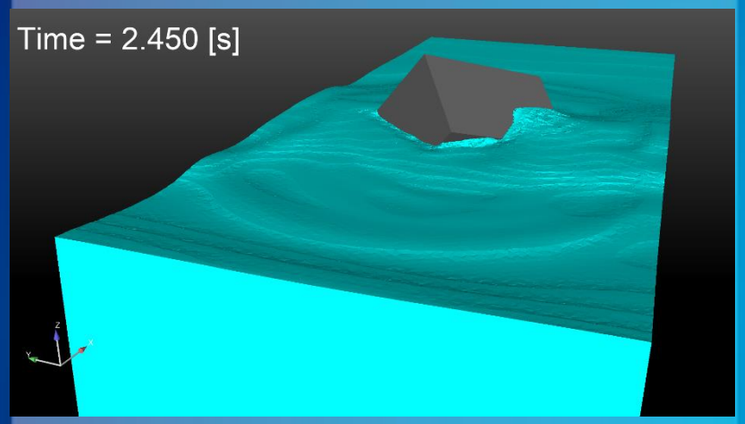


ennovaCFD For OpenFOAM v1.10.1 Overset Mesh Analysis Setup Instructions

IDAJ Corporation
Analytical Technology 8 Dept.

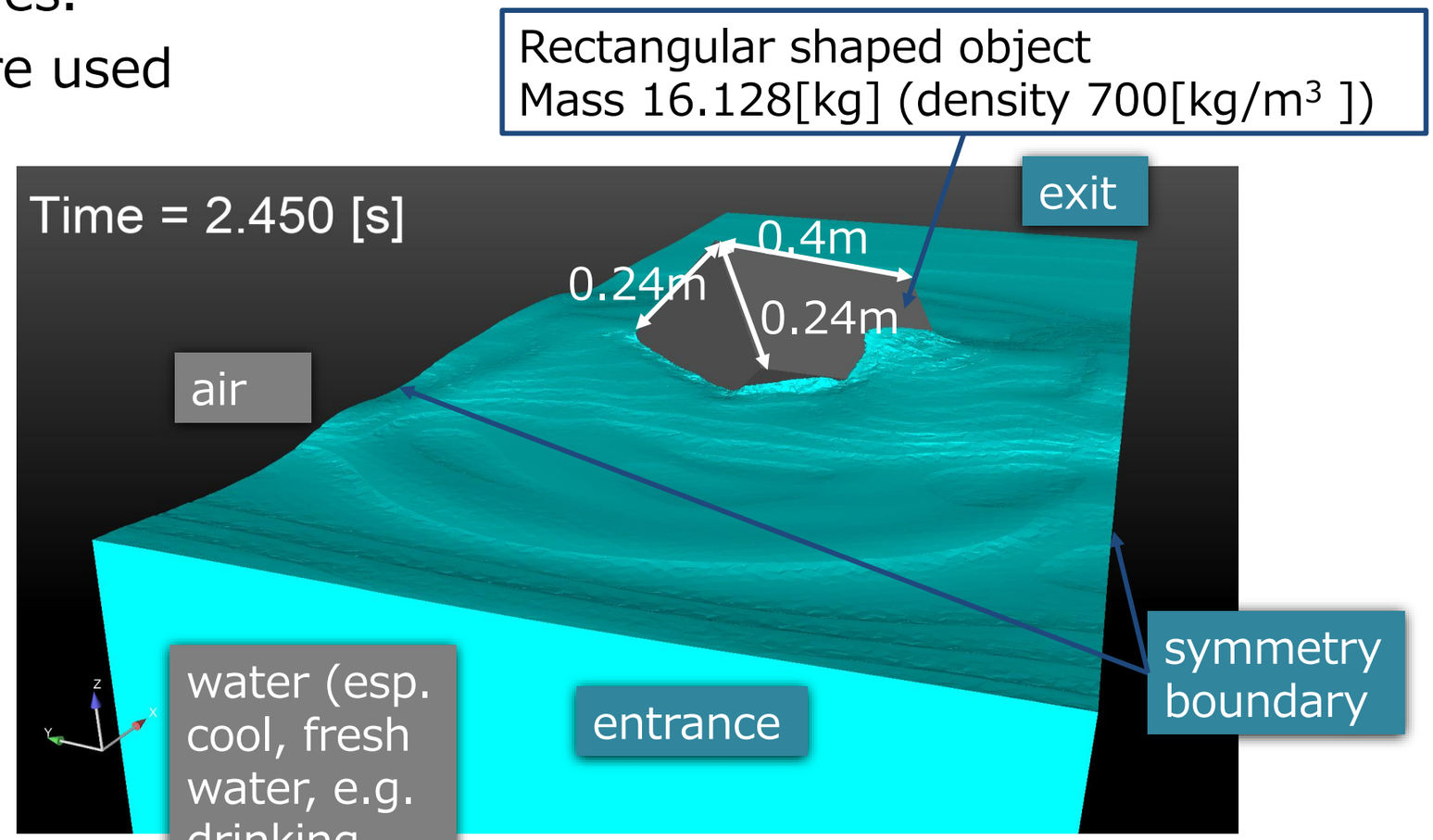
Target solver version: OpenFOAM v2012-v2312

