*ICFD

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

*ICFD_BOUNDARY_CONJ_HEAT

*ICFD_BOUNDARY_CONVECTION_TEMP

*ICFD_BOUNDARY_FLUX_TEMP

*ICFD_BOUNDARY_FREESLIP

*ICFD_BOUNDARY_FSI

*ICFD_BOUNDARY_FSI_EXCLUDE

*ICFD_BOUNDARY_FSI_FIXED

*ICFD_BOUNDARY_FSI_ONEWAY

*ICFD_BOUNDARY_FSWAVE

*ICFD_BOUNDARY_GROUND

*ICFD_BOUNDARY_NAVIERSLIP

*ICFD_BOUNDARY_NONSLIP

*ICFD_BOUNDARY_PERIODIC

*ICFD_BOUNDARY_PRESCRIBED_LEVELSET

*ICFD_BOUNDARY_PRESCRIBED_MOVEMESH

*ICFD_BOUNDARY_PRESCRIBED_PRE

*ICFD_BOUNDARY_PRESCRIBED_SPTRANSP_CONC

*ICFD_BOUNDARY_PRESCRIBED_TEMP

*ICFD_BOUNDARY_PRESCRIBED_TURBULENCE

*ICFD_BOUNDARY_PRESCRIBED_VEL

*ICFD_BOUNDARY_PRESCRIBED_VISCOELASTIC

*ICFD_BOUNDARY_WEAKVEL

*ICFD_BOUNDARY_WINDKESSEL

*ICFD_CONTROL_ADAPT

*ICFD_CONTROL_ADAPT_SIZE

*ICFD_CONTROL_ADVECTION

*ICFD_CONTROL_BACKFLOW

*ICFD_CONTROL_CONJ

*ICFD_CONTROL_DEM_COUPLING

*ICFD_CONTROL_EMBEDSHELL

*ICFD_CONTROL_FSI

*ICFD_CONTROL_GAP

*ICFD_CONTROL_GENERAL

*ICFD_CONTROL_IMPOSED_MOVE

*ICFD_CONTROL_LEVELSET

*ICFD_CONTROL_LOAD

*ICFD_CONTROL_MESH

*ICFD_CONTROL_MESH_MOV

*ICFD_CONTROL_MONOLITHIC

*ICFD_CONTROL_OUTPUT

*ICFD_CONTROL_OUTPUT_SUBDOM

*ICFD_CONTROL_OUTPUT_VAR

*ICFD_CONTROL_PARTITION

*ICFD_CONTROL_POROUS

*ICFD_CONTROL_STEADY

*ICFD_CONTROL_SURFMESH

*ICFD_CONTROL_TAVERAGE

*ICFD_CONTROL_TIME

*ICFD_CONTROL_TRANSIENT

*ICFD_CONTROL_TURB_SYNTHESIS

*ICFD_CONTROL_TURBULENCE

*ICFD_DATABASE_AVERAGE

*ICFD_DATABASE_DRAG

*ICFD_DATABASE_FLUX

*ICFD_DATABASE_FLUXSURF

*ICFD_DATABASE_FORCE_DEM

*ICFD_DATABASE_GOA

*ICFD_DATABASE_HTC

*ICFD_DATABASE_NODEAVG

*ICFD_DATABASE_NODOUT

*ICFD_DATABASE_NTEMPOUT

*ICFD_DATABASE_POINTAVG

*ICFD_DATABASE_POINTOUT

*ICFD_DATABASE_RESIDUALS

*ICFD_DATABASE_SSOUT

*ICFD_DATABASE_SSOUT_EXCLUDE

*ICFD_DATABASE_TEMP

*ICFD_DATABASE_TIMESTEP

*ICFD_DATABASE_TPD

*ICFD_DATABASE_TWINBUILDER

*ICFD_DATABASE_UINDEX

*ICFD_DATABASE_WETNESS

*ICFD_DEFINE_HEATSOURCE

*ICFD_DEFINE_NONINERTIAL

*ICFD_DEFINE_POINT

*ICFD_DEFINE_POROUS_REGION

*ICFD_DEFINE_RESIDENCETIMESOURCE

*ICFD_DEFINE_SOURCE

*ICFD_DEFINE_SPTRANSPSOURCE

*ICFD_DEFINE_TRANSFORM

*ICFD_DEFINE_TURBSOURCE

*ICFD_DEFINE_WAVE_DAMPING

*ICFD_INITIAL

*ICFD_INITIAL_LEVELSET

*ICFD_INITIAL_SPTRANSP

*ICFD_INITIAL_TEMPNODE

*ICFD_INITIAL_TURBULENCE

*ICFD_MAT

*ICFD_MODEL_NONNEWT

*ICFD_MODEL_POROUS

*ICFD_MODEL_SPECIES_TRANSPORT

*ICFD_MODEL_VISCOELASTIC

*ICFD_PART

*ICFD_PART_VOL

*ICFD_SECTION

*ICFD_SET_NODE

*ICFD_SOLVER_SPLIT

*ICFD_SOLVER_TOL_FSI

*ICFD_SOLVER_TOL_LSET

*ICFD_SOLVER_TOL_MMOV

*ICFD_SOLVER_TOL_MOM

*ICFD_SOLVER_TOL_MONOLITHIC

*ICFD_SOLVER_TOL_PRE

*ICFD_SOLVER_TOL_TEMP

*ICFD_BOUNDARY_CONJ_HEAT

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

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

Card 1	1	2	3	4	5	6	7	8
Variable	PID	CTYPE	VAL	SFLCID				
Type	I	I	F	I				
Default	none	0	0.	0				

VARIABLE	DESCRIPTION
PID	PID of the fluid surface in contact with the solid
CTYPE	Contact type: EQ.0: Constraint approach EQ.1: Mortar contact
VAL	Optional temperature drop if CTYPE = 0 or interface heat transfer coefficient if CTYPE = 1 (high value by default to ensure perfect contact).
SFLCID	Load curve ID used to describe scale factor on VAL value as a function of time; see *DEFINE_CURVE, *DEFINE_CURVE_FUNCTION, or *DEFINE_FUNCTION. If a *DEFINE_FUNCTION is used, the following parameters are allowed: f(x, y, z, vx, vy, vz, temp, pres, time).

*ICFD_BOUNDARY_CONVECTION_TEMP

Purpose: Impose a heat transfer coefficient on the boundary expressed as $h = \frac{q}{T_s - T_b}$

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

Card 1	1	2	3	4	5	6	7	8
Variable	PID	HLCID	HSF	TBLCID	TBSF			
Type	I	I	F	I	F			
Default	none	none	1.	none	1.0			

VARIABLE	DESCRIPTION
PID	PID for a fluid surface.
HLCID	Load curve ID to describe the heat transfer coefficient value versus time, see *DEFINE_CURVE,*DEFINE_CURVE_FUNCTION or *DEFINE_FUNCTION. If a DEFINE_FUNCTION is used, the following parameters are allowed: $f(x, y, z, vx, vy, vz, temp, pres, time)$.
HSF	Load curve scale factor applied on the heat transfer coefficient value. (default = 1.0)
TBLCID	Load curve ID to describe the environment (i.e bulk) temperature value versus time, see *DEFINE_CURVE,*DEFINE_CURVE_-FUNCTION or *DEFINE_FUNCTION. If a DEFINE_FUNCTION is used, the following parameters are allowed: $f(x, y, z, vx, vy, vz, temp, pres, time)$.
TBSF	Load curve scale factor applied on the environment value. (default = 1.0)

*ICFD_BOUNDARY_FLUX_TEMP

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

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

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

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

*ICFD_BOUNDARY_FREESLIP

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

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

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

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

*ICFD_BOUNDARY_FSI

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

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

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

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

*ICFD_BOUNDARY_FSI_EXCLUDE

Purpose: Specify which solid part IDs are excluded from the FSI search. No forces coming from the fluid will be transmitted to those parts.

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

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

VARIABLE	DESCRIPTION
PID	Part ID of a part from the solid mechanics problem that is to be excluded from the FSI analysis

*ICFD_BOUNDARY_FSI_FIXED

Purpose: Define fixed fluid surfaces that will be considered for contact with the solid surfaces for FSI. This keyword is similar to *ICFD_BOUNDARY_FSI, except the fluid surface cannot move. This restriction allows a solid surface to "slide" over the fluid and for the exchange of data, such as temperature in CHT applications. This keyword should not be defined if *ICFD_CONTROL_FSI is not defined.

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

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

VARIABLE	DESCRIPTION
PID	Part ID of the fluid surface that can be considered for contact with the solid domain

*ICFD_BOUNDARY_FSI_ONEWAY

Purpose: Specify which solid part IDs experience one-way FSI coupling. These part IDs follow the structural motion to capture the structural deformation.

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

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

VARIABLE	DESCRIPTION
PID	Part ID of a part from the solid mechanics problem for which only one-way FSI coupling occurs
IOWC	Indicates the coupling direction to the solver: EQ.1: The solid mechanics solver transfers displacements to the fluid solver. EQ.2: The fluid solver transfers stresses to the solid mechanics solver.

*ICFD_BOUNDARY_FSWAVE

Purpose: Impose a wave inflow boundary condition.

Card Summary:

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

Card 1. This card is required.

PID	WTYPE	H0	WAMP	WLENG	WMAX	SFLCID	WANG

Card 2. This card is included if WTYPE = 7.

WPEAK							

Data Card Definitions:

Include as many of this card and/or sets of this card with the next as needed. This input ends at the next keyword ("*") card.

Card 1	1	2	3	4	5	6	7	8
Variable	PID	WTYPE	H0	WAMP	WLENG	WMAX	SFLCID	WANG
Type	I	I	F	F	F	F	I	F
Default	none	none	none	none	none	none	↓	none

VARIABLE	DESCRIPTION
PID	PID for a fluid surface
WTYPE	Wave type: EQ.1: Stokes wave of first order EQ.2: Stokes wave of second order EQ.3: Stokes wave of fifth order EQ.4: Solitary wave EQ.5: Irregular waves using JONSWAP spectrum

VARIABLE	DESCRIPTION
	EQ.6: Irregular waves using One Parameter Pierson-Moskowitz spectrum EQ.7: Irregular waves using Two Parameter Pierson-Moskowitz spectrum
H0	Water level (from the bottom of the channel) for the unperturbed condition
WAMP	Wave amplitude or height for WTYPE = 1 and 4. Significant wave height for WTYPE = 5, 6, and 7.
WLENG	WTYPE.LE.2: Wave length WTYPE.EQ.3: Wave period WTYPE.EQ.4: Not used WTYPE.GE.5: Minimum wave frequency in spectrum (rad/sec)
WMAX	Maximum wave frequency in spectrum (rad/sec) for WTYPE = 5, 6, and 7. Angle between the boundary and the incident waves (in degrees) for WTYPE = 3.
SFLCID	Scale factor LCID on the wave amplitude for WTYPE = 1, 2 and 3. Number of wave modes (default = 1024) for WTYPE = 5, 6, and 7.
WANG	Angle between incoming wave direction and x-axis for z- and y-aligned gravity vector, or angle between incoming wave direction and y-axis for x-aligned gravity vector.

Card included for WTYPE = 7 only

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

VARIABLE	DESCRIPTION
WPEAK	Peak wave frequency in spectrum [rad/sec] for WTYPE = 7.

Remarks:

1. **Peak Wave Frequency for WTYPE = 6.** For the irregular waves using the One Parameter Pierson-Moskowitz spectrum, the peak wave frequency in the spectrum LS-DYNA calculates the peak wave frequency with:

$$0.4\sqrt{\frac{g}{H_s}}$$

Here g the gravity and H_s is the significant wave height input with WAMP.

*ICFD_BOUNDARY_GROUND

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

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

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

VARIABLE	DESCRIPTION
PID	PID of the fluid surface where a ground boundary condition is applied.

*ICFD_BOUNDARY_NAVIERSLIP

Purpose: Specify the fluid boundary with the Navier slip boundary condition.

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

Card 1	1	2	3	4	5	6	7	8
Variable	PID	FRIC						
Type	I	F						
Default	none	0.						

VARIABLE	DESCRIPTION
PID	PID of the fluid surface where a Navier boundary condition is applied
FRIC	Friction coefficient. If a negative value is entered, it will refer to a load curve ID used to describe the friction coefficient value as a function of time; see *DEFINE_CURVE, *DEFINE_CURVE_FUNCTION, or *DEFINE_FUNCTION. If a *DEFINE_FUNCTION is used, the following parameters are allowed: f(x, y, z, vx, vy, vz, temp, pres, time).

Remarks:

1. **About the Navier condition.** It is similar to the regular free slip condition, except a local source term is added based on the choice of the friction coefficient. Since this extra shear is a function of the fluid's velocity, a smaller time step may be needed to reduce the effects of the introduced nonlinearity.

*ICFD_BOUNDARY_NONSLIP

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

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

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

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

*ICFD_BOUNDARY_PERIODIC

Purpose: Impose various kinds of constraints between two fluid surfaces.

Card 1	1	2	3	4	5	6	7	8
Variable	PID	PTYPE	PID2	PDLCID	AXE	PTID	ANGLE	
Type	I	I	I	I	I	I	F	
Default	none	none	none	none	none	none	none	

VARIABLE	DESCRIPTION
PID	Part ID for a fluid surface
PTYPE	Boundary type: EQ.1: Periodic rotation boundary condition EQ.2: Periodic reflective boundary condition EQ.3: Sliding mesh boundary condition
PID2	Part ID for the second surface mesh. The boundary condition selected with PTYPE will be applied between PID and PID2. See Remark 1.
PDLCID	Optional load curve ID to describe the pressure drop value between PID and PID2 as a function of time. This curve can be specified with *DEFINE_CURVE,*DEFINE_CURVE_FUNCTION, or *DEFINE_FUNCTION. For *DEFINE_FUNCTION, the following parameters are allowed: f(x,y,z,vx,vy,temp,pres,time).
AXE	The meaning of AXE depends on PTYPE. It only applies for PTYPE = 1 and 3. For the periodic rotation boundary condition (PTYPE = 1): EQ.1: Rotation around *X*-axis EQ.2: Rotation around *Y*-axis EQ.3: Rotation around *Z*-axis For the sliding mesh boundary condition (PTYPE = 3): EQ.0: The contact distance between two faces of PID and PID2 is based on the characteristic local element size.

VARIABLE	DESCRIPTION
	EQ.1: The contact distance between two faces of PID and PID2 is based on the characteristic local element size scaled by a factor given by ANGLE.
	EQ.2: The contact distance between two faces of PID and PID2 is based on the length given by ANGLE.
PTID	Origin point ID for PTYPE = 1 and PTYPE = 2. See *ICFD_DEFINE_POINT.
ANGLE	Rotation angle for PTYPE = 1. Characterizes contact distance for PTYPE = 3 and AXE ≠ 0.

Remarks:

1. **Selection of PID and PID2.** When the two meshes are of different densities, we recommend selecting the finer mesh to be PID and the coarser mesh to be PID2.

*ICFD_BOUNDARY_PRESCRIBED_LEVELSET

Purpose: Prescribe the fluid height on a boundary.

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

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

VARIABLE	DESCRIPTION
PID	PID of the fluid surface where a fluid height will be imposed.
PTID	Point ID specifying the origin of the fluid surface. See *ICFD_DEFINE_POINT.
AXE	Global axis specifying the direction of the fluid: EQ.1: X-axis EQ.2: Y-axis EQ.3: Z-axis

*ICFD_BOUNDARY_PRESCRIBED_MOVEMESH_{OPTION1}

Available options for *OPTION1* include:

DR

Purpose: Allows the fluid surface nodes to translate in specific directions using an ALE approach. This feature is helpful in piston-type applications or for avoiding large mesh deformation in certain cases.

Include as many cards as needed. This input ends at the next keyword ("*") card. If the DR keyword option is active, then only one occurrence of card 1 is allowed.

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

Include this card for the DR keyword option. When the dynamic relaxation phase includes the ICFD solver, this second line allows the application of different parameters during that phase. See IDR of *ICFD_CONTROL_GENERAL.

Card 2	1	2	3	4	5	6	7	8
Variable		DRDOFX	DRDOFY	DRDOFZ				
Type		I	I	I				
Default		1	1	1				

VARIABLE	DESCRIPTION
PID	PID for a fluid surface
DOFX/DOFY/DOFZ	Degrees of freedom in the *X*, *Y*, and *Z* directions: EQ.0: Degree of freedom left free (surface nodes can translate in the chosen direction) EQ.1: Prescribed degree of freedom (surface nodes are restrained)

VARIABLE	DESCRIPTION
DRDOFX/DRDOFY/ DRDOFZ	Same as DOFX, DOFY, and DOFZ but applies to the mesh displacement during the dynamic relaxation phase.

*ICFD_BOUNDARY_PRESCRIBED_PRE

Purpose: Impose a fluid pressure on the boundary.

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

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

VARIABLE	DESCRIPTION
PID	PID for a fluid surface
LCID	Load curve ID to describe the pressure value as a function of time. See *DEFINE_CURVE, *DEFINE_CURVE_FUNCTION, or *DEFINE_FUNCTION. If using a *DEFINE_FUNCTION, the following parameters are allowed: f(x, y, z, vx, vy, vz, temp, pres, time).
SF	Load curve scale factor. The default is 1.0.
DEATH	Time at which the imposed motion/constraint is removed: EQ.0.0: Default set to 1028
BIRTH	Time at which the imposed pressure is activated starting from the initial abscissa value of the curve
ISO	Adds an additional isovelocity condition to the system on top of the user-defined imposed pressure (see Remark 1): EQ.0: Off EQ.1: On

Remarks:

1. **Adding an isovelocity condition.** Flow instability is a well known numerical issue that can occur with imposed pressure conditions, particularly in complex FSI cases or when the global flow changes direction during the transient run (inlet transitions into outlet and vice versa). Using ISO = 1 adds an isovelocity

condition which makes the velocity uniform across that surface. The objective is an attempt to provide an elegant form of constraining the flow and preventing numerical issues in cases where this hypothesis can be considered physically valid. This feature imposes an additional numerical cost.

*ICFD_BOUNDARY_PRESCRIBED_SPTRANSP_CONC

Purpose: Specify the concentration of the transported species at the boundaries.

Card 1	1	2	3	4	5	6	7	8
Variable	PID	LCID1	CONC1	LCID2	CONC2			
Type	I	I	F	I	F			
Default	none	none	0.0	none	0.0			

VARIABLE	DESCRIPTION
PID	Part ID of the boundary with the concentration
LCID*i*	Load curve ID for the curve giving the concentration of species *i* at the boundary as a function of time. Note that all the transported species must have a load curve defined.
CONC*i*	Constant concentration at the boundary for species *i*. If nonzero, LCID*i* is ignored.

*ICFD_BOUNDARY_PRESCRIBED_TEMP

Purpose: Impose a fluid temperature on the boundary.

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

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

VARIABLE	DESCRIPTION
PID	PID for a fluid surface.
LCID	Load curve ID to describe the temperature value versus time; see *DEFINE_CURVE, *DEFINE_CURVE_FUNCTION or *DEFINE_-FUNCTION. . If a DEFINE_FUNCTION is used, the following parameters are allowed: $f(x,y,z,vx,vy,vz,temp,pres,time)$.
SF	Load curve scale factor. (default = 1.0)
DEATH	Time at which the imposed temperature is removed: EQ.0.0: default set to 10E28
BIRTH	Time at which the imposed temperature is activated starting from the initial abscissa value of the curve

*ICFD_BOUNDARY_PRESCRIBED_TURBULENCE

Purpose: Optional keyword for strongly imposing turbulence quantities when you select a RANS turbulence model. See *ICFD_CONTROL_TURBULENCE. This keyword is intended for modifying the default boundary conditions at the inlet.

