*MESH

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

*MESH_BL

*MESH_BL_SYM

*MESH_EMBEDSHELL

*MESH_INTERF

*MESH_NODE

*MESH_SIZE

*MESH_SIZE_SHAPE

*MESH_SURFACE_ELEMENT

*MESH_SURFACE_NODE

*MESH_SURFACE_NULL

*MESH_VOLUME

*MESH_VOLUME_ELEMENT

*MESH_VOLUME_NODE

*MESH_VOLUME_PART

*MESH_BL

Purpose: Define a boundary-layer mesh as a refinement on the volume-mesh. The boundary layer mesh is constructed by subdividing elements near the surface.

Boundary Layer Cards. Define as many cards as are necessary. The next keyword ("*") card terminates this input.

Card 1	1	2	3	4	5	6	7	8
Variable	PID	NELTH	BLTH	BLFE	BLST	BLDR		
Type	I	I	F	F	I	I		
Default	none	none	0.	0.	0	0		

VARIABLE	DESCRIPTION
PID	Part identifier for the surface element
NELTH	Number of elements normal to the surface (in the boundary layer) is NELTH + 1. See Remark 5.
BLTH	Boundary layer mesh thickness if BLST = 1 or 2. Growth scale factor if BLST = 3. Ignored if BLST = 0. See Remark 5.
BLFE	Distance between the wall and the first volume mesh node if BLST = 3. Scaling coefficient if BLST = 1 or 2. Ignored if BLST =0. See Remark 5.
BLST	Boundary layer mesh generation strategy: EQ.0: $2^{NELTH+1}$ subdivisions based on surface mesh size (default). See Remark 1. EQ.1: Power law using BLTH and NELTH with BLFE as a scale factor. See Remark 2 and Figure 8-1. EQ.2: Geometric series based on BLTH and BLFE. See Remark 3 and Figure 8-2. EQ.3: Repartition following a growth scale factor (BLTH). See Remark 4 and Figure 8-3.
BLDR	Boundary layer mesh generation during the dynamic relaxation phase when present (see *ICFD_CONTROL_GENERAL):

VARIABLE	DESCRIPTION
	EQ.0: On. No distinction for boundary layer mesh generation between the dynamic relaxation and transient phases.
	EQ.1: Off. Boundary layer mesh is not generated during the dynamic relaxation phase.

Remarks:

1. **BLST = 0.** For BLST = 0, for every additional NELTH, the automatic volume mesher will divide the elements closest to the surface by two so that the smallest element in the boundary layer mesh will have an aspect ratio of $2^{\text{NELTH}+1}$. A default boundary layer mesh thickness based on the surface mesh size will be chosen.

2. **BLST = 1.** For a constant repartition of the nodes in the boundary layer, use BLST = 1 with BLFE = 1. For BLST = 1, starting from the wall, the position of node n in the normal direction is given by:

$$X_n = \left(\frac{n}{\text{NELTH}+1}\right)^{[5\times(1-\text{BLFE})]} \frac{\text{BLTH}}{\sum_{i=1}^{\text{NELTH}+1}\left[i/(\text{NELTH}+1)\right]^{[5\times(1-\text{BLFE})]}}$$

3. **BLST = 2.** Setting BLFE = 1 makes BLST = 2 equivalent to BLST = 0 except that BLST = 2 allows you to specify the boundary layer thickness instead of automatically using the local surface mesh size. For BLST = 2, starting from BLTH from the wall, each newly inserted node will have its location closer to the wall, following this law:

$$X_n = (0.5 \times \text{BLFE})^n \times \text{BLTH}(1 - 0.5 \times \text{BLFE})$$

4. **BLST = 3.** For BLST = 3, starting from the wall, the position of node n in the normal direction is given by:

$$X_n = \sum_{i=0}^{n} \text{BLFE} \times \text{BLTH}^i \text{ with } 0 \le n \le \text{NELTH}$$

5. **Dynamically changing boundary layer.** For NELTH, BLTH, and BLFE, setting a negative value will point to a time-dependent load curve which enables dynamically changing the boundary layer settings in cases involving remeshing (including returning a 0 value for the number of elements which effectively disables the boundary layer). NELTH and BLTH also accept a *DEFINE_FUNCTION as input with time and the x, y, and z coordinates accepted as arguments.

