, it is used as the prescribed flow direction.

Load Curve Card.

Card 2	1	2	3	4	5	6	7	8
Variable	LC_TOTP	LC_TOTT	LC_PSTAT					
Type	I	I	I					
Remarks	1	1	1					

VARIABLE	DESCRIPTION
LC_TOTP	Load curve ID (see *DEFINE_CURVE) to describe the total pressure as a function of time or function ID (see *DEFINE_FUNCTION) to give the total pressure as a function of position, velocity, temperature, pressure, and time, f(x, y, z, vx, vy, vz, temp, pres, time). EQ.0: Total pressure is a constant with value SF_TOTP.
LC_TOTT	Load curve ID (see *DEFINE_CURVE) to describe the total temperature as a function of time or function ID (see *DEFINE_FUNCTION) to give the total temperature as a function of position, velocity, temperature, pressure, and time, f(x, y, z, vx, vy, vz, temp, pres, time). EQ.0: Total temperature is a constant with value SF_TOTT.
LC_PSTAT	Load curve ID to describe the static pressure as a function of time or function ID to give the static pressure as a function of position,

VARIABLE	DESCRIPTION
	velocity, temperature, pressure, and time, f(x, y, z, vx, vy, vz, temp, pres, time). EQ.0: Static pressure is a constant with value SF_PSTAT.

Scale Factor Card.

Card 3	1	2	3	4	5	6	7	8
Variable	SF_TOTP	SF_TOTT	SF_PSTAT					
Type	F	F	F					
Default	1.0	1.0	1.0					

VARIABLE	DESCRIPTION
SF_TOTP	Scale factor for LC_TOTP
SF_TOTT	Scale factor for LC_TOTT
SF_PSTAT	Scale factor for LC_PSTAT

Remarks:

1. **Consistent boundary values.** On each centroid or set of centroids, the variables (P_{total}, T_{total}, P_{static}) that are given values must be consistent and make the model well-posed (meaning be such that the solution of the model exists, is unique, and is physical).

DUALCESE_BOUNDARY_PRESCRIBED_OUTLET_EXMFRATE_*OPTION

Available options include:

MSURF

SEGMENT_SET

Purpose: For the dual CESE compressible flow solver, set outlet boundary values using reference pressure, reference temperature, and exit mass flow rate. Specifying an arbitrary outlet flow direction is an option. Boundary values are applied at the centroid of elements connected with this boundary.

The MSURF option should be used when the dual CESE mesh has been created using *MESH cards. The SEGMENT_SET card should be used when *DUALCESE_ELE2D or *DUALCESE_ELE3D cards specify the dual 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.

Card Summary:

Card 1a. Include this card if the MSURF keyword option is used.

MSPID	IWALL	DIR_X	DIR_Y	DIR_Z			

Card 1b. Include this card if the SEGMENT_SET keyword option is used.

SSID	IWALL	DIR_X	DIR_Y	DIR_Z			

Card 2. This card is required.

LC_PREF	LC_TREF	LC_MFRT					

Card 3. This card is required.

SF_PREF	SF_TREF	SF_MFRT					

Data Card Definitions:

Surface Part Card. Include this card when the MSURF keyword option is active.

Card 1a	1	2	3	4	5	6	7	8
Variable	MSPID	IWALL	DIRX	DIRY	DIRZ			
Type	I	I	F	F	F			
Default	none	0	0.0	0.0	0.0			

VARIABLE	DESCRIPTION
MSPID	Mesh surface part ID that is referenced by *MESH_SURFACE_ELEMENT cards
IWALL	Artificial wall flag: EQ.0: Do not treat as an artificial wall if there is flow recirculation at this outlet boundary. EQ.1: Do treat this outlet boundary as an artificial wall whenever flow recirculation exists there.
DIRX, DIRY, DIRZ	If this vector is nonzero, it is used as the prescribed flow direction.

Set Card. Include this card when the SEGMENT_SET keyword option is active.

Card 1b	1	2	3	4	5	6	7	8
Variable	SSID	IWALL	DIRX	DIRX	DIRY			
Type	I	I	F	F	F			
Default	none	0	0.0	0.0	0.0			

VARIABLE	DESCRIPTION
SSID	ID for the segment set created with *DUALCESE_SEGMENTSET
IWALL	Artificial wall flag: EQ.0: Do not treat as an artificial wall if there is flow recirculation at this outlet boundary. EQ.1: Do treat this outlet boundary as an artificial wall whenever flow recirculation exists there.
DIRX, DIRY, DIRZ	If this vector is nonzero, it is used as the prescribed flow direction.

Load Curve Card.

Card 2	1	2	3	4	5	6	7	8
Variable	LC_PREF	LC_TREF	LC_MFRT					
Type	I	I	I					
Remarks	1	1	1					

VARIABLE	DESCRIPTION
LC_PREF	Load curve ID (see *DEFINE_CURVE) to describe the reference pressure as a function of time or function ID (see *DEFINE_FUNCTION) to give the reference pressure as a function of position, velocity, temperature, pressure, and time, f(x, y, z, vx, vy, vz, temp, pres, time). EQ.0: Reference pressure is a constant with value SF_PREF.
LC_TREF	Load curve ID (see *DEFINE_CURVE) to describe the reference temperature as a function of time or function ID (see *DEFINE_FUNCTION) to give the reference temperature as a function of position, velocity, temperature, pressure, and time, f(x, y, z, vx, vy, vz, temp, pres, time). EQ.0: Reference temperature is a constant with value SF_TREF.
LC_MFRT	Load curve ID to describe the exit mass flow rate as a function of time or function ID to give the exit mass flow rate as a function of

VARIABLE	DESCRIPTION
	position, velocity, temperature, pressure, and time, f(x, y, z, vx, vy, vz, temp, pres, time). EQ.0: exit mass flow rate is a constant with value SF_MFRT.

Scale Factor Card.

Card 3	1	2	3	4	5	6	7	8
Variable	SF_PREF	SF_TREF	SF_MFRT					
Type	F	F	F					
Default	1.0	1.0	1.0					

VARIABLE	DESCRIPTION
SF_PREF	Scale factor for LC_PREF
SF_TREF	Scale factor for LC_TREF
SF_MFRT	Scale factor for LC_MFRT

Remarks:

1. **Consistent boundary values.** On each centroid or set of centroids, the variables (P_{ref}, T_{ref}, Exit Mass Flow Rate) that are given values must be consistent and make the model well-posed (meaning be such that the solution of the model exists, is unique, and is physical).

DUALCESE_BOUNDARY_PRESCRIBED_OUTLET_FARFIELD_*OPTION

Available options include:

MSURF

SEGMENT_SET

Purpose: For the dual CESE compressible flow solver, set outlet boundary values using static pressure, static temperature, and Mach number. Specifying an arbitrary outlet flow direction is an option. Boundary values are applied at the centroid of elements connected with this boundary.

The MSURF option should be used when the dual CESE mesh has been created using *MESH cards. The SEGMENT_SET card should be used when *DUALCESE_ELE2D or *DUALCESE_ELE3D cards specify the dual 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.

Card Summary:

Card 1a. Include this card if the MSURF keyword option is used.

MSPID	IWALL	DIR_X	DIR_Y	DIR_Z			

Card 1b. Include this card if the SEGMENT_SET keyword option is used.

SSID	IWALL	DIR_X	DIR_Y	DIR_Z			

Card 2. This card is required.

LC_PSTAT	LC_TSTAT	LC_MACH					

Card 3. This card is required.

SF_PSTAT	SF_TSTAT	SF_MACH					

Data Card Definitions:

Surface Part Card. Include this card when the MSURF keyword option is active.

Card 1a	1	2	3	4	5	6	7	8
Variable	MSPID	IWALL	DIRX	DIRY	DIRZ			
Type	I	I	F	F	F			
Default	none	0	0.0	0.0	0.0			

VARIABLE	DESCRIPTION
MSPID	Mesh surface part ID that is referenced by *MESH_SURFACE_ELEMENT cards
IWALL	Artificial wall flag: EQ.0: Do not treat as an artificial wall if there is flow recirculation at this outlet boundary. EQ.1: Do treat this outlet boundary as an artificial wall whenever flow recirculation exists there.
DIRX, DIRY, DIRZ	If this vector is nonzero, it is used as the prescribed flow direction.

Set Card. Include this card when the SEGMENT_SET keyword option is active.

Card 1b	1	2	3	4	5	6	7	8
Variable	SSID	IWALL	DIRX	DIRX	DIRY			
Type	I	I	F	F	F			
Default	none	0	0.0	0.0	0.0			

VARIABLE	DESCRIPTION
SSID	ID for the segment set created with *DUALCESE_SEGMENTSET
IWALL	Artificial wall flag: EQ.0: Do not treat as an artificial wall if there is flow recirculation at this outlet boundary. EQ.1: Do treat this outlet boundary as an artificial wall whenever flow recirculation exists there.
DIRX, DIRY, DIRZ	If this vector is nonzero, it is used as the prescribed flow direction.

Load Curve Card.

Card 2	1	2	3	4	5	6	7	8
Variable	LC_PSTAT	LC_TSTAT	LC_MACH					
Type	I	I	I					
Remarks	1	1	1					

VARIABLE	DESCRIPTION
LC_PSTAT	Load curve ID (see *DEFINE_CURVE) to describe the static pressure as a function of time or function ID (see *DEFINE_FUNCTION) to give the static pressure as a function of position, velocity, temperature, pressure, and time, f(x, y, z, vx, vy, vz, temp, pres, time). EQ.0: Static pressure is a constant with value SF_PSTAT.
LC_TSTAT	Load curve ID (see *DEFINE_CURVE) to describe the static temperature as a function of time or function ID (see *DEFINE_FUNCTION) to give the static temperature as a function of position, velocity, temperature, pressure, and time, f(x, y, z, vx, vy, vz, temp, pres, time). EQ.0: Static temperature is a constant with value SF_TSTAT.
LC_MACH	Load curve ID to describe the Mach number as a function of time or function ID to give the Mach number as a function of position,

VARIABLE	DESCRIPTION
	velocity, temperature, pressure, and time, f(x, y, z, vx, vy, vz, temp, pres, time). EQ.0: Mach number is a constant with value SF_MACH.

Scale Factor Card.

Card 3	1	2	3	4	5	6	7	8
Variable	SF_PSTAT	SF_TSTAT	SF_MACH					
Type	F	F	F					
Default	1.0	1.0	1.0					

VARIABLE	DESCRIPTION
SF_PSTAT	Scale factor for LC_PSTAT
SF_TSTAT	Scale factor for LC_TSTAT
SF_MACH	Scale factor for LC_MACH

Remarks:

1. **Consistent boundary values.** On each centroid or set of centroids, the variables (P_{static}, T_{static}, Mach number) that are given values must be consistent and make the model well-posed (meaning be such that the solution of the model exists, is unique, and is physical).

DUALCESE_BOUNDARY_PRESCRIBED_OUTLET_PRESSURE_*OPTION

Available options include:

MSURF

SEGMENT_SET

Purpose: For the dual CESE compressible flow solver, set outlet boundary values using static pressure. Specifying an arbitrary outlet flow direction is an option. Boundary values are applied at the centroid of elements connected with this boundary.

The MSURF option should be used when the dual CESE mesh has been created using *MESH cards. The SEGMENT_SET card should be used when *DUALCESE_ELE2D or *DUALCESE_ELE3D cards specify the dual 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.

Card Summary:

Card 1a. Include this card if the MSURF keyword option is used.

MSPID	IWALL	DIR_X	DIR_Y	DIR_Z			

Card 1b. Include this card if the SEGMENT_SET keyword option is used.

SSID	IWALL	DIR_X	DIR_Y	DIR_Z			

Card 2. This card is required.

LC_PSTAT							

Card 3. This card is required.

SF_PSTAT							

Data Card Definitions:

Surface Part Card. Include this card when the MSURF keyword option is active.

Card 1a	1	2	3	4	5	6	7	8
Variable	MSPID	IWALL	DIRX	DIRY	DIRZ			
Type	I	I	F	F	F			
Default	none	0	0.0	0.0	0.0			

VARIABLE	DESCRIPTION
MSPID	Mesh surface part ID that is referenced by *MESH_SURFACE_ELEMENT cards
IWALL	Artificial wall flag: EQ.0: Do not treat as an artificial wall if there is flow recirculation at this outlet boundary. EQ.1: Do treat this outlet boundary as an artificial wall whenever flow recirculation exists there.
DIRX, DIRY, DIRZ	If this vector is nonzero, it is used as the prescribed flow direction.

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

Card 1b	1	2	3	4	5	6	7	8
Variable	SSID	IWALL	DIRX	DIRX	DIRY			
Type	I	I	F	F	F			
Default	none	0	0.0	0.0	0.0			

VARIABLE	DESCRIPTION
SSID	ID for the segment set created with *DUALCESE_SEGMENTSET
IWALL	Artificial wall flag: EQ.0: Do not treat as an artificial wall if there is flow recirculation at this outlet boundary. EQ.1: Do treat this outlet boundary as an artificial wall whenever flow recirculation exists there.
DIRX, DIRY, DIRZ	If this vector is nonzero, it is used as the prescribed flow direction.

Card 2	1	2	3	4	5	6	7	8
Variable	LC_PSTAT							
Type	I							
Remarks	1							

VARIABLE	DESCRIPTION
LC_PSTAT	Load curve ID (see *DEFINE_CURVE) to describe the total pressure as a function of time or function ID (see *DEFINE_FUNCTION) to give the total pressure as a function of position, velocity, temperature, pressure, and time, f(x, y, z, vx, vy, vz, temp, pres, time). EQ.0: Total pressure is a constant with value SF_PSTAT.

Card 3	1	2	3	4	5	6	7	8
Variable	SF_PSTAT							
Type	F							
Default	1.0							

VARIABLE	DESCRIPTION
SF_PSTAT	Scale factor for LC_PSTAT

Remarks:

1. **Consistent boundary values.** On each centroid or set of centroids, the variable (P_{static}) that is given a value must be consistent and make the model well-posed (meaning be such that the solution of the model exists, is unique, and is physical).

DUALCESE_BOUNDARY_PRESCRIBED_PHASE_CHANGE_*OPTION

Available options are:

MSURF

SEGMENT_SET

Purpose: For the dual CESE compressible flow solver, set boundary values for velocity, density, pressure, temperature, and other values in the phase-change multiphase model. Boundary values are applied at the centroid of outside elements connected with this boundary.

The MSURF option should be used when the dual CESE mesh has been created using *MESH cards. The SEGMENT_SET option card should be used when *DUALCESE_-ELE2D or *DUALCESE_ELE3D cards specify the dual 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. Cards 2 and 3 provide load curve IDs.

3. Cards 4 and 5 provide scale factors.

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

Card Summary:

Card 1a. This card is included for the MSURF keyword option.

MSPID		DIRX	DIRY	DIRZ			

Card 1b. This card is included for the SEGMENT_SET keyword option.

SSID		DIRX	DIRY	DIRZ			

Card 2. This card is required.

LCIDU	LCIDV	LCIDW	LCIDRHO	LCIDP	LCIDT	LCIDY1	LCIDY2

Card 3. This card is required.

LCIDY3							

Card 4. This card is required.

SFU	SFV	SFW	SFRHO	SFP	SFT	SFY1	SFY2

Card 5. This card is required.

SFY3							

Data Card Definitions:

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

Card 1a	1	2	3	4	5	6	7	8
Variable	MSPID		DIRX	DIRY	DIRZ			
Type	I		F	F	F			
Default	none		0.0	0.0	0.0			

VARIABLE	DESCRIPTION
MSPID	Mesh surface part ID that is referenced by *MESH_SURFACE_ELEMENT cards
DIRX, DIRY, DIRZ	If nonzero, vector giving the prescribed flow direction

Set Card. Card 1 used when the SEGMENT_SET keyword option is active.