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

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

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

VARIABLE	DESCRIPTION
KS/CS	Roughness physical height and roughness constant. When defined, the global values of *ICFD_CONTROL_TURBULENCE are replaced for this surface part.

Remarks:

1. **Turbulence Kinetic Energy.** At the inlet, the relationship between the turbulence kinetic energy, k, and the turbulence intensity, I, is given by:

$$k = \frac{3}{2}\left(U_{\text{avg}}^2 I^2\right) \ .$$

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

2. **Turbulence Dissipation Rate.** At the inlet, if you specify the turbulent dissipation rate using a length scale, l, the following relationship will be used:

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

By default, the solver estimates a length scale based on the total height of the channel.

If you specify the turbulent viscosity ratio, $r = \mu_t/\mu$, the following relationship will be used:

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

3. **Specific Dissipation Rate.** At the inlet, if you specify the specific dissipation rate using a length scale, l, the following relationship will be used:

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

By default, the solver estimates a length scale based on the total height of the channel.

If you specify the turbulence viscosity ratio, $r = \mu_t/\mu$, the following relationship will be used:

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

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

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

*ICFD_BOUNDARY_PRESCRIBED_VEL

Purpose: Impose the fluid velocity on the boundary.

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

Card 1	1	2	3	4	5	6	7	8
Variable	PID	DOF	VAD	LCID	SF	VID	DEATH	BIRTH
Type	I	I	I	I	F	I	F	F
Default	none	none	1	none	1.	0	10^{28}	0.0

VARIABLE	DESCRIPTION
PID	PID for a fluid surface.
DOF	Applicable degrees of freedom: EQ.1: x- degree of freedom EQ.2: y- degree of freedom EQ.3: z degree of freedom EQ.4: Normal direction degree of freedom
VAD	Velocity flag: EQ.1: Linear velocity EQ.2: Angular velocity EQ.3: Parabolic velocity profile EQ.4: Activates synthetic turbulent field on part. See *ICFD_CONTROL_TURB_SYNTHESIS.
LCID	Load curve ID used to describe motion value versus time, see *DEFINE_CURVE, *DEFINE_CURVE_FUNCTION, or *DEFINE_FUNCTION. If a DEFINE_FUNCTION is used, the following parameters are allowed: f(x, y, z, vx, vy, vz, temp, pres, time). For steady state the motion value is a function of the number of iterations instead of time.
SF	Load curve scale factor. (default = 1.0)

VARIABLE	DESCRIPTION
VID	Point ID for angular velocity application point, see *ICFD_DEFINE_POINT.
DEATH	Time at which the imposed motion/constraint is removed: EQ.0.0: default set to 10^{28}
BIRTH	Time at which the imposed motion/constraint is activated starting from the initial abscissa value of the curve

*ICFD_BOUNDARY_PRESCRIBED_VISCOELASTIC

Purpose: Specify the conformation tensor at the boundaries.

Card 1	1	2	3	4	5	6	7	8
Variable	PID	LCIDXX	LCIDYY	LCIDZZ	LCIDXY	LCIDXZ	LCIDYZ	
Type	I	I	I	I	I	I	I	
Default	none	none	none	none	none	none	none	

VARIABLE	DESCRIPTION
PID	Part ID of the boundary with the prescribed conformation tensor
LCID*ij*	Load curve IDs for the conformation tensor components

*ICFD_BOUNDARY_WEAKVEL

Purpose: Specify the fluid boundary with a non-slip boundary condition which is imposed in a weak form.

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

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

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

*ICFD_BOUNDARY_WINDKESSEL

Purpose: Impose the pressure function with circuit parameters where an analogy is made between pressure and scalar potential and between flow rate and current intensity. Such conditions are frequently encountered in hemodynamics.

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

Optional Card. This card defines additional parameters for the Windkessel circuits depending on WTYPE.

Card 2	1	2	3	4	5	6	7	8
Variable	P2LCID	C2	R3	P0	P1			
Type	I	F	F	F	F			
Default	0	0.	0.	0.	0.			

VARIABLE	DESCRIPTION
PID	PID for a fluid surface
WTYPE	Circuit type. See remarks and figures.
R1/C1/L1/R 2/C2/R3	Parameters (resistances, inductances, capacities) for the different circuits. See remarks and figures.
P2LCID	ID of load curve describing the behavior of $P_2(t)$ as a function of time when applicable. See Remarks 2, 3, and 4 and Figures 7-2, 7-3, and 7-4.
QPLCID	ID of load curve describing the behavior of $Q_p(t)$ as a function of time when applicable. See Remark 4 and Figure 7-4.

VARIABLE	DESCRIPTION
P0/P1	Initial pressures at circuit junctions when applicable. See remarks and figures.

Remarks:

1. **WTYPE = 1 and 2.** See Figure 7-1 for a diagram of the circuit. The equation solved at the boundary is:

$$P(t) = P_0(t) + \text{SF} \times Q \times R_2$$
$$P_0'(t) = -\frac{1.0}{R_1 C_1} P_0(t) + \text{SF} \times \left[L_1 Q'' + \frac{Q}{C_1} + \frac{L_1}{R_1 C_1} Q' \right]$$

with Q, the flow rate calculated by the solve, and P, the pressure. Time derivatives are marked using Lagrange notation. By convention, the flow rate has a negative value if the fluid goes out of the fluid domain (outlet) and a positive value if it goes in (inlet). The scale factor, SF, applied on the circuit equation's flow-rate-dependent terms enables recovering the desired behavior of the Windkessel circuit. SF = -1.0 for WTYPE = 1, and SF = 1.0 for WTYPE = 2.

For WTYPE = 1 and 2, R_1, R_2 and L_1 are optional and can be left as 0.

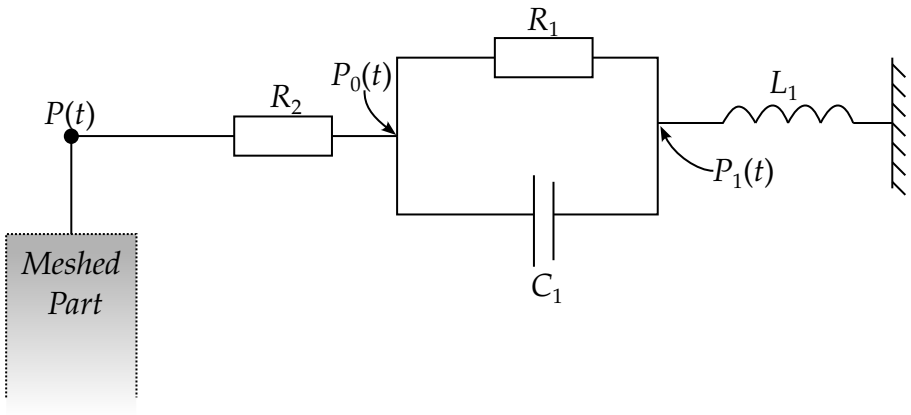


Figure 7-1. WTYPE = 1 and 2 circuit

2. **WTYPE = 3 and 4.** See Figure 7-2 for a diagram of the circuit. The equation solved at the boundary is:

$$P(t) = P_0(t) + \text{SF} \times Q \times R_1$$
$$P_0'(t) = -\frac{1.0}{R_2 C_1} P_0(t) + \frac{1.0}{R_2 C_1} P_1(t) + \text{SF} \times \frac{Q}{C_1}$$
$$P_1'(t) = \frac{1.0}{R_2 C_2} P_0(t) - \frac{R_3 + R_2}{R_3 R_2 C_2} P_1(t) + P_2'(t)$$

with Q and SF as described in Remark 1. SF = -1.0 for WTYPE = 3, and SF = 1.0 for WTYPE = 4. R_1 and R_3 are optional for this type of circuit. If R_3 is set to 0., that branch of the circuit is removed (open circuit). A negative value can be entered for R_3 which points to a *DEFINE_FUNCTION. The function accepts the following parameters: f(time, dt, flux , pre). In the function, flux is the current flow rate, Q, and pre is the current pressure, $P(t)$. With this method, R_3can be set to a specific value or left as 0.0 based on the sign of flux, thus modeling diode behavior.

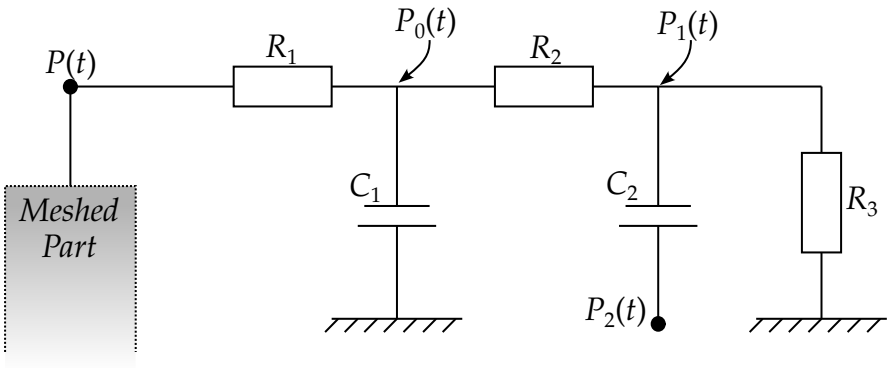


Figure 7-2. WTYPE = 3 and 4 circuit

3. **WTYPE = 7 and 8.** See Figure 7-3 for a diagram of the circuit. The equation solved at the boundary is:

$$P(t) = P_0(t) + \text{SF} \times Q \times R_1$$
$$P_0'(t) = -\frac{1.0}{R_2 C_1} P_0(t) + \text{SF} \times \frac{Q}{C_1} + \frac{1.0}{R_2 C_1} P_2(t)$$

with Q and SF as described in Remark 1. SF = -1.0 for WTYPE = 7, and SF = 1.0 for WTYPE = 8. For this type of circuit, R_1, R_2 , and $P_2(t)$ are optional.

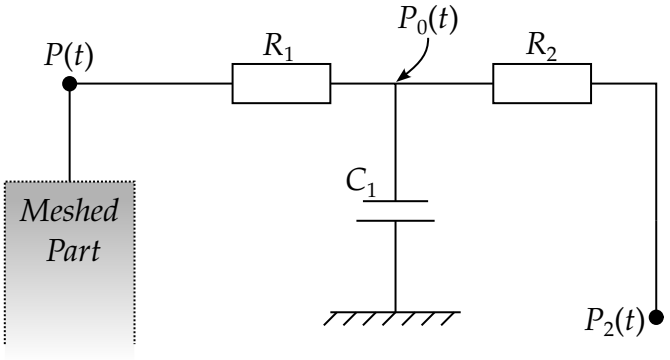


Figure 7-3. WTYPE = 7 and 8 circuit

4. **WTYPE = 9 and 10.** See Figure 7-4 for a diagram of the circuit. The equation solved at the boundary is:

$$P(t) = \frac{R_3}{R_2+R_3}P_0(t) + \mathrm{SF} \times Q \times \frac{R_3R_2}{R_2+R_3} + \frac{R_2}{R_2+R_3}P_2(t)$$

$$P_0'(t) = -\frac{R_1+R_2+R_3}{(R_2+R_3)R_1C_2}P_0(t) + \frac{1.0}{R_1C_2}P_1(t) + \frac{R_3}{(R_2+R_3)}\mathrm{SF} \times \frac{Q}{C_2} - \frac{P_2(t)}{C_2} \times \frac{1.0}{R_2+R_3}$$

$$P_1'(t) = \frac{1.0}{R_1C_1}P_0(t) - \frac{1.0}{R_1C_1}P_1(t) + \frac{Q_p(t)}{C_1}$$

with Q and SF as described in Remark 1. SF = -1.0 for WTYPE = 9, and SF = 1.0 for WTYPE = 10. For this type of circuit, R_2, $P_2(t)$, $Q_p(t)$, and R_3 are optional. If R_3 is set to 0.0, that branch of the circuit is removed (open circuit). A negative value can be entered for R_3 which points to a *DEFINE_FUNCTION. The function accepts the following parameters: f(time, dt, flux , pre). In the function, flux is the current flow rate, Q, and pre is the current pressure, $P(t)$. With this method, R_3can be set to a specific value or left as 0.0 based on the sign of flux, thus modeling diode behavior.

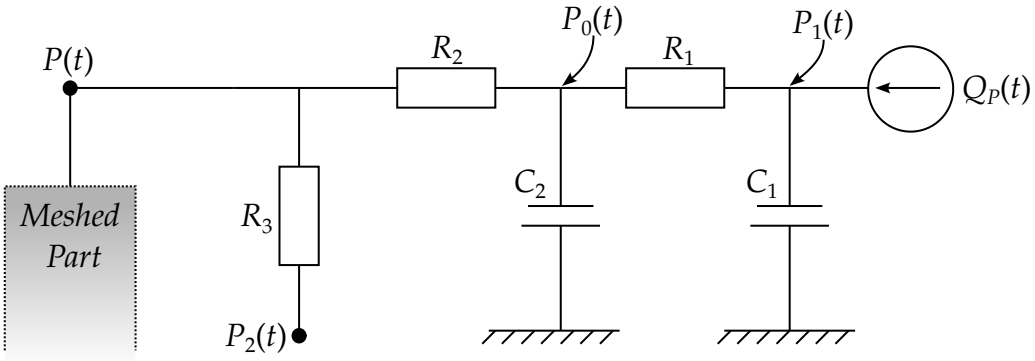


Figure 7-4. WTYPE = 9 and 10 circuit

5. **WTYPE = 6.** WTYPE = 6 enables implementing user-defined circuit laws by compiling the usermat versions of LS-DYNA. The routine that can be modified is called `icfd_usercircuit` and can be found in dyn21icfd.f.

*ICFD_CONTROL_ADAPT

Purpose: Activate the adaptive mesh refinement feature. The solver uses an a-posteriori error estimator to compute a new mesh size bounded by the user to satisfy a maximum perceptual global error.

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

Optional Card.

Card 2	1	2	3	4	5	6	7	8
Variable	DT							
Type	F							
Default	0.0							

VARIABLE	DESCRIPTION
MINH	Minimum mesh size allowed to the mesh generator. The resulting mesh will not have an element smaller than MINH even if the minimum size does not satisfy the maximum error.
MAXH	Maximum mesh size
ERR	Maximum perceptual error allowed in the whole domain
MTH	Specify if the mesh size is computed based on function error or gradient error: EQ.1: Function error (default) EQ.2: Gradient error EQ.3: Relative error EQ.4: Gradient relative error

VARIABLE	DESCRIPTION
NIT	Number of iterations before a remeshing is forced: LT.0: |NIT| is a load curve ID giving the number of iterations before a remeshing as a function of time. EQ.0: Do not remesh. GT.0: Number of iterations before a forced remeshing.
VAR	Specify which variable is taken into account for the error calculation: LT.0: Adaptive meshing is based only on the level-set function. Elements of size MINH are targeted over a length |VAR| on each side of the interface. EQ.0: Velocity, pressure, and level-set function are taken into account. EQ.1: Remove the level-set function from the error calculation. EQ.2: Remove the pressure from the error calculation. EQ.3: Remove both the pressure and the level-set function from the error calculation. Only the fluid velocity will, therefore, remain. EQ.5: For immersed interface methods, adapt the mesh only at the interface of the immersed surfaces.
KIS	Keep initial mesh size: EQ.0: Turned off. The remeshing process will ignore the initial mesh size in the volume. EQ.1: Turned on. Whenever a remeshing occurs, the new local mesh size will not be allowed to be substantially coarser than the one from the previous mesh. The object is to diminish the excessive coarsening that can occur between two remeshings.
DT	Optional time step to control the remeshing frequency. A negative value points to a load curve ID giving the remeshing time step as a function of time.

*ICFD_CONTROL_ADAPT_SIZE

Purpose: Control the remeshing of elements. This keyword enables considering the element quality and distortion instead of only checking for inverted elements.

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

Optional Card. When including ICFD during the dynamic relaxation phase, this second line enables applying different parameters during that phase. See IDR of *ICFD_CONTROL_GENERAL.

Card 2	1	2	3	4	5	6	7	8
Variable	DRASIZE	DRNIT	DRKIS					
Type	I	I	I					
Default	0	optional	0					

VARIABLE	DESCRIPTION
ASIZE	Remesh criteria flag: EQ.0: Only remesh in cases where elements invert. EQ.1: Remesh if elements invert or if element quality deteriorates.
NIT	Number of iterations before a remeshing is forced. If a negative integer is entered, then a load curve function of time defines NIT.
KIS	Keep initial mesh size: EQ.0: Turned off. The remeshing process will ignore the initial mesh size in the volume. EQ.1: Turned on. Whenever a remeshing occurs, the new local mesh size will not be allowed to be substantially coarser

VARIABLE	DESCRIPTION
	than the one from the previous mesh. The object is to diminish the excessive coarsening that can occur between two remeshes.
DRASIZE	Same as ASIZE but for dynamic relaxation
DRNIT	Same as NIT but for dynamic relaxation
DRKIS	Same as KIS but for dynamic relaxation

*ICFD_CONTROL_ADVECTION

Purpose: Modify the default advection schemes for different parts of the solver like Navier Stokes, Level Set, thermal, etc.

Card 1	1	2	3	4	5	6	7	8
Variable				SLLS	SLNS	SLT	SLST	SLRT
Type				I	I	I	I	I
Default				0	0	0	0	0

VARIABLE	DESCRIPTION
SLLS	Advection scheme for the Level Set solver: EQ.0: Default advection scheme EQ.1: Changes the default advection scheme in the Level Set solver to use a linear semi-Lagrangian approach (Min and Gibou [2006]). EQ.2: Changes the default advection scheme in the Level Set solver to use a higher-order semi-Lagrangian approach.
SLNS	Advection scheme for the Navier-Stokes solver: EQ.0: Default advection scheme EQ.1: Changes the default advection scheme in the Navier-Stokes solver to use a semi-Lagrangian approach (Min and Gibou [2006]).
SLT	Advection scheme for the Thermal solver: EQ.0: Default advection scheme EQ.1: Changes the default advection scheme in the Thermal solver to use a semi-Lagrangian approach.
SLST	Advection scheme for the Species Transport solver: EQ.0: Default advection scheme EQ.1: Changes the default advection scheme in the Species Transport solver to use a semi-Lagrangian approach.

VARIABLE	DESCRIPTION
SLRT	Advection scheme for the Residence Time solver: EQ.0: Default advection scheme EQ.1: Changes the default advection scheme in the Residence Time solver to use a semi-Lagrangian approach.

References:

Min, Chohong, and Frédéric Gibou. "A second order accurate projection method for the incompressible Navier–Stokes equations on non-graded adaptive grids." Journal of Computational Physics 219.2 (2006): 912-929.

*ICFD_CONTROL_BACKFLOW

Purpose: Modify default values for backflow stabilization.

Card 1	1	2	3	4	5	6	7	8
Variable	BFOR	SF						
Type	I	F						
Default	0	1.0						

VARIABLE	DESCRIPTION
BFOR	Set the backflow stabilization formulation: EQ.0: Default stabilization dependent on spatial velocity gradients. EQ.1: The stabilization adds a temporal velocity gradient which could be necessary for added stabilization.
SF	Scale factor to increase the stabilization if needed. A very small value larger than zero minimizes the effect.

