Ennova Technologies

Ennova Release 1.11
Explainer Video : OpenFOAM Versions

07th January 2025

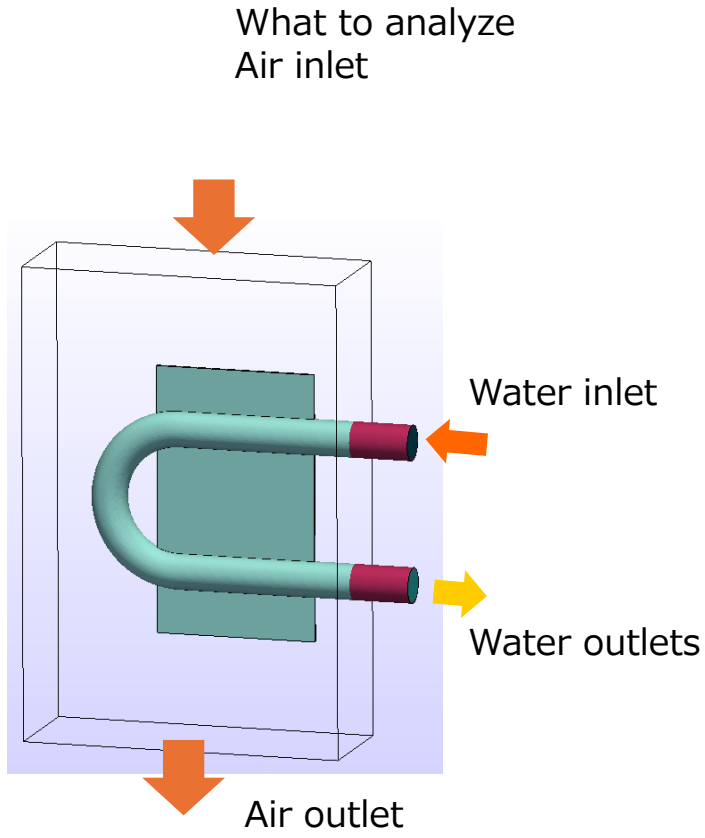
Ennova Technologies

ennova-cfd.com

Fluid-solid thermal coupling analysis

Overview of CHT Analysis

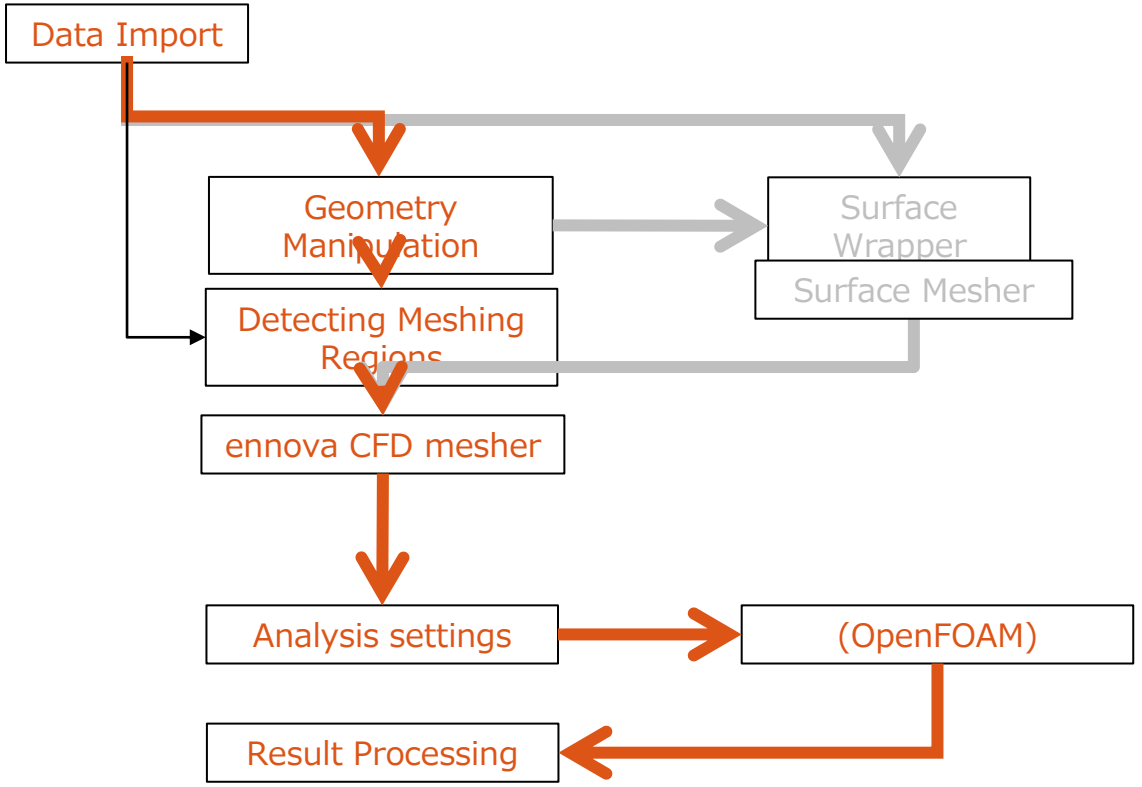
- This procedure manual describes the procedure for creating the shape, setting up the analysis, performing the analysis, and processing the results of the heat flow analysis of a heat exchanger with three regions: air, water, and solids, using ennovaCFD v1.10.1 and the chtMultiRegionSimpleFoam solver, which is a steady-state solid-state fluid coupled analysis solver of OpenFOAM.
- The target of analysis is the heat exchanger shown in the figure on the right.
- Fluid regions (air, water) and solid regions are modeled using ennovaCFD's polyhedral mesh.
- The total number of meshes is about 240,000 meshes, and the generation time is about 5 minutes.



Fluid-solid thermal coupling analysis

Overview of CHT Analysis

- With ennovaCFD, you can create and analyze analysis models in various flows. In this procedure, we will create a model according to the following flow.



Fluid-solid thermal coupling analysis

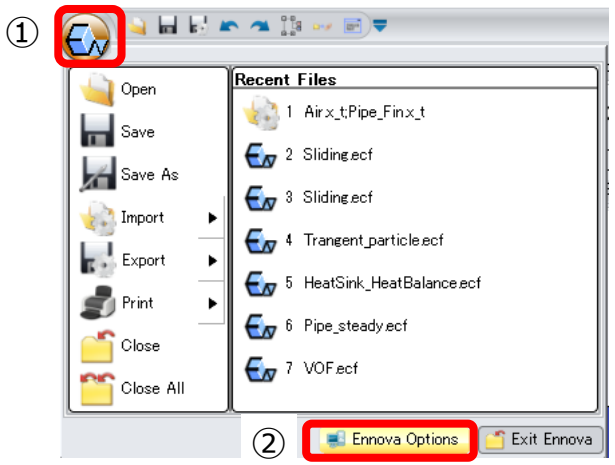
Launching ennova CFD

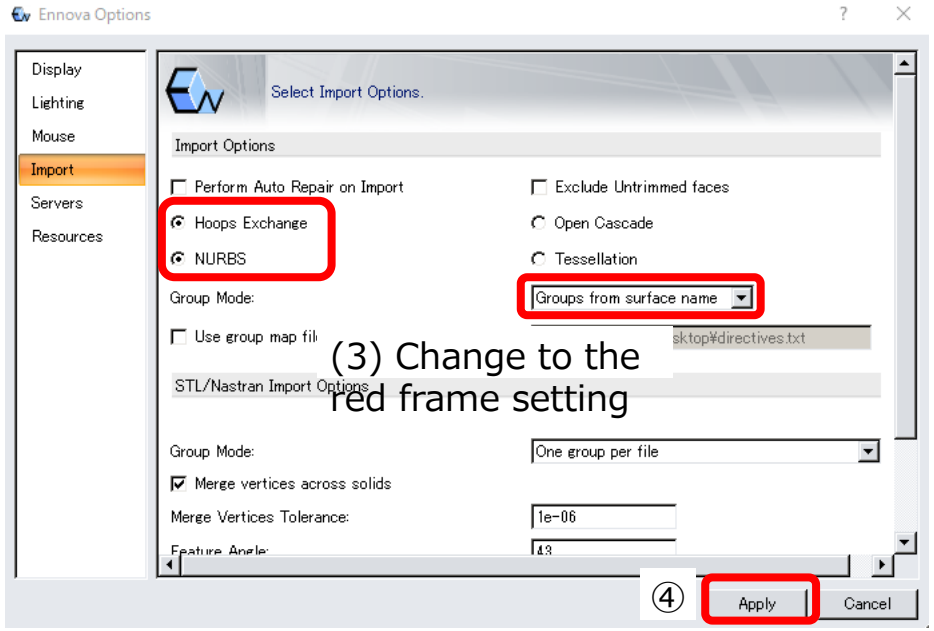
- Launch ennova CFD.
- If you are using Windows-PC, there is an icon for booting in the installation folder.
- If you are using Linux-PC, start it with the ennova command in the installation folder.
- If a license error occurs, it is necessary to set environment variables, etc., so please consult the license administrator or our support.

Fluid-solid thermal coupling analysis

File Import

- Before importing, change the import options.

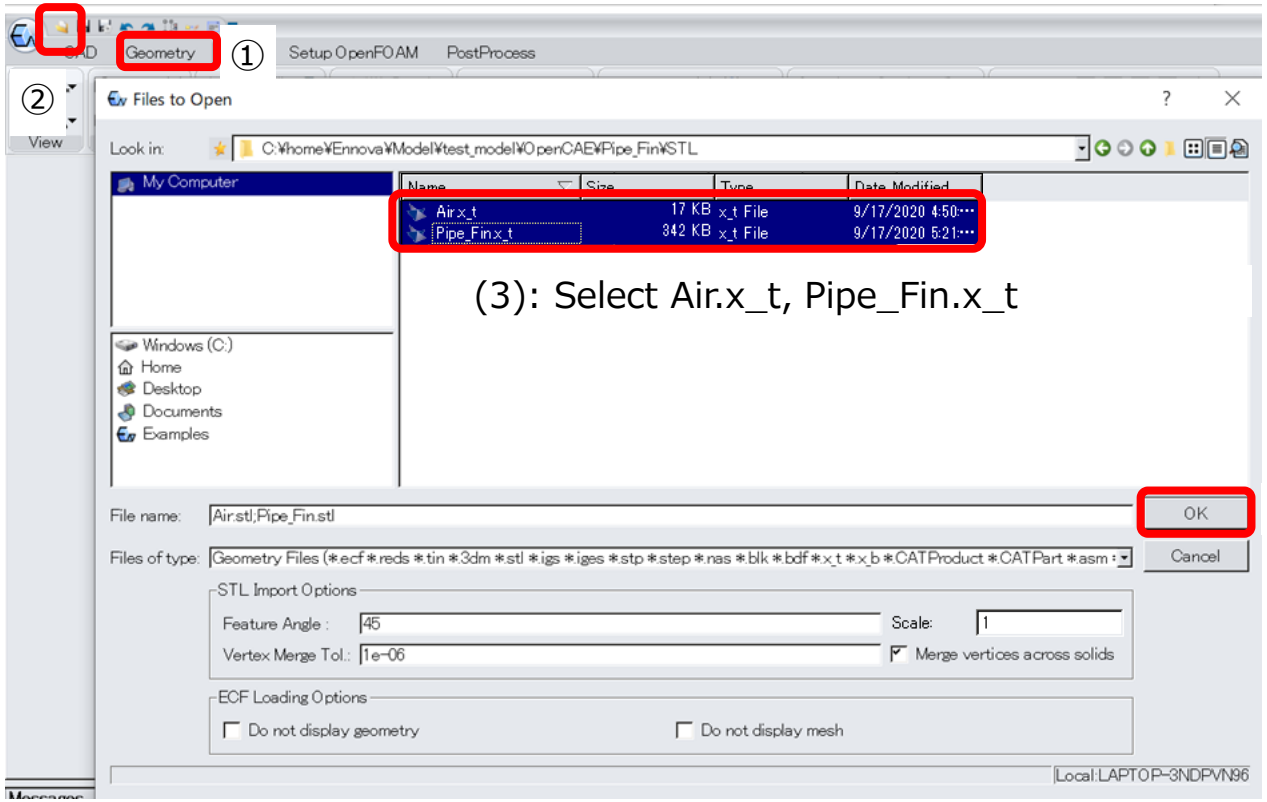




Fluid-solid thermal coupling analysis

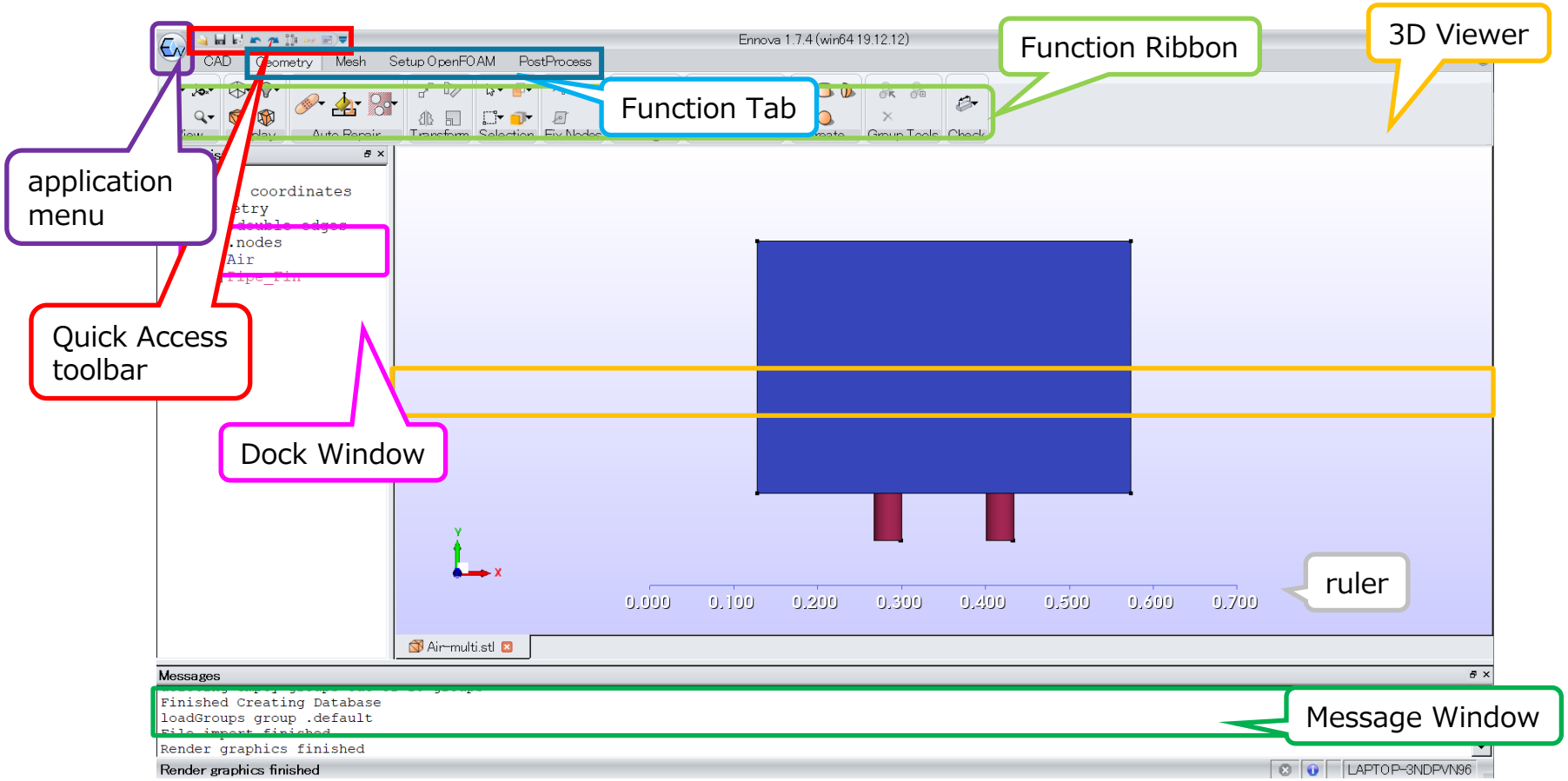
File Import

- In this case, we have prepared Parasolid data, so select the file and import it.
  In the Functions tab, select "Geometry".