Card 1b	1	2	3	4	5	6	7	8
Variable	SSID		DIRX	DIRY	DIRZ			
Type	I		F	F	F			
Default	none		0.0	0.0	0.0			

VARIABLE	DESCRIPTION
SSID	Segment set ID for the segment set created with *DUALCESE_-SEGMENTSET
DIRX, DIRY, DIRZ	If nonzero, vector giving the prescribed flow direction

Card 2	1	2	3	4	5	6	7	8
Variable	LCIDU	LCIDV	LCIDW	LCIDRHO	LCIDP	LCIDT	LCIDY1	LCIDY2
Type	I	I	I	I	I	I	I	I
Default	0	0	0	0	0	0	0	0
Remarks	1	1	1	1	1	1	1	1

VARIABLE	DESCRIPTION
LCIDU	Load curve or defined function ID to describe the *xx*-component of the velocity as a function of time or a function of position, velocity, temperature, pressure, and time, f(x, y, z, vx, vy, vz, temp, pres, time), respectively. EQ.0: The *x*-component of velocity is a constant with value SFU. EQ.-1: The *x*-component of velocity is computed by the solver.
LCIDV	Load curve or defined function ID to describe the *yy*-component of the velocity as a function of time or a function of position, velocity, temperature, pressure, and time, f(x, y, z, vx, vy, vz, temp, pres, time), respectively. EQ.0: The *y*-component of velocity is a constant with value SFV. EQ.-1: The *y*-component of velocity is computed by the solver.
LCIDW	Load curve or defined function ID to describe the *zz*-component of the velocity as a function of time or a function of position, velocity, temperature, pressure, and time, f(x, y, z, vx, vy, vz, temp, pres, time), respectively.

VARIABLE	DESCRIPTION
	EQ.0: The *z*-component of velocity is a constant with value SFW. EQ.-1: The *z*-component of velocity is computed by the solver.
LCIDRHO	Load curve or defined function ID to describe the mixture density as a function of time or a function of position, velocity, temperature, pressure, and time, f(x, y, z, vx, vy, vz, temp, pres, time), respectively. EQ.0: The density is a constant with value SFRHO. EQ.-1: The density is computed by the solver.
LCIDP	Load curve or defined function ID to describe the pressure as a function of time or a function of position, velocity, temperature, pressure, and time, f(x, y, z, vx, vy, vz, temp, pres, time), respectively. EQ.0: The pressure is a constant with value SFP. EQ.-1: The pressure is computed by the solver.
LCIDT	Load curve or defined function ID to describe the temperature as a function of time or a function of position, velocity, temperature, pressure, and time, f(x, y, z, vx, vy, vz, temp, pres, time), respectively. EQ.0: The temperature is a constant with value SFT. EQ.-1: The temperature is computed by the solver.
LCIDY1	Load curve or defined function ID to describe the mass fraction of material 1 as a function of time or a function of position, velocity, temperature, pressure, and time, f(x, y, z, vx, vy, vz, temp, pres, time), respectively. EQ.0: The mass fraction is a constant with value SFY1. EQ.-1: The mass fraction is computed by the solver.
LCIDY2	Load curve or defined function ID to describe the mass fraction of material 2 as a function of time or a function of position, velocity, temperature, pressure, and time, f(x, y, z, vx, vy, vz, temp, pres, time), respectively. EQ.0: The mass fraction is a constant with value SFY2. EQ.-1: The mass fraction is computed by the solver.

Card 3	1	2	3	4	5	6	7	8
Variable	LCIDY3							
Type	I							
Default	0							
Remarks	1							

VARIABLE	DESCRIPTION
LCIDY3	Load curve or defined function ID to describe the mass fraction of material 3 as a function of time or a function of position, velocity, temperature, pressure, and time, f(x, y, z, vx, vy, vz, temp, pres, time), respectively. EQ.0: The mass fraction is a constant with value SFY3. EQ.-1: The mass fraction is computed by the solver.

Card 4	1	2	3	4	5	6	7	8
Variable	SFU	SFV	SFW	SFRHO	SFP	SFT	SFY1	SFY2
Type	F	F	F	F	F	F	F	F
Default	1.0	1.0	1.0	1.0	1.0	1.0	1.0	1.0

VARIABLE	DESCRIPTION
SFU	Scale factor for LCIDU
SFV	Scale factor for LCIDV
SFW	Scale factor for LCIDW
SFRHO	Scale factor for LCIDRHO
SFP	Scale factor for LCIDP

VARIABLE	DESCRIPTION
SFT	Scale factor for LCIDT
SFY1	Scale factor for LCIDY1
SFY2	Scale factor for LCIDY2

Card 5	1	2	3	4	5	6	7	8
Variable	SFY3							
Type	F							
Default	1.0							

VARIABLE	DESCRIPTION
SFY3	Scale factor for LCIDY3

Remarks:

1. **Consistent Boundary Values.** On each centroid or set of element centroids, the variables $(v_x, v_y, v_z, \rho, P, T, \ldots)$ that are given values must be consistent and make the model well-posed (meaning be such that the solution of the model exists, is unique, and is physical).

DUALCESE_BOUNDARY_PRESCRIBED_TWO-PHASE_*OPTION

Available options include:

MSURF

SEGMENT_SET

Purpose: For the dual CESE compressible flow solver, set boundary values for velocity, density, pressure, temperature, and other values in the two-phase multiphase model. Boundary values are applied at the centroid of elements connected with this boundary. OPTION = SEGMENT_SET is for user defined meshes whereas OPTION = MSURF is associated with the automatic volume mesher (See *MESH keywords).

That is, the MSURF option is used when the dual CESE mesh has been created using *MESH cards. The SEGMENT_SET card is used when *DUALCESE_ELE2D or *DUALCESE_ELE3D cards are used to define the dual 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 provides load curve IDs.

3. Card 3 provides scale factors.

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

Card Summary:

Card 1a. This card is included if the keyword option is set to MSURF.

MSPID	IDCOMP	DIRX	DIRY	DIRZ			

Card 1b. This card is included if the keyword option is set to SSID.

SSID	IDCOMP	DIRX	DIRY	DIRZ			

Card 2. This card is required.

LC_Z1	LC_U	LC_V	LC_W	LC_D1	LC_D2	LC_P	LC_T

Card 3. This card is required.

SF_Z1	SF_U	SF_V	SF_W	SF_D1	SF_D2	SF_P	SF_T

Data Card Definitions:

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

Card 1a	1	2	3	4	5	6	7	8
Variable	MSPID	IDCOMP	DIRX	DIRY	DIRZ			
Type	I	I	F	F	F			
Default	none	none	0.0	0.0	0.0			

VARIABLE	DESCRIPTION
MSPID	Mesh surface part ID that is referenced by *MESH_SURFACE_ELEMENT cards
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 [Not yet available].
DIRX, DIRY, DIRZ	If this vector is nonzero, then it is used as the prescribed flow direction.

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

Card 1b	1	2	3	4	5	6	7	8
Variable	SSID	IDCOMP	DIRX	DIRY	DIRZ			
Type	I	I	F	F	F			
Default	none	none	0.0	0.0	0.0			

VARIABLE	DESCRIPTION
SSID	ID for the segment set created with *DUALCESE_SEGMENTSET
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 [Not yet available].
DIRX, DIRY, DIRZ	If this vector is nonzero, then it is used as the prescribed flow direction.

Load Curve Card. See Remark 1.

Card 2	1	2	3	4	5	6	7	8
Variable	LC_Z1	LC_U	LC_V	LC_W	LC_D1	LC_D2	LC_P	LC_T
Type	I	I	I	I	I	I	I	I

VARIABLE	DESCRIPTION
LC_Z1	Load curve or defined function ID to describe the volume fraction of material 1 as a function of time or a function of position, velocity, temperature, pressure, and time, f(x, y, z, vx, vy, vz, temp, pres, time), respectively. EQ.0: The volume fraction is a constant with value SF_Z1. EQ.-1: The volume fraction is computed by the solver.
LC_U	Load curve or defined function ID to describe the x-component of the velocity as a function of time or a function of position, velocity, temperature, pressure, and time, f(x, y, z, vx, vy, vz, temp, pres, time), respectively. EQ.0: The x-component of velocity is a constant with value SF_U. EQ.-1: The x-component of velocity is computed by the solver.
LC_V	Load curve or defined function ID to describe the y-component of the velocity as a function of time or a function of position, velocity, temperature, pressure, and time, f(x, y, z, vx, vy, vz, temp, pres, time), respectively.

VARIABLE	DESCRIPTION
	EQ.0: The y-component of velocity is a constant with value SF_-V. EQ.-1: The y-component of velocity is computed by the solver.
LC_W	Load curve or defined function ID to describe the z-component of the velocity as a function of time or a function of position, velocity, temperature, pressure, and time, f(x, y, z, vx, vy, vz, temp, pres, time), respectively. EQ.0: The z-component of velocity is a constant with value SF_-W. EQ.-1: The z-component of velocity is computed by the solver.
LC_D1	Load curve or defined function ID to describe the density of material 1 as a function of time or a function of position, velocity, temperature, pressure, and time, f(x, y, z, vx, vy, vz, temp, pres, time), respectively. EQ.0: The density is a constant with value SF_D1. EQ.-1: The density is computed by the solver.
LC_D2	Load curve or defined function ID to describe the density of material 2 as a function of time or a function of position, velocity, temperature, pressure, and time, f(x, y, z, vx, vy, vz, temp, pres, time), respectively. EQ.0: The density is a constant with value SF_D2. EQ.-1: The density is computed by the solver.
LC_P	Load curve or defined function ID to describe the pressure as a function of time or a function of position, velocity, temperature, pressure, and time, f(x, y, z, vx, vy, vz, temp, pres, time), respectively. EQ.0: The pressure is a constant with value SF_P. EQ.-1: The pressure is computed by the solver.
LC_T	Load curve or defined function ID to describe the temperature as a function of time or a function of position, velocity, temperature, pressure, and time, f(x, y, z, vx, vy, vz, temp, pres, time), respectively. EQ.0: The temperature is a constant with value SF_T.

VARIABLE	DESCRIPTION
	EQ.-1: The temperature is computed by the solver.

Scale Factor Card.

Card 3	1	2	3	4	5	6	7	8
Variable	SF_Z1	SF_U	SF_V	SF_W	SF_D1	SF_D2	SF_P	SF_T
Type	F	F	F	F	F	F	F	F
Default	1.0	1.0	1.0	1.0	1.0	1.0	1.0	1.0

VARIABLE	DESCRIPTION
SF_Z	Scale factor for LC_Z1
SF_U	Scale factor for LC_U
SF_V	Scale factor for LC_V
SF_W	Scale factor for LC_W
SF_D1	Scale factor for LC_D1
SF_D2	Scale factor for LC_D2
SF_P	Scale factor for LC_P
SF_T	Scale factor for LC_T

Remark:

1. **Consistent Boundary Values.** On each centroid or set of element centroids, the variables ($v_x, v_y, v_z, \rho, P, T, \ldots$) that are given values must be consistent and make the model well-posed (meaning be such that the solution of the model exists, is unique, and is physical).

DUALCESE_BOUNDARY_PRESCRIBED_VN_*OPTION

Available options include:

MSURF

SEGMENT_SET

Purpose: For the dual CESE compressible flow solver, set boundary values for the normal velocity, density, pressure and temperature. Boundary values are applied at the centroid of elements connected with this boundary. This boundary condition differs from *DUALCESE_BOUNDARY_PRESCRIBED in that the normal velocity is prescribed instead of each velocity component.

The MSURF option should be used when the dual CESE mesh has been created using *MESH cards. The SEGMENT_SET option should be used when *DUALCESE_ELE2D or *DUALCESE_ELE3D cards specify the dual 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.

Card Summary:

Card 1a. This card is included if the MSURF keyword option is used.

MSPID	IDCOMP						

Card 1b. This card is included if the SEGMENT_SET keyword option is used.

SSID	IDCOMP						

Card 2. This card is required.

LC_VN			LC_RHO	LC_P	LC_T		

Card 3. This card is required.

SF_VN			SF_RHO	SF_P	SF_T		

Data Card Definitions:

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	MSPID	IDCOMP						
Type	I	I						
Default	none	none						

VARIABLE	DESCRIPTION
MSPID	Mesh surface part ID that is referenced by *MESH_SURFACE_ELEMENT cards
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 [Not yet available].

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

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

VARIABLE	DESCRIPTION
SSID	ID for the segment set created with *DUALCESE_SEGMENTSET

VARIABLE	DESCRIPTION
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 [Not yet available].

Load Curve Card. See Remark 1.

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

VARIABLE	DESCRIPTION
LC_VN	Load curve or function ID to describe the normal velocity as a function of time or a function of position, velocity, temperature, pressure, and time, f(x, y, z, vx, vy, vz, temp, pres, time), respectively. EQ.0: The normal velocity is a constant with value SF_VN. EQ.-1: The normal velocity is computed by the solver.
LC_RHO	Load curve ID to describe the density as a function of time or a function of position, velocity, temperature, pressure, and time, f(x, y, z, vx, vy, vz, temp, pres, time), respectively. EQ.0: The density is a constant with value SF_RHO. EQ.-1: The density is computed by the solver.
LC_P	Load curve ID to describe the pressure as a function of time or a function of position, velocity, temperature, pressure, and time, f(x, y, z, vx, vy, vz, temp, pres, time), respectively. EQ.0: The pressure is a constant with value SF_P. EQ.-1: The pressure is computed by the solver.
LC_T	Load curve ID to describe the temperature as a function of time or a function of position, velocity, temperature, pressure, and time, f(x, y, z, vx, vy, vz, temp, pres, time), respectively. EQ.0: The temperature is a constant with value SF_T. EQ.-1: The temperature is computed by the solver.

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		

VARIABLE	DESCRIPTION
SF_VN	Scale factor for LC_VN
SF_RHO	Scale factor for LC_RHO
SF_P	Scale factor for LC_P
SF_T	Scale factor for LC_T

Remarks:

1. **Consistent Boundary Values.** 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 (meaning be such that the solution of the model exists, is unique, and is physical).

DUALCESE_BOUNDARY_REFLECTIVE_*OPTION

Available options are:

MSURF

SEGMENT_SET

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

The MSURF option should be used when the dual CESE mesh has been created using *MESH cards. The SEGMENT_SET option should be used when *DUALCESE_ELE2D or *DUALCESE_ELE3D cards specify the dual CESE mesh.

Card Summary:

Card 1a. This card is included for the MSURF keyword option. Provide as many cards as necessary. This input ends at the next keyword ("*") card.

MSPID							

Card 1b. This card is included for the SEGMENT_SET keyword option. Provide as many cards as necessary. This input ends at the next keyword ("*") card.

SSID							

Data Card Definitions:

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	MSPID							
Type	I							
Default	none							

VARIABLE	DESCRIPTION
MSPID	Mesh surface part ID that is referenced by *MESH_SURFACE_ELEMENT cards

Set Card. Card 1 format used when the SEGMENT_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	SSID							
Type	I							
Default	none							

VARIABLE	DESCRIPTION
SSID	Segment set ID for the segment set created with *DUALCESE_SEGMENTSET

Remarks:

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

DUALCESE_BOUNDARY_SOLID_WALL_*OPTION1*_*{OPTION2}

For *OPTION1* the choices are:

MSURF

SEGMENT_SET

For *OPTION2* the choices are:

<BLANK>

ROTATE

Purpose: Define a solid wall boundary condition for the dual 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 option should be used when the dual CESE mesh has been created using *MESH cards. The SEGMENT_SET option should be used when *DUALCESE_ELE2D or *DUALCESE_ELE3D cards specify the dual CESE mesh. The ROTATE keyword option allows the boundary condition to rotate around an axis with a variable speed given by a load curve.

Card Summary:

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

Card 1a. This card is included for the MSURF keyword option *without* the ROTATE keyword option.

MSPID	LCID	VX	VY	VZ			

Card 1b. This card is included for the MSURF keyword option *with* the ROTATE keyword option.

MSPID	LCID	XP	YP	ZP	NX	NY	NZ

Card 1c. This card is included for the SEGMENT_SET keyword option *without* the ROTATE keyword option.

SSID	LCID	VX	VY	VZ			

Card 1d. This card is included for the SEGMENT_SET keyword option *with* the ROTATE keyword option.