Table of Contents

Overset mesh analysis

Overview of Overset Mesh Analysis

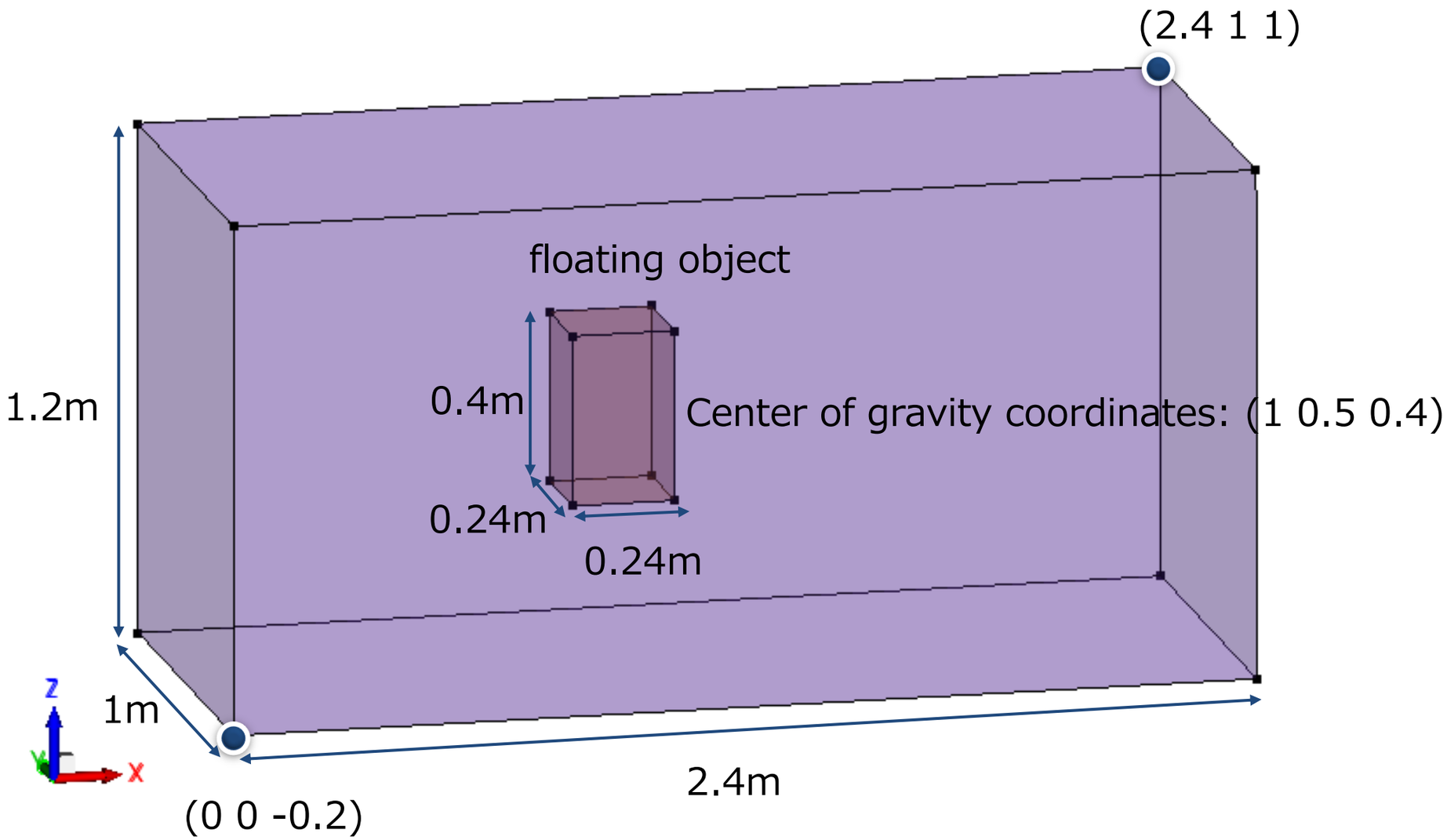
- This procedure describes how to set up an Overset mesh calculation for two-phase flow using OpenFOAM's overInterDyMFoam solver with ennovaCFD for OpenFOAM v1.10.1.
- This is an analysis of 6-DOF motion of a rectangular object floating on the surface of water with waves.
- The following functions are used
  - VOF
  - Overset Mesh
  - 6-DOF motion model
  - wave model