Fluid-solid thermal coupling analysis

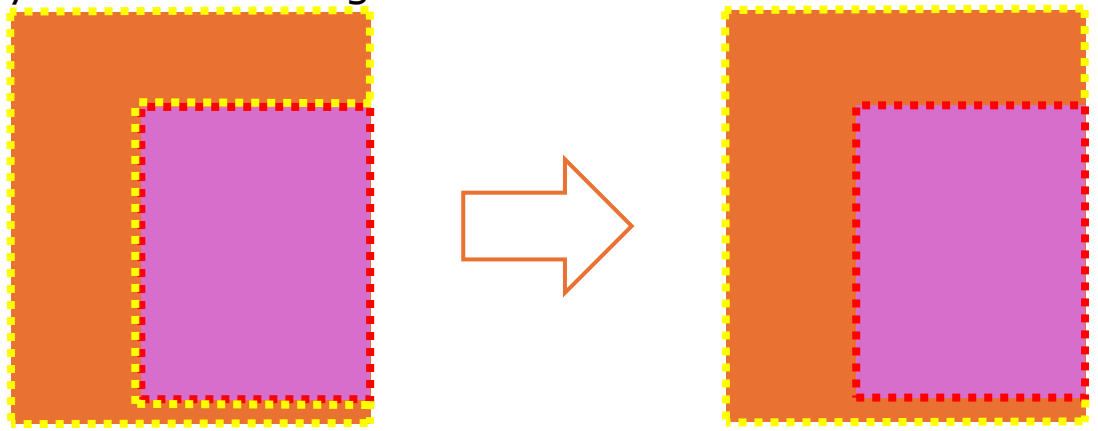
Overview of GUI Operations/Name of GUI



Fluid-solid thermal coupling analysis

About Geometry Editing/Preparing Shapes

- In order to generate a multi-region mesh with ennovaCFD, the following conditions must be met.
- Inter-domain interfaces are shared on a single surface
- Closed surfaces (no single edges)If you run the Find Volume tool of ennovaCFD in the above state, it will copy another interface between regions and automatically detect each region.



A state in which there are two inter-domain interfaces

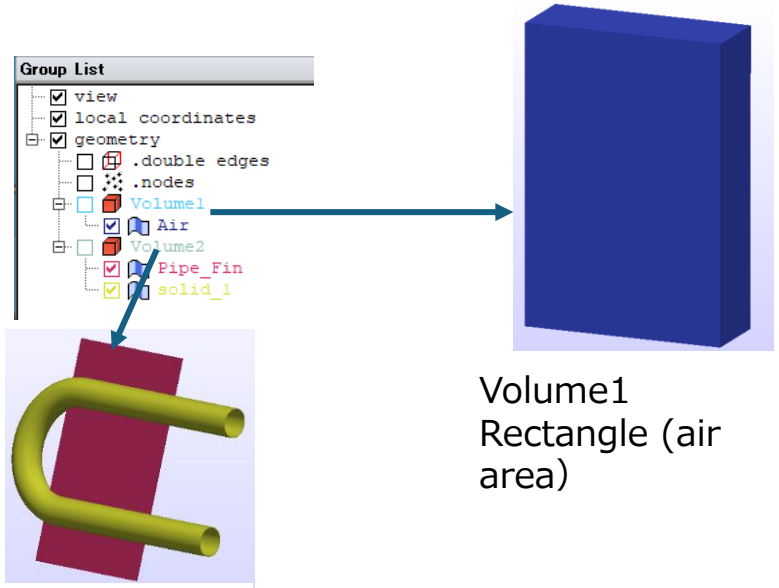
OK

The interface between the regions is shared on one surface, and there is no single edge.

Fluid-solid thermal coupling analysis

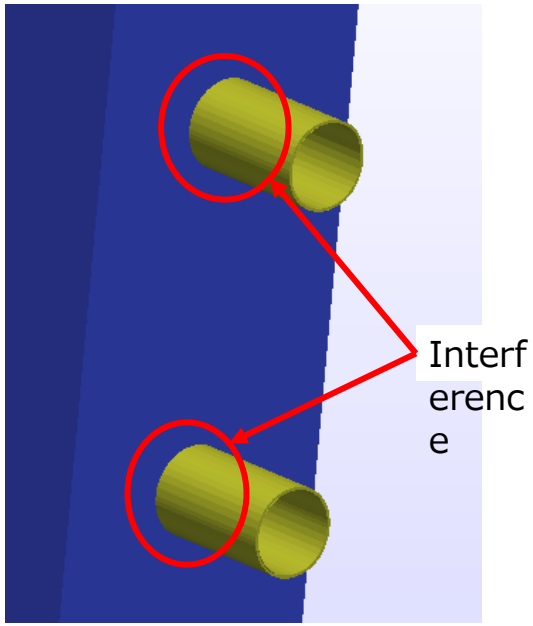
Editing Geometry / Checking Imported Geometry

- The imported shape is a rectangle that simulates the solid shape of the heat exchanger and the air area.
- Since the solid shape and the rectangular shape are interfering, the interfering part is processed.
- If you display the semi-transparency on the next page, it will be easier to check the interference part。



Volume1
Rectangle (air area)

Volume2
Solid shape

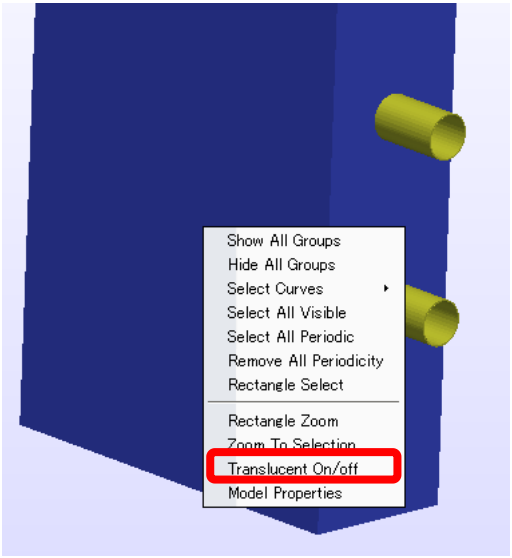


Fluid-solid thermal coupling analysis

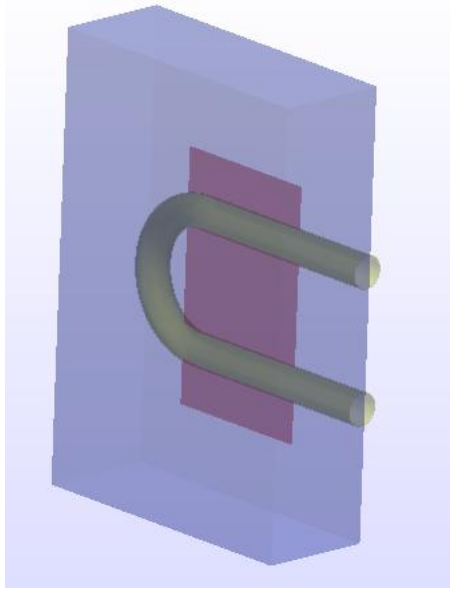
Geometry Editing/Translucency

- With this operation, you can switch between transparent and non-transparent display of the shape.This operation is also valid for mesh viewing.

(1): On the 3D viewer
Right-click mouse to
display menu



② : ”Translucent On/Off

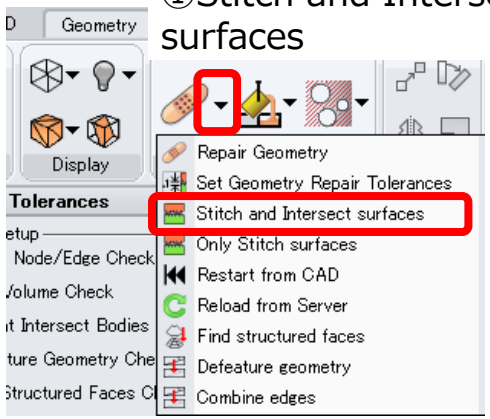


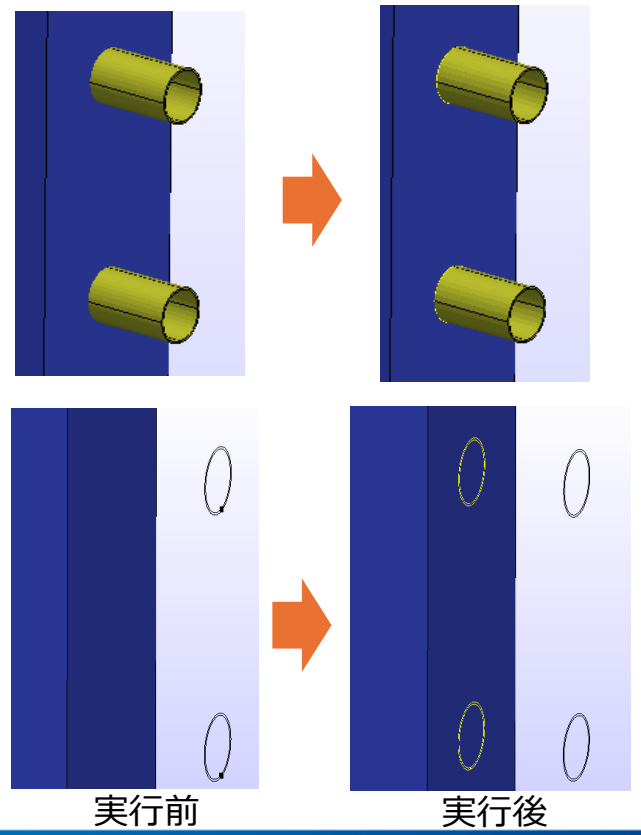
Change geometry to semi-transparent

Fluid-solid thermal coupling analysis

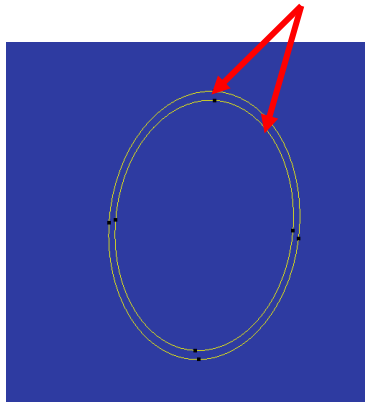
Edit Geometry/Run Stitch Surface

- Execute the geometry editing function "Stitch and Intersect surface".
- Divide the surface of the interference with the split function.





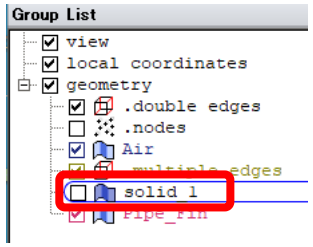
Surfaces are split to create multiple edges



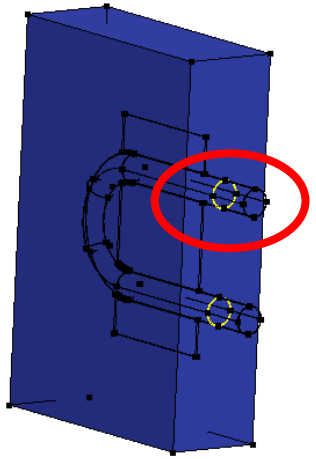
Fluid-solid thermal coupling analysis

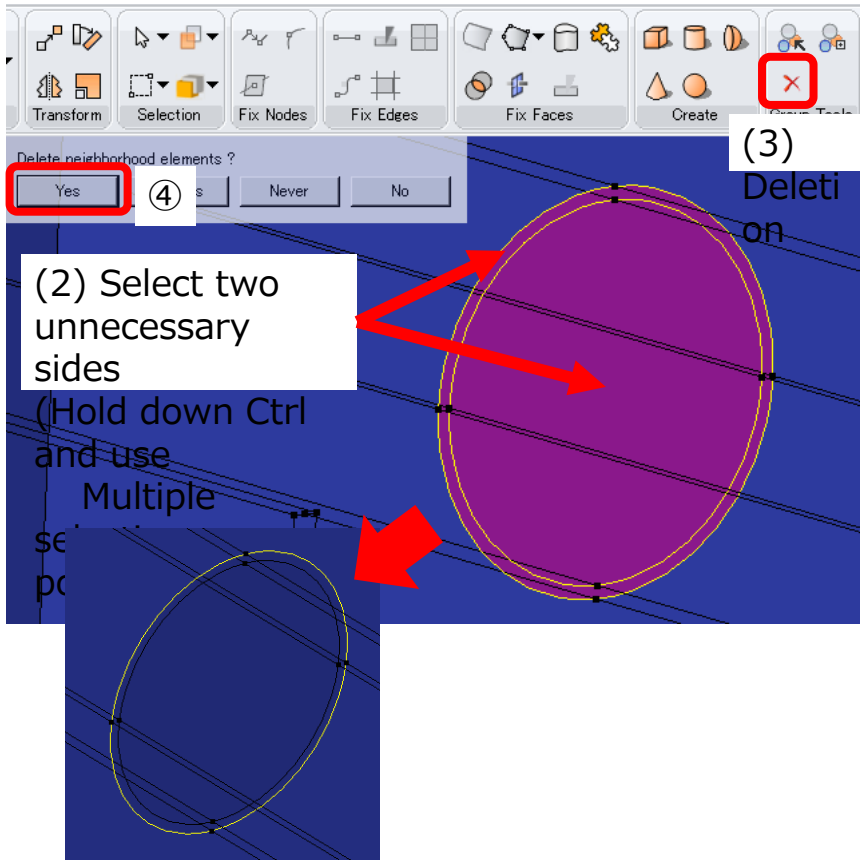
Edit Geometry/Remove Unwanted Faces

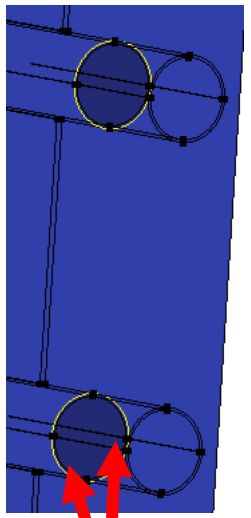
- Remove unwanted surfaces in the air area.



(1) Hide solid_1





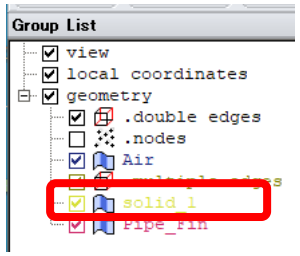


(5) Delete the other place in the same way.

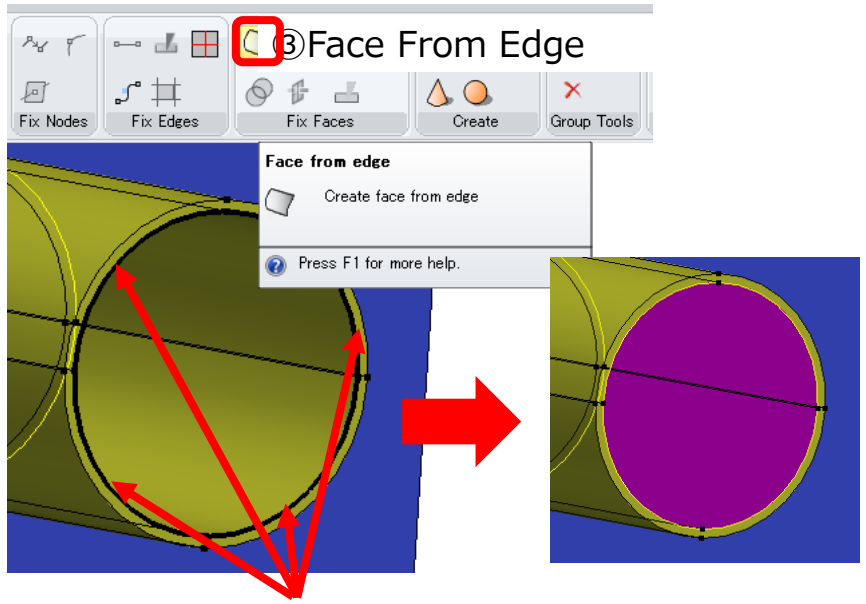
Fluid-solid thermal coupling analysis

Edit Geometry / Create Oil Inlet and Outlet Surfaces

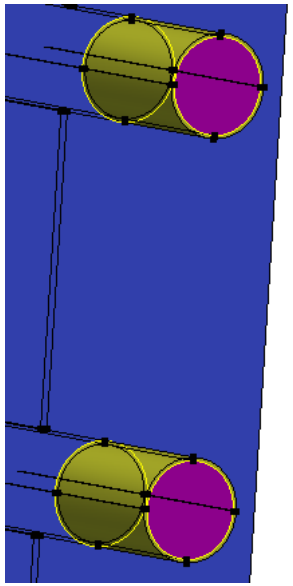
- Use the edges of the solid area to create the inlet and outlet surfaces of the oil。



(1) Display solid_1



(2) Select 4 edges of the oil inflow and outlet surfaces.