SSID	LCID	XP	YP	ZP	NX	NY	NZ

Data Card Definitions:

Surface Part Card without Rotation. Card 1 format used when the MSURF keyword option is active *without* the ROTATE keyword option

Card 1a	1	2	3	4	5	6	7	8
Variable	MSPID	LCID	VX	VY	VZ			
Type	I	I	F	F	F			
Default	none	0	0.0	0.0	0.0			
Remarks		2	2	2	2			

VARIABLE	DESCRIPTION
MSPID	Mesh surface part ID that is referenced by *MESH_SURFACE_ELEMENT cards
LCID	Load curve ID scales the velocity vector specified with (VX, VY, VZ) to give the solid wall boundary movement. If not defined, the solid wall boundary moves with a constant velocity vector specified by (VX, VY, VZ).
VX, VY, VZ	Velocity vector of the solid wall boundary condition: LCID.EQ.0: Constant velocity vector specified with VX, VY, and VZ. LCID.NE.0: VX, VY, and VZ give the velocity vector that is scaled by LCID.

Surface Part Card with Rotation. Card 1 format used when the MSURF keyword option is active *with* the ROTATE keyword option.

Card 1b	1	2	3	4	5	6	7	8
Variable	MSPID	LCID	XP	YP	ZP	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

VARIABLE	DESCRIPTION
MSPID	Mesh surface part ID that is referenced by *MESH_SURFACE_ELEMENT cards
LCID	Load curve ID for specifying the rotating speed frequency in Hz. This input is required.
XP, YP, ZP	Coordinates for a point on the axis of rotation
NX, NY, NZ	Unit vector for specifying the direction of the axis of rotation. This is not used for the 2D case.

Segment Set Card without Rotation. Card 1 format used when the SEGMENT_SET keyword option is active *without* the ROTATE keyword option

Card 1c	1	2	3	4	5	6	7	8
Variable	SSID	LCID	VX	VY	VZ			
Type	I	I	F	F	F			
Default	none	0	0.0	0.0	0.0			
Remarks		2	2	2	2			

VARIABLE	DESCRIPTION
SSID	ID of the segment set created with *DUALCESE_SEGMENTSET

VARIABLE	DESCRIPTION
LCID	Load curve ID scales the velocity vector specified with (VX, VY, VZ) to give the solid wall boundary movement. If not defined, the solid wall boundary moves with a constant velocity vector specified by (VX, VY, VZ).
VX, VY, VZ	Velocity vector of the solid wall boundary condition: LCID.EQ.0: Constant velocity vector specified with VX, VY, and VZ. LCID.NE.0: VX, VY, and VZ give the velocity vector that is scaled by LCID.

Segment Set Card with Rotation. Card 1 format used when the SEGMENT_SET keyword option is active *with* the ROTATE keyword option.

Card 1d	1	2	3	4	5	6	7	8
Variable	SSID	LCID	XP	YP	ZP	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

VARIABLE	DESCRIPTION
SSID	ID of the segment set created with *DUALCESE_SEGMENTSET
LCID	Load curve ID for specifying the rotating speed frequency in Hz. This input is required.
XP, YP, ZP	Coordinates for a point on the axis of rotation
NX, NY, NZ	Unit vector for specifying the direction of the axis of rotation. This is not used for the 2D case.

Remarks:

1. **Boundary Movement Restrictions.** In this solid-wall condition, 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 boundary condition only affects viscous flows (no-slip boundary condition).

2. **Fixed Solid Wall Boundary Condition.** If LCID = 0 and Vx = Vy = Vz = 0.0 (default), this will be a regular solid wall boundary condition.

*DUALCESE_CONTROL_LIMITER

Purpose: Sets some stability parameters used in the dual CESE compressible flow solver on the current dual CESE model.

Card 1	1	2	3	4	5	6	7	8
Variable	IDLMT	ALPHA	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 dual CESE theory manual): EQ.0: Limiter format 1 (re-weighting) EQ.1: Limiter format 2 (relaxing)
ALPHA	Re-weighting coefficient, α (see dual CESE theory manual). Must be ≥ 0.
BETA	Numerical viscosity control coefficient, β (see dual CESE theory manual). $0 \leq \beta \leq 1$.
EPSR	Stability control coefficient, ε (see dual CESE theory manual). Must be ≥ 0.

Remarks:

1. **Re-weighting Coefficient.** Larger values of α give more stability, but less accuracy. Usually $\alpha = 2.0$ or 4.0 will be enough for normal shock problems.

2. **Numerical Viscosity Control Coefficient.** Larger values of β give more stability. For problems with shock waves, $\beta = 1.0$ is recommended.

3. **Stability Control Coefficient.** Larger values of ε give more stability, but less accuracy.

*DUALCESE_CONTROL_MESH_MOV

Purpose: Specify the algorithm for calculating the mesh movement (morphing) of a given DUALCESE part in an FSI problem. This keyword is for the moving mesh version of dual CESE.

Card 1	1	2	3	4	5	6	7	8
Variable	ID	IALG	NINTER	RELERR	MXDISPR			
Type	I	I	I	F	F			
Default	none	9	100	10^{-3}	10^{-2}			

VARIABLE	DESCRIPTION
ID	ID for this mesh motion algorithm
IALG	Mesh motion algorithm: EQ.9: IDW scheme (default)
NITER	Number of linear solver iterations (when using a linear solver specified in IALG). No linear solvers have been implemented at this time, so this field is ignored.
RELERR	Relative error for determining convergence when using a linear solver specified in IALG. No linear solvers have been implemented at this time, so this field is ignored.
MXDISPR	Maximum displacement relative to element size to use as a criterion for avoiding the full calculation of the motion of the DUALCESE part on a given time step. If the full calculation can be avoided, the elements touching an FSI interface are still morphed, but it is assumed that this approximation will not lead to elements that are overly distorted.

*DUALCESE_CONTROL_SOLVER

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

WARNING:	As of version R14, this keyword is deprecated. Instead, in order to select the appropriate equations to solve, you should use *DUALCESE_SOLVER_SELECTION along with the corresponding *DUALCESE_-SOLVER_... card. Also, instead of this card, use *DU-ALCESE_MESH_GEOM for the geometry-related details.

Card 1	1	2	3	4	5	6	7	8
Variable	EQNS	IGEOM	IFRAME	MIXTYPE	IDC	ISNAN		
Type	A	A	A	A	F	I		
Default	EULER	none	FIXED	optional	0.25	0		
Remarks		1			2			

VARIABLE	DESCRIPTION
EQNS	Select the equations being solved with the dual CESE solver: EQ.NS: Navier-Stokes equations EQ.EULER: Euler equations
IGEOM	Sets the geometric dimension: EQ.2D: Two-dimensional (2D) problem EQ.3D: Three-dimensional (3D) problem EQ.AXI: 2D axisymmetric
IFRAME	Choose the frame of reference: EQ.FIXED: Usual non-moving reference frame (default). EQ.ROT: Non-inertial rotating reference frame. IFRAME = ROTATING may also be used.

VARIABLE	DESCRIPTION
MIXTYPE	Select the mix or multiphase model solver (if any): EQ.<blank>: No mix or multiphase model (default) EQ.HYBRID: Hybrid multiphase model solver EQ.TWO-PHASE: Two-phase multiphase solver
IDC	Contact interaction detection coefficient (for FSI problems)
ISNAN	Flag to check for NaN in the dual 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. **Mesh and Boundary Conditions for 2D Problems.** If you want to use the 2D (IGEOM = 2D) or 2D axisymmetric (IGEOM=AXI) solver, the mesh should only be distributed in the *xy*-plane with the boundary conditions given only at the *xy* domain boundaries. Otherwise, a warning message will be given, and the 3D solver will be triggered instead.

   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.

2. **Contact Interaction Detection Coefficient.** 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.

3. **Dual CESE Solver and Restarts.** The dual CESE solver is *not* currently supported for restarts.

*DUALCESE_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	10^{-3}					

VARIABLE	DESCRIPTION
IDDT	Sets the time step option: EQ.0: Fixed time step size of DTINT, the 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). $0.0 < \text{CFL} \leq 1.0$
DTINT	Initial time step size

*DUALCESE_DARCY-FORCHHEIMER_EQ

Purpose: Provide the parameters of the Darcy-Forchheimer equation. This equation is also called the Forchheimer correction of Darcy's Law. This equation models the porous medium for compressible porous media flows, such as supersonic spacecraft parachute flows.

Card 1	1	2	3	4	5	6	7	8
Variable	EQID	KP	BETA					
Type	I	F	F					
Default	none	none	none					

VARIABLE	DESCRIPTION
EQID	Equation ID of the porous medium model (see *DUALCESE_POROUS_SPECIFY_JUMP)
KP	Permeability of the porous medium, k_p. See Remark 1.
BETA	Inertial resistance coefficient of the porous medium, β. See Remark 1.

Remarks:

1. **Darcy-Forchheimer equation.** The Darcy-Forchheimer equation is given by:

$$\frac{\partial P}{\partial n} = \frac{\mu_\mathrm{f}}{k_\mathrm{p}} v_n + \beta \rho_\mathrm{f} v_n$$

where

$\frac{\partial P}{\partial n}$	Pressure gradient (in the normal direction) across the porous medium
ρ_f	Fluid density
μ_f	Fluid dynamic viscosity
v_n	Wall normal velocity of the flow through the porous medium
k_p	Permeablity of the porous media
β	Inertial resistance coefficient

2. **Restrictions for the structure.** The porous media model can only be used in FSI problems with thin shell structures (such as fabric). It is intended for modeling supersonic spacecraft parachutes and airbags.

3. **Porous media model.** Currently, only a jump condition method is available for modeling porous media flow.

*DUALCESE_DATABASE_HISTORY_DRAG_LIFT

Purpose: Enable output of drag and lift-related data for all pieces of structural parts participating in FSI with the dual CESE immersed boundary method (IBM). This data output goes to binary database binout. At most, one instance of this keyword may appear in the input deck.

Output Options Card.

Card 1	1	2	3	4	5	6	7	8
Variable	DT	LCUR	IOOPT	REFDENS	REFVEL	REFPRESS	REFAREA	REFLENG
Type	F	I	I	F	F	F	F	F
Default	0.0	0	0	none	none	none	none	none

Directions Card.

Card 2	1	2	3	4	5	6	7	8
Variable	DRAGDIRX	DRAGDIRY	DRAGDIRZ	LIFTDIRX	LIFTDIRY	LIFTDIRZ		
Type	F	F	F	F	F	F		
Default	1.0	0.0	0.0	0.0	1.0	0.0		

Moment Options Card.

Card 3	1	2	3	4	5	6	7	8
Variable	DRGAREA	MOMNTD	MOMNTL	MOMNTC				
Type	I	I	I	I				
Default	0	0	0	0				

VARIABLE	DESCRIPTION
DT	Time interval between outputs. If DT is zero, no output is generated. DT is ignored if LCUR > 0.
LCUR	Optional curve ID specifying the time interval between outputs. Use *DEFINE_CURVE to define the curve. The abscissa is time, and the ordinate is the time interval between outputs.
IOOPT	Flag to govern the behavior of the output frequency load curve defined by LCUR: EQ.1: When output is generated at time t_n, the next output time t_{n+1} is computed as $$t_{n+1} = t_n + \mathrm{LCUR}(t_n)\ .$$ This is the default behavior. EQ.2: When output is generated at time t_n, the next output time t_{n+1} is computed as $$t_{n+1} = t_n + \mathrm{LCUR}(t_{n+1})\ .$$ EQ.3: Output is generated for each abscissa point in the load curve definition. The actual value of the load curve is ignored.
REFDENS	An appropriate reference density
REFVEL	An appropriate reference velocity
REFPRESS	An appropriate reference pressure
REFAREA	An appropriate reference area
REFLENG	An appropriate reference length
DRAGDIRX	x-component of the drag direction
DRAGDIRY	y-component of the drag direction
DRAGDIRZ	z-component of the drag direction
LIFTDIRX	x-component of the lift direction
LIFTDIRY	y-component of the lift direction
LIFTDIRZ	z-component of the lift direction

VARIABLE	DESCRIPTION
DRGAREA	Flag specifying how to obtain the drag area needed for finding the drag-related coefficients: EQ.0: Use REFAREA in all the calculations requiring an enclosing surface area NE.0: Use LS-DYNA's dynamic calculation of the surface area of each piece/part to compute the various drag-related coefficients. REFAREA scales the computed surface area before being used to calculate the drag-related coefficients.
MOMNTD	Flag determining whether to compute the torque in the drag direction: EQ.0: Do not compute the torque. NE.0: Compute the torque.
MOMNTL	Flag determining whether to compute the torque in the lift direction: EQ.0: Do not compute the torque. NE.0: Compute the torque.
MOMNTC	Flag determining whether to compute the torque in the direction that is the cross product of the drag and lift directions: EQ.0: Do not compute the torque. NE.0: Compute the torque.

*DUALCESE_DATABASE_HISTORY_ELEMENT_SET

Purpose: Enable output of dual CESE solver data at the element centroids of the selected dual CESE mesh elements. This element output goes to binary database binout. At most, one instance of this keyword may appear in the input deck.

Output Options Card.

Card 1	1	2	3	4	5	6	7	8
Variable	ESID	DT	LCUR	IOOPT				
Type	I	F	I	I				
Default	0	0.	0	0				

VARIABLE	DESCRIPTION
ESID	ID of a dual CESE element set (see *DUALCESE_ELEMENTSET)
DT	Time interval between outputs. If DT is zero, no output is generated.
LCUR	Optional curve ID specifying the time interval between outputs. Use *DEFINE_CURVE to define the curve. The abscissa is time, and the ordinate is time interval between outputs.
IOOPT	Flag to govern behavior of the output frequency load curve defined by LCUR:

EQ.1: When output is generated at time t_n, the next output time t_{n+1} is computed as

$$t_{n+1} = t_n + \mathrm{LCUR}(t_n) \ .$$

This is the default behavior.

EQ.2: When output is generated at time t_n, the next output time t_{n+1} is computed as

$$t_{n+1} = t_n + \mathrm{LCUR}(t_{n+1}) \ .$$

EQ.3: Output is generated for each abscissa point in the load curve definition. The actual value of the load curve is ignored.

*DUALCESE_DATABASE_HISTORY_GLOBALS

Purpose: Enable output of global dual CESE solver data. The output goes to binary database binout. At most, one instance of this keyword may appear in the input deck.

Output Options Card.

Card 1	1	2	3	4	5	6	7	8
Variable	DT	LCUR	IOOPT					
Type	F	I	I					
Default	0.0	0	0					

VARIABLE	DESCRIPTION
DT	Time interval between outputs. If DT is zero, no output is generated.
LCUR	Optional curve ID specifying the time interval between outputs. Use *DEFINE_CURVE to define the curve; the abscissa is time, and the ordinate is time interval between outputs.
IOOPT	Flag to govern behavior of the output frequency load curve defined by LCUR:

EQ.1: When output is generated at time t_n, the next output time t_{n+1} is computed as

$$t_{n+1} = t_n + \mathrm{LCUR}(t_n)\ .$$

This is the default behavior.

EQ.2: When output is generated at time t_n, the next output time t_{n+1} is computed as

$$t_{n+1} = t_n + \mathrm{LCUR}(t_{n+1})\ .$$

EQ.3: Output is generated for each abscissa point in the load curve definition. The actual value of the load curve is ignored.

*DUALCESE_DATABASE_HISTORY_NODE_SET

Purpose: Enable output of dual CESE solver data at the selected dual CESE mesh nodes. This nodal output goes to binary database binout. At most, one instance of this keyword may appear in the input deck.

Output Options Card.

Card 1	1	2	3	4	5	6	7	8
Variable	NSID	DT	LCUR	IOOPT				
Type	I	F	I	I				
Default	0	0.	0	0.				

VARIABLE	DESCRIPTION
NSID	ID of a dual CESE node set (see *DUALCESE_NODESET)
DT	Time interval between outputs. If DT is zero, no output is generated.
LCUR	Optional curve ID specifying the time interval between outputs. Use *DEFINE_CURVE to define the curve; the abscissa is time, and the ordinate is time interval between outputs.
IOOPT	Flag to govern behavior of the output frequency load curve defined by LCUR:

EQ.1: When output is generated at time t_n, the next output time t_{n+1} is computed as

$$t_{n+1} = t_n + \mathrm{LCUR}(t_n) \ .$$

This is the default behavior.