*ICFD_CONTROL_CONJ

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

Card 1	1	2	3	4	5	6	7	8
Variable	CTYPE							TSF
Type	I							F
Default	0							none

VARIABLE	DESCRIPTION
CTYPE	Indicates the thermal coupling type. EQ.0: Robust and accurate monolithic coupling where the temperature field are solved simultaneously between the fluid and the structure. EQ.1: Weak thermal coupling. The fluid passes the heat flux to the solid at the fluid-structure interface and the solid returns the temperature which is applied as a Dirichlet condition.
TSF	Thermal Speedup Factor. This factor multiplies all thermal parameters present in the heat equation with units of time in the denominator (e.g., thermal conductivity, convection heat transfer coefficients). It is used to artificially time scale the thermal problem. A negative value will refer to a time dependent load curve.

Remarks:

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

*ICFD_CONTROL_DEM_COUPLING

Purpose: Activate coupling between the ICFD and DEM solvers.

Card 1	1	2	3	4	5	6	7	8
Variable	CTYPE	BT	DT	SF	MAXVEL	DTYPE	SFF	FORM
Type	I	F	F	F	F	I	F	I
Default	0	0.	10^{28}	1.	none	0	1.0	0

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

VARIABLE	DESCRIPTION
CTYPE	Indicates the coupling direction of the solvers: EQ.0: Two-way coupling between the fluid and the solid particles. EQ.1: One-way coupling. The DEM particles transfer their location to the fluid solver. EQ.2: One-way coupling. The fluid solver transfers forces to the DEM particles.
BT	Birth time for the DEM coupling
DT	Death time for the DEM coupling
SF	Scale factor applied to the force transmitted by the fluid to the structure
MAXVEL	Maximal fluid velocity that can be used for the calculation of the fluid force passed on to the DEM particle. This is to avoid having spurious velocities in the fluid causing very high and unrealistic

VARIABLE	DESCRIPTION
	forces on the DEM particles which may lead to a crash.
DTYPE	Drag calculation type: EQ.0: Constant C_d value 0.5 scaled by SF EQ.1: Formula for C_d calculation from Cheng [2009] based on the local Reynolds number value scaled by SF. See Remark 1.
SFF	Scale factor applied to the force transmitted by the structure to the fluid
FORM	Type of formulation used in the coupling: EQ.0: The force at the particle is based on a velocity drag value. EQ.2: The force is computed using the fluid pressure gradient.
NITBS	Number of time steps between particle searches. The default is 50 in explicit analysis and 1 in implicit. Increasing this number can reduce the computational time at the expense of possible less accurate particle tracking.

Remarks:

1. **Coefficient of drag by Cheng [2009].** The calculation for C_d is:

$$C_d = \frac{24}{\mathrm{Re}} \times (1 + 0.27 \times \mathrm{Re})^{0.43} + 0.47 \times \left(1 - \exp\left(-0.04 \times \mathrm{Re}^{0.38}\right)\right)$$

See Cheng [2009] for details.

References:

[1] Cheng, N.-S., “Comparison of formulas for drag coefficient and settling velocity of spherical particles,” Powder Technology, 189, 395-398 (2009).

*ICFD_CONTROL_EMBEDSHELL

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

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

VARIABLE	DESCRIPTION
GTYPE	Gap type. Defines the criteria for selecting a distance to build the gap between the embedded nodes and the newly generated : EQ.0: Automatic and based on the surface mesh size multiplied by a scale factor given by DIST. Default method. EQ.1: Specific gap size given by the user and defined by DIST.
DIST	Distance value if GTYPE = 1 or scale factor value if GTYPE = 0.
TPS	Triple Point Seal. Allows to control the fluid escape through triple points EQ.0: Off. EQ.1: On. The triple points of embedded shells in contact to walls or among each other are sealed and no flow goes through them.

*ICFD_CONTROL_FSI

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

Card 1	1	2	3	4	5	6	7	8
Variable	OWC	BT	DT	IDC	LDICSF	XPROJ		
Type	I	F	F	F	I	I		
Default	0	0	10^{28}	0.25	0	0		

Card 2	1	2	3	4	5	6	7	8
Variable	NSUB				VFORC			
Type	I				I			
Default	none				0			

VARIABLE	DESCRIPTION
OWC	Indicates the coupling direction to the solver: EQ.0: Two-way coupling. Loads and displacements are transferred across the FSI interface, and the full non-linear problem is solved. It gives weak FSI coupling when coupled to explicit mechanical solver and strong FSI coupling when coupled to implicit mechanical solver. EQ.1: One-way coupling. The solid mechanics solver transfers displacements to the fluid solver. EQ.2: One-way coupling. The fluid solver transfers stresses to the solid mechanics solver. EQ.3: Two-way coupling. It causes weak coupling (no sub-stepping) with the implicit mechanical solver.
BT	Birth time for the FSI coupling. Before BT the fluid solver will not pass any loads to the structure, but it will receive displacements

VARIABLE	DESCRIPTION
	from the solid mechanics solver.
DT	Death time for the FSI coupling. After DT the fluid solver will not transfer any loads to the solid mechanics solver, but the fluid will continue to deform with the solid.
IDC	Interaction detection coefficient. See Remark 1.
LCIDSF	Optional load curve ID to apply a scaling factor on the forces transferred to the solid: GT.0: Load curve ID for scale factor as a function of iterations LT.0: |LCIDSF| is a load curve ID for scale factor as a function of time.
XPROJ	Projection of the nodes of the CFD domain that are at the FSI interface onto the structural mesh (see Remark 2): EQ.0: No projection EQ.1: Projection
NSUB	Optional limit on the number of FSI fluid subiterations. This avoids the sometimes unneeded excessive number of FSI subiterations when the fluid and very light structures (like parachutes) develop a resonance-like mode inside the FSI subiterations (coupling iterations).
VFORC	Add viscous tangential forces to the FSI coupling: EQ.0: Only normal pressure forces are used. EQ.1: Viscous forces are added to normal forces. EQ.2: Viscous forces are recomputed at every sub-step during the nonlinear iterations and added to normal forces.

Remarks:

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

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

where h is the size of the fluid mesh, H is the size of the solid mechanics mesh, and IDC is a detection coefficient criteria with IDC = 0.25 by default. In the

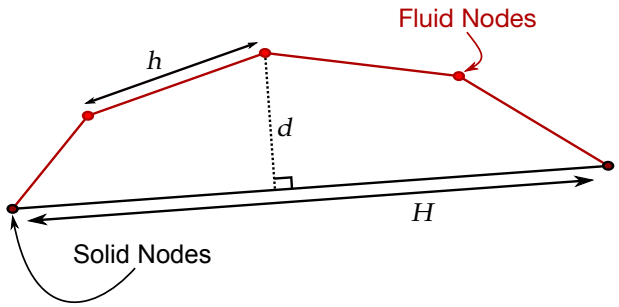


Figure 0-1. Geometry of FSI contact.

majority of cases, this default value is sufficient to ensure FSI interaction. However, it can happen in special cases that the fluid and solid geometries have curvatures that differ too much (such as pipe flows in conjugate heat transfer applications). In such cases, a bigger IDC value may be needed. This flag should be handled with care.

2. **Rotation and projection of nodes.** XPROJ = 1 is recommended for cases with rotation.

*ICFD_CONTROL_GAP

Purpose: Activate the gap closure treatment that deals with flow blockage when surfaces come into contact. In ICFD, surface meshes are not allowed to collide or penetrate each other. Rather, activation of the gap closure feature triggers a flow blockage between two surfaces in close proximity based on a user defined contact distance. There are two types of flow treatment available. The default is a strong exclusion of the fluid elements within the gap. The second option is a porous media description for the fluid nodes inside the contact gap.

Card 1	1	2	3	4	5	6	7	8
Variable	HGAP	PGAP	PERM					
Type	F	I	F					
Default	none	0	1.e-5					

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
HGAP	Threshold distance. If the distance between surfaces is less than this value, the flow motion is blocked in the gap. Currently there is only one universal value of HGAP for all the surfaces listed in Card 2.
PGAP	Flag for flow treatment of fluid elements in the gap: EQ.0: Element exclusion contact treatment EQ.1: Porous media description
PERM	Permeability coefficient. Its value determines the ratio between the fluid and porous representation in the contact region. Low values induce a Darcy description.

VARIABLE	DESCRIPTION
PID*n*	Part IDs of the surfaces involved in the gap closure treatment

*ICFD_CONTROL_GENERAL

Purpose: Specify the type of ICFD analysis.

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

VARIABLE	DESCRIPTION
ATYPE	Analysis type: EQ.-2: Turn off the ICFD solver but leave level set, thermal, etc. Velocities can be set using the define function. It is useful for debugging models or for simple thermal / level set cases that do not need a full ICFD solution. EQ.-1: Turn off the ICFD solver after the initial keyword reading EQ.0: Transient analysis (default) EQ.1: Steady-state analysis
MTYPE	Solving method type: EQ.0: Fractional step method EQ.1: Monolithic solve EQ.2: Potential flow solve
DVCL	Divergence cleaning flag: EQ.0: Initialize the solution with divergence cleaning (default) EQ.1: No divergence cleaning EQ.2: Initial divergence cleaning using potential flow EQ.4: Initial divergence cleaning using steady-state solver
RDVCL	Remeshing divergence cleaning: EQ.0: No divergence cleaning after remesh (default) EQ.1: Divergence cleaning after each remeshing step

VARIABLE	DESCRIPTION
SOLCL	Solver control: EQ.0: The solver automatically detects if the analysis is 2D or 3D based on element connectivity (default). EQ.1: Turns on the 2D axisymmetric solver. EQ.2: 2D axisymmetric solver for FSI cases that use section shell type 15 for solid parts.
IDR	Flag to include ICFD during the structural dynamic relaxation phase: EQ.0: Off EQ.1: Include the ICFD solver during the dynamic relaxation phase. The solver only computes the mesh displacement. It does not find pressure/velocity, but the mesh adapts and follows the displacement of the FSI parts.

*ICFD_CONTROL_IMPOSED_MOVE

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

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

Optional Card. Rotational velocity components using Euler or Tait-Bryan angles (see Remark 1).

Card 2	1	2	3	4	5	6	7	8
Variable	ALPHAL	BETAL	GAMMAL	ALPHAG	BETAG	GAMMAG	VADR	IANG
Type	I	I	I	I	I	I	I	I
Default	0	0	0	0	0	0	0	0

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

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

Optional Card. When defined, Cards 2 and 3 are ignored. With this card, rotation is imposed around a point using the velocity of a second point.

Card 4	1	2	3	4	5	6	7	8
Variable	PTID0	AXE	NID					
Type	I	I	I					
Default	0	0	0					

VARIABLE	DESCRIPTION
PID	Part ID. This can be any part ID referenced in *ICFD_PART or *ICFD_PART_VOL. If PID = 0, then the whole volume mesh will be used.
LCVX, LCVY, LCVZ	Load curve IDs for the velocity/displacements in the three global directions (X, Y, Z). To use a *DEFINE_FUNCTION, see Remark 4.
VADT	Velocity/displacements flag for translation components: EQ.0: Prescribe velocity EQ.1: Prescribe displacements
IDR	Flag determining whether to only impose mesh displacement during dynamic relaxation phase (see IDR of *ICFD_CONTROL_GENERAL): EQ.0: Off. EQ.1: On. Apply imposed mesh movement only during the dynamic relaxation phase.
ALPHAL, BETAL, GAMMAL	Load curves IDs for the three angle rotational velocities/displacements in the local reference frame (see Remarks 1 and 2). To use a *DEFINE_FUNCTION, see Remark 4.
ALPHAG, BETAG, GAMMAG	Load curve IDs for the three angle rotational velocities/displacements in the global reference frame (see Remarks 1 and 2). To use a *DEFINE_FUNCTION, see Remark 4.

VARIABLE	DESCRIPTION
VADR	Velocity/displacements flag for rotation components: EQ.0: Prescribe velocity EQ.1: Prescribe displacements
IANG	Rotation matrix type: EQ.0: Euler angles using $\mathbf{Z}(\alpha)\mathbf{X}(\beta)\mathbf{Z}(\gamma)$ intrinsic rotations. EQ.1: Tait-Bryan angles using $\mathbf{Z}(\alpha)\mathbf{X}(\beta)\mathbf{Y}(\gamma)$ intrinsic rotations.
PTID	Point ID for the origin of the local reference frame. If not defined, the barycenter of the volume mesh will be used
X1, Y1, Z1	Three components of the local reference X1 axis. If not defined, the global X axis will be used. See Remark 2.
X2, Y2, Z2	Three components of the local reference X2 axis. If not defined, the global Y axis will be used. See Remark 2.
PTIDO	Point ID (See *ICFD_DEFINE_POINT) for the center of rotation.
AXE	Rotation axis: EQ.1: X-axis EQ.2: Y-axis EQ.3: Z-axis
NID	ICFD surface node ID for the rotational velocity. If the node is static, no rotation will occur. See Remark 3.

Remarks:

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

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

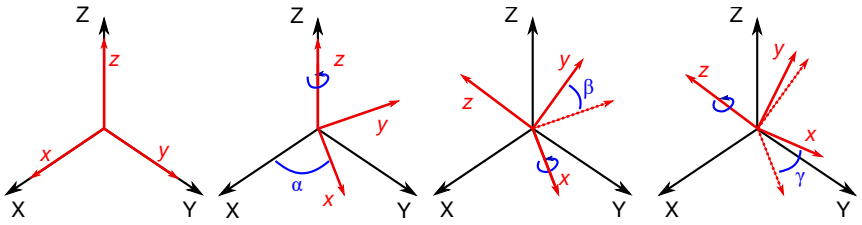


Figure 7-5. A rotation represented by Euler angles (α, β, γ) using $\mathbf{Z}(\alpha)\mathbf{X}(\beta)\mathbf{Z}(\gamma)$ intrinsic rotations.

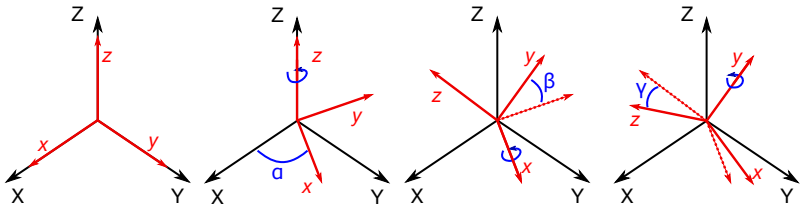


Figure 7-6. A rotation represented by Tait-Bryan angles (α, β, γ) using $\mathbf{Z}(\alpha)\mathbf{X}(\beta)\mathbf{Y}(\gamma)$ intrinsic rotations

However, a different definition of the elemental rotation matrices and their multiplication order can be adopted. The ICFD solver uses the following approach and rotation matrix for the Euler angles:

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

Here, $s_\alpha = \sin(\alpha)$, and $c_\beta = \cos(\beta)$.

For the Tait-Bryan approach, the following rotation matrix is used:

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

2. **Local coordinate systems.** It is possible to have the ICFD parts or ICFD_PART_VOLs rotate around the global reference frame but also to define and use a local reference frame by defining its point of origin and two of its vectors $\mathbf{v}_1 = (\text{X1}, \text{Y1}, \text{Z1})$ and $\mathbf{v}_2 = (\text{X2}, \text{Y2}, \text{Z2})$ (note that $\mathbf{v}_1$ and $\mathbf{v}_2$ should be orthogonal). The third vector is, then, in the direction of $\mathbf{v}_1 \times \mathbf{v}_2$. See Figure 7-5.

3. **Purpose of NID.** We developed this feature for rotating problems involving FSI and sliding mesh. For example, the airflow can push the blades of a wind turbine and the rotation of the sliding mesh can be prescribed as function of the blade rotation speed.

4. ***DEFINE_FUNCTION.** For each of the load curves, a *DEFINE_FUNCTION can be used. If a *DEFINE_FUNCTION is used, the following parameters are allowed: f(x, y, z, vx, vy, vz, temp, visc, pres, time, dt).

*ICFD_CONTROL_LEVELSET

Purpose: Modify default values for the level set solver.

Card 1	1	2	3	4	5	6	7	8
Variable	LSRST	LSINL		LSMTH			SGC	AST
Type	I	I		F			I	I
Default	20	0		0.0			0	0

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

VARIABLE	DESCRIPTION
LSRST	Frequency the level set distance function is re-initialized. The default value is every 20 time steps.
LSINL	Set the level set to be positive at a velocity inlet: EQ.0: Let the level set algorithm compute the value (default). EQ.1: Force a positive level set value at the inlet.
LSMTH	Scale factor for level set smoothness. GT.0.0: Add smoothness to the free surface. Small values like 0.1 are reasonable. The optimal value could be problem dependent.
SGC	Smoother approximate of gradient and curvature: EQ.0: No smoothing added EQ.1: Linear gradient and curvature EQ.-1: Least-squares-based gradient and curvature

VARIABLE	DESCRIPTION
	EQ.2: Least-squares gradient smoothing
AST	Advanced formulation for surface tension: EQ.0: Smoothed Dirac EQ.1: Smoothed Heaviside EQ.2: Laplace-Beltrami
SRL	Smoother reinitialization method for the level set: EQ.0: Standard, geometric-based EQ.1: Linear closest point algorithm EQ.2: Higher-order closest point algorithm

*ICFD_CONTROL_LOAD

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

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

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

*ICFD_CONTROL_MESH

Purpose: Modify default values for automatic volume mesh generation.

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

Optional card. This card is optional.

Card 2	1	2	3	4	5	6	7	8
Variable	AVER	SFBL						
Type	I	F						
Default	14	1.0						

VARIABLE	DESCRIPTION
MGSF	Mesh Growth Scale Factor. It specifies the maximum mesh size that the volume mesher is allowed to use when generating the volume mesh based on the mesh surface element sizes defined in *MESH_SURFACE_ELEMENT. See Remark 1.
MSTRAT	Mesh generation strategy (see Remark 2): EQ.0: Mesh generation based on Delaunay criteria EQ.1: Mesh generation based on octree
2DSTRUC	Flag to decide between an unstructured mesh generation strategy in 2D or a structured mesh strategy: EQ.0: Structured mesh EQ.1: Unstructured mesh

VARIABLE	DESCRIPTION
NRMSH	Flag to turn off any remeshing (see Remark 3): EQ.0: Remeshing possible EQ.1: Remeshing not allowed
AVER	Automatic Volume Mesher version (see Remark 4): EQ.14: Version 14 EQ.16: Version 16
SFBL	Scale factor that controls the speed of boundary layer inflation. If the boundary layer is much larger in size than the surface mesh, we recommend a value less than one.

Remarks:

1. **MGSF.** For MGSF, values between 1 and 2 are allowed. Values closer to 1 will result in a finer volume mesh (1 means the volume mesh is not allowed to be coarser than the element size from the closest surface meshes) and values closer to 2 will result in a coarser volume mesh (2 means the volume can use elements as much as twice as coarse as those from the closest surface mesh). MGSF has a fixed value of 1 in 2D.

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

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

4. **Version.** Version 14 is the default version used for the ICFD solver automatic volume mesher. Version 16 is now supported and available as option. In some cases, it can yield an approximatively 20% mesh generation speed gain.

*ICFD_CONTROL_MESH_MOV

Purpose: Choose the type of algorithm for mesh movement.