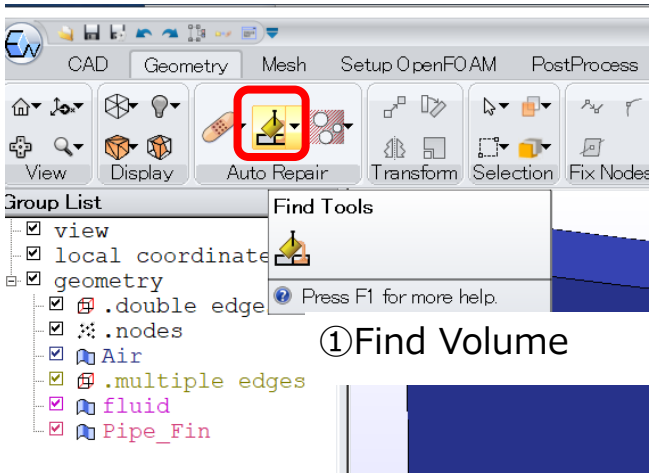


Generate a face, and create another face in the same way

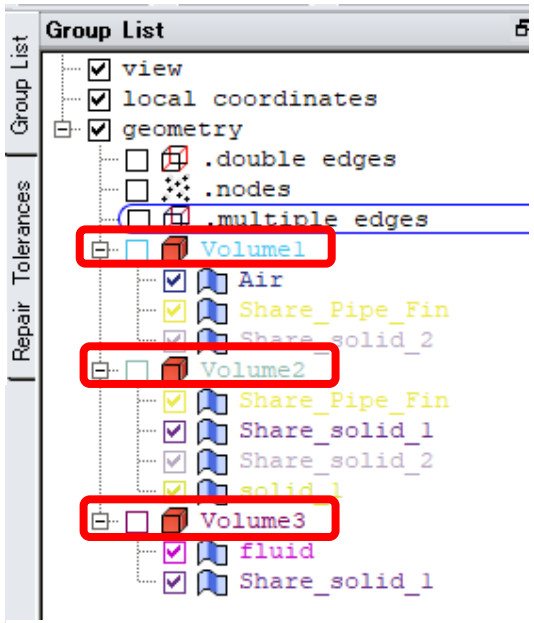
Fluid-solid thermal coupling analysis

Geometry Editing: Finding Regions (Performing Find Volume))

- Now that you are ready to run the Find Volume tool, execute it.
- Three volumes (analysis regions) of air, water, and solids are extracted.



①Find Volume

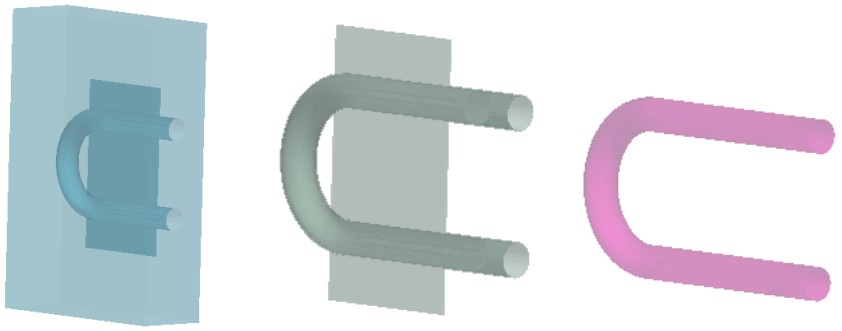


(2) Three volumes are extracted, and the domain interface is automatically duplicated as a part named Share~.

Fluid-solid thermal coupling analysis

Edit Geometry/Rename Region

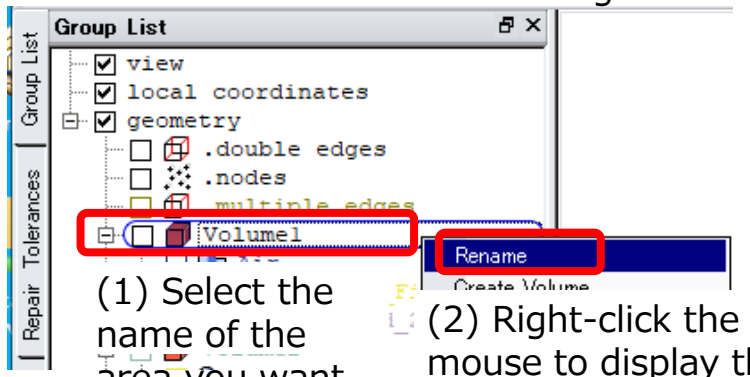
- Change the name of Volume 1~3 to a more understandable name.



Volume1　　Volume2　　Volume 3

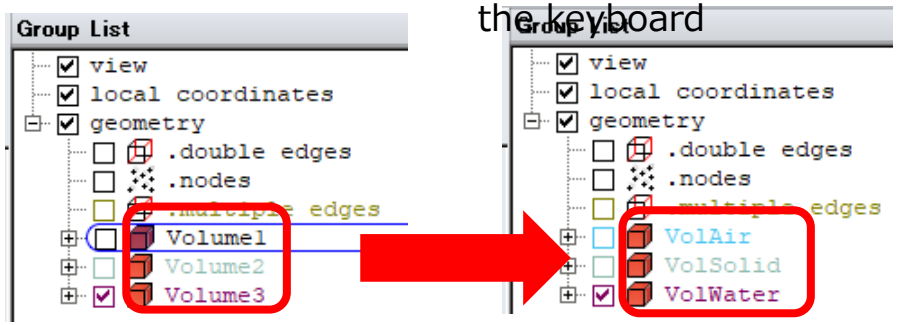
VolAir、VolSolid 、VolWater

How to rename a region



(1) Select the name of the area you want to rename.

(2) Right-click the mouse to display the maneuver and select "Rename".
Enter a new name from the keyboard

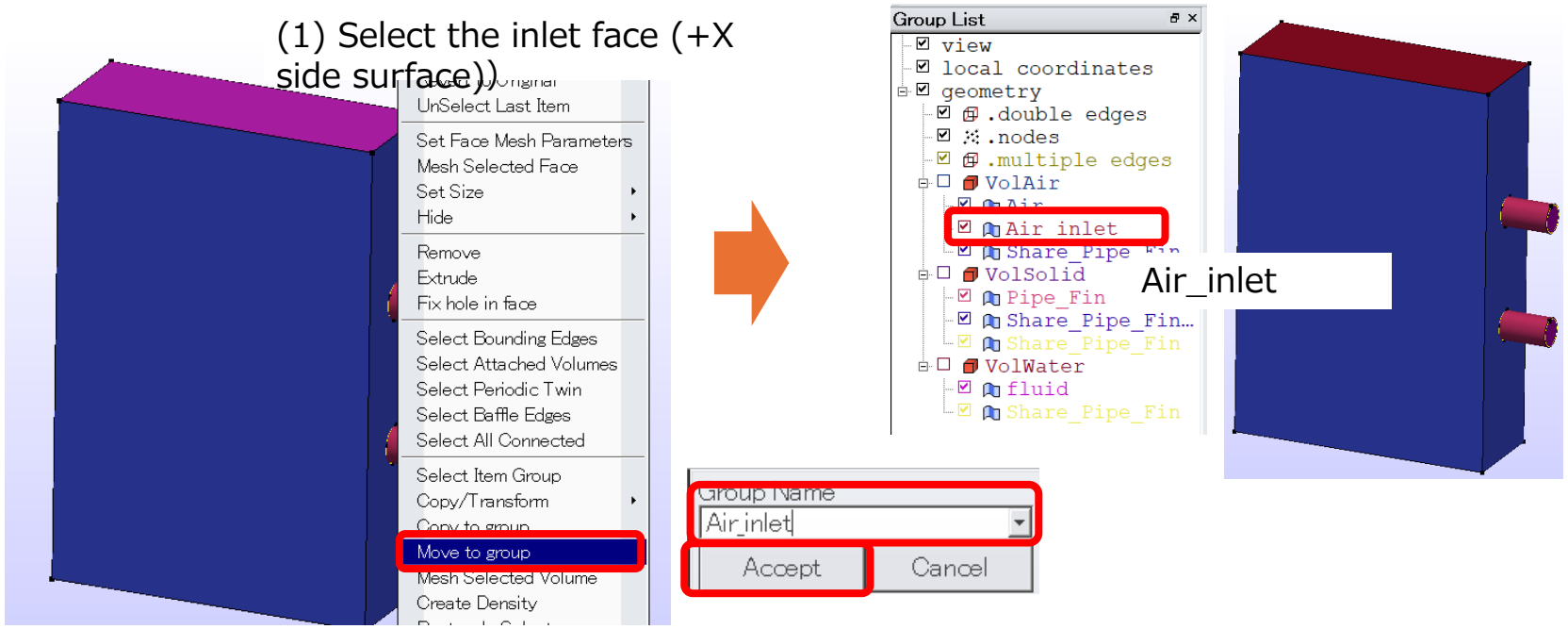


Fluid-solid thermal coupling analysis

Edit Geometry / Rename Boundaries (Air, Water Boundaries)

- Change the name of the air inlet surface.

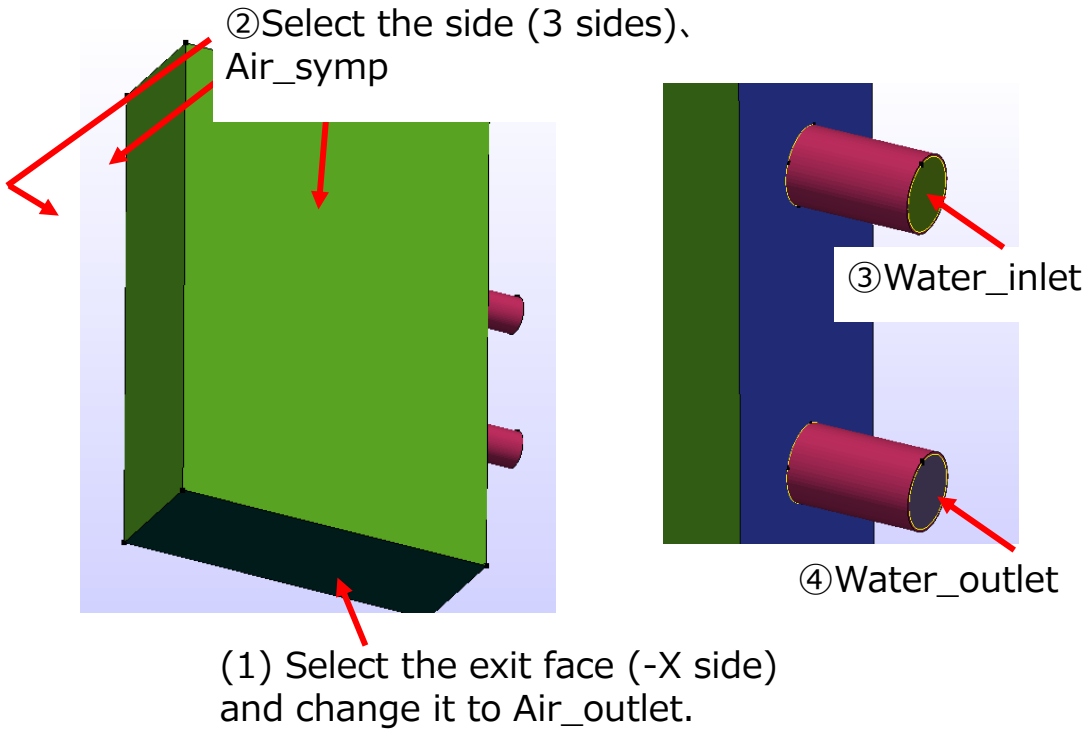
(1) Select the inlet face (+X side surface))



Fluid-solid thermal coupling analysis

Edit geometry / rename boundaries (air, water boundaries)

- Similarly, change the names of the air outlet surface (-X side face) and side side.
- Also change the water inlet surface (+X side) and water outlet surface (-X side).



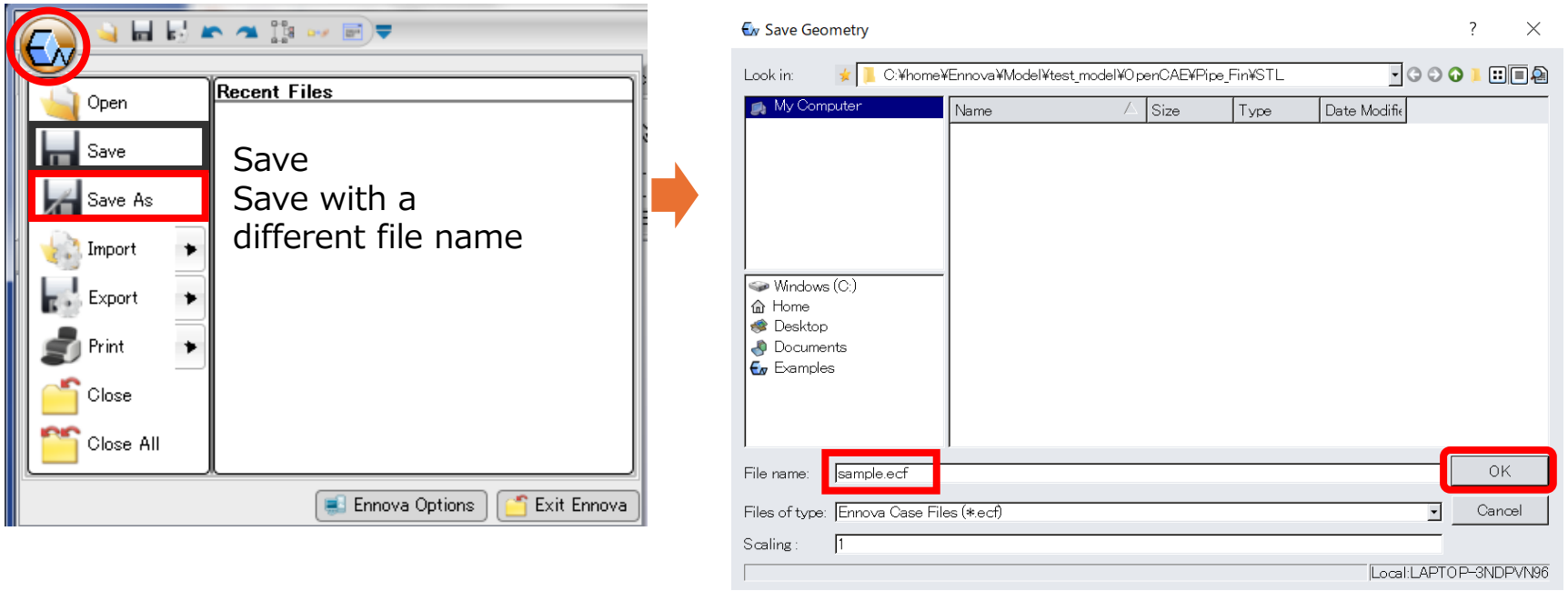
Name of air and water inlet and outlet surfaces

部位	パーツ名称
空気入口面	Air_inlet
空気出口面	Air_outlet
空気対称面	Air_symp
水入口面	Water_inlet
水出口面	Water_outlet

Fluid-solid thermal coupling analysis

Save

- Save your work to a file.
- ennovaCFD stores all the information of the pre-step in an .ecf file.
- From the Applications menu, select Save As and save the file as sample.ecf (the name is arbitrary).