EQ.2: When output is generated at time t_n, the next output time t_{n+1} is computed as

$$t_{n+1} = t_n + \mathrm{LCUR}(t_{n+1}) \ .$$

EQ.3: Output is generated for each abscissa point in the load curve definition. The actual value of the load curve is ignored.

*DUALCESE_DATABASE_HISTORY_POINT_SET

Purpose: Enable output of dual CESE solver data at the selected points inside the dual CESE mesh. This point-based output goes to binary database binout. At most, one instance of this keyword may appear in the input deck.

Output Options Card.

Card 1	1	2	3	4	5	6	7	8
Variable	PNTSID	DT	LCUR	IOOPT				
Type	I	F	I	I				
Default	0	0.	0	0.				

VARIABLE	DESCRIPTION
PNTSID	ID of a dual CESE point set (see *DUALCESE_POINTSET)
DT	Time interval between outputs. If DT is zero, no output is generated.
LCUR	Optional curve ID specifying the time interval between outputs. Use *DEFINE_CURVE to define the curve; the abscissa is time, and the ordinate is time interval between outputs.
IOOPT	Flag to govern behavior of the output frequency load curve defined by LCUR:

EQ.1: When output is generated at time t_n, the next output time t_{n+1} is computed as

$$t_{n+1} = t_n + \mathrm{LCUR}(t_n) \ .$$

This is the default behavior.

EQ.2: When output is generated at time t_n, the next output time t_{n+1} is computed as

$$t_{n+1} = t_n + \mathrm{LCUR}(t_{n+1}) \ .$$

EQ.3: Output is generated for each abscissa point in the load curve definition. The actual value of the load curve is ignored.

*DUALCESE_DATABASE_HISTORY_SEGMENT_SET

Purpose: Enable output of dual CESE solver data averaged on each of the specified dual CESE segment sets. This segment set-based output goes to binary database binout. At most, one instance of this keyword may appear in the input deck.

Output Options Card.

Card 1	1	2	3	4	5	6	7	8
Variable	DT	LCUR	IOOPT					
Type	F	I	I					
Default	0.0	0	0					

Segment Sets Card. Define as many cards as necessary. Input ends at the next keyword ("*") card.

Card 2	1	2	3	4	5	6	7	8
Variable	SSID1	SSID2	SSID3	SSID4	SSID5	SSID6	SSID7	SSID8
Type	I	I	I	I	I	I	I	I
Default	none	none	none	none	none	none	none	none

VARIABLE	DESCRIPTION
DT	Time interval between outputs. If DT is zero, no output is generated.
LCUR	Optional curve ID specifying the time interval between outputs. Use *DEFINE_CURVE to define the curve; the abscissa is time, and the ordinate is time interval between outputs.
IOOPT	Flag to govern behavior of the output frequency load curve defined by LCUR:

VARIABLE	DESCRIPTION
	EQ.1: When output is generated at time t_n, the next output time t_{n+1} is computed as $$t_{n+1} = t_n + \mathrm{LCUR}(t_n)\ .$$ This is the default behavior.
	EQ.2: When output is generated at time t_n, the next output time t_{n+1} is computed as $$t_{n+1} = t_n + \mathrm{LCUR}(t_{n+1})\ .$$
	EQ.3: Output is generated for each abscissa point in the load curve definition. The actual value of the load curve is ignored.
SSID*i*	*i*th dual CESE segment set ID (see *DUALCESE_SEGMENTSET). For each of these segment sets, an average value of each dual CESE output variable is output to the binout file at the times selected by the fields in Card 1.

*DUALCESE_D3PLOT

Purpose: Specify the flow variables to be added to the dual CESE d3plot output.

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

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

VARIABLE	DESCRIPTION
FLOW_VAR	Name of a flow variable to output to the d3plot file. The currently supported variables are listed in the table below.

Flow Variables:

This table lists the supported flow variables.

FLOW_VAR	DESCRIPTION
DENSITY	Density
VELOCITY	Velocity
MOMENTUM	Momentum
VORTICITY	Vorticity
TOTAL_ENERGY	Total energy
INTERNAL_ENERGY	Internal energy
PRESSURE	Pressure
TEMPERATURE	Temperature
MACH_NUMBER	Flow Mach number
SCHLIEREN_NUMBER	Quantity for capturing or highlighting the shock structure in a compressible flow

FLOW_VAR	DESCRIPTION
VOID_FRACTION	Void fraction (only for the dual CESE cavitation solver)
VOLUME_FRACTION	Volume fraction of the different materials in a hybrid or two-phase multiphase model
REACTANT_MASS_FRACTION	Mass fraction of the reactant (material α) with respect to the explosive material (material 2) in a hybrid multiphase model
LIQUID_MASS_FRACTION	Mass fraction of the liquid component of the fluid in the phase change model
VAPOR_MASS_FRACTION	Mass fraction of the vapor component of the fluid in the phase change model
OTHER_GASES_MASS_FRACTION	Mass fraction of the other gases in the fluid in the phase change model

*DUALCESE_D3PLOT_FLUID_SSID

Purpose: Generate surface d3plot output for the dual CESE solver on a specified dual CESE mesh segment set. These surfaces may be on the outside of the dual CESE fluid mesh that is in contact with the structural volume element parts.

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

VARIABLE	DESCRIPTION
SSID	ID of a segment set created with *DUALCESE_SEGMENTSET

Dual CESE variables to output. 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	FLOW_VAR							
Type	A							

VARIABLE	DESCRIPTION
FLOW_VAR	Name of a flow variable to output to the d3plot file. The currently supported variables are listed in the table below.

Flow Variables:

This table lists the supported flow variables.

FLOW_VAR	DESCRIPTION
DENSITY	Density
VELOCITY	Velocity
MOMENTUM	Momentum

FLOW_VAR	DESCRIPTION
VORTICITY	Vorticity
TOTAL_ENERGY	Total energy
INTERNAL_ENERGY	Internal energy
PRESSURE	Pressure
TEMPERATURE	Temperature
MACH_NUMBER	Flow Mach number
SCHLIEREN_NUMBER	Quantity for capturing or highlighting the shock structure in a compressible flow
VOID_FRACTION	Void fraction (only for the dual CESE cavitation solver)
VOLUME_FRACTION	Volume fraction of the different materials in a hybrid or two-phase multiphase model
REACTANT_MASS_FRACTION	Mass fraction of the reactant (material α) with respect to the explosive material (material 2) in a hybrid multiphase model

*DUALCESE_ELE2D

Purpose: Define three and four node elements.

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. Choose a unique number with respect to other elements.
PID	Part ID, see *DUALCESE_MESH_PART.
N1	Nodal point 1
N2	Nodal point 2
N3	Nodal point 3
N4	Nodal point 4

*DUALCESE_ELE3D

Purpose: Define three-dimensional fluid volume elements. These can be 4 node tetrahedra, 5 node pyramids, 6 node wedges (prisms), and 8 node hexahedra.

Card 1	1	2	3	4	5	6	7	8	9	10
Variable	EID	PID	N1	N2	N3	N4	N5	N6	N7	N8
Type	I	I	I	I	I	I	I	I	I	I
Default	none	none	none	none	none	none	none	none	none	none
Remarks	1									

VARIABLE	DESCRIPTION
EID	Element ID. A unique number must be chosen.
PID	Part ID, see *DUALCESE_MESH_PART.
N1	Nodal point 1
N2	Nodal point 2
N3	Nodal point 3
⋮	⋮
N8	Nodal point 8

Remarks:

1. **Node Numbering.** Four, five, six, and eight node elements are allowed as numbered below. This ordering must be followed, or code termination will occur during the initialization phase with a negative volume message. In the case of a pyramid element, the base of the pyramid must follow the ordering used for the hexahedron. See *ELEMENT_SOLID for a figure showing the positions of the nodes in 4, 6, and 8 node elements.

    <u>4-noded tetrahedron</u>   N1, N2, N3, N4, N4, N4, N4, N4

    <u>5-noded pyramid</u>   N1, N2, N3, N4, N5, N5, N5, N5

<u>6-noded pentahedron</u>	N1, N2, N3, N4, N5, N5, N6, N6
<u>8-noded hexahedron</u>	N1, N2, N3, N4, N5, N6, N7, N8

*DUALCESE_ELEMENTSET

Purpose: Define a set of dual CESE mesh elements.

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

Element ID Cards. List of elements in the set, where the element IDs are defined with *DUALCESE_ELE2D or *DUALCESE_ELE3D cards. 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	EID1	EID2	EID3	EID4	EID5	EID6	EID7	EID8
Type	I	I	I	I	I	I	I	I

VARIABLE	DESCRIPTION
ESID	Set ID. All dual CESE element sets should have a unique set ID.
EID*i*	Element ID *i*

*DUALCESE_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	RHOVAP	RHOLIQ	AVAP	ALIQ	MUVAP	MULIQ	PSATVAP
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 ID
RHOVAP	Density of the saturated vapor, ρ_{vap}
RHOLIQ	Density of the saturated liquid, ρ_{liq}
AVAP	Sound speed of the saturated vapor, a_{vap}
ALIQ	Sound speed of the saturated liquid, a_{liq}
MUVAP	Dynamic viscosity of the vapor, μ_{vap}
MULIQ	Dynamic viscosity of the liquid, μ_{liq}
PSATVAP	Pressure of the saturated vapor, P_{SatVap}

Remarks:

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

*DUALCESE_EOS_COCHRAN_CHAN

Purpose: Define a Cochran-Chan type of EOS that provides a means to represent a condensed phase explosive in a dual CESE multiphase model.

Card 1	1	2	3	4	5	6	7	8
Variable	EOSID	A	B	EPS1	EPS2	GAMMA0	RHO0	E0
Type	I	F	F	F	F	F	F	F
Default	none	none	none	none	none	none	none	0.0

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

VARIABLE	DESCRIPTION
EOSID	Equation of state ID for the dual CESE solver
A	Model parameter (in pressure units), A
B	Model parameter (in pressure units), B
EPS1	Model constant (dimensionless), ε_1
EPS2	Model constant (dimensionless), ε_2
GAMMA0	Gruneisen coefficient
RHO0	Initial or reference density, ρ_0
E0	Represents the heat of detonation released during the reactions, or the constant rate of afterburn energy added (E0 = 0.0 is the default), e_0
CV	Heat capacity, C_v

Remarks:

The Cochran-Chan EOS like the JWL EOS (*DUALCESE_EOS_JWL) is a type of Mie-Gruneisen EOS. The equations of state of a Mie-Gruneisen form are given by:

$$P(\rho, e) = P_{\mathrm{ref}} + \Gamma(\rho)\rho[e - e_{\mathrm{ref}}(\rho)]$$

Here $\Gamma(\rho)$ is the Gruneisen coefficient. For the Cochran-Chan EOS reference pressure and energy are given by:

$$\begin{aligned} P_{ref}(\rho) &= A\left(\frac{\rho_0}{\rho}\right)^{-\varepsilon_1} - B\left(\frac{\rho_0}{\rho}\right)^{-\varepsilon_2} \\ e_{\mathrm{ref}}(\rho) &= \frac{A}{\rho_0(1-\varepsilon_1)}\left(\frac{\rho_0}{\rho}\right)^{1-\varepsilon_1} + \frac{B}{\rho_0(1-\varepsilon_2)}\left(\frac{\rho_0}{\rho}\right)^{1-\varepsilon_2} - e_0 \end{aligned}$$

*DUALCESE_EOS_COOLPROP

Purpose: Define an equation of state (EOS) to be evaluated using the COOLPROP EOS library

Note that the COOLPROP library is not provided by ANSYS. You need to download a 64-bit version of the shared library from a public repository, such as:

> https://sourceforge.net/projects/coolprop/files/CoolProp/6.3.0/shared_library/Linux/64bit/

WARNING:	Since the *MODULE capability is not yet working in the Windows build of LS-DYNA, do *not* attempt to use a Windows DLL version of the COOLPROP shared library.

To use the COOLPROP shared library with this keyword card, load this shared library into LS-DYNA using the *MODULE capability. The following *MODULE card needs to appear before a *DUALCESE_MODEL card (not inside the file hierarchy of any file specified with a *DUALCESE_MODEL card):

```
*MODULE_LOAD
UserA                  DUALCESE COOLPROP
 < path to installed COOLPROP shared library >
```

Card Summary:

Card 1. This card is required.

EOSID	NCOMP	TYPE	PHASE	TABULAR			

Card 2. Include as many cards as needed to specify mole fractions for the NCOMP components of the fluid.

MOL_FR1	MOL_FR2	MOL_FR3	MOL_FR4	MOL_FR5	MOL_FR6	MOL_FR7	MOL_FR8

Card 3. Include this card when the TABULAR field is active on Card 1.

N_T	N_DEN	DEN_LOW	DEN_HIGH	T_LOW	T_HIGH		

Card 4. This card is required.

FLUIDNAME

Data Card Definitions:

Card 1	1	2	3	4	5	6	7	8
Variable	EOSID	NCOMP	TYPE	PHASE	TABULAR			
Type	I	I	A	A	A			
Default	none	none	none	GAS	optional			

VARIABLE	DESCRIPTION
EOSID	ID for this EOS
NCOMP	Number of components in the fluid composition
TYPE	Fluid type: EQ.PURE: A single component fluid (default) EQ.PSEUDOPURE: A predefined fluid mixture EQ.MIXTURE: A fluid mixture with NCOMP components
PHASE	Phase of the fluid. EQ.GAS: Gas phase EQ.LIQUID: Liquid phase
TABULAR	Type of lookup tables to build for this EOS: EQ.<BLANK>: No table lookup (default) EQ.P_EIN: Build tables of pressure and internal energy, both as a function of density and temperature.

COOLPROP Parameters by Fluid Component. Repeat this card as many times as needed to input mole fractions for the NCOMP components of the fluid.

Card 2	1	2	3	4	5	6	7	8
Variable	MOL_FR1	MOL_FR2	MOL_FR3	MOL_FR4	MOL_FR5	MOL_FR6	MOL_FR7	MOL_FR8
Type	F	F	F	F	F	F	F	F

VARIABLE	DESCRIPTION
MOL_FR*i*	Mole fraction of the i^{th} component

COOLPROP EOS Table Density and Temperature Ranges. This card is included when the TABULAR option on Card 1 is active.

Card 3	1	2	3	4	5	6	7	8
Variable	N_T	N_DEN	DEN_LOW	DEN_HIGH	T_LOW	T_HIGH		
Type	I	I	F	F	F	F		
Default	none	none	none	none	none	none		
Remarks	1	1	2	2	2	2		

VARIABLE	DESCRIPTION
N_T	Number of temperature values in the tables
N_DEN	Number of density values (on a log scale) in the tables
DEN_LOW	Minimum density available in the tables (in model units)
DEN_HIGH	Maximum density available in the tables (in model units)
T_LOW	Minimum temperature available in the tables (in model units)
T_HIGH	Maximum temperature available in the tables (in model units)

Name of CoolProp fluid. This card is required.

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

VARIABLE	DESCRIPTION
FLUIDNAME	Name of a fluid that has an EOS in CoolProp. For a list of the supported pure and pseudo-pure fluids, see: http://www.coolprop.org/fluid_properties/PurePseudo-Pure.html#list-of-fluids Note that the predefined fluid mixtures are not supported at this time.

Remarks:

1. **Number of Values in the Lookup Tables.** The number of density and temperature values in the tables should not be too few to give good resolution of the EOS. Note that the cost of building the EOS from these tables rises with these numbers, as well as the computer memory required. Nevertheless, if these numbers are too small (< 20), then the accuracy may suffer, while larger numbers of density and temperature points improves the accuracy.

2. **Valid Value Ranges for the Lookup Tables.** For many equations of state in the CoolProp library, there is a range of valid densities and temperatures. Thus, the low and high limits for the table densities and temperatures should not lie outside these ranges. Please refer to the CoolProp documentation for that information.

*DUALCESE_EOS_IDEAL_GAS

Purpose: Define the coefficients $C_v$and C_p in the equation of state for an ideal gas in the dual CESE fluid solver.