Card 1	1	2	3	4	5	6	7	8
Variable	MMSH	LIM_ITER	RELTOL					
Type	I	I	F					
Default	2	100	10^{-3}					

VARIABLE	DESCRIPTION
MMSH	Mesh motion selector: EQ.-1: Completely shuts off any mesh movement EQ.1: Mesh moves based on the distance to moving walls. EQ.2: Mesh moves by solving a linear elasticity problem using the element sizes as stiffness (default). EQ.3: Mesh uses a Laplacian smoothing with stiffness on edges and from node to opposite faces. Very robust, but costly. EQ.4: Full Lagrangian. The mesh moves with the velocity of the flow. EQ.11: Mesh moves using an implicit ball-vertex spring method. EQ.20: Mesh moves by solving a linear elasticity problem using a constant size. This can be useful to avoid large distortions in rotating problems that involve large discrepancies in mesh sizes (typically in cases involving boundary layer mesh).
LIM_ITER	Maximum number of linear solver iterations for the ball-vertex linear system
RELTOL	Relative tolerance to use as a stopping criterion for the ball-vertex method iterative linear solver (conjugate gradient solver with diagonal scaling preconditioner)

*ICFD_CONTROL_MONOLITHIC

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

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

VARIABLE	DESCRIPTION
SID	Solver ID : EQ.0: Fractional Step Solver. Default. EQ.1: Monolithic Solver.

*ICFD_CONTROL_OUTPUT

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

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

Optional Card.

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

VARIABLE	DESCRIPTION
MSGL	Message level. EQ.0: only time step information is output. EQ.1: first level solver information. EQ.2: full output information with details about linear algebra and convergence steps. EQ.4: full output information is also copied to the messag file.

VARIABLE	DESCRIPTION
OUTL	Output the fluid results in other file formats apart from d3plot. EQ.0: only d3plot output EQ.2: output a file with mesh statistics and the fluid results in OpenDX format. A directory named dx will be created in the work directory where the output files will be written. EQ.6: output a file with mesh statistics and the fluid results in VTK format readable by Paraview. A directory named vtk will be created in the work directory where the output files will be written. EQ.7: output a file with mesh statistic and the fluid results in VTU format readable by Paraview. A directory named vtk will be created in the work directory where the output files will be written. EQ.10: output a file with mesh statistic and the fluid results in Fieldview ASCII format. . A directory named fv will be created in the work directory where the output files will be written. Only available in 3D. EQ.11: output a file with mesh statistic and the fluid results in Fieldview binary format. . A directory named fv will be created in the work directory where the output files will be written. Only available in 3D.
DTOUT	Time interval to print the output when OUTL is different than 0.
LSPPOUT	EQ.0: no LSPP output is produced. EQ.1: outputs a file with the automatically created fluid volume mesh in a format compatible for LSPP at each remesh. Also outputs the fluid volume mesh in a format compatible with a subsequent ICFD analysis. EQ.3: Outputs the fluid volume mesh in a format compatible with a subsequent ICFD analysis at each DTOUT.
ITOUT	Iteration interval to print the output, including the d3plot files when the steady state solver is selected (See ICFD_CONTROL_GENERAL).

VARIABLE	DESCRIPTION
PITOUT	Pressure iteration limit output. If the number of pressure iterations in the fractional step solve goes above PITOUT, an extra d3plot will be dumped. This is mainly a debugging feature which can help the user identify problematic areas in the model which often precede a divergence.

.

*ICFD_CONTROL_OUTPUT_SUBDOM

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

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

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

Box Case. Card 2 for Sname = box

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

Sphere Case. Card 2 for Sname = sphere

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

Cylinder Case. Card 2 for Sname = cylinder

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

VARIABLE	DESCRIPTION
SNAME	Shape name. Possibilities include 'box', 'cylinder' and 'sphere'
PMINX, Y, Z]	X, Y, Z for the point of minimum coordinates
PMAX[X, Y, Z]	X, Y, Z for the point of maximum coordinates
CEN-TER[X, Y, Z]	Coordinates of the sphere center in cases where Sname is Sphere
RADIUS	Radius of the sphere if SNAME is *sphere* or of the cross section disk if SNAME is *cylinder*.

*ICFD_CONTROL_OUTPUT_VAR

Purpose: This keyword allows the user to turn off the output of certain CFD variables to reduce the size of the d3plot files.

Card 1	1	2	3	4	5	6	7	8
Variable	VEL	AVGVEL	VORT					
Type	I	I	I					
Default	0	0	0					

Card 2	1	2	3	4	5	6	7	8
Variable	PRE	PREAVG	LSET	QC	CFL			
Type	I	I	I	I	I			
Default	0	0	0	0	0			

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

Card 4	1	2	3	4	5	6	7	8
Variable	KP	EP	MUT	INT	CMU			
Type	I	I	I	I	I			
Default	0	0	0	0	0			

VARIABLE	DESCRIPTION
VEL/AVGVEL/ VORT	Velocity, average velocity, vorticity : EQ.0: Is output. EQ.1: Is not output.
PRE/PREAVG/ LSET/QC/CFL	Pressure, average pressure, levelset, Q criterion, CFL number : EQ.0: Is output. EQ.1: Is not output.
TEMP/ TEMPAVG	Temperature, average temperature : EQ.0: Is output. EQ.1: Is not output.
KP/EP/MUT /INT/CMU	RANS output variables, kinetic energy, diffusion, turbulent viscosity, turbulent intensity, Cmu variable : EQ.0: Is output. EQ.1: Is not output.

.

*ICFD_CONTROL_PARTITION

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

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

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

*ICFD_CONTROL_POROUS

Purpose: This keyword modifies the porous media solve.

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

VARIABLE	DESCRIPTION
PMSTYPE	Indicates the porous media solve type. EQ.0: Anisotropic Generalized Navier-Stokes model for porous media (See *ICFD_MODEL_POROUS) using Fractional step method. EQ.1: Anisotropic Darcy-Forcheimer model using a Monolithic approach for the solve. This method is better suited for very low Reynolds flows through porous media (frequently encountered in Resin Transfer Molding (RTM) applications). See Remark 1.
VELMETH	Method for determining advection velocity: EQ.0: Uses FEM approximation for advection velocity EQ.1: Uses PFEM2 for advection velocity

Remarks:

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

*ICFD_CONTROL_STEADY

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

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

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

*ICFD_CONTROL_SURFMESH

Purpose: Enable automatic surface remeshing. The objective of the remeshing is to improve the mesh quality on the boundaries. It should not be used on a regular basis.

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

VARIABLE	DESCRIPTION
RSRF	Indicates whether or not to perform a surface remeshing: EQ.0: Do not perform surface remeshing. EQ.1: Perform Laplacian smoothing surface remeshing. EQ.2: Perform curvature-preserving surface remeshing.
SADAPT	Indicates whether or not to trigger adaptive surface remeshing: EQ.0: Do not apply adaptive surface remeshing. EQ.1: Apply automatic surface remeshing when quality deteriorates. EQ.2: Apply automatic surface remeshing when quality deteriorates. Keep the initial element size in the case of adaptive mesh refinement using *ICFD_CONTROL_ADAPT throughout the computation.

.

*ICFD_CONTROL_TAVERAGE

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

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

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

*ICFD_CONTROL_TIME

Purpose: Change the default values related to time parameters for the fluid problem.

Card 1	1	2	3	4	5	6	7	8
Variable	TTM	DT	CFL	LCIDSF	DTMIN	DTMAX	DTINIT	TDEATH
Type	F	F	F	I	F	F	F	F
Default	10^{28}	0.0	1.0	0	10^{-9}	10^{28}	↓	10^{28}

Optional card

Card 2	1	2	3	4	5	6	7	8
Variable	DTT							
Type	F							
Default	Rem 2							

Optional card

Card 3	1	2	3	4	5	6	7	8
Variable	DTBL	DTST	DTVISC					
Type	I	I	I					
Default	0	0	0					

Optional card

Card 4	1	2	3	4	5	6	7	8
Variable	IDR	DTDR	CFLDR	LCIDSFDR	DTMINDR	DTMAXDR	DTINITDR	
Type	I	F	F	I	F	F	F	
Default	0	0.0	1.0	0	10^{-9}	10^{28}	0.0	

VARIABLE	DESCRIPTION
TTM	Total time of simulation for the fluid problem
DT	Time step for the fluid problem. If nonzero, the time step will be constant and equal to this value. If set to 0.0, then the time step is automatically computed based on the fluid advection CFL condition. See Remark 1.
CFL	CFL number for DT = 0.0. In general, CFL specifies a scale factor that is applied to the time step. When DT = 0.0, the time step is set to the maximum value satisfying the CFL condition, in which case this scale factor is equal to the *CFL number*.
LCIDSF	Load curve ID specifying the CFL number when DT = 0.0 as a function of time, and more generally LCIDSF specifies the time step scale factor as a function of time.
DTMIN	Minimum time step. When an automatic time step is used and DTMIN is defined, the time step cannot drop below DTMIN. A negative value will refer to a time dependent load curve.
DTMAX	Maximum time step. When an automatic time step is used and DTMAX is defined, the time step cannot increase beyond DTMAX. A negative value will refer to a time dependent load curve.
DTINIT	Initial time step. If not defined, the solver will automatically determine an initial time step based on the flow velocity or dimensions of the problem in cases where there is no inflow.
TDEATH	Death time for the Navier Stokes solve. After TDEATH, the velocity and pressure will no longer be updated. But the temperature and other similar quantities still can.
DTT	Thermal time step. See Remark 2.

VARIABLE	DESCRIPTION
DTBL	Flag to include boundary layer elements in the automatic time step calculation. EQ.0: Exclude boundary layer elements (default). EQ.1: Include boundary layer elements.
DTST	Flag to include surface tension effects (when present) in the automatic time step calculation: EQ.0: Do not take it into account (default). EQ.1: Add surface tension effects. See Remark 3.
DTVISC	Flag to include viscous effects in the automatic time step calculation: EQ.0: Do not take it into account (default). EQ.1: Add viscous term effects. See Remark 4.
IDR	Temporarily overwrite time step settings during dynamic relaxation (DR) phase (see field IDR of *ICFD_CONTROL_GENERAL) EQ.0: Off EQ.1: Overwrite timesteps settings during DR phase
DTDR/[..]/ DTINITDR	Time step parameters equivalent to the fields 2 through 7 on Card 1 used during DR phase.

Remarks:

1. **Courant friedrichs lewy (CFL) condition.** When an automatic time step is selected, the default behavior is to use the following Courant condition:

$$\Delta t \leq \Delta t_{\text{adv}} \leq \frac{l}{V}$$

Here, V is the fluid velocity, and l is the characteristic mesh size. When using an automatic time step, it is recommended to define a minimum and maximum acceptable value. It is also recommended to give an initial value, especially in cases where no velocity boundary condition exists.

2. **Thermal time step.** By default, the heat equation is solved using the same time step as that for the velocity/pressure system. This option allows you to assign a specific time step for the thermal solve. It can be useful in cases where the time

scales are very different between the two domains. When defined, we recommend DTT to always be greater than or equal to the regular CFD time step.

3. **Surface tension effects.** For free surface or bi-phasic cases where surface tension is present and is the dominant force, using DTST = 1 in combination with an automatic time step may be convenient. In such cases, the automatic time step is limited by the following Courant condition:

$$\Delta t \leq \Delta t_{\text{st}} \leq \sqrt{\frac{\tilde{\rho} l^3}{2\pi\sigma}}$$

In this condition, $\tilde{\rho}$ is the average density between the two fluids, l is the mesh size, and σ isthe surface tension value.

4. **Diffusive effects.** In advection-driven flows, the default criteria to estimate an automatic time step is sufficient. However, in applications where the advection term is small compared to the viscous forces (usually cases that combine low Reynolds number with high viscosity value), adding the diffusion Courant condition as an additional constraint may be justified:

$$\Delta t \leq \Delta t_{\text{diff}} \leq \frac{l^2}{\nu}$$

Here, l is the mesh size, and ν is the kinematic viscosity.

*ICFD_CONTROL_TRANSIENT

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

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

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

*ICFD_CONTROL_TURBULENCE

Purpose: Modify the default values for the turbulence model.

Card Summary:

Card 1. This card is required.

TMOD	SUBMOD	WLAW	KS	CS		TWLAW	TYPLUS

Card 2a. This card is read if TMOD = 1. It is optional.

CE1	CE2	SIGMAEPS	SIGMAK	CMU	CCUT		

Card 2b. This card is read if TMOD = 2 or 3. It is optional.

Cs							

Card 2c.1. This card is read if TMOD = 4. It is optional.

GAMMA	BETA01	SIGMAW1	SIGMAK1	BETA0ST	CCUT		

Card 2c.2. This card is read if TMOD = 4. It is optional.

A1	BETA02	SIGMAW2	SIGMAK2	CL			

Card 2d. This card is read if TMOD = 5. It is optional.

CB1	CB2	SIGMANU	CNU1	CW1	CW2		

Data Card Definitions:

Card 1	1	2	3	4	5	6	7	8
Variable	TMOD	SUBMOD	WLAW	KS	CS		TWLAW	TYPLUS
Type	I	I	I	F	F		I	F
Default	0	1	1	0.	0.		none	none

VARIABLE	DESCRIPTION
TMOD	Indicates what turbulence model will be used.

VARIABLE	DESCRIPTION
	EQ.0: Turbulence model based on a variational multiscale approach is used by default. EQ.1: RANS k - ε approach (see Remark 1) EQ.2: LES Smagorinsky or dynamic sub-grid scale model EQ.3: LES Wall adapting local eddy-viscosity (WALE) model EQ.4: RANS k - w approach (see Remark 2) EQ.5: RANS Spalart-Allmaras approach
SUBMOD	Turbulence sub-model. For RANS k - ε approach (TMOD = 1): EQ.1: Standard model EQ.2: Realizable model For LES Smagorinsky or dynamic sub-grid model (TMOD = 2): EQ.1: Smagorinsky model (see Remark 6) EQ.2: Dynamic model (see Remark 7). For RANS k - ω approach (TMOD = 4): EQ.1: Standard Wilcox 98 model. EQ.2: Standard Wilcox 06 model. EQ.3: SST Menter 2003.
WLAW	Law of the wall ID if a RANS turbulence model is selected (see Remark 4): EQ.1: Standard classic law of the wall (default for TMOD = 1) with linear blending between the log and linear regions. EQ.2: Standard Launder and Spalding law of the wall EQ.4: Nonequilibrium Launder and Spalding law of the wall EQ.5: Automatic classic law of the wall
KS	Roughness physical height, only used for RANS turbulence models.
CS	Roughness constant, only used for RANS turbulence models.
TWLAW	Thermal law of the wall flag (see Remark 8): EQ.0: No thermal law of the wall activated.

VARIABLE	DESCRIPTION
	EQ.1: Thermal law of the wall
TYPLUS	Thermal Y+ value (Y_{+t}). If Y_{+t} is not defined, the solver will automatically estimate its value using $Y_{+tc} = Y_{+c}/\mathrm{Pr}^{1./3.}$ with $Y_{+c} = 11.225$ the critical Y_+ value and Pr the Prandtl number.

RANS k – ε Card. Optional card if TMOD = 1. Optional card read if TMOD = 1. See Remark 1.

Card 2a	1	2	3	4	5	6	7	8
Variable	CE1	CE2	SIGMAE	SIGMAK	CMU	CCUT		
Type	F	F	F	F	F	F		
Default	1.44	1.92	1.3	1.0	0.09	-1.		

VARIABLE	DESCRIPTION
CEPS1	k - ε model constant, $C_{1\varepsilon}$
CEPS2	k - ε model constant, $C_{2\varepsilon}$
SIGMAEPS	k - ε model constant, σ_ε
SIGMAK	k - ε model constant, σ_k
CMU	k - ε model constant, C_μ
CCUT	k - ε model constant, C_{cut}

LES Card. Optional card read if TMOD = 2 or 3.

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

VARIABLE	DESCRIPTION
Cs	Smagorinsky constant if TMOD = 2 and SUBMOD = 1 or WALE constant if TMOD = 3

RANS $k-\omega$ Card. Optional card read if TMOD = 4. See Remark 2.

Card 2c.1	1	2	3	4	5	6	7	8
Variable	GAMMA	BETA01	SIGMAW1	SIGMAK1	BETA0ST	CCUT		
Type	F	F	F	F	F	F		
Default	1.44	0.072	2.	2.	0.09	-1.		

RANS $k-\omega$ Card. Optional card read if TMOD = 4. See Remark 2.

Card 2c.2	1	2	3	4	5	6	7	8
Variable	A1	BETA02	SIGMAW2	SIGMAK2	CL			
Type	F	F	F	F	F			
Default	0.31	0.0828	2	2	0.875			

VARIABLE	DESCRIPTION
GAMMA	k -ω model constant, γ
BETA01	k -ω model constant, β_{01}
SIGMAW1	k -ω model constant, $\sigma_{\omega 1}$
SIGMAK1	k -ω model constant, σ_{k1}
BETA0ST	k -ω model constant, β_0^*
CCUT	k -ω model constant, C_{cut}
A1	k -ω model constant, a_1
BETA02	k -ω model constant, β_{02}
SIGMAW2	k -ω model constant, $\sigma_{\omega 2}$

VARIABLE	DESCRIPTION
SIGMAK2	k -ω model constant, σ_{k2}
CL	k -ω model constant, C_l

RANS Spalart-Allmaras Card. Optional card read if TMOD = 5.