Fluid-solid thermal coupling analysis

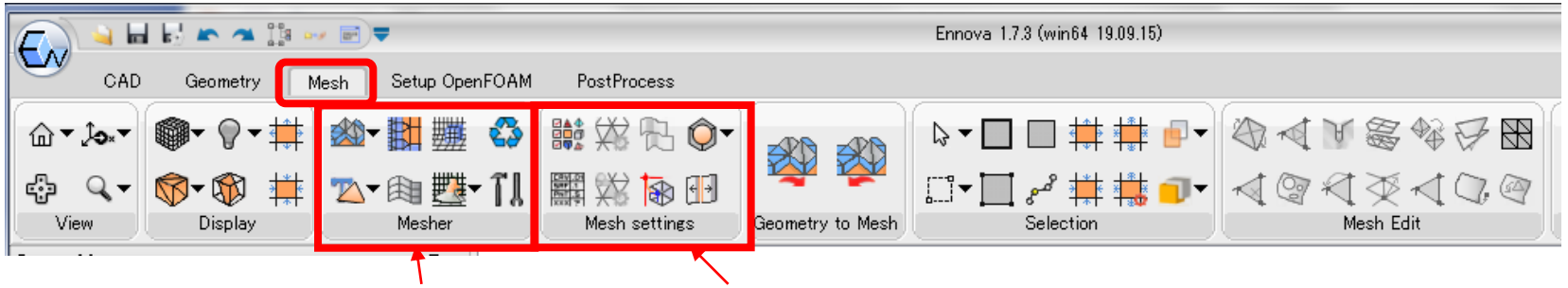
Overview of Mesh Creation and Settings

- ennovaCFD enables the following mesh control:
- Global sizing
- The mesh size that applies to the entire model.
- Set the maximum, minimum, curvature size, layer size, and number of layers.
- Local sizing
- This is the mesh size applied to each part.
- Set the maximum, minimum, size of the curvature section, and layer On/Off.
- Spatial mesh sizing
- Set the DensityBox (the range is controlled by a rectangle or a cylinder).
- The mesh size used in this model is as follows.
- Global Size
- Maximum size=0.01 [m] , Minimum size=0.0025 [m]
- Number of layers = 2 layers, Initial height = 0.0003, Ratio = 1.0

Fluid-solid thermal coupling analysis

Meshing/Mesh Tab Descriptions

- Mesh settings and mesher execution are done in the Mesh tab.



Mesher Execution Menu

Mesh sizing menu

topologyBasedMesh Run Button
This is the mesher used in this work.

Global Sizing Menu

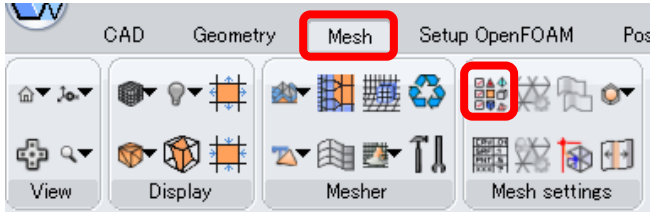
Local sizing menu

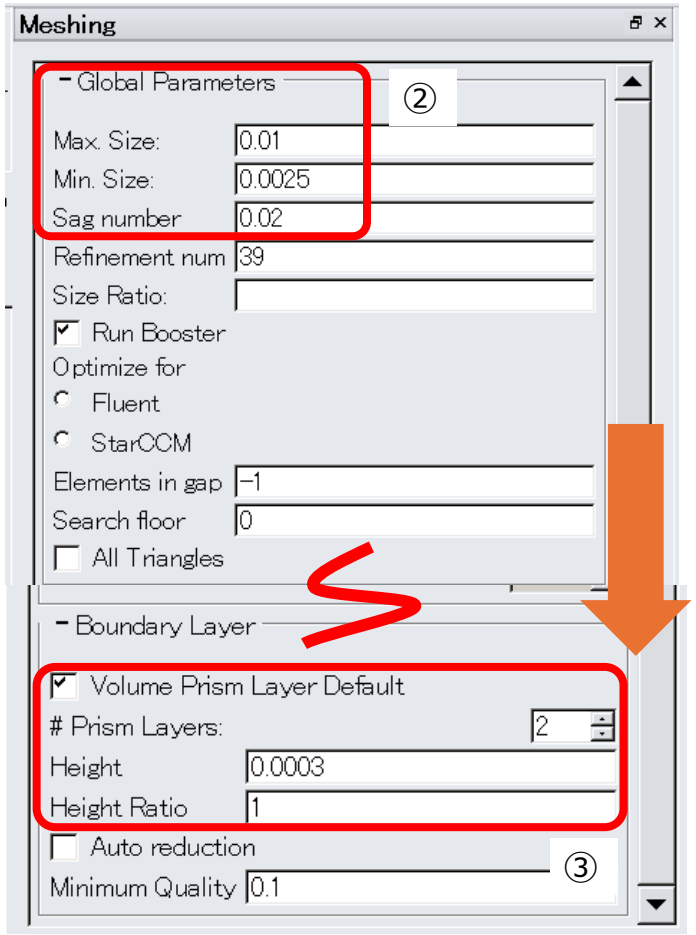
Spatial mesh sizing menu

Fluid-solid thermal coupling analysis

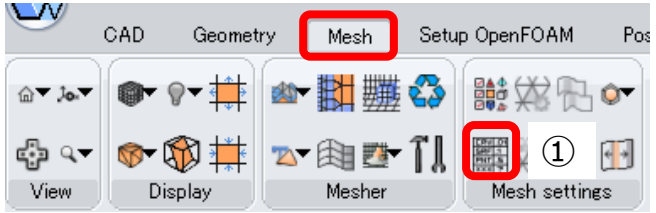
Meshing/Global Sizing Panel

- Sets the global mesh size.
- Change the red text settings.
- Do not change any other settings.
  - Max Size. : 0.01
  - Min Size. : 0.0025
  - Sag number : 0.02
  - Sag number is the distance between the mesh surface and the curved surface, Refinment num is the number of divisions on the circumference, and the mesh setting applied to the curved surface.
  - Sag number and Refinment num are interlocked parameters.
  - Volume Prism Layer Default : On
  - #Prism Layers : 2
  - Height : 0.0003
  - Height Ratio : 1.0(Thickness ratio after the second layer)



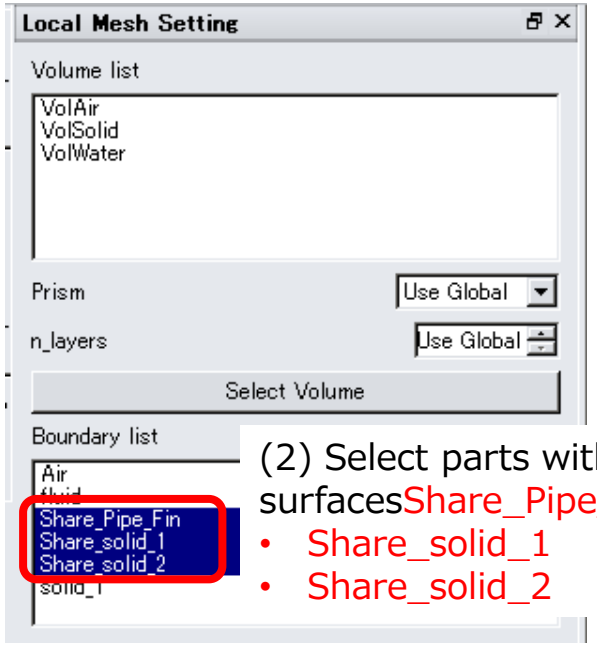


Fluid-solid thermal coupling analysis



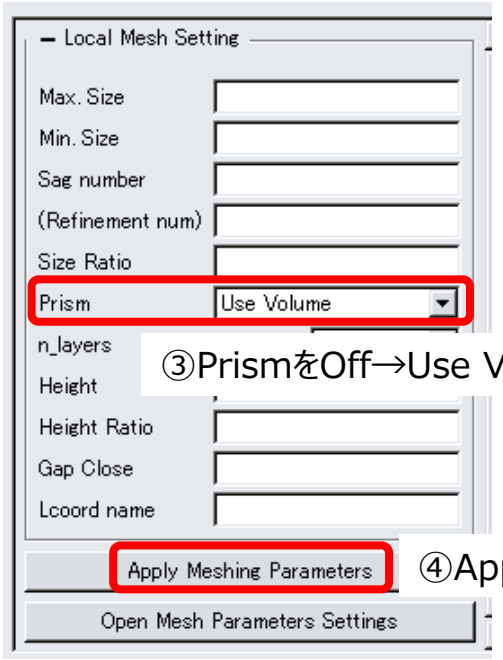
Mesh Creation/Local Sizing Panel

- In the Local Sizing panel, turn on the Solid Surface Layer setting (the default is Layer OFF).
- We won't use it this time, but you can set the maximum, minimum, etc. for each part.



(2) Select parts with solid surfacesShare_Pipe_Fin

- Share_solid_1
- Share_solid_2



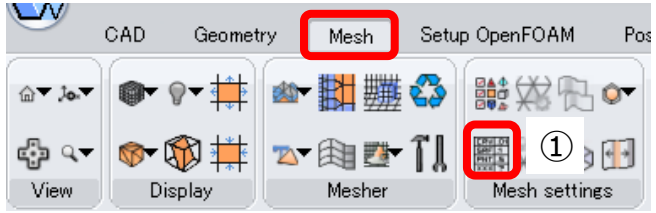
③PrismをOff→Use Volume

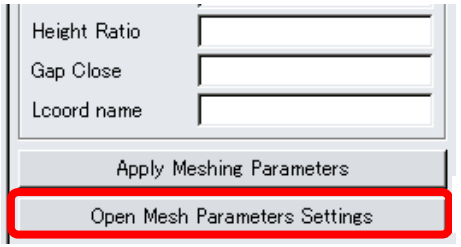
④Apply

Fluid-solid thermal coupling analysis

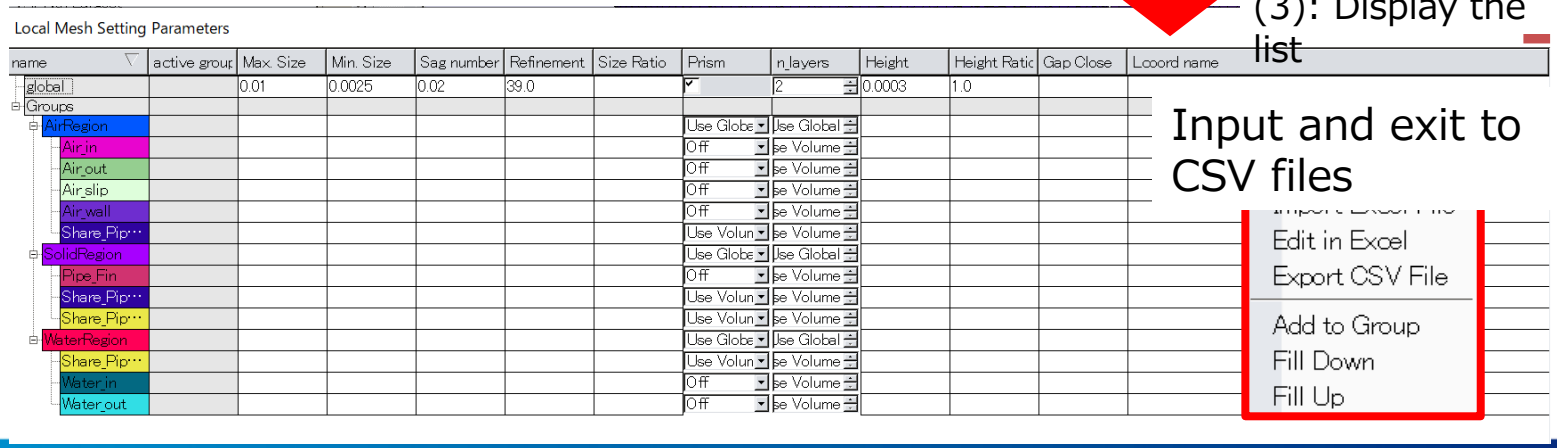
Mesh Creation/Local Sizing Panel (List List)

- You can see a list of local sizing settings in a spreadsheet.
- You can list the mesh sizes with the Open Mesh Parameters Settings button.
- If you right-click on the panel, you can output (input) the settings to a CSV file。





(3): Display the list

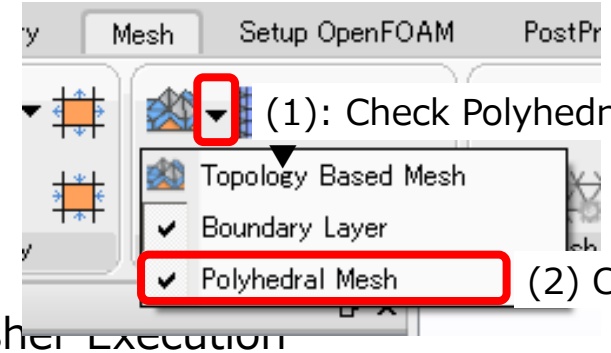


Input and exit to CSV files

Fluid-solid thermal coupling analysis

Meshing/Meshing

- Polyhedral Mesh Settings
- Check the flags of the Polyhedral mesh.

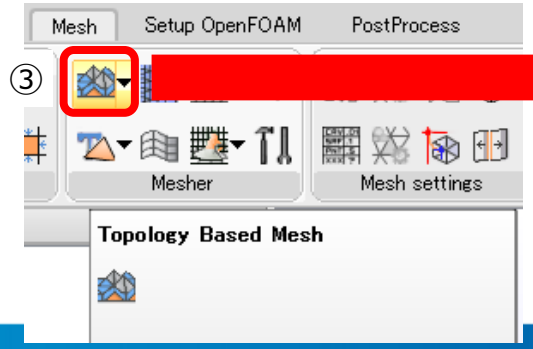


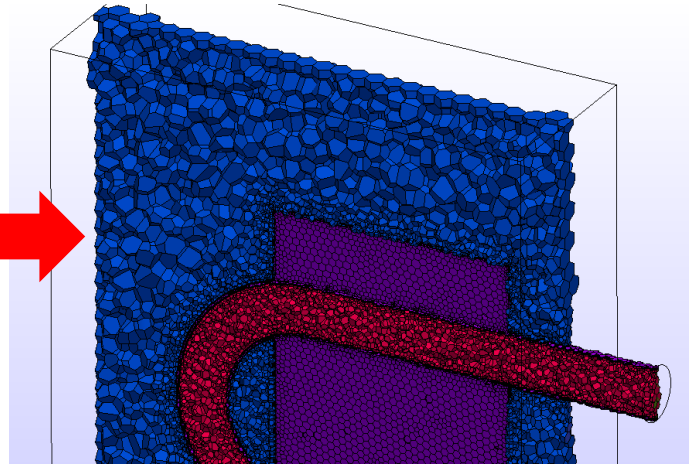
(1): Check Polyhedral Mesh from

(2) Check :P olyhedral Mesh

Button to interrupt meshing
 lisplayed at the bottom
ıeft of the window)

- Mesher Execution
- Run topologyBasedMesh to create a polyhedral mesh.

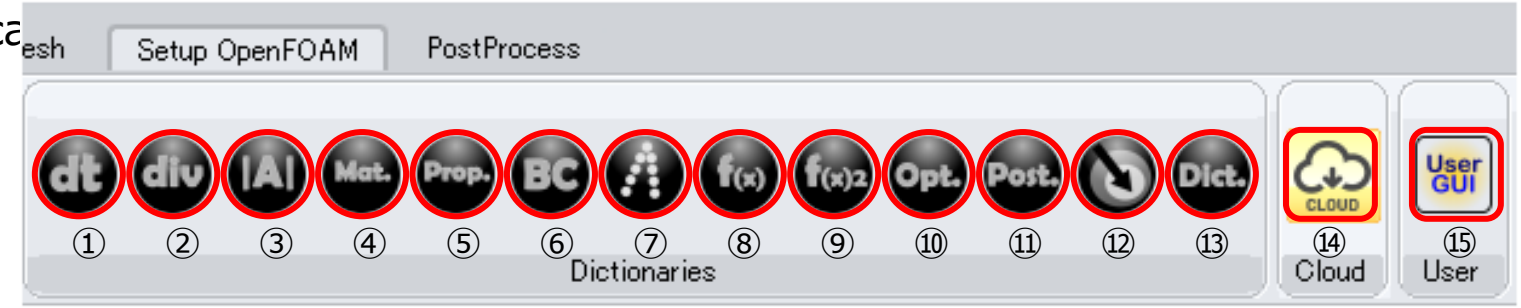




Fluid-solid thermal coupling analysis

Setup OpenFOAM

- OpenFOAM calculation settings are configured in the Setup OpenFOAM tab. The menus are roughly categorized for each OpenFOAM dictionary file. The tasks are ca



(1) controlDict Panel
- Solver, version used
- Region division method for parallel computing
- Time increments and frequency of output results
・Turbulence model and other settings

(2) fvSchemes Panel
- Discrete schemes for each term Configuring

(3) fvSolution Panel
・Linear solver
・Relaxation factor and other settings

(4) Materials Panel
・Physical properties of materials Configuring

(5) Constant Panel
・Initial conditions
・Physical properties
・CellZone and other settings

(6) caseProperties Panel
・Boundary conditions Configuring

(7) Lagrangian Panel
- Particle settings

(8) Functions Panel
・function Configuring

(9) functions2 panel
・Function2 Configuring

(10) fvOptions Panel
・fvOptions Configuring

(11) postProcessingパネル
・caseDicts の設定

(12) Optimisationパネル
Optimisationの設定

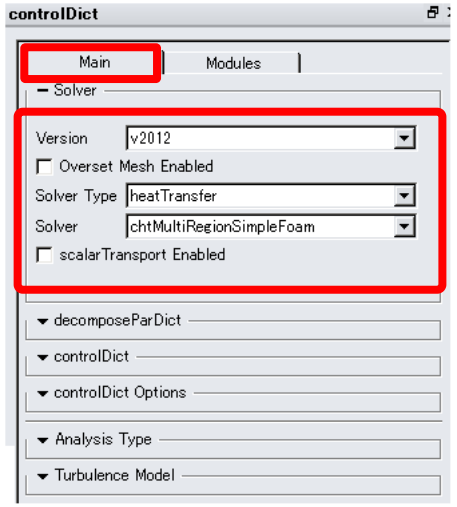
(13) writeDictパネル
OpenFOAMファイル一式の出力

(14) Cloud Panel
Calculation execution by cloud server

(15) UserGUI Panel
Add GUI by user

Fluid-solid thermal coupling analysis

Analysis Settings/controlDict Panel > Main Tab



- Specify the solver version to be used and the basic settings for calculations.
- Change the red text settings.
- Some menus are not displayed by the selected version.
- Solver
  - Version*1 : Optional selection
  - Overset Mesh Enabled : Checkoff
  - Solver Type*2 : heatTransfer
  - Solver*3 : chtMultiRegionSimpleFoam
  - scalarTransport Enabled : Checkoff

*1: Select the version of OpenFOAM to be used for calculations. Some features may not be available in all versions.