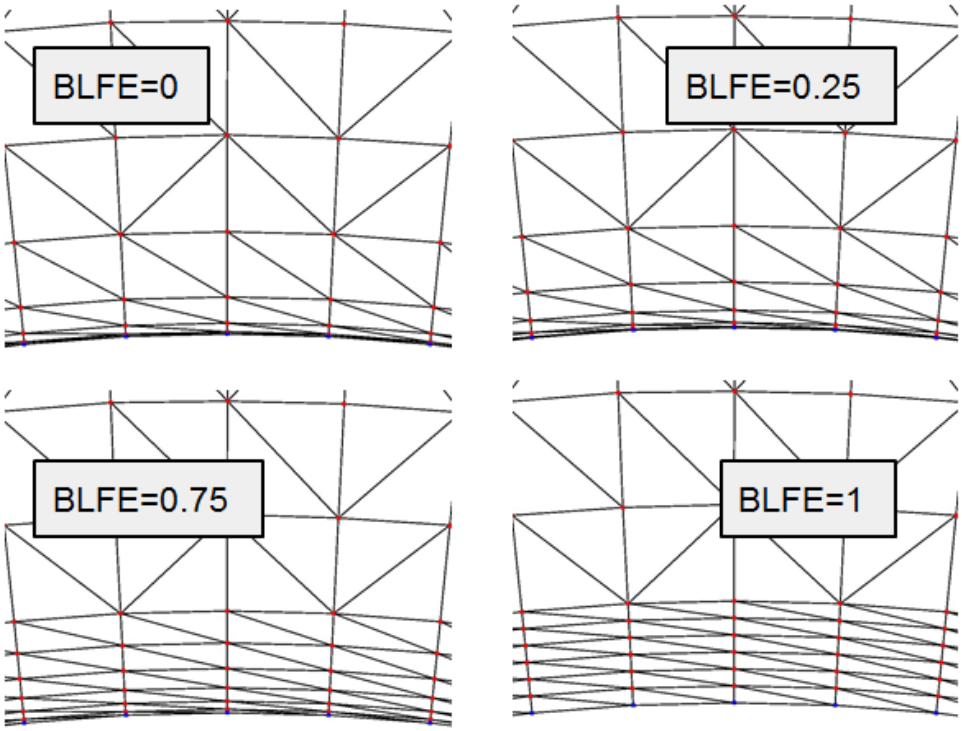

Figure 8-1. BLST = 1 example

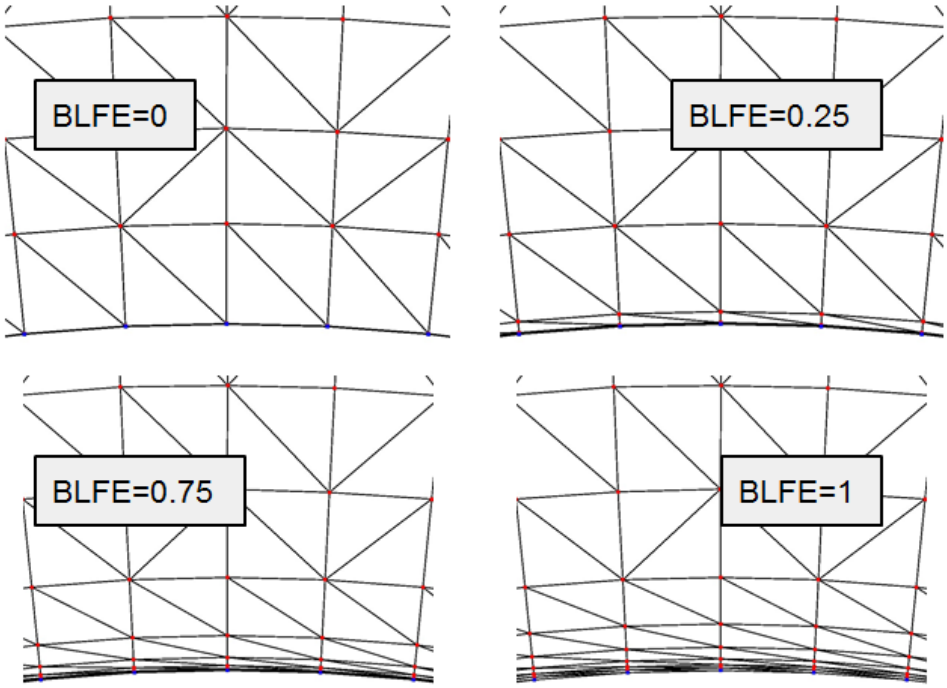

Figure 8-2. BLST = 2 example

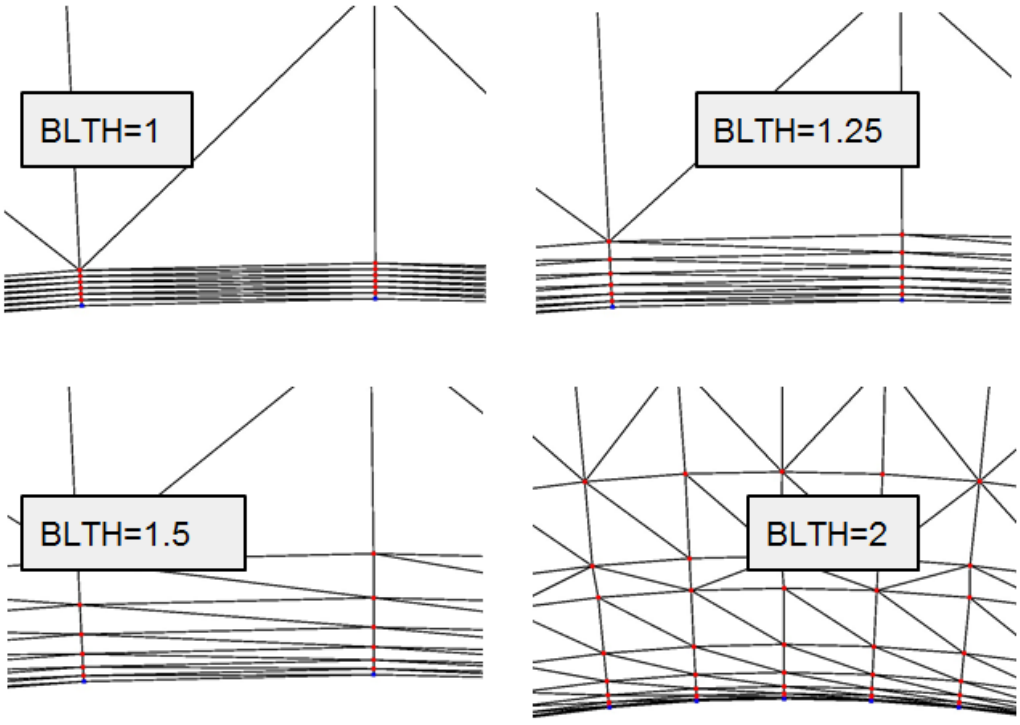

Figure 8-3. BLST = 3 example

*MESH_BL_SYM

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

Boundary Layer with Symmetry Condition Cards. Define as many cards as necessary. The next "*" card terminates the input.

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

VARIABLE	DESCRIPTION
PID1, …	Part identifiers for the surface element. This is the surface with symmetry.

*MESH_EMBEDSHELL

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

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

Define as many cards as are necessary based on the number of PIDs (the next "*" card terminates the input.)