Card 2d	1	2	3	4	5	6	7	8
Variable	CB1	CB2	SIGMANU	CNU1	CW1	CW2		
Type	F	F	F	F	F	F		
Default	0.1355	0.622	0.66	7.2	0.3	2.0		

VARIABLE	DESCRIPTION
CB1	Spalart-Allmaras constant, C_{b1}
CB2	Spalart-Allmaras constant, C_{b2}
SIGMANU	Spalart-Allmaras constant, σ_ν
CNU1	Spalart-Allmaras constant, $C_{\nu 1}$
CW1	Spalart-Allmaras constant, C_{w1}
CW2	Spalart-Allmaras constant, C_{w2}

Remarks:

1. **k -ε model (TMOD = 1).** For the Standard k - ε model, the following two equations are solved for the turbulent kinetic energy (k) and the turbulent dissipation (ε):

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

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

Here P_k is the k production term (see Remark 3), P_b is the production term due to buoyancy and S_k and S_e are the user defined source terms. P_k and P_b are expressed as:

$$P_k = \frac{\mu_t}{\rho} S^2$$
$$P_b = \frac{\beta \mu_t}{\rho \mathrm{Pr}_t} g_i \frac{\partial T}{\partial x_i}$$

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

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

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

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

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

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

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

In the above,

$$U^* = \sqrt{\Omega_{ij}\Omega_{ij} + S_{ij}S_{ij}}$$
$$A_0 = 4.04$$
$$A_s = \sqrt{6} cos\left(\frac{1}{3} cos^{-1}\left(\sqrt{6} \frac{S_{ij}S_{jk}S_{ki}}{(S_{ij}S_{ij})^{3/2}} \right) \right)$$

Note that in this case, the constant value C_μ that can be input by you serves as the limiting value that C_μ can take. By default, $C_\mu = 0.09$ so:

$$0.0009 < C_\mu < 0.09$$

2. ***k* - *ω* model (TMOD = 4).** For the Standard Wilcox 06 k - ω model, the following two equations are solved for the turbulent kinetic energy and the specific turbulent dissipation rate, respectively k and ω:

$$\frac{\partial k}{\partial t} + \frac{\partial (k u_i)}{\partial x_i} = \frac{\partial}{\partial x_j}\left[\left(\frac{\mu}{\rho} + \frac{\mu_t}{\rho \sigma_{k1}} \right) \frac{\partial k}{\partial x_j} \right] + P_k - \beta^* k\omega + S_k$$
$$\frac{\partial \omega}{\partial t} + \frac{\partial (\omega u_i)}{\partial x_i} = \frac{\partial}{\partial x_j}\left[\left(\frac{\mu}{\rho} + \frac{\mu_t}{\rho \sigma_{\omega 1}} \right) \frac{\partial \varepsilon}{\partial x_j} \right] + \gamma \frac{\omega}{k} P_k - \beta \omega^2 + \sigma_d X_k \omega^2 + S_\omega$$

Here P_k is the k production term (see Remark 3) and S_k and S_ω are the user defined source terms. P_k, β^*, β and σ_d are expressed as:

$$\begin{aligned} P_k &= \frac{\mu_t}{\rho} S^2 \\ \beta^* &= \beta_0^* f_{\beta*} \\ \beta &= \beta_{01} f_\beta \\ \sigma_d &= \begin{cases} 0. & X_k \le 0. \\ 1/8 & X_k > 0. \end{cases} \end{aligned}$$

where

$$\begin{aligned} f_\beta &= \frac{1 + 85 X_\omega}{1 + 100 X_\omega} \\ f_{\beta*} &= 1. \\ X_k &= \frac{1}{\omega^3} \frac{\partial k}{\partial x_j} \frac{\partial \omega}{\partial x_j} \\ X_\omega &= \left| \frac{\Omega_{ij} \Omega_{jk} S_{ki}}{(\beta_0^* \omega)^3} \right| \end{aligned}$$

The turbulent viscosity is then:

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

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

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

The turbulent viscosity is then:

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

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

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

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

Each of the constants, γ, β, σ_k, and σ_ω are now computed by a blend of two constants with a blending function through:

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

The blending function F_1 is defined by:

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

With y the distance to the nearest wall and:

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

The turbulent viscosity is then:

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

with:

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

3. **Production term.** You can activate a limiter on the production term, P_k, for TMOD =1 and 4. If $C_{\mathrm{cut}} \geq 0$ (CCUT), then:

$$P_k = \begin{cases} \min(P_k, C_{\mathrm{cut}} \varepsilon) & \text{if TMOD} = 1 \\ \min(P_k, C_{\mathrm{cut}} \beta_0^* k \omega) & \text{if TMOD} = 4 \end{cases}$$

This is especially common when using the Menter SST 2003 model.

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

a) *Standard Classic (WLAW = 1)* with linear blending when $5 \leq Y^+ \leq 60$.

$$U^+_{blend}(Y^+) = \left(1 - \alpha(Y^+)\right) \times Y^+ + \alpha(Y^+) \times (1/k \ln(EY^+))$$
$$\alpha(Y^+) = \frac{Y^+ - 5}{60 - 5}$$
$$Y^+ = \frac{\rho y U_\tau}{\mu}$$
$$U^+ = \frac{U}{U_\tau}$$
$$U_\tau = \sqrt{\frac{\tau_w}{\rho}}$$

This is the default for TMOD = 1.

b) *Standard Launder and Spalding (WLAW = 2).*

$$U^* = \begin{cases} \frac{1}{\kappa}\ln(EY^*) & \text{if } Y^* > 11.225 \\ Y^* & \text{otherwise} \end{cases}$$
$$Y^* = \frac{\rho {C_\mu}^{1/4} k^{1/2} y}{\mu}$$
$$U^* = \frac{U {C_\mu}^{1/4} k^{1/2}}{{U_\tau}^2}$$
$$U_\tau = \sqrt{\frac{\tau_w}{\rho}}$$

c) *Nonequilibrium Launder and Spalding (WLAW = 4).* The nonequilibrium laws of the wall modify the expression of the velocity at the wall making it sensitive to the pressure gradient :

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

with:

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

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

d) *Automatic Classic (WLAW = 5).* The automatic wall law attempts to blend the viscous and log layers to better account for the transition zone. In the buffer region, we have :

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

This is the recommended approach for TMOD = 4.

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

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

If we introduce the non-dimensional roughness height:

$$K^+ = \frac{\rho K_s {C_\mu}^{1/4} k^{1/2}}{\mu} \ ,$$

we have:

$$\Delta B = \begin{cases} 0 & \text{for } K^+ \le 2.25 \\ \frac{1}{\kappa}\ln\left(\frac{K^{\pm 2.25}}{87.75} + C_s K^+\right) \times \sin\left(0.4258(\ln K^+ - 0.811)\right) & \text{for } 2.25 < K^+ \le 90.0 \\ \frac{1}{\kappa}\ln(1 + C_s K^+) & \text{for } 90. < K^+ \end{cases}$$

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

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

7. **LES dynamic model.** The LES dynamic model is based on the model originally proposed by Germano et. al. (1991) and improved by Lilly (1992), with localization on Cs by Piomelli and Liu (1995).

8. **Thermal law of the wall.** When the thermal law of the wall is activated, the turbulent heat flux will be calculated as an additional output variable:

$$Q_t = \rho C_p \frac{U_\tau}{T_+}\,(T_s - T_c)$$

$$T_+ = \begin{cases} \mathrm{Pr}_t Y_+ & \text{if } Y_{+t} \le Y_{+tc} \\ \frac{\mathrm{Pr}_t}{\vartheta}\log(Y_+) + \left(3.85\mathrm{Pr}_t^{1.\overline{3}\cdot} - 1.3\right)^2 + 2.12\log(\mathrm{Pr}_t) & \text{otherwise} \end{cases}$$

*ICFD_CONTROL_TURB_SYNTHESIS

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

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

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

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

Remarks:

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

*ICFD_DATABASE_AVERAGE

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

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

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

Remarks:

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

*ICFD_DATABASE_DRAG_{OPTION}

Available options include

VOL

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

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

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

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

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

Remarks:

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

2. The output contains:

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

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

   where P is the pressure and A the surface area.

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

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

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

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

*ICFD_DATABASE_FLUX

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

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

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

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

Remarks:

1. **Database Name.** The file name for this database is icfd_flux.dat.

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

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

the average pressure, called "Pre-avg,"

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

and the total area, called "Areatot."

*ICFD_DATABASE_FLUX_SURF

Purpose: Enable the computation of the flow rate and average pressure over a given surface (icfd part) of the model which is not necessary part of the analysis but lies in the fluid volume by mapping and interpolating volume results on the given surface.

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

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

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

Remarks:

1. **Database Name.** The file name for this database is icfd_fluxsurf.dat.

*ICFD_DATABASE_FORCE_DEM

Purpose: Enable the computation of the total fluid force that is transferred to the DEM particles over time. The output is in icfd_force_dem.dat.

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

VARIABLE	DESCRIPTION
OUT	Flag to enable computing the fluid forces and generating output: EQ.0: No output is generated. EQ.1: Output is generated.

*ICFD_DATABASE_GOA

Purpose: Enable the computation of the geometric orifice area (GOA), a quantity of interest used to evaluate valve performance. The GOA surface connects the free edges of the leaflets. It is defined in the fluid mesh as surface mesh. If multiple keywords exist in the input deck, separate files are output. It must be used with *MESH_SURFACE_NULL which defines parts where no physics is computed. These parts are only used for geometrical computations or post-processing.

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

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

VARIABLE	DESCRIPTION
PID	Fluid part ID of the fluid surface mesh where the GOA will be computed (see *MESH_SURFACE_NULL)
DTOUT	Output frequency. The default is at every fluid time step.

Remarks:

1. **Database name.** The file name for this database is icfd_goa.dat.

2. **GOA surface.** Define the GOA surface in the fluid mechanics problem with *MESH_SURFACE_NULL. It is unnecessary to define *ICFD_PART for any *MESH_SURFACE_NULL part IDs. The surface is automatically excluded from the FSI coupling.

*ICFD_DATABASE_HTC

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

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

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

Remarks:

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

2. The heat transfer coefficient is defined as follows:

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

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

*ICFD_DATABASE_NODEAVG

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

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

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

Remarks:

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

*ICFD_DATABASE_NODOUT

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

Output Options Card.

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

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

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

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

Remarks:

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

*ICFD_DATABASE_NTEMPOUT

Purpose: Output the temperature at individual nodes in a format consistent with *ICFD_-INITIAL_TEMPNODE to initialize a subsequent ICFD problem.

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

Card 1	1	2	3	4	5	6	7	8
Variable	NID	DTOUT						
Type	I	F						
Default	none	0.0						

VARIABLE	DESCRIPTION
NID	Internal ICFD node ID
DTOUT	Output frequency EQ.0.0: The ICFD timestep will be used.

*ICFD_DATABASE_POINTAVG

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

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

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

Remarks:

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

*ICFD_DATABASE_POINTOUT

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

Output Options Card.

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

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

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

VARIABLE	DESCRIPTION
PSID	Point Set ID.
DTOUT	Time interval to print the output. If DTOUT is equal to 0.0, then the ICFD timestep will be used.
PSTYPE	Point Set type : EQ.0: Fixed points. EQ.1: Tracer points using prescribed velocity. EQ.2: Tracer points using fluid velocity. EQ.3: Tracer points using mesh velocity..
VX, VY, VZ	Constant velocities to be used when PSTYPE = 1
PID	Point ID

VARIABLE	DESCRIPTION
X, Y, Z	Point initial coordinates

Remarks:

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

*ICFD_DATABASE_RESIDUALS

Purpose: This keyword allows the user to output the residuals of the various systems.

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

VARIABLE	DESCRIPTION
RLVL	Residual output level : EQ.0: No output. EQ.1: Only outputs the number of iterations needed for solving the pressure Poisson equation. EQ.2: Outputs the number of iterations for the momentum, pressure, mesh movement and temperature equations. EQ.3: Also gives the residual for each iteration during the solve of the momentum, pressure, mesh movement and temperature equations.

Remarks:

1. The file names for the momentum, pressure, mesh movement and temperature equations are called icfd_residuals.moms.dat, icfd_residuals.pres.dat, icfd_residuals.mmov.dat, and icfd_residuals.temp.dat respectively.

*ICFD_DATABASE_SSOUT

Purpose: Output the pressure load on a structure from the fluid. It can be useful for linear FSI applications, where the structure is made static, and the loads applied by the fluid are retrieved.

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

VARIABLE	DESCRIPTION
OUT	Determines if the solver should retrieve the pressure loads and how to output it: EQ.0: Inactive EQ.1: The fluid solver will collect the segment sets (see *SET_SEGMENT) that are part of a FSI boundary and retrieve the pressure for subsequent print out in an icfd_presseg and icfd_lcsegid pair of files. EQ.2: Same as 1 except the results are collected in a single icfd_presseg / icfd_plcsegid pair of files and the load curves associated with each segment are made functions of time, thus taking the transient nature of the CFD analysis into account. EQ.3: Same as 2 except uses a more memory efficient way of handling and outputting the data.
OUTDT	Frequency of the pressure extraction. If left as 0., the solver will extract the pressure of the fluid on the FSI boundary at every time step. This can lead to additional memory and calculation cost.
LCIDSF	Optional load curve ID to apply a scale factor on the fluid pressure output
POFF	Optional pressure offset on the fluid pressure output

*ICFD_DATABASE_SSOUT_EXCLUDE

Purpose: This keyword defines which segment set IDs are excluded from the SSOUT search. No forces coming from the fluid will be transmitted on those segment sets for output (see ICFD_DATABASE_SSOUT).

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	SSOUTID							
Type	I							
Default	none							

VARIABLE	DESCRIPTION
SSOUTID	Segment Set ID of the solid mechanics problem which is to be excluded from the output of the fluid forces on the solid boundaries.

*ICFD_DATABASE_TEMP

Purpose: This keyword enables the computation of the average temperature and the heat flux over given parts of the model. If multiple keywords are given, separate files are output.

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

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

VARIABLE	DESCRIPTION
PID	Part ID of the surface where the average temperature and heat flux will be computed.
DTOUT	Output frequency. Default is at every fluid timestep.

Remarks:

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

2. Two average temperature are given in the icfd_thermal.dat file: "Temp-avg" and "Temp-sum". The average temperature is calculated using the local node area as weighting factor,

$$T_{\text{avg}} = \frac{\sum_i^N T_i A_i}{\sum_i^N A_i},$$

whereas, the sum is not weighted by area

$$T_{\text{sum}} = \frac{\sum_i^N T_i}{N}$$

If the mesh is regular, the two values will be of similar value. The icfd_thermal.dat output file also includes the average heat flux, the total surface area, and the average heat transfer coefficients (See *ICFD_DATABASE_HTC).

*ICFD_DATABASE_TIMESTEP

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

Output Options Card.

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

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

Remarks:

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

2. Outputs the run's ICFD timestep versus the timestep calculated using the ICFD CFL condition as criteria (autotimestep). This can be useful in cases using a fixed timestep where big mesh deformations and/or big fluid velocity changes occur in order to track how that fixed timestep value compares to the reference auto-timestep.

*ICFD_DATABASE_TPD

Purpose: Enable the output of the transvalvular pressure difference (TPD) data. The TPD is computed by averaging the fluid pressure at two spheres upstream and downstream of a valve and computing the difference between them.

Card 1	1	2	3	4	5	6	7	8
Variable	PID	X1	Y1	Z1	X2	Y2	Z2	
Type	I	F	F	F	F	F	F	
Default	none	0.	0.	0.	0.	0.	0.	

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

Card 2	1	2	3	4	5	6	7	8
Variable	DTOUT	RAD1	RAD2					
Type	F	F	F					
Default	↓	0.	0.					

VARIABLE	DESCRIPTION
PID	Part ID used to plot multiple files if multiple *ICFD_DATABASE_-TPD keywords are defined.
X1, X2, X3	Coordinates of the center of the upstream sphere where the upstream average pressure is computed.
Y1, Y2, Y3	Coordinates of the center of the downstream sphere where the downstream average pressure is computed.
DTOUT	Time interval to print the output. If DTOUT is equal to 0.0, then the ICFD time step is used.
R1	Radius of the upstream sphere
R2	Radius of the downstream sphere

Remarks:

1. **Output filename.** The filename for this database is icfd_tpd.dat.

2. **Computing TPD.** The TPD is computed as:

$$\mathrm{TPD}(t^n) = P_{\mathrm{up}}(t^n) - P_{\mathrm{down}}(t^n)$$

*ICFD_DATABASE_TWINBUILDER

Purpose: Enable the output of ICFD data to be consumed by the Ansys Twin Builder static or dynamic ROM module. If this keyword is active, the solver automatically creates a directory tree compatible with the ROM module. If the model is transient, the dynamic ROM format is used. If the model is steady, the static ROM format is used. In the dynamic case, DTOUT in *ICFD_CONTROL_OUTPUT determines the output frequency. The output scenarios are identified using the job ID for each run. The job IDs should go from 1 to the number of scenarios used to train the ROM. See Remark 1.

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

Card 1	1	2	3	4	5	6	7	8
Variable	OUTRP	INEX1	INEX2	INEX3	INEX4	INEX5	INEX6	INEX7
Type	A	A	A	A	A	A	A	A
Default	none	none	none	none	none	none	none	none

VARIABLE	DESCRIPTION
OUTRP	Name of the output variable or response. All the fields are nodal. EQ.pres: Pressure EQ.apres: Time average pressure EQ.vel: Velocity EQ.avel: Time average velocity EQ.lset: Level set EQ.temp: Temperature EQ.atemp: Time average temperature
INEX1-7	Name of the input variables or excitations. Field values like pressure, velocity, and temperature are nodal arrays of boundary conditions. In the case of material properties, the values written to the excitation's files are scalars. A combination of up to seven excitations can be used for each output. EQ.pres: Pressure values at imposed pressure boundary conditions. EQ.vel: Velocity values at inflow boundary conditions.

VARIABLE	DESCRIPTION
	EQ.temp: Temperature values at imposed temperature boundary conditions.
	EQ.rho: Fluid density
	EQ.mu: Fluid viscosity
	EQ.hc: Heat capacity
	EQ.tc: Thermal conductivity

Remarks:

1. **Output structure.** This database creates a tree structure of directories that contain the different scenarios to be used to train the ROMs. The root directory name is `tb_s` for a static ROM and `tb_d` for a dynamic ROM. They are created in the location where the executable file is. The scenarios are numbered according to the job ID of the process. The user is responsible for providing numerical job IDs which are recommended to start from 1 and increase to the number of scenarios.

*ICFD_DATABASE_UINDEX

Purpose: This keyword allows the user to have the solver calculate the uniformity index (See Remark 1).

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

VARIABLE	DESCRIPTION
OUT	Determines if the solver should calculate the uniformity index. EQ.0: Off. EQ.1: On.

Remarks:

1. **Uniformity Index.** The uniformity index is a post treatment quantity which measures how uniform the flow is through a given section. It is especially useful in internal aerodynamics cases. It is expressed as :

$$\gamma = 1 - \frac{1}{2nA}\sum_{i=1}^{n}\left[\frac{\sqrt{(u_i - \bar{u})^2}}{\bar{u}}A_i\right]$$

with A_i, the local cell area, A the total section area, u_i the local velocity, $\bar{u}$ the average velocity through the section, and n the number of cells.

Values close to 0 means that the flow is very unevenly distributed. This can be used to identify bends, corners or turbulent effects. Values close to 1 imply smooth or equally distributed flow through the surface.

*ICFD_DATABASE_WETNESS

Purpose: For free surface problems, output the wetted area as a function of time for a given surface part. It is output to an ASCII file beginning with icfd_wetness.

Card 1	1	2	3	4	5	6	7	8
Variable	SPID	DTOUT						
Type	I	F						
Default	none	0.						

VARIABLE	DESCRIPTION
SPID	Part ID of the surface where the wetted area will be computed
DTOUT	Time interval to print the output. If DTOUT is equal to 0.0, then the ICFD time step will be used.

Remarks:

1. **Wetted Area.** The ICFD solver identifies a surface as wet if its level set value is positive.

*ICFD_DEFINE_HEATSOURCE

Purpose: Define a volumetric heat source for evaluating the heat equation.

Card 1	1	2	3	4	5	6	7	8
Variable	HSID	LCID	ISHAPE	R	PTID1	PTID2		
Type	I	I	I	F	I	I		
Default	none	none	none	none	none	none		

VARIABLE	DESCRIPTION
HSID	Heat source ID
LCID	Load curve ID specifying the evolution of the heat source term function of time for the *X*, *Y*, and *Z* degrees-of-freedom (see *DEFINE_CURVE,*DEFINE_CURVE_FUNCTION, or *DEFINE_FUNCTION). If using a *DEFINE_FUNCTION, the following arguments are allowed: f(x, y, z, vx, vy, vz, temp, pres, time).
ISHAPE	Shape of the volumetric heat source: EQ.1: Box shape EQ.2: Cylinder shape EQ.3: Sphere shape
R	Radius of the cylinder if ISHAPE = 2 or radius of the sphere if ISHAPE = 3
PTID1	Point ID (see *ICFD_DEFINE_POINT) whose meaning depends on ISHAPE: ISHAPE.EQ.1: Minimum coordinates of the box ISHAPE.EQ.2: Point in the center of one base of the cylinder ISHAPE.EQ.3: Center of the sphere
PTID2	Point ID (see *ICFD_DEFINE_POINT) whose meaning depends on ISHAPE:

VARIABLE	DESCRIPTION
	ISHAPE.EQ.1: Maximum coordinates of the box
	ISHAPE.EQ.2: Point in the center of the other base of the cylinder

.