*2: Select the calculation method.

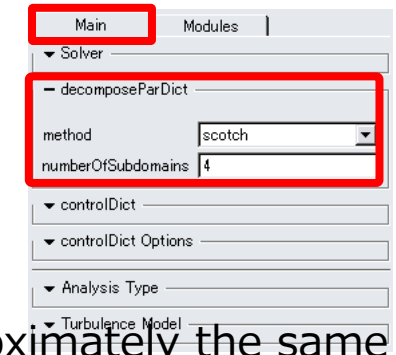
*3: Select the solver name to be used for the calculation.

As for the calculation method and solver name, it is necessary to know what kind of calculation the analyst will perform and select the appropriate item.

Continued on the next page

Fluid-solid thermal coupling analysis

Analysis Settings/controlDict Panel > Main Tab

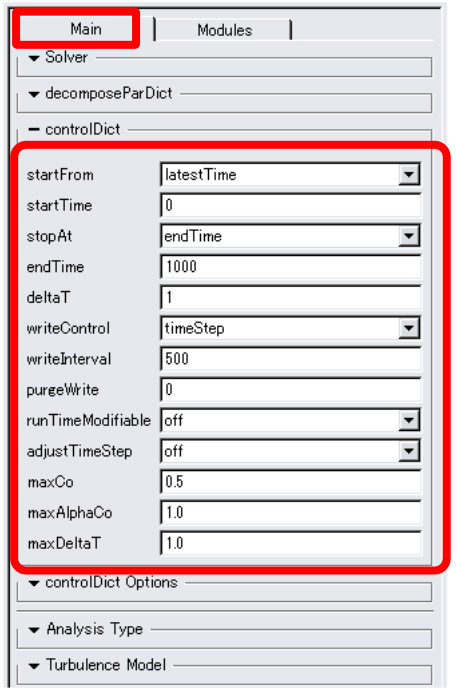


- Set up parallel processing.
- decomposeParDict
- Method: Scotch
- How to divide the number of meshes so that the number of meshes is approximately the same for each CPU
- HPC Num :4
- Specifies the number of CPUs available for calculation.

Continued to the next page〈

Fluid-solid thermal coupling analysis

Analysis Settings/controlDict Panel > Main Tab



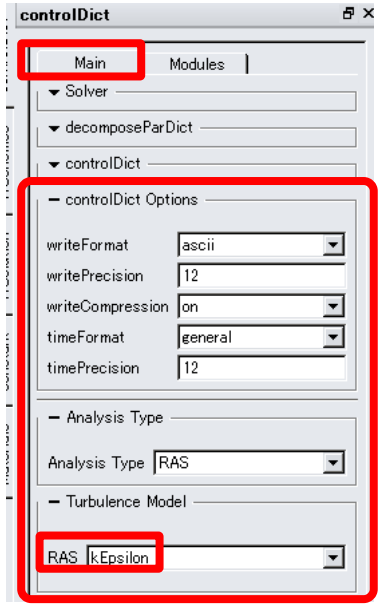
- Set the iteration, the frequency of output results, and so on.
- This panel uses the default settings and does not change them.
- controlDict
  - Start From : latestTime
  - Start Step : 0
  - stopAt : endTime
  - endTime : 1000
  - deltaT : 1
  - Write Control : timeStep
  - Write Interval : 500
  - Purge Write : 0

If the specified number of times is exceeded, old results will be deleted, and all "0" will be saved.runTimeModifiable : off

  - adjustTimeStep : off
  - maxCo : 0.5
  - maxAlphaCo : 1.0
  - maxDeltaT : 1.0

Continued to the next page〈

Fluid-solid thermal coupling analysis



Analysis Settings/controlDict Panel > Main Tab

- Set the calculation specifications, select the turbulence
- model, and so on.
- contorlDict Options
  - writeFormat : ascii
  - writePrecision : 12
  - writeCompression : on
  - timeFormat : general
  - timePrecision : 12
- Analysis Type
  - Analysis Type : RAS

Choice of laminar or turbulent flow calculationTurbulence Model

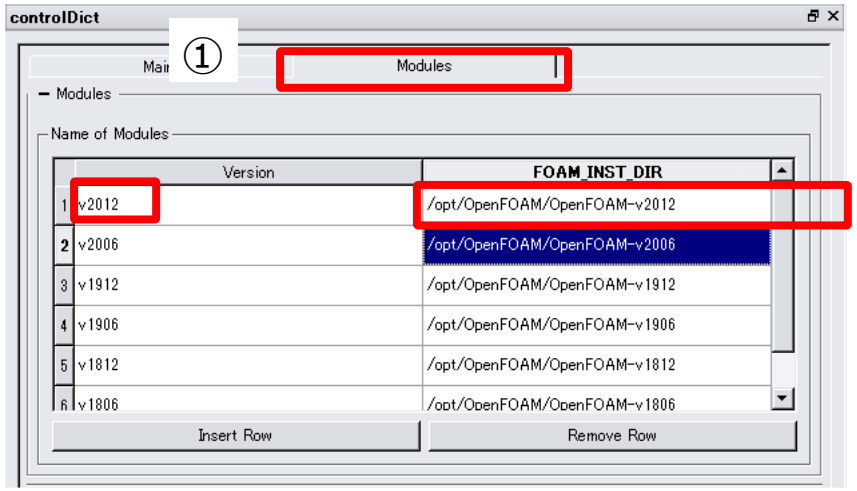
- RAS : kEpsilon

Choosing a Turbulence Model

Fluid-solid thermal coupling analysis

Analysis Settings/controlDict Panel> Modules Tab

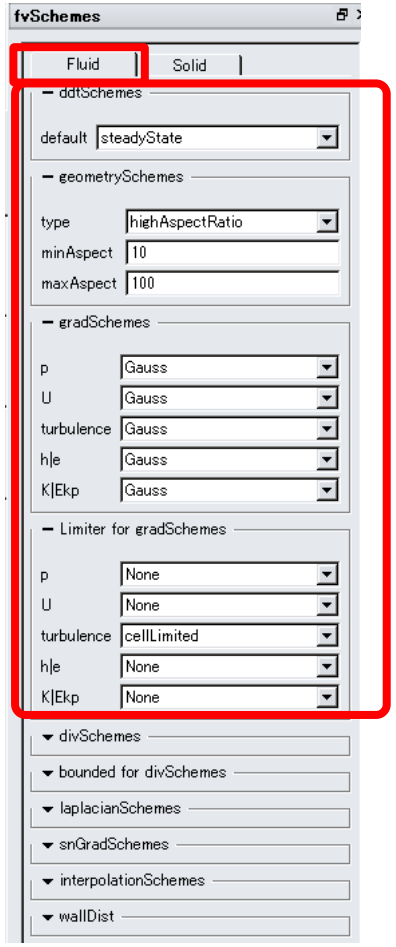
- If the installation environment of ennovaCFD and OpenFOAM is the same, you can perform OpenFOAM calculations from the GUI of ennovaCFD by setting the OpenFOAM installation folder in the FOAM_INST_DIR.
- Please see page 59 for how to perform the calculations.
- If the installation environment of ennovaCFD and OpenFOAM is different, no FOAM_INST_DIR configuration is required.
- Please see page 60 for how to perform the calculations.



Fluid-solid thermal coupling analysis

Analysis Settings/fvSchemes Panel> Fluid Tab

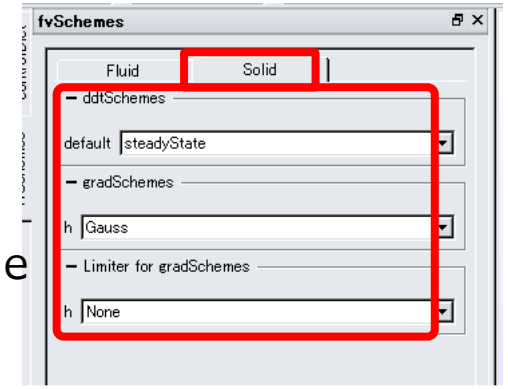
- Sets the difference scheme to be used for fluid calculations.
- This panel uses the default settings and does not change them.
- ddtSchemes(Time Term Scheme)
- geometrySchemes
- gradSchemes(Gradient Term Scheme)
- Limiter for gradSchemes(Gradient Term Scheme)
- divSchemes
- bounded for divSchemes
- laplacianSchemes
- snGradSchemes
- interpolationaSchemes
- wallDist



Continued to the next page〈

Fluid-solid thermal coupling analysis

Analysis Settings/ fvSchemes Panel

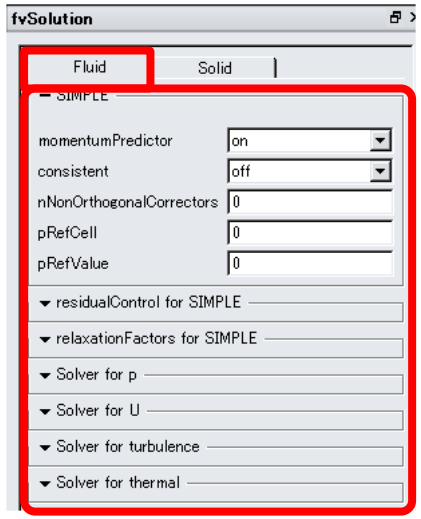


- Sets the difference scheme to be used for solid-state calculations.
- This panel uses the default settings and does not change them.
- ddtSchemes(Time Term Scheme)
  - default : steadyState
- gradSchemes(Gradient Term Scheme)
  - h : Gauss
- Limiter for gradSchemes (Gradient Term Scheme)
  - h : None

Fluid-solid thermal coupling analysis

Analysis Settings/fvSolution Panel > Fluid Tab

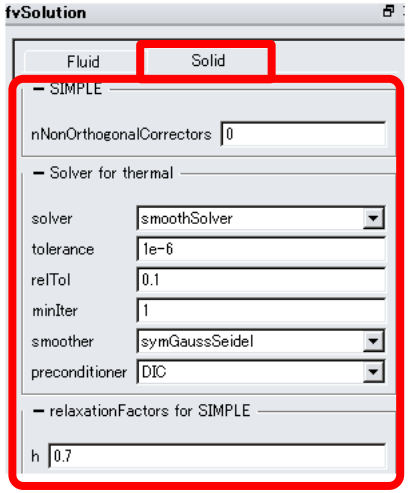
- Set the SIMPLE algorithm on the fluid side and the linear solver for each variable.
- This panel uses the default settings and does not change them.
- SIMPLE
- residualControl for SIMPLE (Convergence judgment value)
- relaxationFactors for SIMPLE (Relaxation factor)
- Solver for p
- Solver for U
- Solver for turbulence
- Solver for thermal



次頁へ続く

Fluid-solid thermal coupling analysis

Analysis Settings/fvSolution Panel > Solid Tab



- Set the SIMPLE algorithm on the solid side and the linear solver for each variable.
- This analysis is calculated with the default settings, so it will not be changed.
- SIMPLE
  - nNonOrthogonalCorrectors : 1 - 2
- Solver for thermal
  - solver : smoothSolver
  - tolerance : 1e-6
  - relTol : 0.1
  - minIter : 1
  - smoother : symGaussSeidel
  - preconditioner : DIC

Continued to the next page

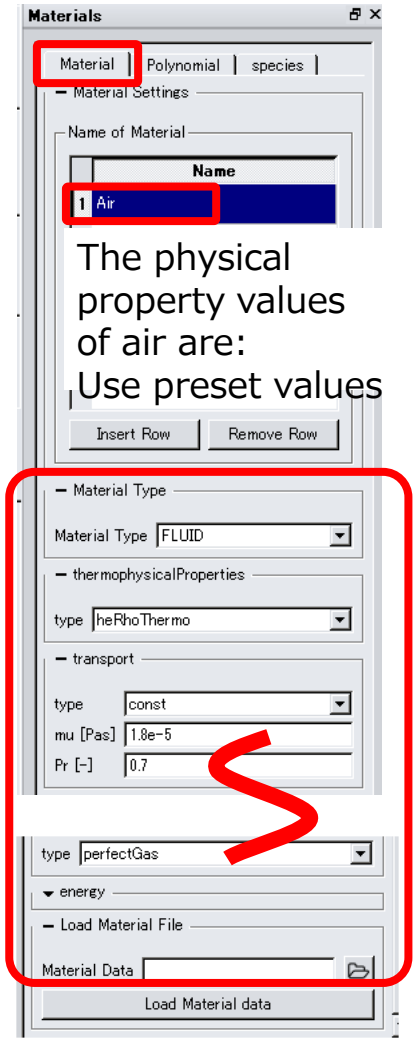
- residualFactors for SIMPLE(Relaxation factor)
  - h : 0.7

Fluid-solid thermal coupling analysis

Analysis Settings/Constant Panel > Material Tab

- Set the physical properties to be used in the calculation.
- Check the panel and set the following physical properties.
- Fluid: Air > use default settings
- Fluid: Create a new >
- Solid: Newly created > Al (aluminum)
- The physical property values used are as follows.