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

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

*MESH_INTERF

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

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

Define as many cards as are necessary based on the number of PIDs. This input ends at the next keyword ("*") card.

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

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

*MESH_NODE

Purpose: Define a fluid node and its coordinates. These nodes are used in the mesh generation process by the *MESH_SURFACE_ELEMENT keyword, or as user defined volume nodes by the *MESH_VOLUME_ELEMENT keyword.

Node Cards. Include one additional card for each node. This input ends at the next keyword ("*") card.

Card 1	1	2	3	4	5	6	7	8	9	10
Variable	NID	X		Y		Z				
Type	I	F		F		F				
Default	none	0		0		0				

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

Remarks:

1. The data card format for the *MESH_NODE keyword is identical to *NODE.
2. The *MESH_NODE keyword supersedes *MESH_SURFACE_NODE, which was for surfaces nodes as well as *MESH_VOLUME_NODE for, which was for volume nodes in user defined.

*MESH_SIZE

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

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

Define as many cards as are necessary based on the number of PIDs (the next "*" card terminates the input.).

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

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

*MESH_SIZE_SHAPE

Purpose: Defines a local mesh size in specific zones corresponding to given geometrical shapes (box, sphere, cylinder, and polynomial). The solver automatically applies the conditions specified during the generation of the volume mesh. This zone does not need to be entirely defined in the volume mesh. In the polynomial case, we recommend defining several zones for better mesh size control.

Card Summary:

Card Sets. Add as many sets of these cards as needed. This input ends with the next keyword ("*") card.

Card 1. This card is required.

SNAME	FORCE	METHOD	BT	DT			

Card 2a. Include this card if SNAME = box and METHOD = 0.

MSIZE	PMINX	PMINY	PMINZ	PMAXX	PMAXY	PMAXZ	

Card 2b. Include this card if SNAME = sphere and METHOD = 0.

MSIZE	RADIUS	CENTERX	CENTERY	CENTERZ			

Card 2c. Include this card if SNAME = cylinder and METHOD = 0.

MSIZE	RADIUS	PMINX	PMINY	PMINZ	PMAXX	PMAXY	PMAXZ

Card 2d. Include this card if SNAME = pol and METHOD = 0.

MSIZE	X	Y	Z	NX	NY	NZ	

Card 2e. Include this card if METHOD = 1.

MSIZE	RADIUS	PTID1	PTID2		RMIN		

Card 2f. Include this card if METHOD = 2.

MSIZE	RADIUS	NID1	NID2				

Data Card Definitions:

Card 1	1	2	3	4	5	6	7	8
Variable	SNAME	FORCE	METHOD	BT	DT			
Type	A	I	I	F	F			
Default	none	0	0	0.	10^{12}			

VARIABLE	DESCRIPTION
SNAME	Shape name. Possibilities include box, cylinder, pol, and sphere.
FORCE	Flag to keep the mesh size criteria even after performing remeshing: EQ.0: Exclude boundary layer elements (default). EQ.1: Include boundary layer elements.
METHOD	Specifies which method to use when defining the second card: EQ.0: Default, directly input the coordinates. EQ.1: Define the coordinates with *ICFD_DEFINE_POINT IDs. See Remark 1. EQ.2: Define the coordinates by using user node IDs from solid mechanics. This method is only available when a solid mechanics problem is present. See Remark 1.
BT/DT	Birth and death time of the mesh size area in cases where remeshing occurs.