*ICFD_DEFINE_POROUS_REGION

Purpose: Define moving isotropic porous regions. The permeability can be a constant value or defined through a time-dependent load curve or *DEFINE_FUNCTION. Alternatively (using SENSFLG ≠ 0), this keyword optionally enables pressure sensors to specify the permeability as a function of the pressure drop across the porous domain length (see Remark 1). *ICFD_DEFINE_POINTs determine the locations of the sensors. The first sensor is upstream of the region, and the second sensor downstream. The primary application of this feature is modeling the closing and opening of pressure-driven valves without the need for more complex moving parts (like leaflets). If points PTID1 and PTID2 (and sensors SENSID1 and SENSID2) are attached to a moving part, the points move accordingly.

Card 1	1	2	3	4	5	6	7	8
Variable	PRID	ISHAPE	R	PTID1	PTID2	PERM	LCID	SENSFLG
Type	I	I	F	I	I	F	I	I
Default	none	none	none	none	none	none	0	0

Sensor Card. Define only if SENSFLG ≠ 0 on Card 1.

Card 2	1	2	3	4	5	6	7	8
Variable	SENSID1	SENSID2	KMIN	KMAX	DLTAPREF	SLOPE		
Type	I	I	F	F	F	F		
Default	none	none	none	none	0	0		

VARIABLE	DESCRIPTION
PRID	Porous region ID
ISHAPE	Shape of the volumetric porous region: EQ.1: Box shape, EQ.2: Cylinder shape, EQ.3: Sphere shape,

VARIABLE	DESCRIPTION
R	Radius of the cylinder if ISHAPE = 2. Radius of the sphere if ISHAPE = 3.
PTID1	Point ID (see *ICFD_DEFINE_POINT) with meaning depending on ISHAPE: ISHAPE.EQ.1: Minimum coordinates of the box ISHAPE.EQ.2: Tail point of the cylinder ISHAPE.EQ.3: Center of the sphere
PTID2	Point ID with meaning depending on ISHAPE (ignored for ISHAPE = 3): ISHAPE.EQ.1: Maximum coordinates of the box ISHAPE.EQ.2: Head point of the cylinder.
PERM	Isotropic permeability, k, of the region. PERM is ignored if LCID ≠ 0 or SENSFLG = 1.
LCID	Load curve ID specifying the evolution of the permeability as a function of time for the X, Y, and Z degrees of freedom; see *DEFINE_CURVE, *DEFINE_CURVE_FUNCTION, or *DEFINE_FUNCTION. If a using a *DEFINE_FUNCTION, the following parameters are allowed: f(x, y, z, vx, vy, vz, temp, pres, time). Ignored if SENSFLG = 1.
SENSFLG	Sensor flag determining whether the permeability is a function of the pressure drop computed between the sensors: EQ.0: Not a function of pressure drop EQ.1: Function of a pressure drop between sensors. See Remark 1.
SENSID1	Sensor (*ICFD_DEFINE_POINT ID) upstream of the porous region
SENSID2	Sensor (ICFD_DEFINE_POINT ID) downstream of the porous region
KMIN	Minimum permeability for the valve (closed position), k_{min}
KMAX	Maximum permeability for the valve (open position), k_{max}
DLTAPREF	Pressure drop that opens/closes the valve, Δp_{ref}

VARIABLE	DESCRIPTION
SLOPE	Slope of the transition from k_{max} to k_{min}, m. See Remark 1.

Remarks:

1. **Permeability as a function of pressure drop.** When SENSFLG = 1, the pressure drop, Δp, between two sensors determines the permeability, k:

$$k = \frac{(k_{max} - k_{min})}{2} \times \left(1 + \tanh\left(\frac{(\Delta p - \Delta p_{ref})}{m}\right)\right) + k_{min}$$

This feature is intended for using the porous region to model the opening and closing of a valve. At a given Δp_{ref}, the permeability switches from k_{min} to k_{max} as Δp increases, modeling opening the valve. The valve closes when the opposite occurs. See Figure 7-7.

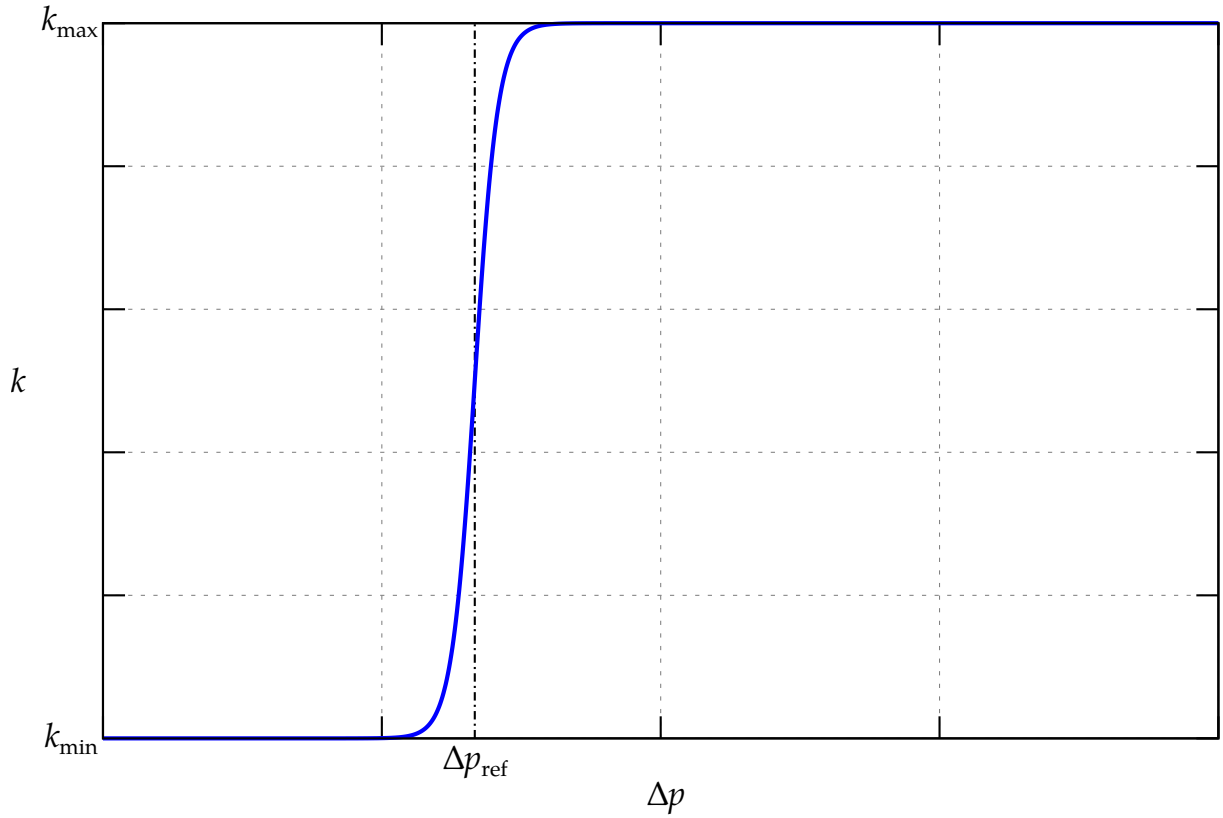


Figure 7-7. Pressure-driven valve law

*ICFD_DEFINE_RESIDENCETIMESOURCE

Purpose: Specify a volumetric residence time (RT) source.

Card 1	1	2	3	4	5	6	7	8
Variable	RTSID	ISHAPE	R	PTID1	PTID2	MASSDIFF	DEATHT	IRT0PBC
Type	I	I	F	I	i	F	F	I
Default	none	none	none	none	none	10^{-6}	↓	0

VARIABLE	DESCRIPTION
RTSID	RT source ID
ISHAPE	Shape of the volumetric RT source: EQ.1: Box EQ.2: Cylinder EQ.3: Sphere
R	Radius of the cylinder if ISHAPE = 2 or radius of the sphere if ISHAPE = 3
PTID1	ID of a point (see *ICFD_DEFINE_POINT) giving the minimum coordinate of the box if ISHAPE = 1, the tail point for the cylinder if ISHAPE = 2, or the origin of the sphere if ISHAPE = 3
PTID2	ID of a point giving the maximum coordinate of the box if ISHAPE = 1 or the head point of the cylinder if ISHAPE = 2.
MASSDIFF	Mass diffusion for the transport equation
DEATHT	End time for the source. EQ.0.0: End time of the simulation
IRT0PBC	Flag for which prescribed boundaries RT = 0 is imposed: EQ.0: Imposed only on boundaries with prescribed velocity EQ.1: Imposed on boundaries with either prescribed velocity or prescribed pressure

*ICFD_DEFINE_SOURCE

Purpose: Define a volumetric external force for the momentum equation solve.

Card 1	1	2	3	4	5	6	7	8
Variable	SID	LCIDX	LCIDY	LCIDZ	SHAPE	R	PTID1	PTID2
Type	I	I	I	I	I	F	I	I
Default	none	none	none	none	none	none	none	none

VARIABLE	DESCRIPTION
SID	Source ID
LCIDX/Y/Z	Load curve IDs specifying the evolution of the volumetric force as a function of time for the three global components
SHAPE	Shape to which the volumetric force is applied: EQ.1: Box EQ.2: Cylinder EQ.3: Sphere
R	Radius of the cylinder or sphere if SHAPE = 2 or 3
PTID1	Point ID (see *ICFD_DEFINE_POINT) whose meaning depends on SHAPE: SHAPE.EQ.1: Minimum coordinates of the box SHAPE.EQ.2: Tail point of the cylinder SHAPE.EQ.3: Origin of the sphere
PTID2	Point ID (see *ICFD_DEFINE_POINT) whose meaning depends on SHAPE: SHAPE.EQ.1: Maximum coordinates of the box SHAPE.EQ.2: Head point of the cylinder

*ICFD_DEFINE_SPTRANSPSOURCE

Purpose: Specify a volumetric species source for the species transport solver.

Card 1	1	2	3	4	5	6	7	8
Variable	SPTRSID	LCID	ISHAPE	R	PTID1	PTID2	MASSDIFF	DEATHT
Type	I	I	I	F	I	i	F	F
Default	none	none	none	none	none	none	10^{-6}	↓

VARIABLE	DESCRIPTION
SPTRSID	Species transport source ID
LCID	Load curve ID specifying the evolution of the species source term as a function of time for the X, Y, and Z degrees of freedom (see *DEFINE_CURVE, *DEFINE_CURVE_FUNCTION, and *DEFINE_FUNCTION). If a *DEFINE_FUNCTION is used, the following parameters are allowed: f(x, y,z, vx, vy, vz, temp, pres, time).
ISHAPE	Shape of the volumetric species source: EQ.1: Box EQ.2: Cylinder EQ.3: Sphere
R	Radius of the cylinder if ISHAPE = 2 or radius of the sphere if ISHAPE = 3
PTID1	ID of a point (see *ICFD_DEFINE_POINT) giving the minimum coordinate of the box if ISHAPE = 1, the tail point for the cylinder if ISHAPE = 2, or the origin of the sphere if ISHAPE = 3
PTID2	ID of a point giving the maximum coordinate of the box if ISHAPE = 1 or the head point of the cylinder if ISHAPE = 2.
MASSDIFF	Mass diffusion for the transport equation
DEATHT	End time for the source. EQ.0.0: End time of the simulation

*ICFD_DEFINE_TRANSFORM

Purpose: Enable applying translations, rotations, and scaling on the initial surface mesh before generating the volume mesh. This feature makes possible quick changes (such as when switching between unit systems) on the geometry without having to manually edit the case.

Card 1	1	2	3	4	5	6	7	8
Variable	PID	TRX	TRY	TRZ	SF			
Type	I	F	F	F	F			
Default	none	0.0	0.0	0.0	1.0			

Optional card

Card 2	1	2	3	4	5	6	7	8
Variable	ORX	ORY	ORZ	NX	NY	NZ	ANGLE	
Type	F	F	F	F	F	F	F	
Default	0.	0.	0.	0.	0.	0.	0.	

VARIABLE	DESCRIPTION
PID	Surface PID on which to apply the transformation. EQ.0: Apply the transformation to the entire model.
TRX/TRY/TRZ	Translation offset in the global X, Y, and Z-directions
SF	Scale factor to be applied on the X, Y, and Z coordinates
ORX/ORY/ORZ	Origin point of the rotation
NX/NY/NZ	Normal for the rotation
ANGLE	Angle of rotation (in degrees)

*ICFD_DEFINE_TURBSOURCE

Purpose: Define an external source for the RANS turbulence equations.

Card 1	1	2	3	4	5	6	7	8
Variable	TSID	LCIDK	LCIDEP	LCIDNU	ISHAPE	R	PTID1	PTID2
Type	I	I	I	I	I	F	I	I
Default	none	none	none	none	none	none	none	none

VARIABLE	DESCRIPTION
TSID	Turbulent external source ID
LCIDK	Load curve ID specifying the evolution of the external source term function of time for the turbulent kinetic energy, k, equation (see *DEFINE_CURVE,*DEFINE_CURVE_FUNCTION, or *DEFINE_FUNCTION). If using a *DEFINE_FUNCTION, the following arguments are allowed: f(x, y, z, vx, vy, vz, temp, pres, time).
LCIDEP	Load curve ID specifying the evolution of the external source term function of time for the turbulent diffusion, ε, or specific rate of dissipation, w, equation (see *DEFINE_CURVE,*DEFINE_CURVE_FUNCTION, or *DEFINE_FUNCTION). If using a *DEFINE_FUNCTION, the following arguments are allowed: f(x, y, z, vx, vy, vz, temp, pres, time).
LCIDNU	Load curve ID specifying the evolution of the external source term function of time for the kinematic eddy turbulent viscosity equation used in the Spalart-Allmaras model (see *DEFINE_CURVE,*DEFINE_CURVE_FUNCTION, or *DEFINE_FUNCTION). If using a *DEFINE_FUNCTION, the following arguments are allowed: f(x, y, z, vx, vy, vz, temp, pres, time).
ISHAPE	Shape of the external source: EQ.1: Box shape EQ.2: Cylinder shape

VARIABLE	DESCRIPTION
	EQ.3: Sphere shape
R	Radius of the cylinder if ISHAPE = 2 or radius of the sphere if ISHAPE = 3
PTID1	Point ID (see *ICFD_DEFINE_POINT) whose meaning depends on ISHAPE: ISHAPE.EQ.1: Minimum coordinates of the box ISHAPE.EQ.2: Point in the center of one base of the cylinder ISHAPE.EQ.3: Center of the sphere
PTID2	Point ID (see *ICFD_DEFINE_POINT) whose meaning depends on ISHAPE: ISHAPE.EQ.1: Maximum coordinates of the box ISHAPE.EQ.2: Point in the center of the other base of the cylinder

.

*ICFD_DEFINE_POINT

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

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

Optional Card 2. Load curve IDS specifying velocity components of translating point

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

Optional Card 3. Load curve IDS and rotation axis of rotating point

Card 2	1	2	3	4	5	6	7	8
Variable	LCIDW	XT	YT	ZT	XH	YH	ZH	
Type	I	F	F	F	F	F	F	
Default	0	none	none	none	none	none	none	

VARIABLE	DESCRIPTION
POID	Point ID.
X/Y/Z	x, y ,z coordinates for the point.

VARIABLE	DESCRIPTION
CONSTPID	Surface Part ID to which the point is constrained. This means that if the selected surface moves, then the localization of the point will update as well.
LCIDX/LCIDY/LCIDZ	The point can be made to translate. Those are the three load curve IDs for the three translation components.
LCIDW	The point can also be made to rotate. This load curve specifies the angular velocity.
XT/YT/ZT	Rotation axis tail point coordinates.
XH/YH/ZH	Rotation axis head point coordinates.

.

*ICFD_DEFINE_NONINERTIAL

Purpose: Define a non-inertial reference frame to avoid heavy mesh distortions and remeshing associated with large-scale rotations. This frame helps when modeling spinning cylinders, wind turbines, and turbomachinery.

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

Card 1	1	2	3	4	5	6	7	8
Variable	W1	W2	W3	R	PTID	L	LCID	RELV
Type	F	F	F	F	I	F	I	I
Default	none	none	none	none	none	none	none	0

VARIABLE	DESCRIPTION
W1, W2, W3	Rotational velocity, ω, along the X, Y, and Z-axes
R	Radius of the rotating reference frame. GT.0.0: Constant value LT.0.0: |R| refers to a *DEFINE_FUNCTION ID. The function supports the following arguments: f(x, y, z, vx, vy, vz, temp, pres, time).

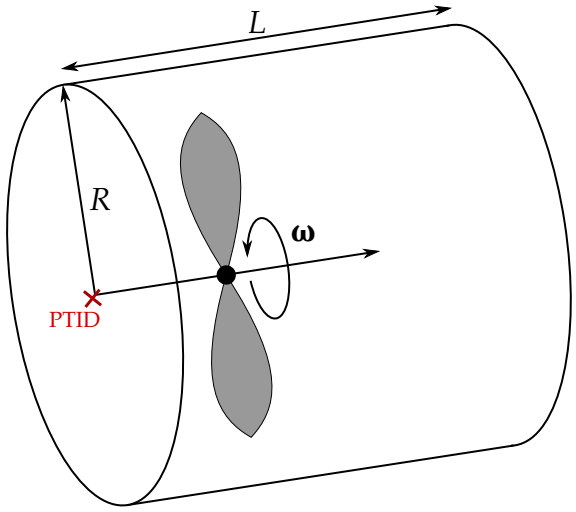


Figure 7-8. Example of using a non-inertial reference frame

VARIABLE	DESCRIPTION
PTID	Starting point ID for the reference frame (See *ICFD_DEFINE_-POINT)
L	Length of the rotating reference frame
LCID	Load curve for scaling factor of ω.
	GT.0: Load curve ID (see *DEFINE_CURVE) for the curve giving the scale factor as a function of time
	LT.0: |LCID| refers to a *DEFINE_FUNCTION ID. The function supports the following arguments: f(x, y, z, vx, vy, vz, temp, pres, time).
RELV	Velocities computed and displayed:
	EQ.0: Relative velocity, only the non-rotating components of the velocity are used and displayed.
	EQ.1: Absolute velocity. All the components of the velocity are used. Useful in cases where several or at least one non-inertial reference frame is combined with an inertial "classical" reference frame.

*ICFD_DEFINE_WAVE_DAMPING

Purpose: This keyword defines a damping zone for free surface waves.

Card 1	1	2	3	4	5	6	7	8
Variable	PID	NID	L	F1	F2	N	LCID	
Type	I	I	F	F	F	I	I	
Default	none	none		10	10	1	none	

VARIABLE	DESCRIPTION
PID	Point ID defining the start of the damping layer.
NID	Normal ID defined using ICFD_DEFINE_POINT and pointing to the outgoing direction of the damping layer.
L	Length of damping layer. If no is value specified, the damping layer will have a length corresponding to five element lengths.
F1/F2	Linear and quadratic damping factor terms.
N	Damping term factor.
LCID	Load curve ID acting as temporal scale factor on damping term.