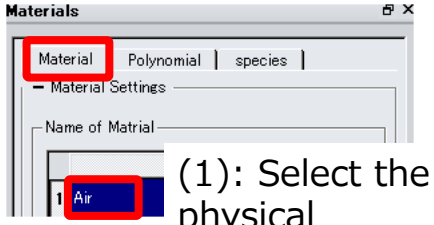
	空気 Air	水 Water	固体（アルミ） Al
thermal conductivity[W/mK]	-	-	237
Viscosity coefficient[Pa･s]	1.8e-5	1.0e-3	-
Pr[-]	0.7	6.98	-
specific heat[J/kgK]	1000	4186	903
molecular weight[kg/kmol]	28.9	18.02	-
density[kg/m3]	perfctGas	1000	2702



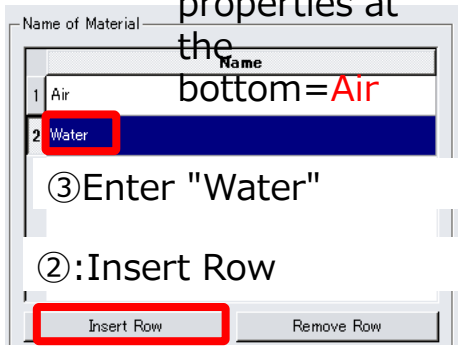
Fluid-solid thermal coupling analysis

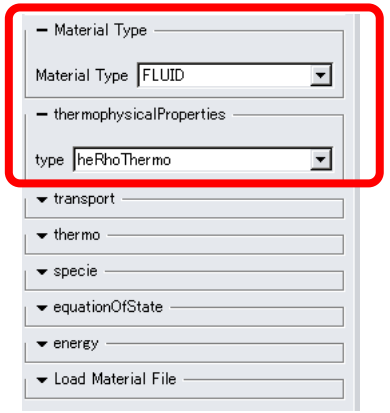
Analysis Settings/Constant Panel > Material Tab

- Add the physical properties of water.
- Add physical properties by the operation shown in the figure on the left.
- Material Type
  - Material Type : FLUID
- thermophysicalProperties
  - type : heRhoThermo



(1): Select the physical properties at the bottom=Air





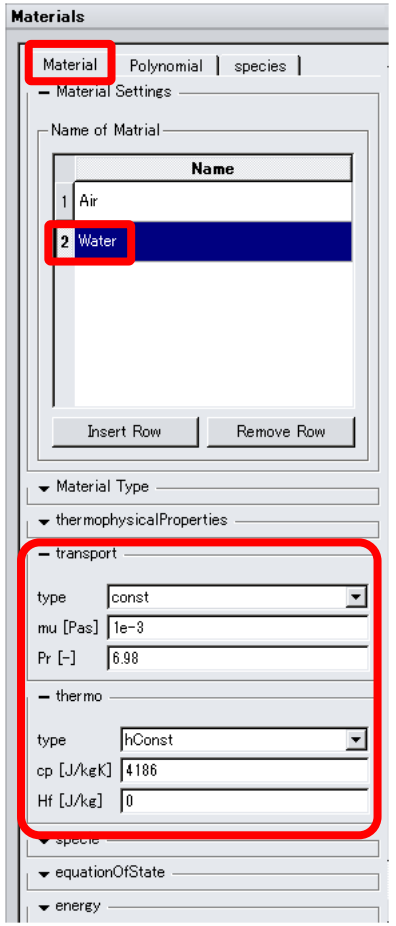
Continued to the next page〈

Fluid-solid thermal coupling analysis

Analysis Settings/Constant Panel > Material Tab

- Add the physical properties of water.
- Transport
  - Type : const
  - Molecular Viscosity[Pa･s] : 1e-3
  - Pr[-] : 6.98
- thermo
  - equationOfState(Density Type) : hConst
  - Specific Heat(Cp) [J/(kg･K)] (Specific Heat) : 4186
  - : 0

	水 Water
粘性係数[Pa･s]	1.0e-3
Pr[-]	6.98
比熱[J/kgK]	4186
分子量[kg/kmol]	18.02
密度[kg/m3]	1000

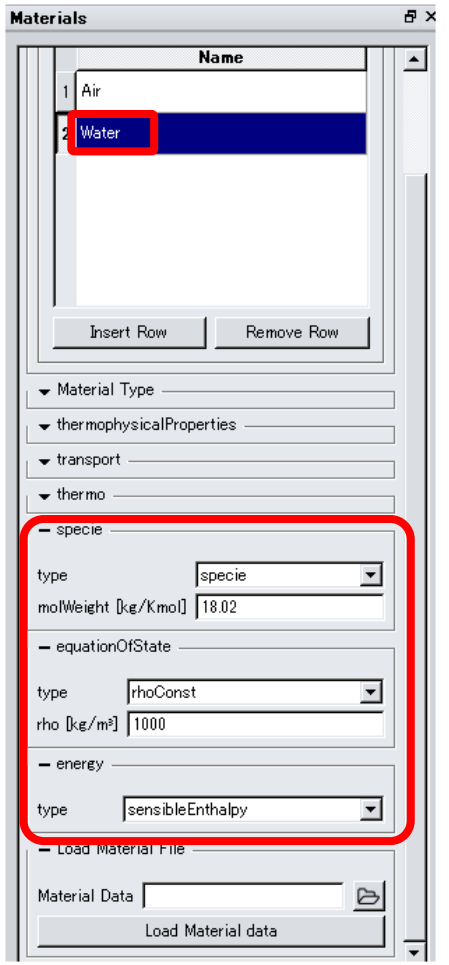


次頁へ続く

Fluid-solid thermal coupling analysis

Analysis Settings/Constant Panel > Material Tab

- Add the physical properties of water.
- spcice
  - type : speice
  - Molecular Weight[kg/kmol](モル質量) : 18.02
- equationOfState
  - type : rhoConst
  - Density[kg/m^3] (密度) : 1000
- energy
  - type :

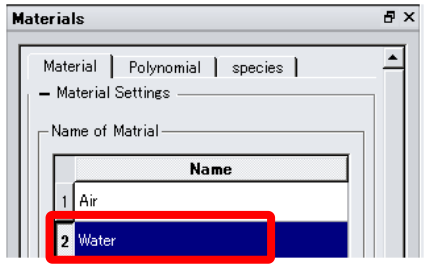


	水 Water
Viscosity coefficient[Pa･s]	1.0e-3
Pr[-]	6.98
specific heat[J/kgK]	4186
molecular weight[kg/kmol]	18.02
density[kg/m3]	1000

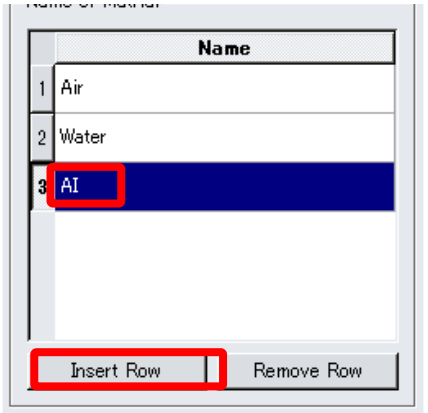
Fluid-solid thermal coupling analysis

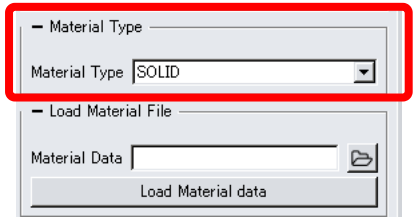
Analysis Settings/Constant Panel > Material Tab

- Add the physical properties of aluminum.
- Add physical properties by the operation shown i figure on the left.
- Material Type
  - Material Type : SOLID
- thermophysicalProperties
  - type :
    heSolidThermo



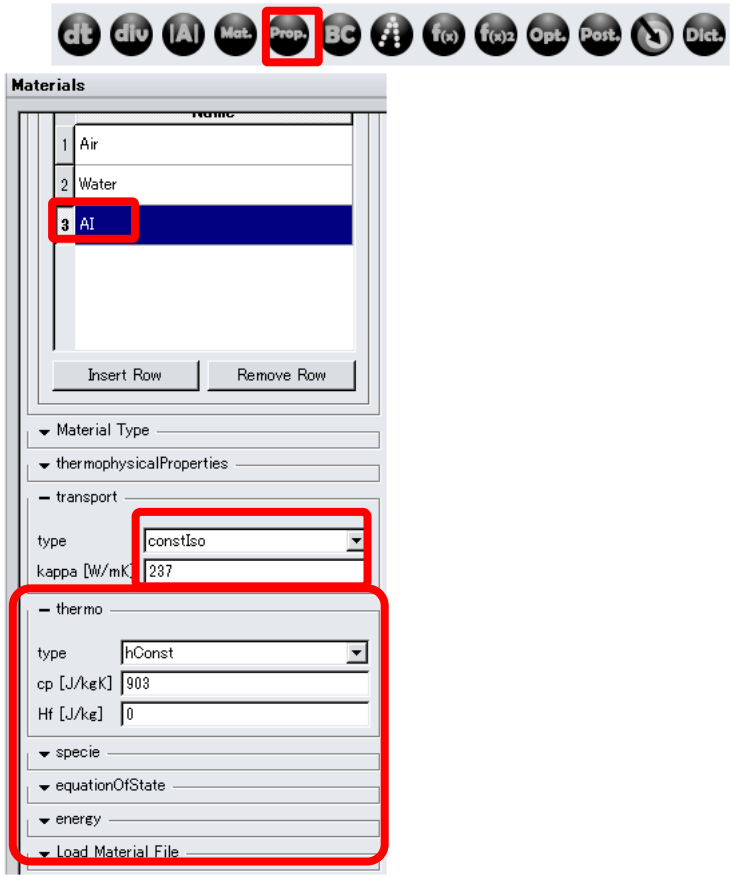
(1): Select the physical properties at the bottom





Continued to the next page〈

Fluid-solid thermal coupling analysis



Analysis Settings/Constant Panel > Material Tab

- Add the physical properties of aluminum.
- Transport
  - Type : constIso
  - kappa [W/(m･k)] : 237
- thermo
  - equationOfState(Density Type) : hConst
  - Specific Heat(Cp) [J/(kg･K)]) : 903
  - Ref.Enthalpy(Hf) [J/kg] : 0

	Solid (aluminum)Al
thermal conductivity[W/mK]	237
specific heat[J/kgK]	903
density[kg/m3]	2702

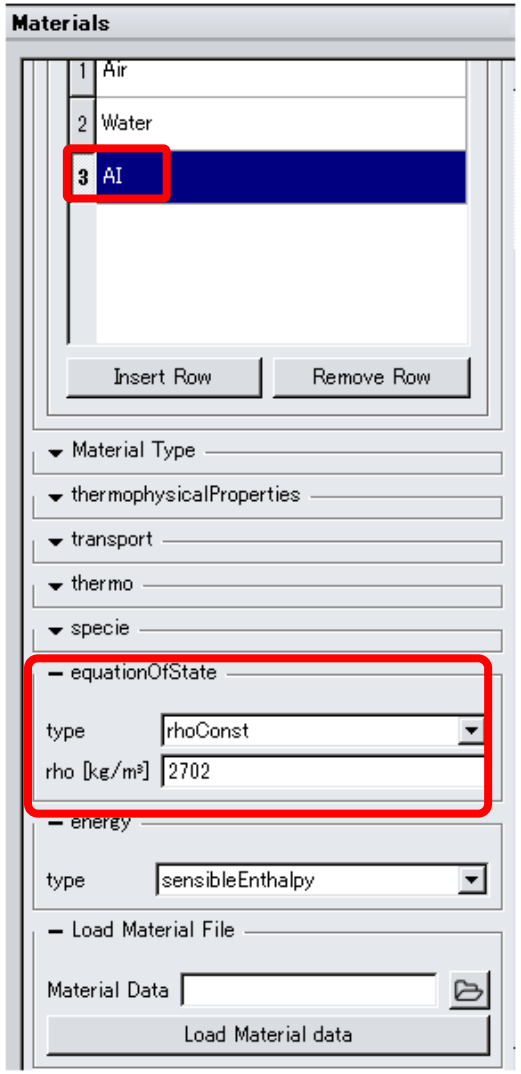
Continued on the next page

Fluid-solid thermal coupling analysis

Analysis Settings/Constant Panel > Material Tab

- Add the physical properties of aluminum.
- equationOfState
  - type : rhoConst
  - Density[kg/m^3] : 2702

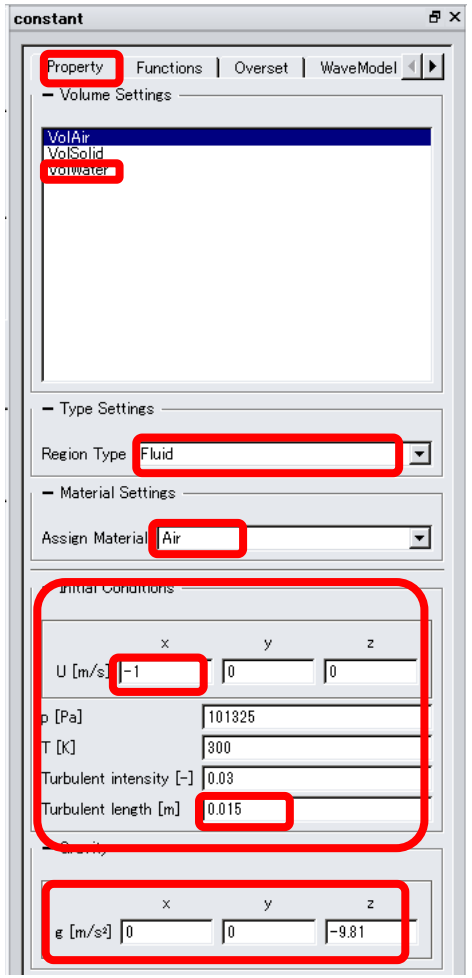
	Solid (aluminum)) Al
thermal conductivity[W/mK]	237
specific heat[J/kgK]	903
density[kg/m3]	2702



Fluid-solid thermal coupling analysis

Analysis Settings/constantProperty

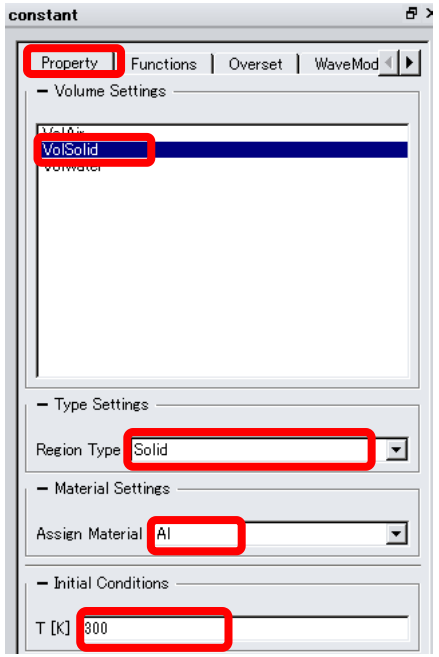
- VolAir(Fluid Region) and give the physical properties of the air.
- Volume Settings
  - domain name : VolAir
- Type Settings
  - Region Type : Fluid
- Material Settings
  - Assign Material :
    Air
- Initial Conditions
  - U[m/s] : X=-1,Y=0,Z=0
  - p[Pa] : 101325
  - T[K] : 300
  - Turbulent Intensity : 0.03
  - Turbulent Length[m] : 0.015
- Gravity g[m/s^2] : X=0,Y=0,Z=-9.81



Fluid-solid thermal coupling analysis

Analysis Settings/constantProperty

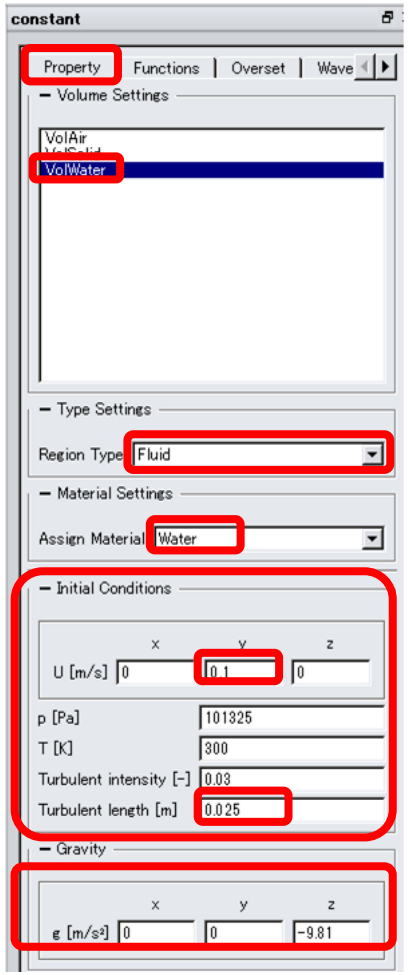
- VolSolid(solid area) and give it the physical properties of aluminum.
- Volume Settings
  - : VolSolid
- Type Settings
  - Region Type : Solid
- Material Settings
  - Assign Material : Al
- Initial Conditions
  - T[K] : 300



Fluid-solid thermal coupling analysis

Analysis Settings/constantProperty

- VolWater(Fluid Region) and give it the physical properties of water.
- Volume Settings
  - domain name : VolWater
- Type Settings
  - Region Type : Fluid
- Material Settings
  - Assign Material : Water
- Initial Conditions
  - U[m/s] : X=0,Y=0.1,Z=0
  - p[Pa] : 101325
  - T[K] : 300
  - Turbulent Intensity : 0.03
  - Turbulent Length[m] : 0.025
- Gravity
  - g[m/s^2] : X=0,Y=0,Z=-9.81



Fluid-solid thermal coupling analysis

Analysis Settings/caseProperties Panel

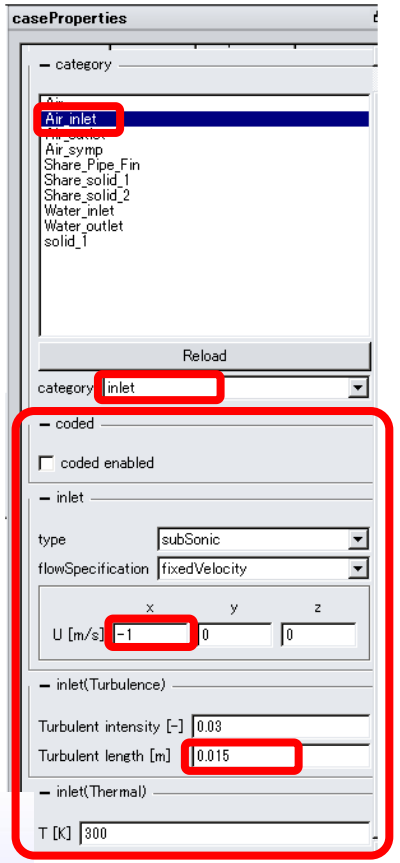
- The boundary conditions to be set are summarized below.
- The solver for compressibility calculations uses pressure in units of [Pa].