Card 1	1	2	3	4	5	6	7	8
Variable	EOSID	CV	CP	E0				
Type	I	F	F	F				
Default	none	717.5	1004.5	0.0				
Remarks		1	1	2				

VARIABLE	DESCRIPTION
EOSID	Equation of state ID
CV	Specific heat at constant volume, C_v
CP	Specific heat at constant pressure, C_p
E0	Represents the heat of detonation released during the reactions, or the constant rate of afterburn energy added (E0 = 0.0 is the default), e_0

Remarks:

1. **Units.** As with other solvers in LS-DYNA, you are responsible for unit consistency. For example, if you want to use dimensionless variables, CV and CP should also be replaced by the corresponding dimensionless ones. If the dual CESE model has a specified system of units either directly from the *DUALCESE_MODEL card or inherited from the overall problem input, then these values need to be given in that unit system.

2. **E0.** E0 is used only with the hybrid multiphase solver where the EOS of the reactant is specified by this ideal gas EOS in the *DUALCESE_EOS_SET card.

*DUALCESE_EOS_INFLATOR1

Purpose: Define an EOS using C_p and C_v 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 ID for the dual CESE solver
CP0, …, CP4	Coefficients of temperature-dependent specific heat at constant pressure $$C_p(T) = C_{p_0} + C_{p_1}T + C_{p_2}T^2 + C_{p_3}T^3 + C_{p_4}T^4$$

VARIABLE	DESCRIPTION
CV0, …, CV4	Coefficients of temperature-dependent specific heat at constant volume $$C_v(T) = C_{v0} + C_{v1}T + C_{v2}T^2 + C_{v3}T^3 + C_{v4}T^4$$

Remarks:

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

*DUALCESE_EOS_INFLATOR2

Purpose: Define an EOS using C_p and C_v thermodynamic expansions for an inflator gas mixture with two temperature ranges, one below 1000 Kelvin, and the other above 1000 Kelvin.

Card Summary:

Card 1. This card is required.

EOSID							

Card 2. This card is required. This card with Card 3 specifies C_p. This card gives the coefficients for T < 1000 K.

CP10	CP11	CP12	CP13	CP14			

Card 3. This card is required. This card gives the coefficients for T > 1000 K.

CP20	CP21	CP22	CP23	CP24			

Card 4. This card is required. This card with Card 5 defines C_v. This card gives the coefficients for T < 1000 K.

CV10	CV11	CV12	CV13	CV14			

Card 5. This card is required. This card gives the coefficients for T > 1000 K.

CV20	CV21	CV22	CV23	CV24			

Data Card Definitions:

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

VARIABLE	DESCRIPTION
EOSID	Equation of state ID for the dual CESE solver

Coefficients for the expansion to determine specific heat at constant pressure for T < 1000 K.

Card 2	1	2	3	4	5	6	7	8
Variable	CP10	CP11	CP12	CP13	CP14			
Type	F	F	F	F	F			
Default	0.	0.	0.	0.	0.			

Coefficients for the expansion to determine specific heat at constant pressure for T > 1000 K.

Card 3	1	2	3	4	5	6	7	8
Variable	CP20	CP21	CP22	CP23	CP24			
Type	F	F	F	F	F			
Default	0.	0.	0.	0.	0.			

VARIABLE	DESCRIPTION
CP10, …, CP14	Coefficients of temperature-dependent specific heat at constant pressure valid for T < 1000 K
CP20, …, CP24	Coefficients of temperature-dependent specific heat at constant pressure valid for T > 1000 K

Cards 2 and 3 give C_p over the two temperature ranges:

$$C_p(T) = \begin{cases} \mathrm{CP10} + \mathrm{CP11} \times T + \mathrm{CP12} \times T^2 + \mathrm{CP13} \times T^3 + \mathrm{CP14} \times T^4 & \text{for } T < 1000\text{ K} \\ \mathrm{CP20} + \mathrm{CP21} \times T + \mathrm{CP22} \times T^2 + \mathrm{CP23} \times T^3 + \mathrm{CP24} \times T^4 & \text{for } T > 1000\text{ K} \end{cases}$$

Card for the Expansion of Specific Heat at Constant Volume. Valid for T < 1000 K

Card 4	1	2	3	4	5	6	7	8
Variable	CV10	CV11	CV12	CV13	CV14			
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 K

Card 5	1	2	3	4	5	6	7	8
Variable	CV20	CV21	CV22	CV23	CV24			
Type	F	F	F	F	F			
Default	0.	0.	0.	0.	0.			

VARIABLE	DESCRIPTION
CV10, …, CV14	Coefficients of temperature-dependent specific heat at constant volume valid for T < 1000 K
CV20, …, CV24	Coefficients of temperature-dependent specific heat at constant volume valid for T > 1000 K

Cards 4 and 5 give C_v over the two temperature ranges:

$$C_v(T) = \begin{cases} \mathrm{CV10} + \mathrm{CV11} \times T + \mathrm{CV12} \times T^2 + \mathrm{CV13} \times T^3 + \mathrm{CV14} \times T^4 & \text{for } T < 1000\text{ K} \\ \mathrm{CV20} + \mathrm{CV21} \times T + \mathrm{CV22} \times T^2 + \mathrm{CV23} \times T^3 + \mathrm{CV24} \times T^4 & \text{for } T > 1000\text{ K} \end{cases}$$

Remarks:

These coefficient expansions for the specific heats over two temperature ranges are generated by the zero-dimensional inflator model solver. See *CHEMISTRY_CONTROL_-INFLATOR and *CHEMISTRY_INFLATOR_PROPERTIES for details related to running that solver.

***DUALCESE_EOS_JWL**

Purpose: Define a JWL-type EOS that provides a means to represent a condensed phase explosive in a dual CESE multiphase model.

Card 1	1	2	3	4	5	6	7	8
Variable	EOSID	A	B	R1	R2	GAMMA0	RHO0	E0
Type	I	F	F	F	F	F	F	F
Default	none	none	none	none	none	none	none	0.0

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

VARIABLE	DESCRIPTION
EOSID	Equation of state ID for the dual CESE solver.
A	Model parameter, A (in pressure units)
B	Model parameter, B (in pressure units)
R1	Model constant, R_1 (dimensionless)
R2	Model constant, R_2 (dimensionless)
GAMMA0	Gruneisen coefficient
RHO0	Initial or reference density, ρ_0
E0	Represents the heat of detonation released during the reactions or the constant rate of afterburn energy added (E0 = 0.0 for standard JWL EOS), e_0
CV	Heat capacity, C_v

Remark:

The equations of state of a Mie-Gruneisen form are given by:

$$P(\rho, e) = P_{\text{ref}} + \Gamma(\rho)\rho[e - e_{\text{ref}}(\rho)]$$

Here $\Gamma(\rho)$ is the Gruneisen coefficient. Equations of state of this type are very popular in condensed phase explosive modeling. Depending on the form of the reference pressure and energy functions, different EOS types can be retrieved. The JWL EOS is one type with reference pressure and energy given by:

$$P_{\text{ref}}(\rho) = A\exp\left(\frac{-R_1\rho_0}{\rho}\right) + B\exp\left(\frac{-R_2\rho_0}{\rho}\right)$$

$$e_{\text{ref}}(\rho) = \frac{A}{\rho_0 R_1}\exp\left(\frac{-R_1\rho_0}{\rho}\right) + \frac{B}{\rho_0 R_2}\exp\left(\frac{-R_2\rho_0}{\rho}\right) - e_0$$

*DUALCESE_EOS_NASG

Purpose: Define a Noble-Abel Stiffened-Gas (NASG) type fluid EOS for use in the phase-change dual CESE solver. See M'etayer and Saurel [2016] for details.

Card 1	1	2	3	4	5	6	7	8
Variable	EOSID	GAMMA	CV	PINF	Q	QP	B	W
Type	I	F	F	F	F	F	F	F
Default	none	none	none	none	none	none	none	none

VARIABLE	DESCRIPTION
EOSID	Equation of state ID
GAMMA	Ratio of specific heats, γ
CV	Specific heat at constant volume
PINF	Parameter for a given phase, p_∞ (dimensions: $\left[\frac{M}{LT^2}\right]$). See Le M'etayer and Saurel [2016].
Q	Parameter for a given phase, q (dimensions: $\left[\frac{L^2}{T^2}\right]$). q is the heat bond for the phase. See Le M'etayer and Saurel [2016].
QP	Parameter for a given phase, q' (dimensions: $\left[\frac{L^2}{T^2\theta}\right]$, where θ represents the dimension of temperature). See Le M'etayer and Saurel [2016].
B	Parameter for a given phase, b (dimensions: $\left[\frac{L^3}{M}\right]$). b indicates the fluid's covolume. See Le M'etayer and Saurel [2016].
W	Molar mass of this fluid

Remarks:

The NASG EOS given by M'etayer and Saurel [2016] has the following form:

$$
\begin{aligned}
p(\nu, e) &= \frac{(\gamma-1)(e-q)}{\nu-b} - \gamma p_\infty \\
T(p, \nu) &= \frac{(\nu-b)(p+p_\infty)}{C_v(\gamma-1)} \\
g(p, T) &= (\gamma C_v - q')T - C_v T \ln\left[\frac{T^\gamma}{(p+p_\infty)^{\gamma-1}}\right] + bp + q \\
c(p, \nu) &= \sqrt{\gamma\nu(p+p_\infty)}
\end{aligned}
$$

where $\gamma, C_v, p_\infty, q, q'$ and b are parameters needed for each phase. ν, e, g and c are for each fluid phase the specific volume, the specific internal energy, the specific Gibbs free energy, and the sound speed, respectively. $g = h - Ts$ with h and s as the specific enthalpy and specific entropy, respectively.

References:

[1] Le M'etayer, O. and R. Saurel, "The Nobel-Abel Stiffened-Gas equation of state," *Physics of Fluids*, 28 (4), (2016).

***DUALCESE_EOS_REFPROP**

Purpose: Define an equation of state (EOS) to be evaluated using the REFPROP EOS library

Note that the REFPROP library is provided by ANSYS; it is REFPROP v10.0 from NIST.

WARNING:	Since the *MODULE capability is not yet working in the Windows build of LS-DYNA, do *not* attempt to use a Windows DLL version of the REFPROP shared library that comes with REFPROP v10.0.

To use the REFPROP shared library with this keyword card, load this shared library into LS-DYNA using the *MODULE capability. The following *MODULE card needs to appear before a *DUALCESE_MODEL card (not inside the file hierarchy of any file specified with a *DUALCESE_MODEL card):

```
*MODULE_LOAD
UserA                  DUALCESE REFPROP
 < path to the installed REFPROP shared library >
```

In addition, for REFPROP to be able to find the appropriate EOS data, *DUALCESE_-EOS_REFPROP_PATH must also be given somewhere inside a *DUALCESE_MODEL file hierarchy to point to the place in your filesystem where REFPROP has been installed.

Card Summary:

Card 1. This card is required.

EOSID	NCOMP	TYPE	PHASE	TABULAR			

Card 2. Include as many cards as needed to specify mole fractions for the NCOMP components of the fluid.

MOL_FR1	MOL_FR2	MOL_FR3	MOL_FR4	MOL_FR5	MOL_FR6	MOL_FR7	MOL_FR8

Card 3. This card is included when the TABULAR field is active on Card 1.

N_T	N_DEN	DEN_LOW	DEN_HIGH	T_LOW	T_HIGH		

Card 4. This card is required.

FLUIDNAME

Data Card Definitions:

Card 1	1	2	3	4	5	6	7	8
Variable	EOSID	NCOMP	TYPE	PHASE	TABULAR			
Type	I	I	A	A	A			
Default	none	none	none	GAS	optional			

VARIABLE	DESCRIPTION
EOSID	ID for this EOS
NCOMP	Number of components in the fluid composition
TYPE	Fluid type: EQ.PURE: A single component fluid (default) EQ.PSEUDOPURE: A predefined fluid mixture EQ.MIXTURE: A fluid mixture with NCOMP components
PHASE	Phase of the fluid: EQ.GAS: Gas phase EQ.LIQUID: Liquid phase
TABULAR	Type of lookup tables to build for this EOS: EQ.<BLANK>: No table lookup (default) EQ.P_EIN: Build tables of pressure and internal energy, both as a function of density and temperature.

REFPROP Parameters by Fluid Component. Repeat this card as many times as needed to input mole fractions for the NCOMP components of the fluid.

Card 2	1	2	3	4	5	6	7	8
Variable	MOL_FR1	MOL_FR2	MOL_FR3	MOL_FR4	MOL_FR5	MOL_FR6	MOL_FR7	MOL_FR8
Type	F	F	F	F	F	F	F	F
Default	none	none	none	none	none	none	none	none

VARIABLE	DESCRIPTION
MOL_FR*i*	Mole fraction of the i^{th} component of the fluid.

.

REFPROP EOS Table Density and Temperature Ranges. This card is included when the TABULAR field is active on Card 1.

Card 3	1	2	3	4	5	6	7	8
Variable	N_T	N_DEN	DEN_LOW	DEN_HIGH	T_LOW	T_HIGH		
Type	I	I	F	F	F	F		
Default	none	none	none	none	none	none		
Remarks	1	1	2	2	2	2		

VARIABLE	DESCRIPTION
N_T	Number of temperature values in the tables
N_DEN	Number of density values (on a log scale) in the table
DEN_LOW	Minimum density available in the tables (in model units)
DEN_HIGH	Maximum density available in the tables (in model units)
T_LOW	Minimum temperature available in the tables (in model units)
T_HIGH	Maximum temperature available in the tables (in model units)

Name of REFPROP fluid. This card is required.

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

VARIABLE	DESCRIPTION
FLUIDNAME	Name of a fluid that has an EOS in REFPROP. For a list of the supported pure and pseudo-pure fluids, see the directory of supported fluids that comes with the REFPROP v10.0 library from ANSYS. Note that the predefined fluid mixtures are not supported at this time.

Remarks:

1. **Number of Values in the Lookup Table.** The number of values of density and temperature axes of the tables should not be too few to give good resolution of the EOS. Note that the cost of building the EOS from these tables rises with these numbers, as well as the computer memory required. Nevertheless, if these numbers are too small (< 20), then the accuracy may suffer, while larger numbers of density and temperature points improves the accuracy.

2. **Valid Value Ranges for the Lookup Tables.** For many equations of state in the REFPROP library, a range of densities and temperatures are valid. Thus, the low and high limits for the table densities and temperatures should not lie outside these ranges. Please refer to the REFPROP documentation for that information.

*DUALCESE_EOS_REFPROP_PATH

Purpose: Provide the file path to the directory where the REFPROP EOS system is installed.

Note that in any problem where a *DUALCESE_EOS_REFPROP card is used, you must also provide a *DUALCESE_EOS_REFPROP_PATH card somewhere inside a *DUALCESE_MODEL file hierarchy to point to the place in your filesystem where REFPROP has been installed so that the appropriate EOS data can be loaded.

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

VARIABLE	DESCRIPTION
PATH	Path giving the directory where the REFPROP data is installed.

*DUALCESE_EOS_SET

Purpose: Define a set of equations of state that are used together to compute the thermodynamic state of a multiphase fluid for the dual CESE solver.

WARNING:	As of version R14, this keyword is deprecated. Instead, use *DUALCESE_SOLVER_HYBRID_MULTIPHASE for the hybrid multiphase case and *DUALCESE_SOLVER_TWO-PHASE_MULTIPHASE for the two-phase multiphase case.

Include one card for each dual CESE multiphase mesh. This input ends at the next keyword ("*") card.

Card 1	1	2	3	4	5	6	7	8
Variable	EOSSID	EOSINID	EOSRCTID	EOSPRDID				
Type	I	I	I	I				
Default	none	none	none	none				

VARIABLE	DESCRIPTION
EOSSID	Set ID of the EOS mixture of a given multiphase fluid
EOSINID	EOS ID of the inert component of the multiphase mixture
EOSRCTID	EOS ID of the reactant phase of the multiphase mixture
EOSPRDID	EOS ID of the product phase of the multiphase mixture

*DUALCESE_EOS_STIFFENED_GAS

Purpose: Define a stiffened gas type fluid EOS for use by the dual CESE solver.