Remarks:

1. The damping is achieved by adding a source term to the momentum equations :

$$s^d = w\ (f_1 + f_2|u|)\ u$$

with w the weight function :

$$w = \frac{e^{\gamma} - 1}{e - 1}$$

and γ the blending function which allows a smooth insertion of the source term in the damping layer :

$$\gamma = \left(\frac{x - x_{sd}}{x_{ed} - x_{sd}}\right)^n$$

x_{sd} and x_{ed} representing the start and end coordinates of the damping zone.

*ICFD_INITIAL

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

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

Card 1	1	2	3	4	5	6	7	8
Variable	PID	Vx	Vy	Vz	T	P		DFUNC
Type	I	F	F	F	F	F		I
Default	none	none	none	none	none	none		0

VARIABLE	DESCRIPTION
PID	Part ID for the volume elements or the surface elements where the values are initialized (see *ICFD_PART_VOL and *ICFD_PART). PID = 0 to assign the initial condition to all nodes at once.
Vx	x coordinate for the velocity.
Vy	y coordinate for the velocity.
Vz	z coordinate for the velocity.
T	Initial temperature.
P	Initial Pressure.
DFUNC	Option to define initial conditions using *DEFINE_FUNCTION EQ.0: Turned off. EQ.1: Turned on. All previous flags for initial velocity, pressure and temperature now refer to *DEFINE_FUNCTION IDs. The following parameters are allowed : $f(x,y,z)$, allowing to define initial profiles function of coordinates.

*ICFD_INITIAL_LEVELSET

Purpose: Define an initial level set surface instead of a multi-fluid domain (replaces the need for *MESH_INTERF).

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	STYPE	NX	NY	NZ	X	Y	Z	INVERT
Type	I	F	F	F	F	F	F	I
Default	none	none	none	none	none	none	none	0

VARIABLE	DESCRIPTION
STYPE	Initial surface type: EQ.0/1: Generated by a section plane. EQ.2: Generated by a box. See Remark 1. EQ.3: Generated by a sphere. EQ.4: Generated by a cylinder.
NX, NY, NZ	X, Y and Z components of the section plane normal if STYPE = 1. Minimum coordinates of the box, P_{min}, if STYPE = 2. NX is the sphere/cylinder radius if STYPE = 3 and STYPE = 4. NY is the cylinder length if STYPE = 4. NZ is the global axis if STYPE = 4 (NZ = 1, 2, or 3 means X-axis, Y-axis, or Z-axis, respectively).
X, Y, Z	X, Y and Z components of the section plane origin point if STYPE = 1 and 4. Maximum coordinates of the box, P_{max}, if STYPE = 2. Coordinates of the sphere origin point if STYPE = 3.
INVERT	Inversion of initial level set: EQ.0: No inversion. Positive level set values are assigned to nodes contained within the volume defined by STYPE. EQ.1: The sign of the initial level set values is reversed.

Remarks:

1. **Box Adjacent to Fluid Boundaries.** When STYPE = 2 is used and the box is adjacent to the fluid boundaries such as during a dam break simulation, the distance from any point in the fluid to the fluid boundary must remain smaller than the distance to the defined box. Therefore, the P_{min} coordinates need to be defined far outside the initial fluid domain.

2. **Multiple Keyword Definitions.** Multiple definitions of this keyword are possible, but the different shapes generated must not intersect for a correct initialization.

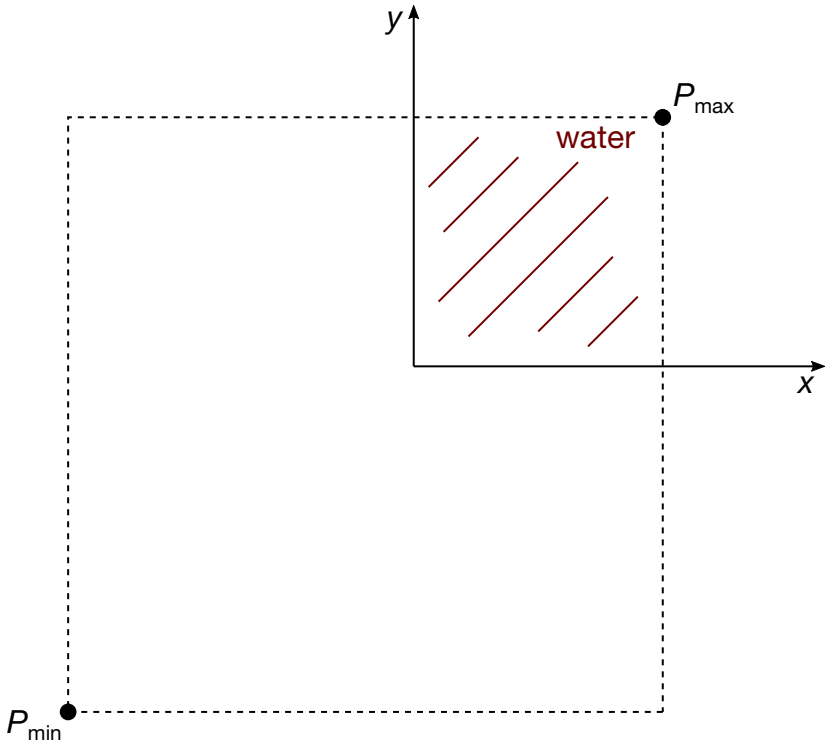


Figure 7-9. 2D Dam breaking example with initial levelset surface defined using STYPE = 2. P_{min} is defined sufficiently far away from the fluid surface boundaries.

*ICFD_INITIAL_SPTRANSP

Purpose: Initialize the concentration of the species being transported within a volume.

Card 1	1	2	3	4	5	6	7	8
Variable	PID	CONC1	IDFUNC1	CONC2	IDFUNC2			
Type	I	F	I	F	I			
Default	0	none	0	none	0			

VARIABLE	DESCRIPTION
PID	Part ID for the volume elements or surface elements where the values are initialized. EQ.0: Assign the initial condition to all nodes at once.
CONC*i*	Initial concentration of species *i*. CONC*i* is a *DEFINE_FUNCTION ID if IDFUNC*i* = 1.
IDFUNC*i*	Flag to define initial conditions using a *DEFINE_FUNCTION: EQ.0: Turned off. EQ.1: Turned on. CONC*i* is a *DEFINE_FUNCTION ID. The following parameters are allowed: f(x, y, z). This allows for defining initial profiles as a function of coordinates.

*ICFD_INITIAL_TEMPNODE

Purpose: Allow the solver to initialize the temperature at individual nodes.

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	NID	TEMP						
Type	I	F						
Default	none	none						

VARIABLE	DESCRIPTION
NID	Internal ICFD node ID
TEMP	Initial temperature value

*ICFD_INITIAL_TURBULENCE

Purpose: Modify the default initial values of the turbulence quantities for a RANS turbulence 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	PID	I	R	K	EW			
Type	I	F	F	F	F			
Default	none	none	none	optional	optional			

VARIABLE	DESCRIPTION
PID	Part ID for the volume elements or the surface elements where the values are initialized (see *ICFD_PART_VOL and *ICFD_PART). Set PID to 0 to assign the initial condition to all nodes at once.
I	Initial turbulence intensity, I
R	Initial turbulence viscosity to laminar viscosity ratio ($r = \mu_{\text{turb}}/\mu$).
K	Initial kinetic energy. When defined, it replaces the choice of I. LT.0.0: |K| refers to a *DEFINE_FUNCTION ID. The following parameters are allowed: $f(x,y,z)$, allowing you to define initial profiles as a function of coordinates.
EW	Initial turbulence specific dissipation rate or dissipation rate depending on the choice of turbulence model. When defined, it replaces the choice of R. LT.0.0: |EW| refers to a *DEFINE_FUNCTION ID. The following parameters are allowed: $f(x,y,z)$, allowing you to define initial profiles as a function of coordinates.

Remarks:

1. **Default Initial Conditions.** If no initial conditions have been assigned to a specific PID, the solver will automatically pick I = 0.05 (5%) and R = 10000.

***ICFD_MAT**_*{OPTION}*

Available options include

TITLE

Purpose: Specify physical properties for the fluid material.

Fluid Material Card Sets:

The Material Fluid Parameters Card is required. If a second card is given, it must be a Thermal Fluid Parameters Card. If the fluid thermal properties are not needed, the second card can be a blank card. With the third card, you can associate the fluid material to a non-Newtonian model, to a porous media model, to a species transport model, and/or to a viscoelastic model (see *ICFD_MODEL_NONNEWT, *ICFD_MODEL_POROUS, *ICFD_MODEL_SPECIES_TRANSPORT, and *ICFD_MODEL_VISCOELASTIC).

Material Fluid Parameters Card.

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

Thermal Fluid Parameters Card. Only to be defined if the thermal problem is solved. Otherwise include a blank card.

Card 2	1	2	3	4	5	6	7	8
Variable	HC	TC	BETA	PRT	HCSFLCID	TCSFLCID		
Type	F	F	F	F	I	I		
Default	0.	0.	0.	0.85	none	none		

Additional fluid models. Only to be defined if the fluid is non-Newtonian, there is a porous media, and/or a species is being transported.

Card 3	1	2	3	4	5	6	7	8
Variable	NNMOID	PMMOID		SPTRID		VID		
Type	I	I		I		I		
Default	optional	optional		optional		optional		

VARIABLE	DESCRIPTION
MID	Material ID
FLG	Flag to choose between fully incompressible, slightly compressible, or barotropic flows: EQ.0: Vacuum (free surface problems only) EQ.1: Fully incompressible fluid.
RO	Flow density
VIS	Dynamic viscosity
ST	Surface tension coefficient
STSFLCID	Load curve ID for scale factor applied on ST as a function of time. See *DEFINE_CURVE, *DEFINE_CURVE_FUNCTION, or *DEFINE_FUNCTION. If a *DEFINE_FUNCTION is used, the following parameters are allowed: f(x, y, z, vx, vy, vz, temp, pres, time).
CA	Contact angle
HC	Heat capacity
TC	Thermal conductivity
BETA	Thermal expansion coefficient used in the Boussinesq approximation for buoyancy
PRT	Turbulent Prandlt number. Only used if K-Epsilon turbulence model selected.

VARIABLE	DESCRIPTION
HCSFLCID	Load curve ID for scale factor applied on HC function of time. See *DEFINE_CURVE, *DEFINE_CURVE_FUNCTION, or *DEFINE_-FUNCTION. If a *DEFINE_FUNCTION is used, the following parameters are allowed: f(x, y, z, vx, vy, vz, temp, pres, time).
TCSFLCID	Load curve ID for scale factor applied on TC function of time. See *DEFINE_CURVE, *DEFINE_CURVE_FUNCTION, or *DEFINE_-FUNCTION. If a *DEFINE_FUNCTION is used, the following parameters are allowed: f(x, y, z, vx, vy, vz, temp, pres, time).
NNMOID	Non-Newtonian model ID. This refers to a non-Newtonian fluid model defined using *ICFD_MODEL_NONNEWT.
PMMOID	Porous media model ID. This refers to a porous media model defined using *ICFD_MODEL_POROUS.
SPTRID	Species transport model ID. This refers to a species transport model defined using *ICFD_MODEL_SPECIES_TRANSPORT.
VID	Viscoelastic model ID. This refers to a viscoelastic model defined using *ICFD_MODEL_VISCOELASTIC.

*ICFD_MODEL_NONNEWT

Purpose: Specify a non-newtonian model or a viscosity law that can associated to a fluid material.

Non-Newtonian Model ID and type.

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

Non-Newtonian Fluid Parameters Card.

Card 2	1	2	3	4	5	6	7	8
Variable	K	N	MUMIN	LAMBDA	ALPHA	TALPHA		
Type	F	F	F	F	F	F		
Default	0.0	0.0	0.0	1.e30	0.0	0.0		

VARIABLE	DESCRIPTION
NNMOID	Non-Newtonian Model ID.
NNID	Non-Newtonian fluid model type : EQ.1 : Power-Law model EQ.2 : Carreau model EQ.3 : Cross model EQ.4 : Herschel-Bulkley model EQ.5 : Cross II model EQ.6 : Sutherland formula for temperature dependent viscosity EQ.7 : Power-Law for temperature dependent viscosity EQ.8 : Viscosity defined by Load Curve ID or Function ID

VARIABLE	DESCRIPTION
K	Consistency index if NNID = 1 and 4. Zero shear Viscosity if NNID = 2,3 and 5.Reference viscosity if NNID = 6 and NNID = 7. Load curve ID or function ID if NNID = 8.
N	Measure of the deviation of the fluid from Newtonian (Power Law index) for NNID = 1,2,3,4,5,7. Not used for NNID = 6 and 8.
MUMIN	Minimum acceptable viscosity value if NNID = 1. Infinite Shear Viscosity if NNID = 2,5.Yielding viscosity if NNID = 4.Not used if NNID = 3,6,7,8.
LAMBDA	Maximum acceptable viscosity value if NNID = 1. Time constant if NNID = 2, 3, 5. Yield Stress Threshold if NNID = 4.Sutherland constant if NNID = 6. Not used if NNID = 7,8.
ALPHA	Activation energy if NNID = 1, 2. Not used if NNID = 3,4,5,6,7,8.
TALPHA	Reference temperature if NNID = 2. Not used if NNID = 1,3,4,5,6,7,8

Remarks:

1. For the Non-Newtonian models, the viscosity is expressed as :

   a) POWER-LAW :

$$\mu = k\dot{\gamma}^{n-1}e^{\alpha T_0/T}$$

$$\mu_{min} < \mu < \mu_{max}$$

   With k the consistency index, n the power law index, α the activation energy, T_0 the initial temperature, T the temperature at any given time t, μ_{min} the minimum acceptable viscosity and μ_{max} the maximum acceptable viscosity.

   b) CARREAU :

$$\mu = \mu_\infty + (\mu_0 - \mu_\infty)\left[1 + (H(T)\dot{\gamma}\lambda)^2\right]^{(n-1)/2}$$

$$H(T) = exp\left[\alpha\left(\frac{1}{T - T_0} - \frac{1}{T_\alpha - T_0}\right)\right]$$

   With μ_∞ the infinite shear viscosity, μ_0 the zero shear viscosity, n the power law index, λ a time constant, α the activation energy, T_0 the initial temperature, T

the temperature at any given time t and T_α the reference temperature at which $H(T) = 1$.

c) CROSS :

$$\mu = \frac{\mu_0}{1 + (\lambda\dot{\gamma})^{1-n}}$$

With μ_0 the zero shear viscosity, n the power law index and λ a time constant.

d) HERSCHEL-BULKLEY :

$$\mu = \mu_0 \; if \; (\dot{\gamma} < \tau_0/\mu_0)$$

$$\mu = \frac{\tau_0 + k[\dot{\gamma}^n - (\tau_0/\mu_0)^n]}{\dot{\gamma}}$$

With k the consistency index, τ_0 the Yield stress threshold, μ_0 the yielding viscosity and n the power law index.

e) CROSS II :

$$\mu = \mu_\infty + \frac{\mu_0 - \mu_\infty}{1 + (\lambda\dot{\gamma})^n}$$

With μ_0 the zero shear viscosity, μ_∞ the infinite shear viscosity, n the power law index and λ a time constant.

2. For the temperature dependent viscosity models, the viscosity is expressed as :

a) SUTHERLAND's LAW :

$$\mu = \mu_0 (\frac{T}{T_0})^{3/2} \frac{T_0 + S}{T + S}$$

With μ_0 a reference viscosity, T_0 the initial temperature (which therefore must not be 0.), T the temperature at any given time t and S Sutherland's constant.

b) POWER LAW :

$$\mu = \mu_0 (\frac{T}{T_0})^n$$

With μ_0 a reference viscosity, T_0 the initial temperature (which therefore must not be 0.), T the temperature at any given time t and n the power law index.

3. For NNID = 8, a load curve function of time, a curve function or a function can be used. If it references a DEFINE_FUNCTION, the following arguments are allowed $f(x, y, z, vx, vy, vz, temp, pres, shear, time)$.

*ICFD_MODEL_POROUS

Purpose: Specify a porous media model.

Card Summary:

Card 1. This card is required.

PMMOID	PMMTYPE	FORM	RHOCP	KAPPA			

Card 2a. This card is included if PMMTYPE = 1, 2, or 8.

POR	PER	FF		PSFLCID			

Card 2b. This card is included if PMMTYPE = 3 or 10.

POR	TH		FABTH	PVLCID			

Card 2c. This card is included if PMMTYPE = 4, 6, or 7.

POR							

Card 2d. This card is included if PMMTYPE = 5.

POR	THX	THY	THZ	PVLCIDX	PVLCIDY	PVLCIDZ	

Card 2e. This card is included if PMMTYPE = 11.

POR	ALPHA	BETA					

Card 3. This card is included if PMMTYPE = 4, 5, 6, or 7

KXP	KYP	KZP					

Card 4a. This card is included if PMMTYPE = 4 or 6.

PROJXPX	PROJXPY	PROJXPZ	PROJYPX	PROJYPY	PROJYPZ		

Card 4b. This card is included if PMMTYPE = 5 or 7.

PID1REF	PID2REF						

Data Card Definitions:

Card 1	1	2	3	4	5	6	7	8
Variable	PMMOID	PMMTYPE	FORM	RHOCP	KAPPA			
Type	I	I	I	F	F			
Default	none	none	0	0.0	0.0			

VARIABLE	DESCRIPTION
PMMOID	Porous media model ID
PMMTYPE	Porous media model type: EQ.1: Isotropic porous media - Ergun correlation EQ.2: Isotropic porous media - Darcy-Forchheimer model EQ.3: Isotropic porous media - permeability defined through pressure-velocity data EQ.4: Anisotropic porous media. Fixed local reference frame (see Figure 7-10). EQ.5: Anisotropic porous media model - moving local reference frame and permeability vector in local reference frame (x', y', z') defined by three pressure-velocity curves. EQ.6: Anisotropic porous media model - moving local reference frame and permeability vector constant. EQ.7: Anisotropic porous media model - moving local reference frame and permeability vector constant. This model differs from PMMTYPE = 6 in the way the local reference frame is moved. EQ.8: Main parachute model to be used jointly with *MESH_EMBEDSHELL for the parachute surface. Similar to PMMTYPE = 2. EQ.10: Parachute model to be used jointly with *MESH_EMBEDSHELL where the fabric permeability and Forchheimer factor are computed from the pressure-velocity curves of experimental data given by a *LOAD_CURVE. Similar to PMMTYPE = 3.

VARIABLE	DESCRIPTION
	EQ.11: Parachute model similar to PMMTYPE = 8, but pressure gradient is directly defined by coefficients α and β as: $\frac{\Delta P(u_x)}{\Delta x} = \alpha u_x + \beta {u_x}^2$.
FORM	Porous media formulation: EQ.0: Classical (default) EQ.2: Interstitial velocity
RHOCP	Density of the structure multiplied by the specific heat of the structure
KAPPA	Thermal conductivity of the structure

Porous Media Parameters Card (PMMTYPE = 1, 2, and 8). This card is included PMMTYPE = 1, 2, or 8.

Card 2a	1	2	3	4	5	6	7	8
Variable	POR	PER	FF		PSFLCID			
Type	F	F	F		I			
Default	0.	0.	0.		optional			

VARIABLE	DESCRIPTION
POR	Porosity, ε
PER	Permeability, κ
FF	Forchheimer factor to be defined if PMMTYPE = 2 or 8.
PSFLCID	Optional permeability scale factor load curve ID, *DEFINE_CURVE_FUNCTION ID or *DEFINE_FUNCTION ID. If a *DEFINE_FUNCTION is used, the following parameters are allowed: f(x, y, z, vx, vy, vz, temp, pres, time).