Overset mesh analysis

Overview of Overset Mesh Analysis

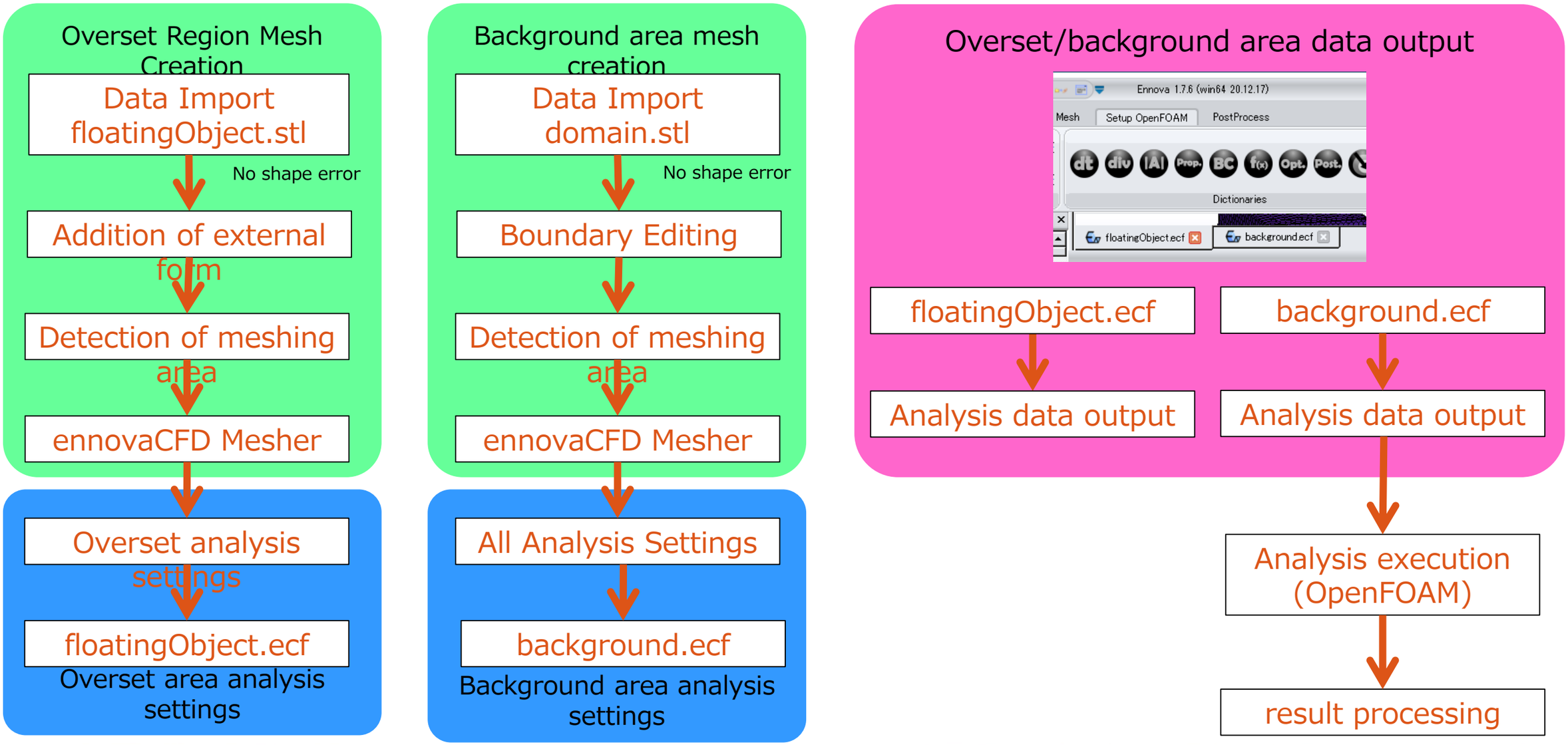
- The geometry of the analytical model is shown in the figure below.
  - Overset area: Mesh area in motion with floating objects
  - background area: static background mesh area



Overset mesh analysis

Overview of Overset Mesh Analysis

- Overset mesh calculation flow
  - When using ennovaCFD for OpenFOAM to set up an Overset mesh calculation, two ecf files are created and the analysis data output is done by opening both ecf files in the same GUI.