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

VARIABLE	DESCRIPTION
EOSID	Equation of state ID for this dual CESE solver EOS
GA	Adiabatic exponent, γ. Must be > 1.0.
BT	Reference pressure, β. Must be ≥ 0.0

Remark:

The stiffened gas equation of state:

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

provides a fundamental characterization of material properties of fluids. Here e is the internal energy per unit mass, ρ is the density, and γ and β are two thermodynamic constants. γ and β can be determined by a fitting procedure from laboratory data. A typical set of parameter values for water are: $\gamma = 7$ and $\beta = 3000$ atm while for human blood are: $\gamma = 5.527$ and $\beta = 614.6$ MPa. In addition to the modelling of a liquid, it is often used to describe other type of materials, including many compressible solids of practical importance.

*DUALCESE_EOS_VAN_DER_WAALS_GENERALIZED

Purpose: Define a Van Der Waals generalized type fluid EOS for use in the dual CESE solver.

Card 1	1	2	3	4	5	6	7	8
Variable	EOSID	A	B	GA	BT			
Type	I	F	F	F	F			
Default	none	none	none	none	none			

VARIABLE	DESCRIPTION
EOSID	Equation of state ID
A	van der Waals gas constant for molecular cohesive forces, a
B	van der Waals gas constant for the finite size of molecules, b
GA	Ratio of specific heats, γ. Must be > 1.0.
BT	Reference pressure, β. Must be ≥ 0.0. See *DUALCESE_EOS_-STIFFENED_GAS.

Remark:

The generalized Van der Waals equation of state can be written as:

$$p(\rho, e) = \frac{(\gamma - 1)}{1 - b\rho}\left(\rho e - \beta + a\rho^2\right) - \left(\beta + a\rho^2\right)$$

Here e denotes the specific internal energy, γ is the ratio of specific heats ($\gamma > 1$), β is a reference pressure, and the quantities a and b are the van der Waals gas constants for molecular cohesive forces and the finite size of molecules, respectively ($a \geq 0$, $0 \leq b < 1/\rho$). This EOS is often used to deal with possible real-gas effect (without phase transition) when both the temperature and pressure are high. When $\beta = 0$, the generalized van der Waals equation of state becomes a standard Van der Waals equation of state. If $a = b = 0$, the stiffened gases EOS will be recovered.

References:

[1] G. Allaire, S. Clerc, S. Kokh, A five-equation model for the simulation of interfaces between compressible fluids. J. Comp. Phys. 181 (2) (2002) 577-616.

[2] K.M. Shyue, A fluid-mixture type algorithm for compressible multicomponent flow with van der Waals equation of state, J. Comp. Phys. 156, 43 (1999)

*DUALCESE_FSI_EXCLUDE

Purpose: Provide a list of mechanics solver parts that are not involved in the dual CESE FSI calculation. This keyword is intended for increasing computational efficiency by excluding parts that will not involve significant FSI interactions with the dual 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
PID*n*	IDs of mechanics parts that will be excluded from the FSI interaction calculation with the dual CESE solver

*DUALCESE_FSI_EXCLUDE_PARTSET

Purpose: Provide a list of mechanics solver part sets for parts that are not involved in the dual CESE FSI calculation. This keyword is intended for increasing computational efficiency by excluding parts that will not involve significant FSI interactions with the dual 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	PSID1	PSID2	PSID3	PSID4	PSID5	PSID6	PSID7	PSID8
Type	I	I	I	I	I	I	I	I
Default	none	none	none	none	none	none	none	none

VARIABLE	DESCRIPTION
PSID*n*	IDs of mechanics part sets that will be excluded from the FSI interaction calculation with the dual CESE solver

*DUALCESE_GRAVITY

Purpose: Apply a gravity load to the dual CESE fluid. It is only avilable for single fluid solvers (meaning EQNS = EULER or NS) for both FSI (including FSI-IBM and FSI-MMM) and no FSI. Please note that this keyword must be accompanied by *LOAD_BODY_VECTOR, *LOAD_BODY_X, *LOAD_BODY_Y, or *LOAD_BODY_Z. However, these *LOAD_BODY_... keywords cannot be in the dual CESE model portion of the input deck (see Dual CESE Keyword Deck Structure for details).

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

VARIABLE	DESCRIPTION
IGRAV	Flag for applying a gravity load: EQ.0: No gravity load applied (default) EQ.1: Apply gravity load

Remarks:

1. **Including gravity.** In most compressible cases, the gravity terms are not needed (the default is no gravity). Including gravity only has a significant effect for certain cases, such as buoyancy driven flow, stratified flow, etc.

*DUALCESE_INCLUDE_MODEL

Purpose: Provide the filename of a file containing additional keywords belonging to a dual CESE model. Any number of these *DUALCESE_INCLUDE_MODEL keywords may be used in a single dual CESE model, where at the top level the overall model begins with a *DUALCESE_MODEL card.

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

VARIABLE	DESCRIPTION
FILENAME	Filename of the keyword file containing more of the dual CESE model. This card is only allowed inside a file that is given in one instance of a *DUALCESE_MODEL keyword card.

*DUALCESE_INITIAL

Purpose: Specify constant initial conditions for flow variables at the centroid of each dual CESE fluid element.

Card 1	1	2	3	4	5	6	7	8
Variable	U	V	W	RH	P	T		IFUNC
Type	F	F	F	F	F	F		I
Default	0	0.0	0.0	1.225	0.0	0.0		none

VARIABLE	DESCRIPTION
U, V, W	x, y, and z velocity components, respectively
RHO	Density, ρ
P	Pressure, P
T	Temperature, T
IFUNC	Option to define initial conditions using *DEFINE_FUNCTION cards: EQ.0: Not in use. EQ.1: All values for initial velocity, pressure, density, and temperature now refer to *DEFINE_FUNCTION IDs. In these functions, the following parameters are allowed: f(x,y,z), meaning that each variable's initial profile is a function of position.

Remarks:

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

2. **Applicable Elements.** These initial conditions will be applied in those elements that have not been assigned a value by *DUALCESE_INITIAL_*OPTION* cards for individual elements or sets of elements.

*DUALCESE_INITIAL_SET

Purpose: Specify initial conditions for the flow variables at the centroid of each element in an element subset of the dual CESE mesh.

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

Card 2	1	2	3	4	5	6	7	8
Variable	U	V	W	RHO	P	T		
Type	F	F	F	F	F	F		
Default	none	none	none	none	none	none		

VARIABLE	DESCRIPTION
ESID	Element set ID (see *DUALCESE_ELEMENTSET)
IFUNC	Option to define initial conditions using *DEFINE_FUNCTION cards: EQ.0: Not in use. EQ.1: All values for initial velocity, pressure, density, and temperature now refer to *DEFINE_FUNCTION IDs. In these functions, the following parameters are allowed: f(x,y,z), meaning that each variable's initial profile is a function of position.
U, V, W	x, y, and z velocity components, respectively
RHO	Density, ρ
P	Pressure, P

VARIABLE	DESCRIPTION
T	Temperature, T

Remarks:

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

2. **Keyword Priority.** The priority of this card is higher than *DUALCESE_INITIAL. Thus, if an element is assigned an initial value by this card, *DUALCESE_INITIAL will no longer apply to that element.

*DUALCESE_INITIAL_HYBRID

Purpose: Specifies values to use for velocity, pressure, etc. for problem initialization of a hybrid multiphase model on the dual CESE mesh. Note that these values can be overridden in some mesh elements by use of the *DUALCESE_INITIAL_HYBRID_SET card.

The hybrid multiphase model involves two materials, separated by a material surface. These materials do not mix. The first material is inert while the second material is an explosive mixture, composed of reactants and products. This model is useful for simulations with high explosives. See Michael and Nikiforakis 2016 for details about this model.

Card 1	1	2	3	4	5	6	7	8
Variable	Z1	RA	UIC	VIC	WIC	RHO1	RHO_A	RHO_B
Type	F	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	PIC	TIC		IFUNC				
Type	F	F		I				
Default	none	none		none				

VARIABLE	DESCRIPTION
Z1	Volume fraction of material 1 (or color function). This is usually a value of 0 or 1. For numerical stability, however, use a very small value instead of zero.
RA	Mass fraction of the reactant (material α) with respect to material 2 (the explosive mixture)
UIC, VIC, WIC	Multiphase flow velocity components in the x, y, and z-directions, respectively
RHO1	Density of material 1

VARIABLE	DESCRIPTION
RHO_A	Density of the reactant (material α)
RHO_B	Density of the product (material β)
PIC	Equilibrium multifluid pressure
TIC	Equilibrium multifluid temperature
IFUNC	Option to define initial conditions using *DEFINE_FUNCTION cards: EQ.0: Not in use. EQ.1: All values for initial velocity, pressure, and temperature now refer to *DEFINE_FUNCTION IDs. In these functions, the following parameters are allowed: f(x,y,z), meaning that each variable's initial profile is a function of position.

*DUALCESE_INITIAL_HYBRID_SET

Purpose: Specifies values to use for velocity, pressure, etc. for problem initialization of a hybrid multiphase model in an element subset of the dual CESE mesh. The values specified here override the values set on *DUALCESE_INITIAL_HYBRID for the element subset.

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

Card 2	1	2	3	4	5	6	7	8
Variable	Z1	RA	UIC	VIC	WIC	RHO1	RHO_A	RHO_B
Type	F	F	F	F	F	F	F	F
Default	none	none	none	none	none	none	none	none

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

VARIABLE	DESCRIPTION
ESID	Element set ID (see *DUALCESE_ELEMENTSET)
IFUNC	Option to define initial conditions using *DEFINE_FUNCTION cards:

VARIABLE	DESCRIPTION
	EQ.0: Not in use. EQ.1: All values for initial velocity, pressure, and temperature now refer to *DEFINE_FUNCTION IDs. In these functions, the following parameters are allowed: f(x,y,z), meaning that each variable's initial profile is a function of position.
Z1	Volume fraction of material 1 (or color function). This is usually a value of 0 or 1. For numerical stability, however, use a small value instead of 0.
RA	Mass fraction of the reactant (material α) with respect to material 2 (the explosive mixture)
UIC, VIC, WIC	Multiphase flow velocity components in the x, y, and z-directions, respectively
RHO1	Density of material 1
RHO_A	Density of the reactant (material α)
RHO_B	Density of the product (material β)
PIC	Equilibrium multifluid pressure
TIC	Equilibrium multifluid temperature

*DUALCESE_INITIAL_PHASE_CHANGE

Purpose: Specify values to use for velocity, pressure, etc. for problem initialization of a phase-change multiphase model on the dual CESE mesh. Note that these values can be overridden in some mesh elements by use of the *DUALCESE_INITIAL_PHASE_CHANGE_SET card.

The phase-change multiphase model involves three fluids: a liquid, its vapor, and other gases.

Card 1	1	2	3	4	5	6	7	8
Variable	UIC	VIC	WIC	RHO	PIC	TIC	Y1	Y2
Type	F	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	Y3			IFUNC				
Type	F			I				
Default	none			0				

VARIABLE	DESCRIPTION
UIC, VIC, WIC	Multiphase flow velocity components in the x, y, and z-directions, respectively
RHO	Mixture density
PIC	Equilibrium multifluid pressure
TIC	Equilibrium multifluid temperature
Y1	Mass fraction of fluid 1
Y2	Mass fraction of fluid 2

VARIABLE	DESCRIPTION
Y3	Mass fraction of fluid 3
IFUNC	Option to define initial conditions with *DEFINE_FUNCTION: EQ.0: Not in use. EQ.1: All values for initial velocity, pressure, and temperature now refer to *DEFINE_FUNCTION IDs. In these functions, the following parameters are allowed: f(x, y, z). Thus, each variable's initial profile is a function of position.

*DUALCESE_INITIAL_PHASE_CHANGE_SET

Purpose: Specify values to use for velocity, pressure, etc. for problem initialization of a phase- change multiphase model in an element subset of the dual CESE mesh. The values specified here override the values set on *DUALCESE_INITIAL_PHASE_CHANGE for the element subset.

Card 1	1	2	3	4	5	6	7	8
Variable	ESID	IFUNC						
Type	I	I						
Default	none	0						

Card 2	1	2	3	4	5	6	7	8
Variable	UIC	VIC	WIC	RHO	PIC	TIC	Y1	Y2
Type	F	F	F	F	F	F	F	F
Default	none	none	none	none	none	none	none	none

Card 3	1	2	3	4	5	6	7	8
Variable	Y3							
Type	F							
Default	none							

VARIABLE	DESCRIPTION
ESID	Element set ID (see *DUALCESE_ELEMENTSET)
IFUNC	Option to define initial conditions with *DEFINE_FUNCTION: EQ.0: Not in use.

VARIABLE	DESCRIPTION
	EQ.1: All values for initial velocity, pressure, and temperature now refer to *DEFINE_FUNCTION IDs. In these functions, the following parameters are allowed: f(x, y, z). Thus, each variable's initial profile is a function of position.
UIC, VIC, WIC	Multiphase flow velocity components in the x, y, and z-directions, respectively
RHO	Mixture density
PIC	Equilibrium multifluid pressure
TIC	Equilibrium multifluid temperature
Y1	Mass fraction of fluid 1
Y2	Mass fraction of fluid 2
Y3	Mass fraction of fluid 3

*DUALCESE_INITIAL_TWO-PHASE

Purpose: Specify values to use for velocity, pressure, etc. for problem initialization of a two-phase multifluid model on the dual CESE mesh. Note that these values can be overridden in some mesh elements by use of the *DUALCESE_INITIAL_TWO-PHASE_SET card.

Card 1	1	2	3	4	5	6	7	8
Variable	Z1	UIC	VIC	WIC	RHO_1	RHO_2	PIC	TIC
Type	F	F	F	F	F	F	F	F
Default	none	none	0.	0.	0.	none	none	none

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

VARIABLE	DESCRIPTION
Z1	Volume fraction of material 1 (or color function)
UIC, VIC, WIC	Multiphase flow velocity components in the x, y, and z-directions respectively.
RHO_1	Density of fluid 1
RHO_2	Density of fluid 2
PIC	Equilibrium multifluid pressure
TIC	Equilibrium multifluid temperature
IFUNC	Option to define initial conditions using *DEFINE_FUNCTION cards:

VARIABLE	DESCRIPTION
	EQ.0: Not in use.
	EQ.1: All values for initial velocity, pressure, and temperature now refer to *DEFINE_FUNCTION IDs. In these functions, the following parameters are allowed: f(x,y,z), meaning that each variable's initial profile is a function of position.

*DUALCESE_INITIAL_TWO-PHASE_SET

Purpose: Specify values to use for velocity, pressure, etc. for problem initialization of a two-phase multifluid model in an element subset of the dual CESE mesh.

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

Card 2	1	2	3	4	5	6	7	8
Variable	Z1	UIC	VIC	WIC	RHO_1	RHO_2	PIC	TIC
Type	F	F	F	F	F	F	F	F
Default	none	none	none	none	none	none	none	none

VARIABLE	DESCRIPTION
ESID	Element set ID (see *DUALCESE_ELEMENTSET)
IFUNC	Option to define initial conditions using *DEFINE_FUNCTION cards: EQ.0: Not in use. EQ.1: All values for initial velocity, pressure, and temperature now refer to *DEFINE_FUNCTION IDs. In these functions, the following parameters are allowed: f(x,y,z), meaning that each variable's initial profile is a function of position.
Z1	Volume fraction of material 1 (or color function)
UIC, VIC, WIC	Multiphase flow velocity components in the x, y, and z-directions, respectively

VARIABLE	DESCRIPTION
RHO_1	Density of material 1
RHO_2	Density of material 2
PIC	Equilibrium multiphase flow pressure
TIC	Equilibrium multiphase flow temperature

*DUALCESE_MAT_GAS

Purpose: Define the fluid (gas) properties in a viscous flow for the dual CESE solver. In this model, the dynamic viscosity is determined using Sutherland's formula for viscosity, and the thermal conductivity is determined using the Prandtl Number.

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 ID
C1, C2	Two coefficients in the Sutherland's formula for viscosity: $$\mu = \frac{C_1 T^{3/2}}{T + C_2} \ .$$ Here 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	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. **Inviscid flows.** C1 and C2 are only used to calculate the viscosity in viscous flows, so for inviscid flows, this material card is not needed.

2. **Thermal coupling.** The Prandtl number is used to extract the thermal conductivity. It is only needed when thermal coupling with the structure is being done. However, the dual CESE does not include a conjugate heat transfer capability in the Dual CESE solver.