Box Case. Card 2 for SNAME = box and METHOD = 0

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

VARIABLE	DESCRIPTION
MSIZE	Mesh size that needs to be applied in the zone of the shape. A negative value points to a *DEFINE_FUNCTION. If using a *DEFINE_FUNCTION, the following parameters are allowed: f(x, y, z, time).
PMIN*i*	x, y, or z value for the point of minimum coordinates
PMAX*i*	x, y, or z value for the point of maximum coordinates

Sphere Case. Card 2 for SNAME = sphere and METHOD = 0

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

VARIABLE	DESCRIPTION
MSIZE	Mesh size that needs to be applied in the zone of the shape
RADIUS	Radius of the sphere
CENTER*i*	x, y, and z coordinates of the sphere's center

Cylinder Case. Card 2 for SNAME = cylinder and METHOD = 0

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

VARIABLE	DESCRIPTION
MSIZE	Mesh size that needs to be applied in the zone of the shape
RADIUS	Radius of the cylinder
PMIN*i*	*x*, *y*, or *z* value for the point of minimum coordinates
PMAX*i*	*x*, *y*, or *z* value for the point of maximum coordinates

Polynomial Case. Card 2 for SNAME = pol and METHOD = 0

Cards 2d	1	2	3	4	5	6	7	8
Variable	MSIZE	X	Y	Z	NX	NY	NZ	
Type	F	F	F	F	F	F	F	
Default	none	none	none	none	none	none	none	

VARIABLE	DESCRIPTION
MSIZE	Mesh size that needs to be applied in the zone of the shape
X/Y/Z	Coordinates of starting point
NX/NY/NZ	Direction in which the mesh size will be applied

METHOD = 1 Case. Card 2 for METHOD = 1.

Cards 2e	1	2	3	4	5	6	7	8
Variable	MSIZE	RADIUS	PTID1	PTID2		RMIN		
Type	F	F	I	I		F		
Default	none	none	none	none		0.		

VARIABLE	DESCRIPTION
MSIZE	Mesh size that needs to be applied in the zone of the shape defined by SNAME

VARIABLE	DESCRIPTION
RADIUS	Radius of the sphere if SNAME is sphere or radius of the cylinder if SNAME is cylinder. Ignored otherwise.
PTID1	Point ID 1, referring to a *ICFD_DEFINE_POINT. It replaces PMIN*i* for box and cylinder, X/Y/Z for pol, and CENTER for sphere.
PTID2	Point ID 2. It is ignored if SNAME is sphere. It replaces PMAX*i* for box and cylinder and NX/NY/NZ for pol.
RMIN	Optional radius available if SNAME is sphere or cylinder to define an internal sphere/cylinder where the *MESH_SIZE_SHAPE does not apply. When defined, RMIN should be smaller than RADIUS.

METHOD = 2 Case. Card 2 for METHOD=2

Cards 2f	1	2	3	4	5	6	7	8
Variable	MSIZE	RADIUS	NID1	NID2				
Type	F	F	I	I				
Default	none	none	none	none				

VARIABLE	DESCRIPTION
MSIZE	Mesh size that needs to be applied in the zone of the shape defined by SNAME
RADIUS	Radius of the sphere if SNAME is sphere or radius of the cylinder if SNAME is cylinder. Ignored otherwise.
NID1/NID2	User node IDs that come from the solid mechanics problem. Equivalent usage to PTID1 and PTID2. NID1 replaces PMIN*i* for box and cylinder, X/Y/Z for pol, and CENTER for sphere. NID2 is ignored if SNAME is sphere. It replaces PMAX*i* for box and cylinder and NX/NY/NZ for pol.

Remarks:

1. **Moving coordinates.** METHOD = 2 and 3 have the advantage of allowing the region to move since the regions move with *ICFD_DEFINE_POINTs and nodes

from the solid mechanics' problem, respectively. Thus, these methods provide control over how the mesh size area evolves as a function of time (in cases where remeshing occurs).

*MESH_SURFACE_ELEMENT

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

Each solver that supports the *MESH volume mesher for building its volume mesh uses the PID given for each surface element specified with this keyword differently.