Porous Media Parameters Card (PMMTYPE = 3 and 10). This card is included if PMMTYPE = 3 or 10.

Card 2b	1	2	3	4	5	6	7	8
Variable	POR	TH		FABTH	PVLCID			
Type	F	F		F	I			
Default	0.	0.		0.	none			

VARIABLE	DESCRIPTION
POR	Porosity, ε
TH	Probe thickness if PMMTYPE = 3
FABTH	Fabric thickness if PMMTYPE = 10
PVLCID	Pressure as a function of velocity load curve ID

Porous Media Parameters Card (PMMTYPE = 4, 6, and 7). This card is included if PMMTYPE = 4, 6, or 7.

Card 2c	1	2	3	4	5	6	7	8
Variable	POR							
Type	F							
Default	0.							

VARIABLE	DESCRIPTION
POR	Porosity, ε

Porous Media Parameters Card (PMMTYPE = 5). This card is included if PMMTYPE = 5.

Card 2d	1	2	3	4	5	6	7	8
Variable	POR	THX	THY	THZ	PVLCIDX	PVLCIDY	PVLCIDZ	
Type	F	F	F	F	I	I	I	
Default	0.	0.	0.	0.	none	none	none	

VARIABLE	DESCRIPTION
POR	Porosity, ε
THX	Probe thickness, Δx
THY	Probe thickness, Δy
THZ	Probe thickness, Δz
PVLCIDX	Load curve ID for pressure as a function of velocity in the global X-direction
PVLCIDY	Load curve ID for pressure as a function of velocity in the global Y-direction
PVLCIDZ	Load curve ID for pressure as a function of velocity in the global Z-direction

Porous Media Parameters Card (PMMTYPE = 11). This card is included if PMMTYPE = 11.

Card 2e	1	2	3	4	5	6	7	8
Variable	POR	ALPHA	BETA					
Type	F	F	F					
Default	0.	0.	0.					

VARIABLE	DESCRIPTION
POR	Porosity, ε
ALPHA	Coefficient, α
BETA	Coefficient, β

Permeability Vector Card in local reference frame. Only to be defined if the porous media is anisotropic (PMMTYPE = 4, 5, 6, 7).

Card 3	1	2	3	4	5	6	7	8
Variable	KXP	KYP	KZP					
Type	F	F	F					
Default	0.	0.	0.					

VARIABLE	DESCRIPTION
KXP, KYP, KZP	Permeability vector in local reference frame (x', y', z'). Those values become scale factors if PMMTYPE = 5.

Projection of Local Vectors in Global Reference Frame. This card is defined if PMMTYPE = 4 or 6.

Card 4a	1	2	3	4	5	6	7	8
Variable	PROJXPX	PROJXPY	PROJXPZ	PROJYPX	PROJYPY	PROJYPZ		
Type	F/I	F/I	F/I	F/I	F/I	F/I		
Default	0./0	0./0	0./0	0./0	0./0	0./0		

VARIABLE	DESCRIPTION
PROJXPX, PROJXPY, PROJXPZ	Projection of the local permeability vector, **x**′, in the global reference frame, (x, y, z). If PMMTYPE = 6, PROJXPX, PROJXPY, and PRPJXPZ become load curve IDs, so the coordinates of the local **x**′ vector can change in time.

VARIABLE	DESCRIPTION
PROJYPX, PROJYPY, PROJYPZ	Projection of the local permeability vector, **y**′, in the global reference frame, (x,y,z). If PMMTYPE = 6, PROJYPX, PROJYPY, and PRPJYPZ become load curve IDs, so the coordinates of the local **y**′ vector can change in time.

Local Reference Frame Vectors. This card is defined if PMMTYPE = 5 or 7.

Card 4b	1	2	3	4	5	6	7	8
Variable	PID1REF	PID2REF						
Type	I	I						
Default	0	0.						

VARIABLE	DESCRIPTION
PID1REF, PID2REF	Two local reference frame vectors are defined by the coordinates of the two-point IDs defined by PID1REF and PID2REF. (See *ICFD_DEFINE_POINT). Since those points can be made to move, it is therefore possible to define a moving reference frame for the anisotropic porous media domain.

Remarks:

1. **Generalized Flow Equations in a Porous Media.** Let ε be the porosity and κ be the permeability of the porous media. Then,

$$\varepsilon = \frac{\text{void volume}}{\text{total volume}} \ .$$

u_i, the volume averaged velocity field, can then be defined in terms of the fluid velocity field, u_{if}, as:

$$u_i = \varepsilon u_{if} \ .$$

The generalized flow equations of momentum and mass conservation can be expressed as:

$$\frac{\partial u_i}{\partial x_i} = 0$$

$$\frac{\rho}{\varepsilon}\left[\frac{\partial u_i}{\partial t} + \frac{\partial}{\partial x_j}\left(\frac{u_i u_j}{\varepsilon}\right)\right] = -\frac{1}{\varepsilon}\frac{\partial (P\varepsilon)}{\partial x_i} + \frac{\mu}{\varepsilon}\left(\frac{\partial^2 u_i}{\partial x_j \partial x_j}\right) + \rho g_i - D_i$$

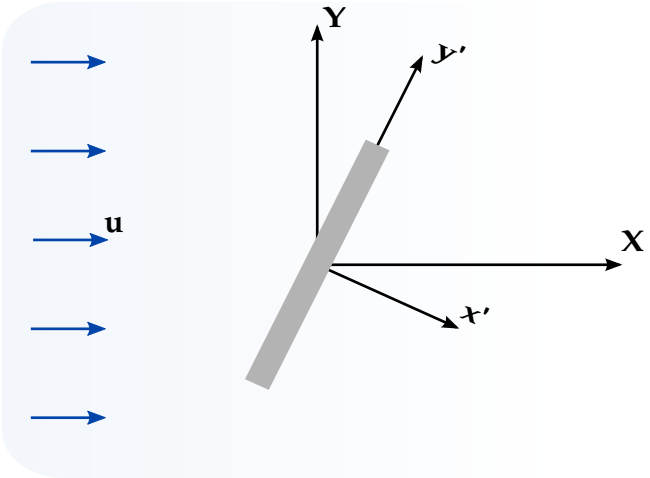


Figure 7-10. Anisotropic porous media vectors definition (PMMTYPE = 4, 5, 6, and 7). The vectors **X** and **Y** are the global axes; $\mathbf{x}'$ and $\mathbf{y}'$ define the system for the primed coordinate(x', y', z').

where D_i are the forces exerted on the fluid by the porous matrix (see Remarks 2 and 3).

2. **Porous Forces for Isotropic Models.** For the isotropic model, the porous forces are a function of the matrix porosity and its permeability. For the isotropic case, three models are available:

   a) *Model 1 (Ergun correlation).*

$$D_i = \frac{\mu u_i}{\kappa} + \frac{1.75\rho|U|}{\sqrt{150}\sqrt{\kappa}\varepsilon^{3/2}} u_i$$

   b) *Model 2 (Darcy-Forcheimer).*

$$D_i = \frac{\mu u_i}{\kappa} + \frac{F\varepsilon\rho|U|}{\sqrt{\kappa}} u_i$$

   c) *Model 3.* Using the $\Delta P - V$ experimental data. In this case, it is assumed that the pressure-velocity curve was obtained by applying a pressure difference or pressure drop on both ends of a porous slab of thickness Δx with porous properties κ and ε. It then becomes possible for the solver to fit that experimental curve with a quadratic polynomial of the form $\Delta P(u_x) = \alpha {u_x}^2 + \beta u_x$. Once α and β are known, it is possible to estimate D_i.

3. **Anisotropic Darcy-Forcheimer Term.** The anisotropic (see Figure 7-10) version of the Darcy-Forcheimer term can be written as:

$$D_i = \mu B_{ij} \mu_j + F\varepsilon|U|C_{ij}u_j$$
$$B_{ij} = \left(K_{ij}\right)^{-1}$$
$$C_{ij} = \left(K_{ij}\right)^{-1/2}$$

Here K_{ij} is the anisotropic permeability tensor.

*ICFD_MODEL_SPECIES_TRANSPORT

Purpose: Specify a species transport model that can be associated with a fluid material.

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

Card 2	1	2	3	4	5	6	7	8
Variable	MASSDIF1	LCIDDIF1	MASSDIF2	LCIDDIF1				
Type	F	I	F	I				
Default	none	0	none	0				

VARIABLE	DESCRIPTION
SPTRID	Species transport model ID
SPTRTYPE	Species transport model type: EQ.1: Passive species transport EQ.2: Passive two-species transport
MASSDIF*i*	Mass diffusion of the i^{th} species for the transport equation. Ignored if LCIDDIF*i* is defined.
LCIDDIF*i*	Load curve ID for the curve giving the mass diffusion of the i^{th} species as a function of time. It is optional.

*ICFD_MODEL_VISCOELASTIC

Purpose: Specify a viscoelastic model that can be associated with a fluid material.

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

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

VARIABLE	DESCRIPTION
VID	Viscoelastic model ID
VTYPE	Viscoelastic model type: EQ.1: Oldroyd-B model
PVISC	Polymeric viscosity
RTIME	Relaxation time

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

Available options include

TITLE

Purpose: Define parts for this incompressible flow solver.

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

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

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

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

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

***ICFD_PART_VOL**_{OPTION}

Available options include

TITLE

Purpose: This keyword assigns material properties to the nodes enclosed by surface ICFD parts.

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

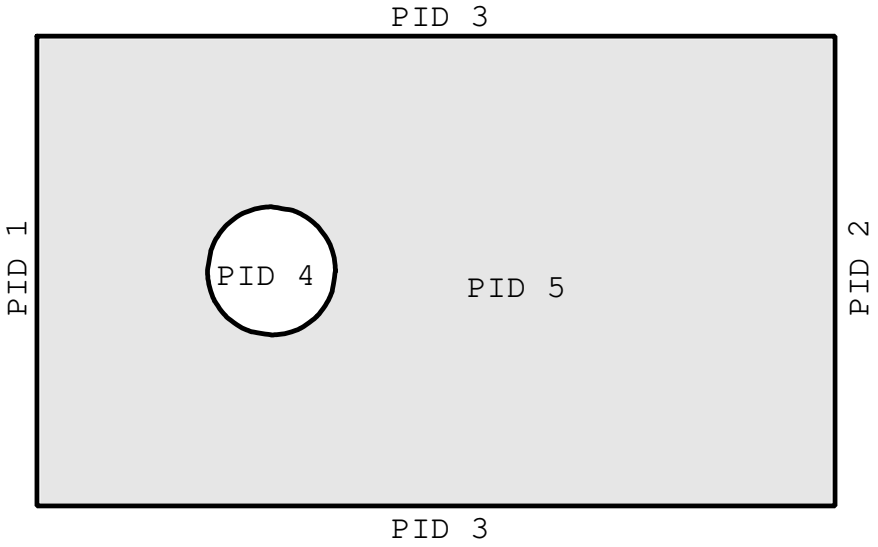
Title	1	2	3	4	5	6	7	8
Variable	HEADING							
Type	A							
Default	none							

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

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

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

VARIABLE	DESCRIPTION
PID	Part identifier for fluid volumes.
SECID	Section identifier defined by the *ICFD_SECTION card.
MID	Material identifier.
SPID1, …	Part IDs for the surface elements that define the volume mesh.




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

*ICFD_SECTION

Purpose: Define a section for the incompressible flow solver.

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

VARIABLE	DESCRIPTION
SID	Section identifier.

*ICFD_SET_NODE_LIST

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

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

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

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

VARIABLE	DESCRIPTION
SID	Set ID
PID	Associated Part ID.
NID1, …	Node IDs

Remarks:

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

*ICFD_SOLVER_SPLIT

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

Card 1	1	2	3	4	5	6	7	8
Variable	NIT	TOL						
Type	I	F						
Default	1	10^{-3}						

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

*ICFD_SOLVER_TOL_FSI

Purpose: This keyword allows the user to change the default tolerance values for the Newton Raphson loop in the strong FSI analysis. *Care should be taken when deviating from the default values*.

Card 1	1	2	3	4	5	6	7	8
Variable	ATOL	RTOL		MAXIT				
Type	F	F		I				
Default	10^{-5}	10^{-5}		1000				

VARIABLE	DESCRIPTION
ATOL	Absolute convergence criteria. Convergence is achieved when $\text{Residual}_{i+1} - \text{Residual}_i \leq \text{ATOL}$. If a negative integer is entered, then that value will be used as a load curve ID for ATOL.
RTOL	Relative convergence criteria. Convergence is achieved when $(\text{Residual}_{i+1} - \text{Residual}_i)/\text{Residual}_{\text{initial}} \leq RTOL$. If a negative integer is entered, then that value will be used as a load curve ID for RTOL.
MAXIT	Maximum number of iterations allowed to achieve convergence. If a negative integer is entered, then that value will be used as a load curve ID for MAXIT.

*ICFD_SOLVER_TOL_LSET

Purpose: This keyword allows the user to change the default tolerance values for the advection equation for levelset. *Care should be taken when deviating from the default values*.

Card 1	1	2	3	4	5	6	7	8
Variable	ATOL	RTOL		MAXIT				
Type	F	F		I				
Default	10^{-8}	10^{-8}		1000				

VARIABLE	DESCRIPTION
ATOL	Absolute convergence criteria. Convergence is achieved when $\text{Residual}_{i+1} - \text{Residual}_i \leq \text{ATOL}$. If a negative integer is entered, then that value will be used as a load curve ID for ATOL.
RTOL	Relative convergence criteria. Convergence is achieved when $(\text{Residual}_{i+1} - \text{Residual}_i)/\text{Residual}_{\text{initial}} \leq RTOL$. If a negative integer is entered, then that value will be used as a load curve ID for RTOL.
MAXIT	Maximum number of iterations allowed to achieve convergence. If a negative integer is entered, then that value will be used as a load curve ID for MAXIT.

*ICFD_SOLVER_TOL_MMOV

Purpose: This keyword allows the user to change the default tolerance values for the mesh movement algorithm. *Care should be taken when deviating from the default values*.

Card 1	1	2	3	4	5	6	7	8
Variable	ATOL	RTOL		MAXIT		DISPTOL		
Type	F	F		I		F		
Default	1e-8	1e-8		1000		0.		

VARIABLE	DESCRIPTION
ATOL	Absolute convergence criteria. Convergence is achieved when $\text{Residual}_{i+1} - \text{Residual}_i \leq \text{ATOL}$. If a negative integer is entered, then that value will be used as a load curve ID for ATOL.
RTOL	Relative convergence criteria. Convergence is achieved when $(\text{Residual}_{i+1} - \text{Residual}_i)/\text{Residual}_{\text{initial}} \leq \text{RTOL}$. If a negative integer is entered, then that value will be used as a load curve ID for RTOL.
MAXIT	Maximum number of iterations allowed to achieve convergence. If a negative integer is entered, then that value will be used as a load curve ID for MAXIT.
DISPTOL	Element deformation tolerance before a matrix reassembly is triggered. Default is 0. which means any element deformation detected will automatically trigger a matrix reassembly. Higher values will potentially save calculation times at the expense of accuracy.

*ICFD_SOLVER_TOL_MOM

Purpose: This keyword allows the user to change the default tolerance values for the momentum equation solve. *Care should be taken when deviating from the default values*.

Card 1	1	2	3	4	5	6	7	8
Variable	ATOL	RTOL		MAXIT				
Type	F	F		I				
Default	10^{-8}	10^{-8}		1000				

VARIABLE	DESCRIPTION
ATOL	Absolute convergence criteria. Convergence is achieved when $\text{Residual}_{i+1} - \text{Residual}_i \leq \text{ATOL}$. If a negative integer is entered, then that value will be used as a load curve ID for ATOL.
RTOL	Relative convergence criteria. Convergence is achieved when $(\text{Residual}_{i+1} - \text{Residual}_i)/\text{Residual}_{\text{initial}} \leq \text{RTOL}$. If a negative integer is entered, then that value will be used as a load curve ID for RTOL.
MAXIT	Maximum number of iterations allowed to achieve convergence. If a negative integer is entered, then that value will be used as a load curve ID for MAXIT.

*ICFD_SOLVER_TOL_MONOLITHIC

Purpose: This keyword allows the user to change the default tolerance values for the monolithic solver. *Care should be taken when deviating from the default values*.

Card 1	1	2	3	4	5	6	7	8
Variable	ATOL	RTOL		MAXIT				
Type	F	F		I				
Default	10^{-8}	10^{-8}		1000				

VARIABLE	DESCRIPTION
ATOL	Absolute convergence criteria. Convergence is achieved when $\text{Residual}_{i+1} - \text{Residual}_i \leq \text{ATOL}$. If a negative integer is entered, then that value will be used as a load curve ID for ATOL.
RTOL	Relative convergence criteria. Convergence is achieved when $(\text{Residual}_{i+1} - \text{Residual}_i)/\text{Residual}_{\text{initial}} \leq \text{RTOL}$. If a negative integer is entered, then that value will be used as a load curve ID for RTOL.
MAXIT	Maximum number of iterations allowed to achieve convergence. If a negative integer is entered, then that value will be used as a load curve ID for MAXIT.

*ICFD_SOLVER_TOL_PRE

Purpose: Change the default tolerance values for the Poisson equation for pressure. *Care should be taken when deviating from the default values*.

Card 1	1	2	3	4	5	6	7	8
Variable	ATOL	RTOL		MAXIT	PREID	PTOL		
Type	F	F		I	I	F		
Default	10^{-8}	10^{-8}		1000	2	10^{-3}		

VARIABLE	DESCRIPTION
ATOL	Absolute convergence criteria. Convergence is achieved when $\text{Residual}_{i+1} - \text{Residual}_i \leq \text{ATOL}$. If a negative integer is entered, then that value will be used as a load curve ID for ATOL.
RTOL	Relative convergence criteria. Convergence is achieved when $(\text{Residual}_{i+1} - \text{Residual}_i)/\text{Residual}_{\text{initial}} \leq \text{RTOL}$. If a negative integer is entered, then that value will be used as a load curve ID for RTOL.
MAXIT	Maximum number of iterations allowed to achieve convergence. If a negative integer is entered, then that value will be used as a load curve ID for MAXIT.
PREID	Choice of Preconditioner for the Conjugate Gradient Solve: EQ.1: Diagonal Preconditioner EQ.2: Incomplete LU factorization EQ.5: Global MUMPS factorization
PTOL	Preconditioner tolerance (a.k.a Drop Tolerance if PREID = 2)

*ICFD_SOLVER_TOL_TEMP

Purpose: This keyword allows the user to change the default tolerance values for the heat equation. To be handled with great care.

Card 1	1	2	3	4	5	6	7	8
Variable	ATOL	RTOL		MAXIT				
Type	F	F		I				
Default	1e-8	1e-8		1000				

VARIABLE	DESCRIPTION
ATOL	Absolute convergence criteria. Convergence is achieved when $Residual_{i+1} - Residual_i \leq ATOL$. If a negative integer is entered, then that value will be used as a load curve ID for ATOL.
RTOL	Relative convergence criteria. Convergence is achieved when $(Residual_{i+1} - Residual_i)/Residual_{initial} \leq RTOL$. If a negative integer is entered, then that value will be used as a load curve ID for RTOL.
MAXIT	Maximum number of iterations allowed to achieve convergence. If a negative integer is entered, then that value will be used as a load curve ID for MAXIT.