3. **Unit consistency.** As with other solvers in LS-DYNA, you are responsible for unit consistency. For example, if dimensionless variables are used, C_1 and C_2 should be replaced by the corresponding dimensionless ones. If the dual CESE model has a specified system of units either directly from the *DUALCESE_-MODEL card or inherited from the overall problem input, then these values need to be given in that unit system. Also, note that the formulas here require the temperature be given in either Kelvin or Rankine units.

*DUALCESE_MAT_GAS_0

Purpose: Define the fluid (gas) properties in a viscous flow for the dual CESE solver. In this model, the dynamic viscosity and thermal conductivity are specified constants.

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 ID
MU	Fluid dynamic viscosity. MU = 1.81×10^{-5} kg/ms for air at 15°C.
K	Thermal conductivity of the fluid

Remarks:

1. **Fields that depend on the problem's physics.** The viscosity is only used in viscous flows, so it is not necessary to define it for inviscid flows. The thermal conductivity is only used when coupling is activated between the structure and thermal solver to calculate the heat transfer. However, the Dual CESE solver does not include a conjugate heat transfer capability.

2. **Unit consistency.** As with other solvers in LS-DYNA, you are responsible for unit consistency. For example, if dimensionless variables are used, MU should be replaced by the corresponding dimensionless one. If the dual CESE model has a specified system of units either directly from the *DUALCESE_MODEL card or inherited from the overall problem input, then these values need to be given in that unit system.

*DUALCESE_MAT_GAS_2

Purpose: Define the fluid (gas) properties in a viscous flow for the dual CESE solver. This model determines dynamic viscosity and thermal conductivity by combining Sutherland's formula with the Power law for dilute gases.

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 ID
MU0 / SMU	Two coefficients appearing in the equation derived by combining Sutherland's formula with the Power law for dilute gases: $$\frac{\mu}{\mu_0} = \left(\frac{T}{T_0}\right)^{3/2} \frac{T_0 + S_\mu}{T + S_\mu} \ .$$ Here μ is the dynamic viscosity, 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: $$\frac{k}{k_0} = \left(\frac{T}{T_0}\right)^{3/2} \frac{T_0 + S_k}{T + S_k} \ .$$ Here 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, $$k_0 = 0.0241\ \ \mathrm{W/m}\,, \qquad S_k = 194\ \mathrm{K}$$
T0	Reference temperature, T_0. The default value (273.0) is for air in K.

Remarks:

1. **Fields that depend on the problem's physics.** The viscosity is only used for viscous flows, so it is not necessary to define it for inviscid flows. The thermal conductivity is only used to calculate the heat transfer between the structure and the thermal solver when coupling is activated. However, the dual CESE solver does not include a conjugate heat transfer capability.

2. **Unit consistency.** As with other solvers in LS-DYNA, you are responsible for unit consistency. For example, if dimensionless variables are used, MU should be replaced by the corresponding dimensionless one. If the dual CESE model has a specified system of units either directly from the *DUALCESE_MODEL card or inherited from the overall problem input, then these values need to be given in that unit system. Also, note that the formulas here require the temperature be given in either Kelvin or Rankine units.

*DUALCESE_MAT_LIQUID

Purpose: Define the fluid (liquid) properties in a viscous flow for the dual CESE solver. The parameters are used to calculate the viscosity of the liquid. This material model is usually accompanied by a liquid EOS (see *DUALCESE_EOS_NASG). This keyword is only available for single fluid, viscous flows for either no FSI or FSI-IBM.

Card 1	1	2	3	4	5	6	7	8
Variable	MID	A	B			K		
Type	I	F	F			F		
Default	none	1.732E-6	1863			0.579		

VARIABLE	DESCRIPTION
MID	Material ID
A, B	Two coefficients in the Andrade's formula for the liquid's viscosity: $$\mu = A \exp\left(\frac{B}{T}\right)$$ Here T is the absolute temperature of the liquid. See Remark 1. The default values taken from the values for water at 10°C: $A = 1.732 \times 10^{-6}\ \mathrm{N \cdot s/m^2}$ and $B = 1863$ K.
K	Thermal conductivity of the liquid. The default value is taken from that for water at 10°C: 0.579 W/m·K.

Remarks:

1. **Properties in Andrade's formula.** The values of A and B need to be determined for the liquid by fitting the equation to experimental viscosity data at different temperatures.

*DUALCESE_MESH_GEOMETRY

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

Card 1	1	2	3	4	5	6	7	8
Variable	GEOM	IDC						
Type	A	F						
Default	none	0.25						
Remarks	1	2						

VARIABLE	DESCRIPTION
GEOM	Sets the geometric dimension: EQ.2D: Two-dimensional (2D) problem EQ.3D: Three-dimensional (3D) problem EQ.AXI: 2D axisymmetric
IDC	Contact interaction detection coefficient (for FSI problems)

Remarks:

1. **Mesh and Boundary Conditions for 2D Problems.** If you want to use the 2D (GEOM = 2D) or 2D axisymmetric (GEOM=AXI) solver, the mesh should only be distributed in the *xy*-plane with the boundary conditions given only at the *xy* domain boundaries. Otherwise, a warning message will be given, and the 3D solver will be triggered instead.

   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.

2. **Contact Interaction Detection Coefficient.** 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.

3. **Dual CESE Solver and Restarts.** The dual CESE solver is *not* currently supported for restarts.

*DUALCESE_MESH_PART

Purpose: Define dual CESE solver mesh parts, that is, specify the dual CESE FSI algorithm to use for a mesh part. In other words, this keyword allows you to restrict the type of solver (immersed boundary FSI, moving mesh FSI, or non-FSI Eulerian) used on a region of a dual CESE mesh. The part ID specified with this keyword corresponds to the second field of Card 1 for either *DUALCESE_ELE2D or *DUALCESE_ELE3D.

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

Card 1	1	2	3	4	5	6	7	8
Variable	PID	FSITYPE	MMSHID					
Type	I	A	I					
Default	none	optional	0					

VARIABLE	DESCRIPTION
PID	Part ID (must be different from any other *DUALCESE_MESH_-PART part ID). It is important to note that *DUALCESE_PART and *DUALCESE_PART_MULTIPHASE cards should not be used when the newer *DUALCESE_MESH_PART card is used.
FSITYPE	FSI type to use on this part: EQ.<BLANK>: If left blank, no FSI performed. EQ.IBM: Immersed boundary FSI solver EQ.MOVMESH: Moving mesh FSI solver (FSITYPE = MMM may also be used for the same effect)
MMSHID	ID for the mesh motion algorithm to use for the moving mesh FSI solver on this part (region of the current dual CESE mesh). This ID refers to the ID of an instantiation of *DUALCESE_CONTROL_-MESH_MOV.

*DUALCESE_MODEL

Purpose: Set the units used by a dual CESE compressible flow problem, along with the name of the file specifying the dual CESE model. There can be any number of such models (each with a separate mesh), and each such model must be in a different file.

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

VARIABLE	DESCRIPTION
UNITSYS	Name of the unit system of this dual CESE model (defined with *UNIT_SYSTEM). EQ.<BLANK>: Use same units as the presumed units of the entire problem.
FILENAME	Filename of the keyword file containing the dual CESE model. Note that only *DUALCESE_... keyword cards are allowed in this file.

*DUALCESE_NODE2D

Purpose: Define a node and its coordinates in the global coordinate system. The nodal point ID must be unique relative to other nodes defined with *DUALCESE_NODE2D or *DUALCESE_NODE3D cards.

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						
Type	I	F		F						
Default	none	0.		0.						

VARIABLE	DESCRIPTION
NID	Node number
X	x coordinate
Y	y coordinate

*DUALCESE_NODE3D

Purpose: Define a node and its coordinates in the global coordinate system. The nodal point ID must be unique relative to other nodes defined with *DUALCESE_NODE3D or *DUALCESE_NODE2D cards.

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 number
X	*x* coordinate
Y	*y* coordinate
Z	*z* coordinate

*DUALCESE_NODESET

Purpose: Define a nodal set of dual CESE mesh nodes.

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

Node ID Cards. Set one value per node in the set. 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

VARIABLE	DESCRIPTION
NSID	Set ID of new node set. All dual CESE node sets should have a unique set ID.
NID*i*	Node ID *i*

*DUALCESE_PART

Purpose: Define dual CESE solver parts, that is, specify the dual CESE material and EOS information for a part. It also provides a means to restrict the type of solver used on a region of a dual CESE mesh, meaning an immersed boundary FSI solver, a moving mesh FSI solver, or a non-FSI Eulerian solver can be specified for just this part.

WARNING:	As of version R14, this keyword is deprecated. Instead, for the part ID and FSI solver details, use *DUALCESE_MESH_PART. For the material properties, use the *DUALCESE_SOLVER_... card corresponding to the choice made with *DUALCESE_SOLVER_SELECTION.

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

Card 1	1	2	3	4	5	6	7	8
Variable	PID	MID	EOSID	FSITYPE	MMSHID			
Type	I	I	I	A	I			
Default	none	none	none	optional	0			

VARIABLE	DESCRIPTION
PID	Part ID (must be different from any other *DUALCESE_PART, *DUALCESE_PART_MULTIPHASE, *DUALCESE_MESH_PART part ID)
MID	Material ID referring to a *DUALCESE_MAT_… material (see Remark 1)
EOSID	Equation of state ID referring to a *DUALCESE_EOS_… EOS
FSITYPE	FSI type to use on this part: EQ.<BLANK>: If left blank, no FSI performed. EQ.IBM: Immersed boundary FSI solver EQ.MOVMESH: Moving mesh FSI solver (FSITYPE = MMM may also be used for the same effect)

VARIABLE	DESCRIPTION
MMSHID	ID for the mesh motion algorithm to use for the moving mesh FSI solver on this part (region of the current dual CESE mesh). This ID refers to a *DUALCESE_CONTROL_MESH_MOV card ID.

Remarks:

1. **MID for Inviscid Flows.** Since material coefficients are only used in viscous flows, MID can be left blank for inviscid flows.

*DUALCESE_PART_MULTIPHASE

Purpose: Define dual CESE multiphase solver parts, that is, specify the dual CESE material and EOS information for a part. It also provides a means to restrict the type of solver used on a region of a dual CESE mesh, meaning an immersed boundary FSI solver, a moving mesh FSI solver, or a non-FSI Eulerian solver can be specified for just this part.

WARNING:	As of version R14, this keyword is deprecated. Instead, for the part ID and FSI solver details, use *DUALCESE_MESH_PART. For the material properties, use the *DUALCESE_SOLVER_... card corresponding to the choice made with *DUALCESE_SOLVER_SELECTION.

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

Card 1	1	2	3	4	5	6	7	8
Variable	PID	REACT_ID	EOSSID	MID	FSITYPE	MMSHID		
Type	I	I	I	I	A	I		
Default	none	none	none	none	optional	0		

VARIABLE	DESCRIPTION
PID	Part ID (must be different from any PID on a *DUALCESE_PART, *DUALCESE_PART_MULTIPHASE, *DUALCESE_MESH_PART card)
REACT_ID	ID of chemical reaction rate model (see *DUALCESE_REACTION_RATE_... cards)
EOSSID	Set ID of multiphase EOS set specification (see *DUALCESE_EOS_SET)
MID	Material ID defined by a *DUALCESE_MAT_… card

VARIABLE	DESCRIPTION
FSITYPE	FSI type to use on this part: EQ.<BLANK>: If left blank, no FSI is performed. EQ.IBM: Immersed boundary FSI solver EQ.MOVMESH: Moving mesh FSI solver (FSITYPE = MMM may also be used for the same effect)
MMSHID	ID of the mesh motion algorithm to use for the moving mesh FSI solver on this part (region of the current dual CESE mesh). This ID refers to a *DUALCESE_CONTROL_MESH_MOV card ID.

*DUALCESE_POINTSET

Purpose: Define a list of points used to output at specified sample times variables from the chosen dual CESE solver to binary database binout.

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

Point Cards. 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	X	Y	Z					
Type	F	F	F					
Default	none	none	none					

VARIABLE	DESCRIPTION
PNTSID	ID for this point set which can be used by *DUALCESE_DATABASE_HISTORY_POINT_SET
X, Y, Z	Coordinates of a point. As many points as desired can be specified

*DUALCESE_POINT_SOURCE

Purpose: For the dual CESE compressible flow solver, define values at a point source (inside the fluid mesh) for mass flow rate, magnitude of velocity, density, pressure, and temperature.

Card Sets:

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

1. Card 1 specifies the position and direction where an injected point source is to be applied.

2. Card 2 reads in load curve IDs.

3. Card 3 reads in scale factors.

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

Card Summary:

Card 1. This card is required.

NSID							

Card 2. This card is required.

LC_MRTE	LC_V	LC_RHO	LC_P	LC_T			

Card 3. This card is required.

SF_MRTE	SF_V	SF_RHO	SF_P	SF_T			

Data Card Definitions:

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

VARIABLE	DESCRIPTION
NSID	The ID of a *DUALCESE_POINT_SOURCE_STRUCTNODE_SET card to determine where the point source values in this card should be set.

Card 2	1	2	3	4	5	6	7	8
Variable	LC_MRTE	LC_V	LC_RHO	LC_P	LC_T			
Type	I	I	I	I	I			
Default	0	0	0	0	0			
Remarks	1	1	1	1	1			

VARIABLE	DESCRIPTION
LC_MRTE	Load curve ID (see *DEFINE_CURVE) to describe the mass flow rate as a function of time. EQ.0: Mass flow rate is a constant with value SF_MRTE. EQ.-1: Mass flow rate is computed by the solver.
LC_V	Load curve ID to describe the magnitude of the velocity as a function of time. EQ.0: The magnitude of velocity is a constant with value SF_V. EQ.-1: The magnitude of velocity is computed by the solver.

VARIABLE	DESCRIPTION
LC_RHO	Load curve ID to describe the density as a function of time. EQ.0: Density is a constant with value SF_RHO. EQ.-1: Density is computed by the solver.
LC_P	Load curve ID to describe the pressure as a function of time. EQ.0: Pressure is a constant with value SF_P. EQ.-1: Pressure is computed by the solver.
LC_T	Load curve ID to describe the temperature as a function of time. EQ.0: Temperature is a constant with value SF_T. EQ.-1: Temperature is computed by the solver.

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

VARIABLE	DESCRIPTION
SF_MRTE	Scale factor for LC_MRTE
SF_V	Scale factor for LC_V
SF_RHO	Scale factor for LC_RHO
SF_P	Scale factor for LC_P
SF_T	Scale factor for LC_T

Remarks:

1. **Consistent flow injection values.** On each point source, the variables ($mrate$, v, ρ, P, T) that are given values must be consistent and make the model well-posed (meaning be such that the solution of the model exists, is unique, and is physical).

*DUALCESE_POINT_SOURCE_TWO-PHASE

Purpose: For the dual CESE compressible two-phase multiphase flow solver, this keyword specifies values at a point source (inside the fluid mesh) for the mass flow rate, volume fraction of material 1, magnitude of the velocity, density of material 1, density of material 2, pressure, and temperature.

Card Sets:

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

1. Card 1 specifies the position and direction where an injected point source is to be applied.

2. Card 2 reads in load curve IDs.

3. Card 3 reads in scale factors.

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

Card Summary:

Card 1. This card is required.

NSID							

Card 2. This card is required.

LC_MRTE	LC_Z1	LC_V	LC_D1	LC_D2	LC_P	LC_T	

Card 3. This card is required.

SF_MRTE	SF_Z1	SF_V	SF_D1	SF_D2	SF_P	SF_T	

Data Card Definitions:

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

VARIABLE	DESCRIPTION
NSID	ID of a *DUALCESE_POINT_SOURCE_STRUCTNODE_SET where the point source values in this card should be set.

Card 2	1	2	3	4	5	6	7	8
Variable	LC_MRTE	LC_Z1	LC_V	LC_D1	LC_D2	LC_P	LC_T	
Type	I	I	I	I	I	I	I	
Default	0	0	0	0	0	0	0	
Remarks	1	1	1	1	1	1	1	

VARIABLE **DESCRIPTION**

LC_MRTE

Load curve ID (see *DEFINE_CURVE) to describe the mass flow rate as a function of time.

EQ.0: Constant mass flow rate with value SF_MRTE

EQ.-1: The solver computes the mass flow rate.