1. For the *ICFD solver, the *ICFD_PART keyword references the surface mesh PIDs.

2. For the *DUALCESE solver, the MSPIDs in *DUALCESE_BOUNDARY_... keywords cards reference the surface element PIDs. Note that when the dual CESE solver uses *MESH_SURFACE_ELEMENT, this card defines which boundary faces belong to each mesh surface PID. In this case, no other mechanism exists for defining these PIDs. Also, when the dual CESE solver is defined with *MESH cards, there should not be any *DUALCESE_SEGMENTSET cards related to the dual CESE mesh.

Surface Element Card. Define as many cards as necessary. The next keyword ("*") card terminates this input.

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

VARIABLE	DESCRIPTION
EID	Element ID. A unique number with respect to all *MESH_SURFACE_ELEMENTS cards.
PID	Mesh surface part ID. A unique identifier for the surface to which this mesh surface element belongs.
N1	Nodal point 1
N2	Nodal point 2

VARIABLE	DESCRIPTION
N3	Nodal point 3
N4	Nodal point 4

Remarks:

1. **Defining Surface Elements.** The convention for defining surface elements is the same as for *ELEMENT_SHELL. In the case of a triangular face, N3 = N4. In 2D N2 = N3 = N4. Note that the accepted card format is 6i8 (not 6i10).

*MESH_SURFACE_NODE

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

*MESH_NODE supersedes this card; so please use *MESH_NODE instead of this card.

Surface Node Cards. Include one card for each node. Include as many cards a necessary. This input ends at the next keyword ("*") card.

Card 1	1	2	3	4	5	6	7	8	9	10
Variable	NID	X		Y		Z				
Type	I	F		F		F				
Default	none	0		0		0				

VARIABLE	DESCRIPTION
NID	Node ID. This NID must be unique within the set of surface nodes.
X	*x* coordinate.
Y	*y* coordinate.
Z	*z* coordinate.

*MESH_SURFACE_NULL

Purpose: Specify a set of surface part IDs to be used during post-processing. The part IDs are defined with the *MESH_SURFACE_ELEMENT keyword. Elements from *MESH_SURFACE_ELEMENT that belong to a *MESH_SURFACE_NULL part are not used by the mesher to construct a volume mesh. These surface elements are not involved in any physics computation.

For the *ICFD solver, the *ICFD_PART keyword referencing the null surface mesh PIDs is not necessary.

Null Surface Element Card. Define as many cards as necessary. The next keyword ("*") card terminates this input.

Card 1	1	2	3	4	5	6	7	8	9	10
Variable	PID1	PID2	PID3	PID4	PID5	PID6				
Type	I	I	I	I	I	I				
Default	none	none	none	none	none	none				

VARIABLE	DESCRIPTION
PID*i*	Mesh surface part ID *i*

*MESH_VOLUME

Purpose: This keyword defines the volume space that will be meshed. The boundaries of the volume are the surfaces defined by *MESH_SURFACE_ELEMENT. The surfaces listed have to be non-overlapping, and should not leave any gaps or open spaces between the surface boundaries. On the boundary between two neighbor surfaces, nodes have to be in common (no duplicate nodes) and should match exactly on the interface. They are defined by the keyword *MESH_SURFACE_NODE. This card will be ignored if the volume mesh is specified by the user and not generated automatically.

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

Define as many cards as are necessary based on the number of PIDs (the next "*" card terminates the input.)

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

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

*MESH_VOLUME_ELEMENT

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

Volume Element Card. Define as many cards as necessary. The next "*" card terminates the input.

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

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

Remarks:

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

*MESH_VOLUME_NODE

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

*MESH_NODE supersedes this card; so please use *MESH_NODE instead of this card.

Volume Node Cards. Include as many cards in the following format as desired. This input ends at the next keyword ("*") card.

Card 1	1	2	3	4	5	6	7	8	9	10
Variable	NID	X		Y		Z				
Type	I	F		F		F				
Default	none	0		0		0				

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

*MESH_VOLUME_PART

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

Mesh Volume Part Card. Include as many cards in the following format as desired. This input ends at the next keyword ("*") card.

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

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