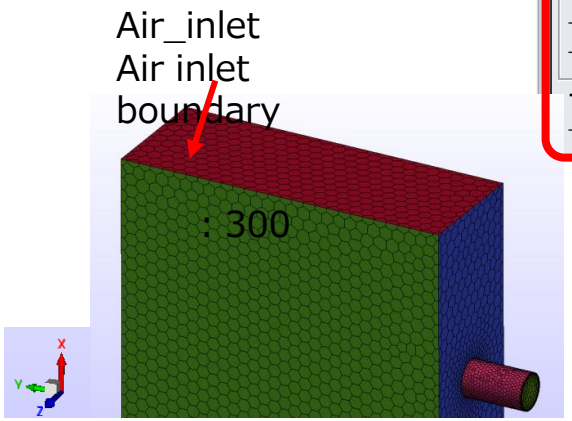
BC	Group	Boundary conditions
Air inlet	Air_inlet	Conditions: Ux=−1.0,Uy=0,Uz=0[m/s] T=300 [K]
Air outlet	Air_outlet	Pressure Regulation Conditionsp=101325 [Pa]
Plane of air symmetry	Air_symp	Symmetry condition
Water inlet	Water_inlet	Conditions: Ux=0,Uy=0.1, Uz=0 [m/s] T=353 [K]
Water outlets	Water_outlet	Pressure stipulates conditions p=101325 [Pa] The conditions for co-service are hot
Oil-solid interface Air-solid interface	Share_Pipe_Fin, Share_solid_1, Share_solid_2	Pressure stipulates conditions p=101325 [Pa] The conditions for co-service are hot

Fluid-solid thermal coupling analysis

Analysis Settings/caseProperties panel> Main tab

- Air_inlet sets the entry speed.
- category
  - Boundary Name : Air_inlet
  - category : inlet
- coded
  - coded enable : unchecked
- inlet
  - type : subsonic
  - flowSpecification : fixedVelocity
  - U[m/s] : X=-1 Y=0
    Z=0
- inlet(Turbulence)
  - Turbulent intensity[-] : 0.03
  - Turbulent length[m] : 0.015
- inlet(Thermal)
  - T[k] : 300

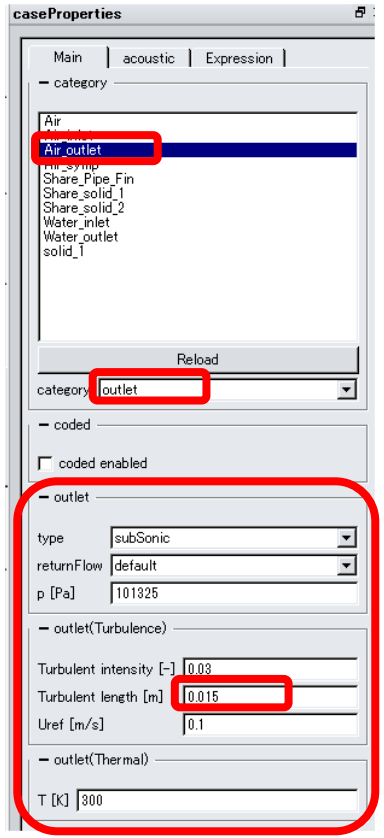


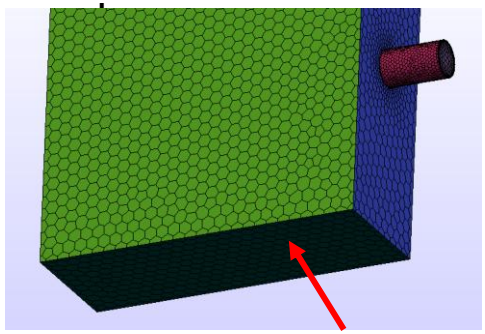


Fluid-solid thermal coupling analysis

Analysis Settings/caseProperties panel> Main tab

- Air_outletsets the outlet pressure.
- category
  - Boundary Name
    : Air_outlet
  - category : outlet
- coded
  - coded enable : チェックオフ
- outlet
  - type :
    subsonic
  - returnFlow : default
  - p[Pa] :
    101325
- outlet(Turbulence)
  - Turbulent intensity[-] : 0.03
  - Turbulent length[m] : 0.015
  - Uref[m/s]
- outlet(Thermal)
  - T[k] : 300



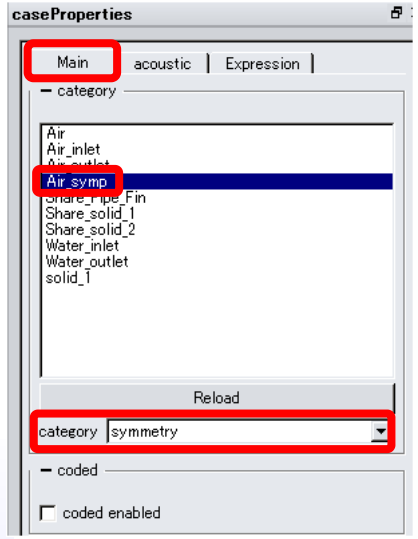


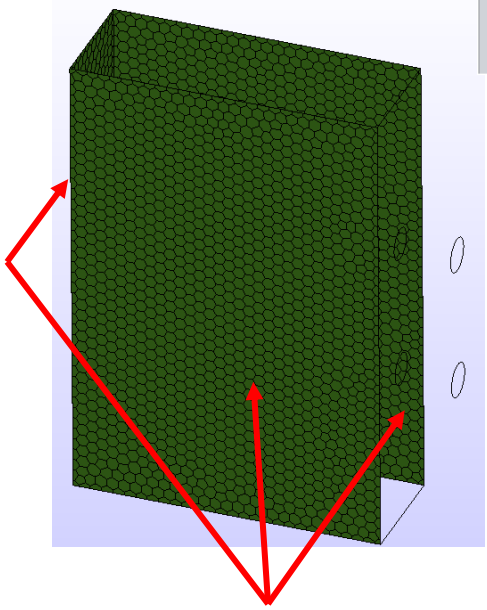
Air outlet surface

Fluid-solid thermal coupling analysis

Analysis Settings/caseProperties panel> Main tab

- Air_symp sets symmetry.
- category
  - Boundary Name : Air_symp
  - category : Symmetry
- coded
  - coded enable : Checkoff

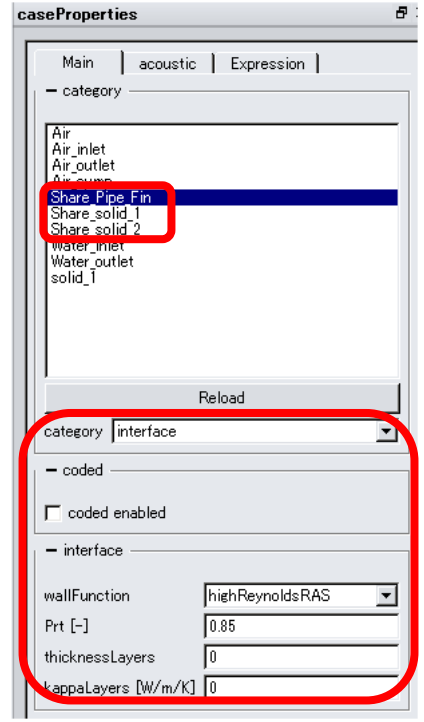




air side

Fluid-solid thermal coupling analysis

Analysis Settings/caseProperties panel> Main tab

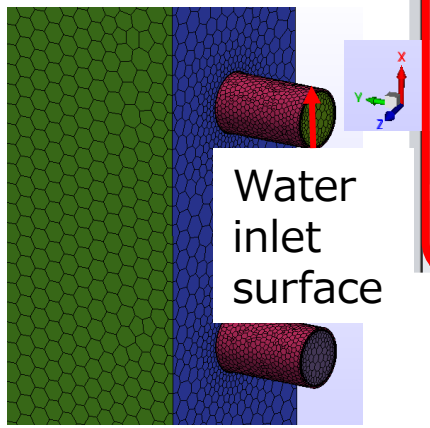


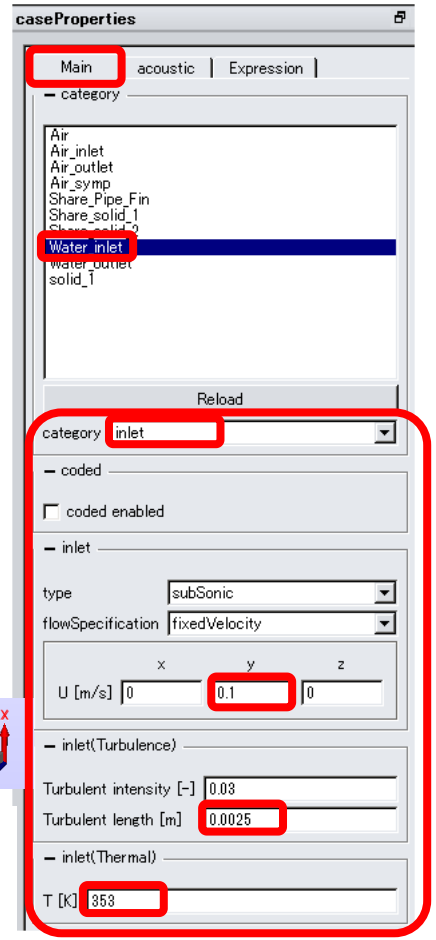
- Share_Pipe_Fin, Share_solid_1, and Share_solid_2 set up conjugate heat transfer at the inter-domain interface (interface).
- category
  - Boundary Name : Share_Pipe_Fin,Share_solid_1,Share_solid_2
  - category : Interface
- coded
  - coded enable : checkoff
- interface
  - wallFunction : highReynoldsRAS
  - Prt[-] : 0.85
  - thicknessLayers : 0
  - kappaLayers[W/m/k] : 0

Fluid-solid thermal coupling analysis

Analysis Settings/caseProperties panel> Main tab

- Water_inlet sets the entry speed.
- category
  - Boundary Name : Water_inlet
  - category : inlet
- coded
  - coded enable : CheckOff
- inlet
  - type : subsonic
  - flowSpecification : fixedVelocity
  - U[m/s] : X=0 Y=0.1
    Z=0
- inlet(Turbulence)
  - Turbulent intensity[-] : 0.03
  - Turbulent length[m] : 0.0025
- inlet(Thermal)
  - T[k] : 353



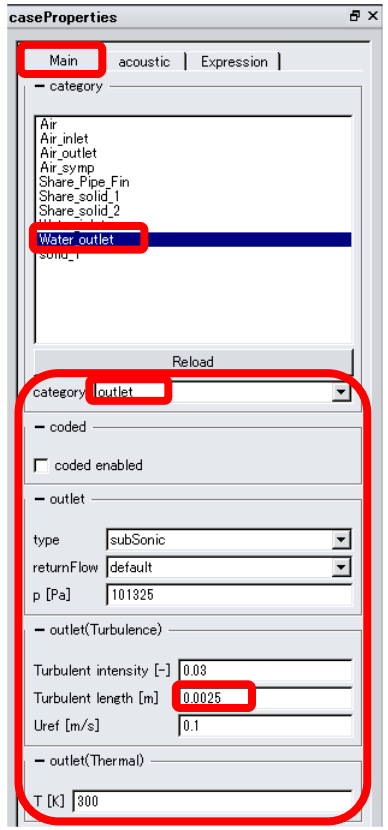


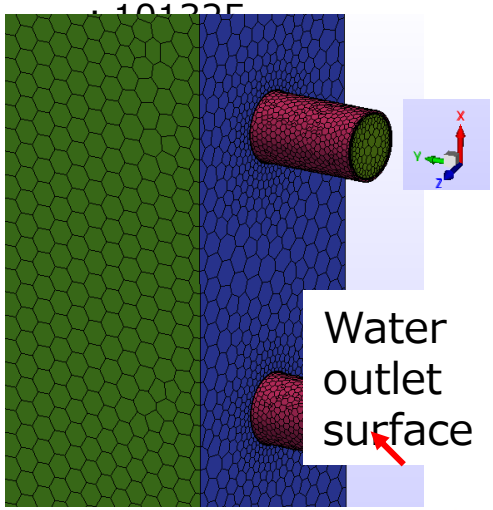
Fluid-solid thermal coupling analysis

Analysis Settings/caseProperties panel> Main tab

- The Water_outlet sets the outlet pressure.
- category
  - Boundary Name : Water_outlet
  - category : outlet
- coded
  - coded enable : unchecked
- outlet
  - type : subsonic
  - returnFlow : default
  - p[Pa]
- outlet(Turbulence)
  - Turbulent intensity[-] : 0.03
  - Turbulent length[m] : 0.0025

  Uref[m/s]
- outlet(Thermal)
  - T[k]

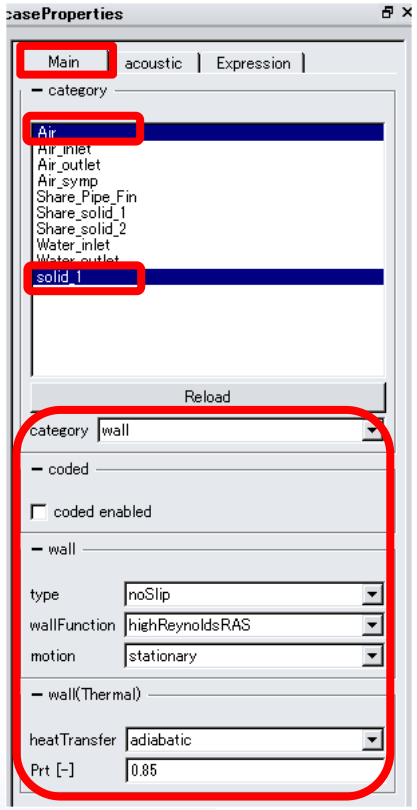


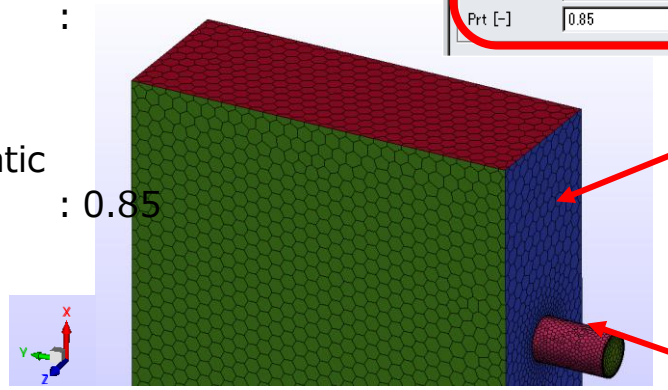


Fluid-solid thermal coupling analysis

Analysis Settings/caseProperties panel> Main tab

- Air, solid_1 sets up a wall with friction.
- It uses the default settings, so no input is required.
- category
  - Boundaries : Air,solid_1
  - category : wall
- coded
  - coded enable :checkoff
- wall
  - type : noSlip
  - wallFunction : highReynoldsRAS
  - motion : stationary
- wall(Thermal)
  - heatTransfer : adiabatic
  - Prt[-] : 0.85