LC_Z1

Load curve ID to describe the magnitude of the volume fraction of material 1 as a function of time.

EQ.0: Constant volume fraction for material 1 with value SF_Z1

EQ.-1: The solver computes the volume fraction of material 1.

VARIABLE	DESCRIPTION
LC_V	Load curve ID to describe the magnitude of the velocity as a function of time. EQ.0: Constant magnitude of velocity with value SF_V EQ.-1: The solver computes the magnitude of velocity.
LC_D1	Load curve ID to describe the density of material 1 as a function of time. EQ.0: Constant density for material 1 with value SF_D1 EQ.-1: The solver computes the density of material 1.
LC_D2	Load curve ID to describe the density of material 2 as a function of time. EQ.0: Constant density for material 2 with value SF_D2 EQ.-1: The solver computes the density of material 2.
LC_P	Load curve ID to describe the pressure as a function of time. EQ.0: Constant pressure is a constant with value SF_P EQ.-1: The solver computes the pressure.
LC_T	Load curve ID to describe the temperature as a function of time. EQ.0: Constant temperature with value SF_T EQ.-1: The solver computes the temperature.

Card 3	1	2	3	4	5	6	7	8
Variable	SF_MRTE	SF_Z1	SF_V	SF_D1	SF_D2	SF_P	SF_T	
Type	F	F	F	F	F	F	F	
Default	1.0	1.0	1.0	1.0	1.0	1.0	1.0	

VARIABLE	DESCRIPTION
SF_MRTE	Scale factor for LC_MRTE
SF_Z1	Scale factor for LC_Z1

VARIABLE	DESCRIPTION
SF_V	Scale factor for LC_V
SF_D1	Scale factor for LC_D1
SF_D2	Scale factor for LC_D2
SF_P	Scale factor for LC_P
SF_T	Scale factor for LC_T

Remarks:

1. **Consistent flow injection values.** On each point source, the variables ($mrate$, $z1, v$, $\rho 1$, $\rho 2$, P, T) that are given values must be consistent and make the model well-posed (meaning be such that the solution of the model exists, is unique, and is physical).

*DUALCESE_POINT_SOURCE_STRUCTNODE_SET

Purpose: Provide the position, direction, and size (area of the orifice) of a flow to be injected into a background flow field.

Node Set ID Card.

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

Orifice Definition 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	NID	VID	ORIFA					
Type	I	I	F					
Default	none	none	1.0					

VARIABLE	DESCRIPTION
NSID	ID of this set of orifice definitions
NID	Structural node that gives the position of the orifice
VID	ID of a vector defined by *DEFINE_VECTOR that gives the direction of the flow away from the position defined by NID
ORIFA	Surface area of the orifice

*DUALCESE_POROUS_MODEL

Purpose: Specify the method for treating porous media flow in porous media fluid-structure interaction. Although the literature contains several methods, currently only the jump condition method is available. Other methods may be added in the future.

Card 1	1	2	3	4	5	6	7	8
Variable	METHOD							
Type	C							
Default	JUMP							

VARIABLE	DESCRIPTION
METHOD	Method for modeling the porous-media interface: EQ.JUMP: Jump method (default). See *DUALCESE_POROUS_SPECIFY_JUMP.

Remarks:

1. **Availability.** The porous media flow capability is available for single-fluid FSI-IBM and two-phase FSI-IBM solvers including for viscous and inviscid flows.

*DUALCESE_POROUS_SPECIFY_JUMP

Purpose: Specify the properties of the porous media used in the jump method (see *DUALCESE_POROUS_MODEL). These properties include the structural part or part set of the porous medium, the thickness of the medium, and the model for mimicking the porosity of the materials.

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

VARIABLE	DESCRIPTION
PID	Structural part ID of the porous medium
THICKNSS	Thickness of the medium
EQID	ID of the equation used to model the porous medium (see *DUALCESE_DARCY-FORCHHEIMER_EQ)

*DUALCESE_REACTION_RATE_IG

Purpose: Define a reaction rate law for the Ignition and Growth model to describe the conversion of reactants to products in the modeling of a condensed phase explosive in a dual CESE multiphase model. See Michael and Nikiforakis 2016 and Tarver 2005 for details about this law.

Card 1	1	2	3	4	5	6	7	8
Variable	REACT_ID	IGN	AA	BB	XX	GROW1	CC	DD
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	YY	GROW2	EE	GG	ZZ	IGMAX	G1MAX	G2MAX
Type	F	F	F	F	F	F	F	F
Default	none	none	none	none	none	none	none	none

VARIABLE	DESCRIPTION
REACT_ID	ID of reaction rate law
IGN	Reaction ignition term parameter, I
AA	Reaction ignition term parameter, a
BB	Reaction ignition term parameter, b
XX	Reaction ignition term parameter, x
GROW1	Reaction growth term parameter, G_1
CC	Reaction growth term parameter, c
DD	Reaction growth term parameter, d
YY	Reaction growth term parameter, y

VARIABLE	DESCRIPTION
GROW2	Reaction completion term parameter, G_2
EE	Reaction completion term parameter, e
GG	Reaction completion term parameter, g
ZZ	Reaction completion term parameter, z
IGMAX	Maximum mass fraction of the product for reaction ignition term, Φ_{IGmax}
G1MAX	Maximum mass fraction of the product for reaction growth term, Φ_{G1max}
G2MAX	Maximum Φ for reaction completion term, Φ_{G2max}

Remarks:

Using the notation of Michael and Nikiforakis 2016, the reaction rate law can be given as:

$$\frac{d\Phi}{dt} = I(1-\Phi)^b(\rho - 1 - a)^x H(\Phi_{\mathrm{IGmax}} - \Phi) + G_1(1-\Phi)^c\Phi^d p^y H(\Phi_{\mathrm{G1max}} - \Phi) + G_2(1-\Phi)^e\Phi^g p^z H(\Phi - \Phi_{\mathrm{G2max}})$$

where H is the Heaviside function. Here Φ is the mass fraction of the products, p is the pressure, and ρ is the density of the explosive mixture. Note that the pressure is assumed to be in equilibrium between the phases. I, G_1, G_2, a, b, c, d, e, g, x, y, and z are constants. They depend on the explosive as well as the part of the detonation process being modeled; see Michael and Nikiforakis 2016 for details.

The constants Φ_{IGmax}, Φ_{G1max}, and Φ_{G2max} in the Heavyside functions indicate when each of the three stages of the reaction are dominant. It was developed by Tarver 2005 to describe the reaction of pressed solid explosives during shock initiation and detonation. Tarver 2005 developed this reaction law for pressed solid explosives to describe the observed reaction stages during shock initiation and detonation See Tarver 2005 for a description of each stage of the reaction.

*DUALCESE_REACTION_RATE_IG_REDUCED

Purpose: Define a reduced form reaction rate law compared to that defined with *DUALCESE_REACTION_RATE_IG for describing the conversion of reactants to products in the modeling of a condensed phase explosive in a dual CESE multiphase model. See Michael and Nikiforakis 2016 for details about this reaction rate law.

Card 1	1	2	3	4	5	6	7	8
Variable	REACT_ID	GROW1	CC	DD	YY	PHI0		
Type	I	F	F	F	F	F		
Default	none	none	none	none	none	none		

VARIABLE	DESCRIPTION
REACT_ID	ID of reaction rate law
GROW1	Reaction growth term parameter, G_1
CC	Reaction growth term parameter, c
DD	Reaction growth term parameter, d
YY	Reaction growth term parameter, y
PHI0	Additional parameter to account for the non-zero amount of reaction when the mass fraction of the products, ϕ, is zero

Remarks:

Michael and Nikiforakis 2016 simplified the Ignition and Growth model reaction law to a pressure dependent law with a single stage to make the reduced model. They excluded the first and third terms from the Ignition and Growth model to makes this law. However, to account for a finite amount of reaction from ignition when ϕ is zero, they added a constant ϕ_0. This reduced law has the form:

$$\frac{d\phi}{dt} = G_1(1-\phi)^c(\phi+\phi_0)^d p^y \quad .$$

All the parameters are the same as the Ignition and Growth model except Φ_0. See *DUALCESE_REACTION_RATE_IG and Michael and Nikiforakis 2016 for details.

*DUALCESE_REACTION_RATE_P_DEPEND

Purpose: Define an explicitly pressure-dependent reaction rate law for describing the conversion of reactants to products in the modeling of a condensed phase explosive in a dual CESE multiphase model. This law is from Banks et al 2008.

Card 1	1	2	3	4	5	6	7	8
Variable	REACT_ID	SIGMA	NU	N				
Type	I	F	F	F				
Default	none	none	none	none				

VARIABLE	DESCRIPTION
REACT_ID	ID of reaction rate law
SIGMA	A positive constant, σ
NU	A positive constant, ν
N	A positive constant, n

Remarks:

Using the notation of Michael and Nikiforakis 2016, this simple pressure dependent reaction rate law by Banks et al 2008 can be stated as:

$$\frac{d\phi}{dt} = \sigma \phi^{\nu} p^{n}$$

Here ϕ is the mass fraction of the products, p is the pressure, and σ, ν, and n are positive constants found from experimental data. As discussed by Banks et al 2008, this type of reaction rate law that depends explicitly on pressure is often preferred for modeling the detonation of solid explosives because the constants can be found by fitting experimental data and other physical variables are difficult to measure.

*DUALCESE_SEGMENTSET

Purpose: Define a set of segments. For three-dimensional geometries, a segment can be triangular or quadrilateral. For two-dimensional geometries, a segment is a line defined by two nodes.

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

Segment Cards. For each segment in the set include one card of this format. Include as many cards as necessary. This input ends at the next keyword ("*") card.

Card 2	1	2	3	4	5	6	7	8
Variable	N1	N2	N3	N4				
Type	I	I	I	I				

VARIABLE	DESCRIPTION
SID	Set ID. All segment sets should have a unique set ID.
N1	Nodal point n_1
N2	Nodal point n_2
N3	Nodal point n_3. To define a line segment, set N3 = N2.
N4	Nodal point n_4. To define a triangular segment, set N4 = N3. To define a line segment, set N4 = N2.

FACE	Hexahedron	Pentahedron	Pyramid	Tetrahedron
1	N1, N5, N8, N4	N1, N2, N5	N1, N4, N3, N2	N1, N2, N4
2	N2, N3, N7, N6	N4, N6, N3	N1, N2, N5	N2, N3, N4

FACE	Hexahedron	Pentahedron	Pyramid	Tetrahedron
3	N1, N2, N6, N5	N1, N4, N3, N2	N2, N3, N5	N1, N3, N2
4	N4, N8, N7, N3	N2, N3, N6, N5	N3, N4, N5	N1, N4, N3
5	N1, N4, N3, N2	N1, N5, N6, N4	N4, N1, N5	
6	N5, N6, N7, N8			

Table 5-1. Face definitions for volume dual CESE elements

*DUALCESE_SOLVER_SELECTION

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

Card 1	1	2	3	4	5	6	7	8
Variable	EQNS							
Type	A							
Default	EULER							
Remark	1							

VARIABLE	DESCRIPTION
EQNS	Select the equations being solved with the dual CESE solver: EQ.CAVITATION: Cavitation solver EQ.EULER: Euler equations EQ.HYBRID: Hybrid multiphase model EQ.NS: Navier-Stokes equations EQ.PHASE-CHNG: Phase change model EQ.TWO-PHASE: Two-phase multiphase model

Remarks:

1. **Dual CESE Solver and Restarts.** The dual CESE solver is *not* currently supported for restarts.

*DUALCESE_SOLVER_CAV_EQNS

Purpose: Set the properties of the cavitating flow mixture being solved with the cavitation solver.

Property Card. Include one card.

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

VARIABLE	DESCRIPTION
EOSID	Equation of state (EOS) ID that must refer to an EOS defined with *DUALCESE_EOS_CAV_HOMOG_EQUILIB

*DUALCESE_SOLVER_EULER_EQNS

Purpose: Set the properties of the gas flow being solved with the Euler equations solver.

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

VARIABLE	DESCRIPTION
EOSID	Equation of state (EOS) ID that refers to an EOS defined by one of the following keywords: *DUALCESE_EOS_IDEAL_GAS, *DUALCESE_EOS_INFLATOR1, or *DUALCESE_EOS_INFLATOR2.

*DUALCESE_SOLVER_HYBRID_MULTIPHASE

Purpose: Define the properties of the fluid and condensed phase materials via equations-of-state that are used together to compute the thermodynamic state in the hybrid multi-phase dual CESE solver. Also, specify the single-step reaction rate model for the chemical reactions.

Card 1	1	2	3	4	5	6	7	8
Variable	EOSINID	EOSRCTID	EOSPRDID	REACT_ID				
Type	I	I	I	I				
Default	none	none	none	none				
Remark	1	1	1	2				

VARIABLE	DESCRIPTION
EOSINID	EOS ID of the inert component of the multiphase mixture
EOSRCTID	EOS ID of the reactant phase of the multiphase mixture
EOSPRDID	EOS ID of the product phase of the multiphase mixture
REACT_ID	ID of chemical reaction rate model

Remarks:

1. **Supported EOS Types.** Each EOS can be one of the following types:

    *DUALCESE_EOS_IDEAL_GAS

    *DUALCESE_EOS_STIFFENED_GAS

    *DUALCESE_EOS_VAN_DER_WAALS_GENERALIZED

    *DUALCESE_EOS_COCHRAN_CHAN

    *DUALCESE_EOS_JWL.

2. **Supported Reaction Rate Types.** The reaction rate can be one of the following types:

*DUALCESE_REACTION_REAT_IG

*DUALCESE_REACTION_RATE_IG_REDUCED

*DUALCESE_REACTION_RATE_P_DEPEND

*DUALCESE_SOLVER_NAVIER_STOKES

Purpose: Set the properties of the fluid flow being solved with the Navier-Stokes equations solver. That is, specify the dual CESE material and EOS information to be used.

Property Card. Include one card.

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

VARIABLE	DESCRIPTION
EOSID	Equation of state (EOS) ID referring to an EOS defined by one of the following card types: *DUALCESE_EOS_IDEAL_GAS, *DUALCESE_EOS_INFLATOR1, or *DUALCESE_EOS_INFLATOR2.
MID	Material ID referring to a *DUALCESE_MAT_… material card (see Remark 1)

Remarks:

1. **MID for Inviscid Flows.** Since material coefficients are only used in viscous flows, MID can be left blank for inviscid flows.

***DUALCESE_SOLVER_PHASE_CHANGE**

Purpose: Define the properties of the fluid and condensed phase materials via equations of state that are used together to compute the thermodynamic state in the phase change multiphase dual CESE solver.

Card 1	1	2	3	4	5	6	7	8
Variable	EOSIDL	EOSIDV	EOSIDO					
Type	I	I	I					
Default	none	none	none					
Remarks	1	1	1					

VARIABLE	DESCRIPTION
EOSIDL	Equation of state (EOS) ID of the liquid component in the multiphase mixture
EOSIDV	Equation of state (EOS) ID of the vapor component in the multiphase mixture
EOSIDO	Equation of state (EOS) ID of the other gases in the multiphase mixture

Remarks:

1. **Available EOS models.** Each EOS can be one of the following types:

   *DUALCESE_EOS_NASG

*DUALCESE_SOLVER_TWO-PHASE_MULTIPHASE

Purpose: Define the properties of the fluid phase materials through equations-of-state that are used together to compute the thermodynamic state in the two-phase multiphase dual CESE solver. This solver does not compute phase changes.

Card 1	1	2	3	4	5	6	7	8
Variable	EOSID1	EOSID2						
Type	I	I						
Default	none	none						
Remarks	1	1						

VARIABLE	DESCRIPTION
EOSID1	EOS ID of the first inert component of the multiphase mixture
EOSID2	EOS ID of the second inert component of the multiphase mixture

Remarks:

1. **Supported EOS Types.** Each EOS can be one of the following types:

   *DUALCESE_EOS_IDEAL_GAS

   *DUALCESE_EOS_STIFFENED_GAS

   *DUALCESE_EOS_VAN_DER_WAALS_GENERALIZED

   *DUALCESE_EOS_COCHRAN_CHAN

   *DUALCESE_EOS_JWL