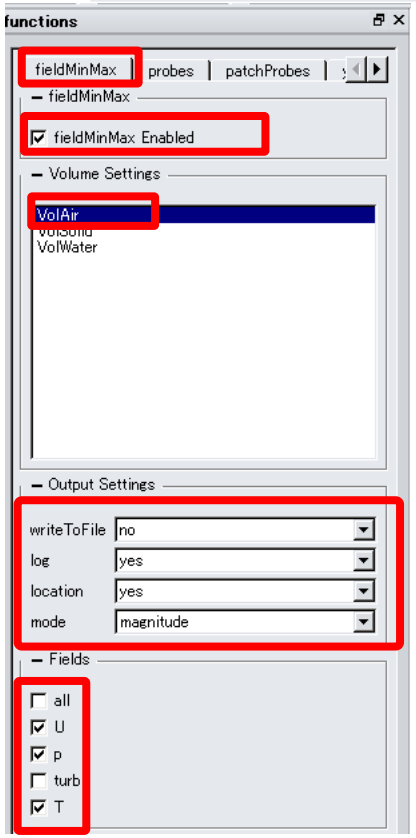


Air
Air
Boundaries

solid_1
Pipe
Boundaries

Fluid-solid thermal coupling analysis

Analysis Settings/functions panel>fieldMinMax tab



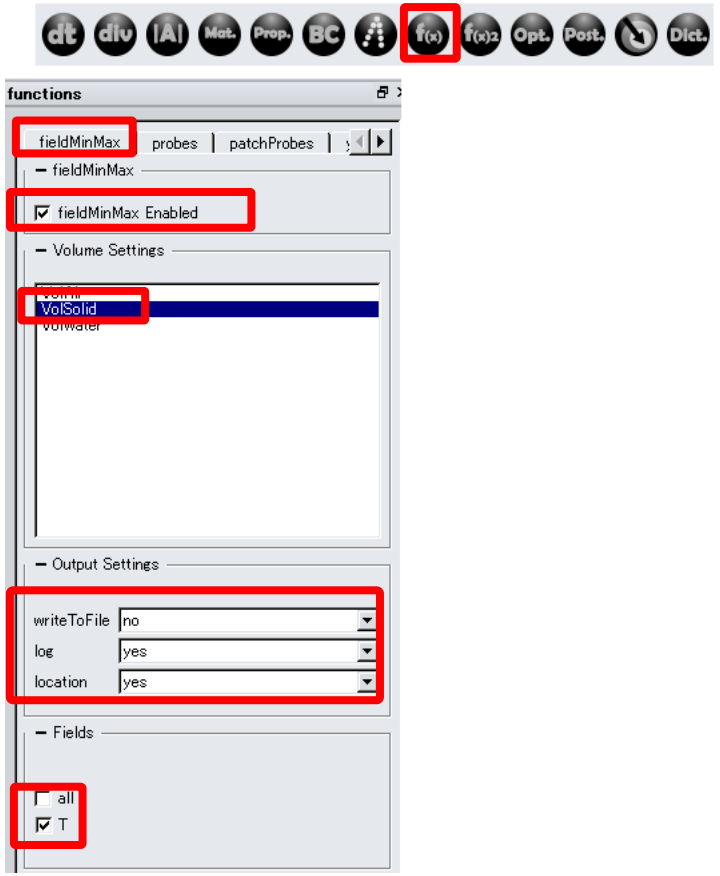
- Sets the record of the maximum/minimum value of the physical quantity being calculated.
- filedMinMax Enabled : on
- Volume Settings
  - domain name : VolAir
- Output Settings
  - writeToFile : no
  - log : yes
  - location : yes
  - mode : magnitude
- Fields
  - all : off
  - U : on
  - p : on
  - turb : off
  - T : on

。

Fluid-solid thermal coupling analysis

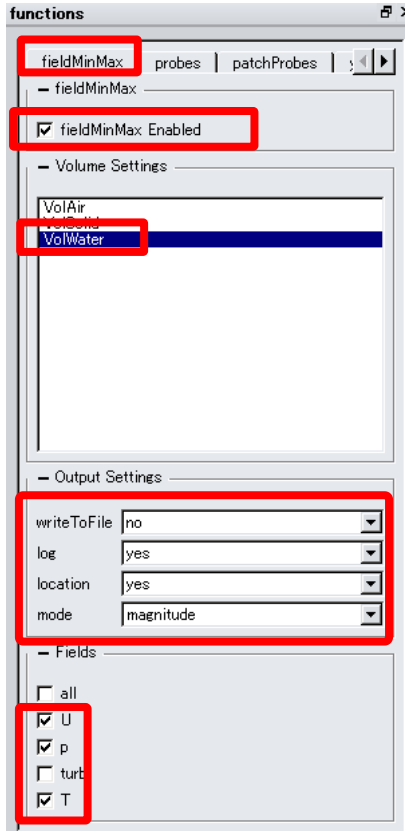
Analysis Settings/functions panel>fieldMinMax tab

- Sets the record of the maximum/minimum value of the physical quantity being calculated.
- filedMinMax Enabled : on
- Volume Settings
  - domain name : VolSolid
- Output Settings
  - writeToFile : no
  - log : yes
  - location : yes
  - mode : magnitude
- Fields
  - all : off
  - T : on



Fluid-solid thermal coupling analysis

Analysis Settings/functions panel>fieldMinMax tab



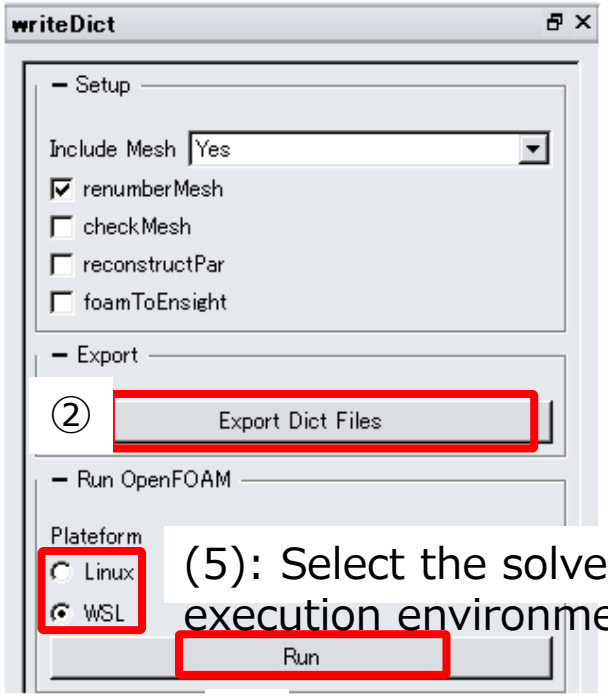
- Sets the record of the maximum / minimum value of the physical quantity being calculated。

- filedMinMax Enabled : on
- Volume Settings
  - domain name : VolWater
- Output Settings
  - writeToFile : no
  - log : yes
  - location : yes
  - mode : magnitude
- Fields
  - all : off
  - U : on
  - p : on
  - turb : off
  - T : on

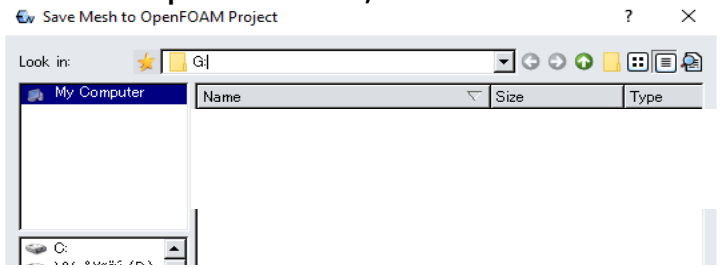
Fluid-solid thermal coupling analysis

Perform Analysis/Execute Direct Calculations > writeDict Panel

- If you have an environment that calculates directly from OpenFOAM, run the calculations with Run.

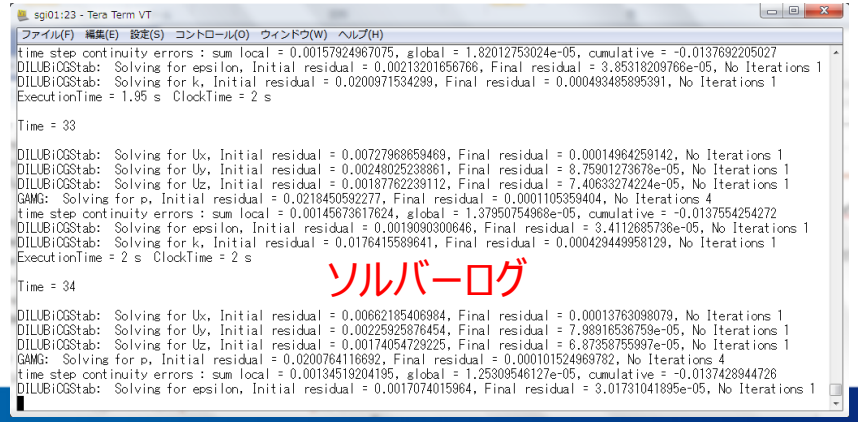


(5): Select the solver execution environment.



(2) Folder name for outputting a set of files for OpenFOAM

③



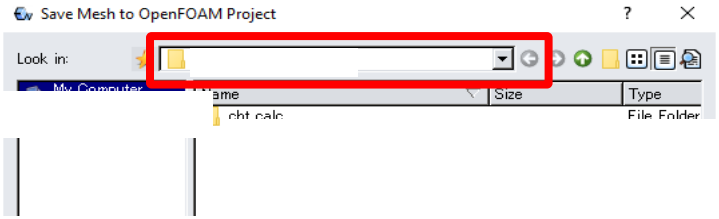
ソルバーログ

Fluid-solid thermal coupling analysis

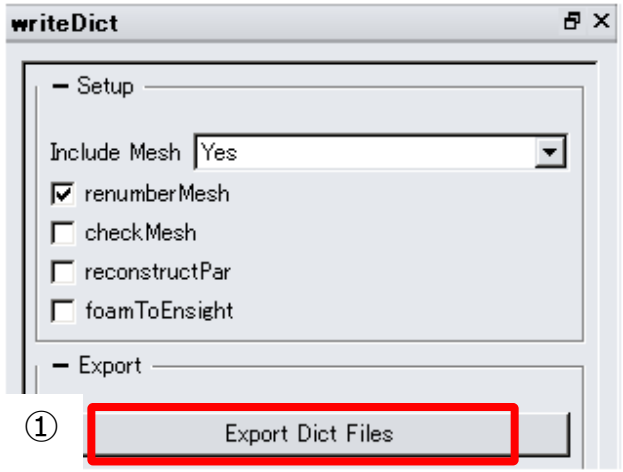
Execute Analysis/writeDict Panel > File Output

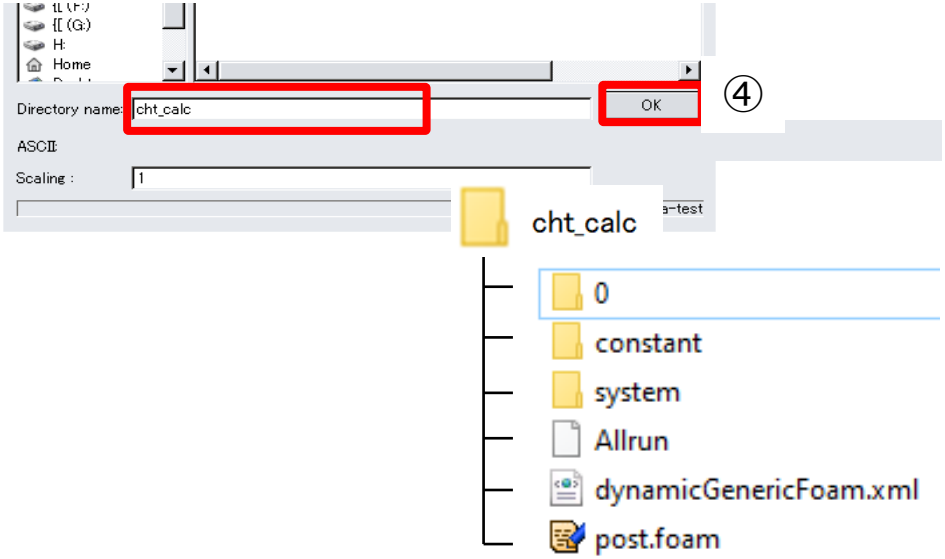
①

- If you do not have an environment to calculate directly from ennovaCFD, output a file and perform the calculations.



set of output files for OpenFOAM



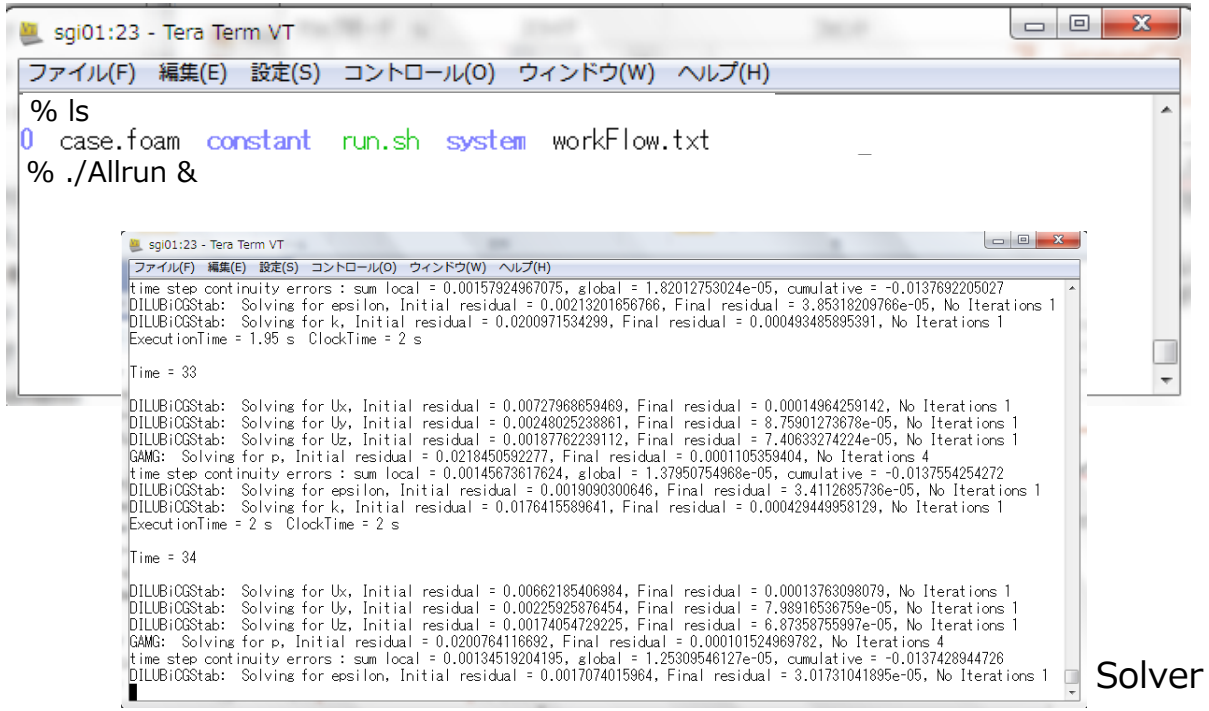


A set of output files for OpenFOAM

Fluid-solid thermal coupling analysis

How to run the analysis / OpenFOAM solver

Copy the set of files for OpenFOAM to an environment (Linux server) where OpenFOAM can be executed and execute Allrun.



Solver Logs

Fluid-solid thermal coupling analysis

Analysis Execution/Execution Progress

- If you set the OpenFOAM environment and execute the Allrun script, the process will be executed according to the settings in ennovaCFD.

```
#!/bin/bash
#------------------------------------------------
# Version: v1812
# FOAM_INST_DIR:
# Check switch ----------------------------------
# isParallel      : 1
# initCondCellZone: 0
# isCHT           : 1
# isRenumber      : 1
# isCheckMesh     : 0
# isReconst       : 0
# isFoamToEnsight : 0
# isRadiation     : 0
# isLocalBlend    : 0
#------------------------------------------------
. $WM_PROJECT_DIR/bin/tools/RunFunctions
# Start OpenFOAM workflow

# changeDictionary
for region in $(foamListRegions)
do
runApplication -s $region changeDictionary -region $region -constant -noFunctionObjects
done

# decomposePar
runApplication decomposePar -allRegions -noFunctionObjects

# Run
for region in $(foamListRegions)
do
runParallel -s $region renumberMesh -overwrite -region $region -noFunctionObjects
done
runParallel $(getApplication)

#----- end of Allrun -----#
```

Execute changeDictionary for each region

decomposeRun renumberMesh for each regionPar

Run renumberMesh for each region

Run Solver