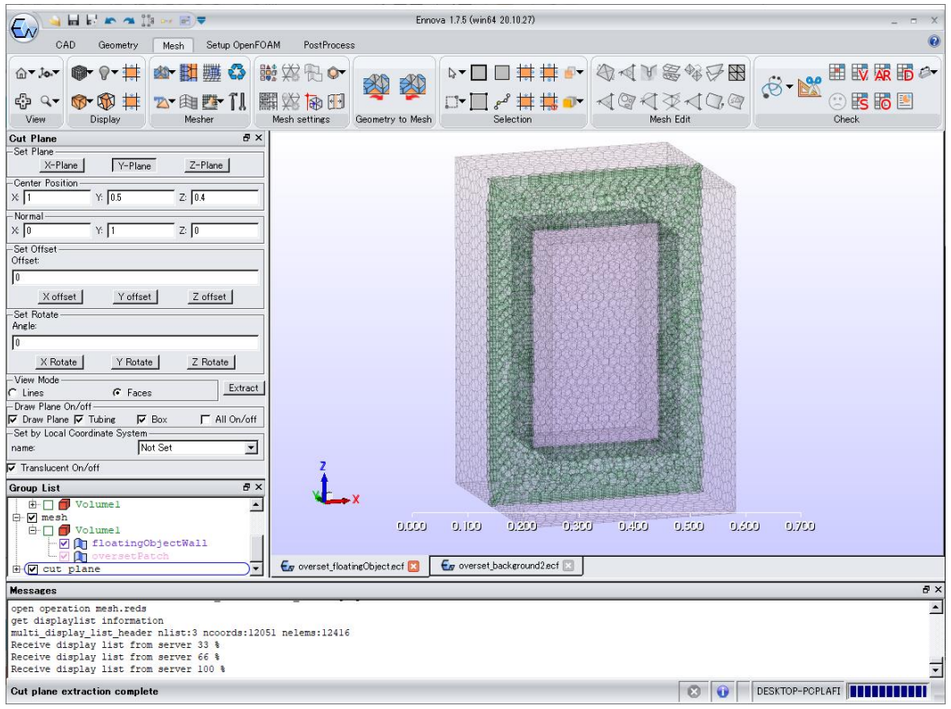


Overset mesh analysis

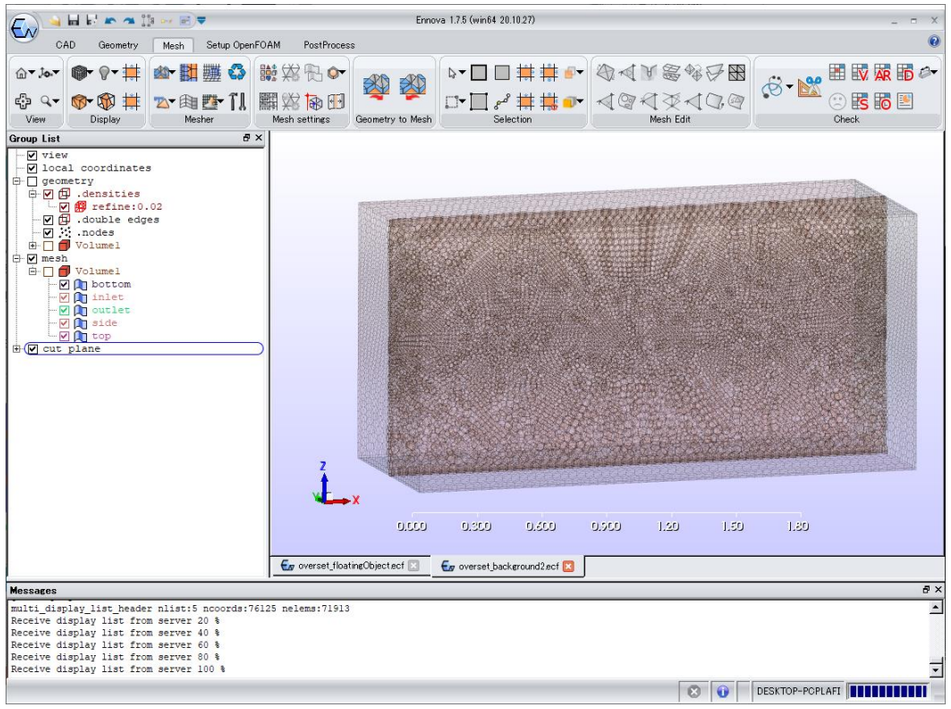
Overview of Overset Mesh Analysis

- Data to be created
  - Generate a mesh for each of the overset and background regions and save it in an ecf file.
  - This procedure creates the two models shown in the figure below.

overset area/floatingObject.ecf



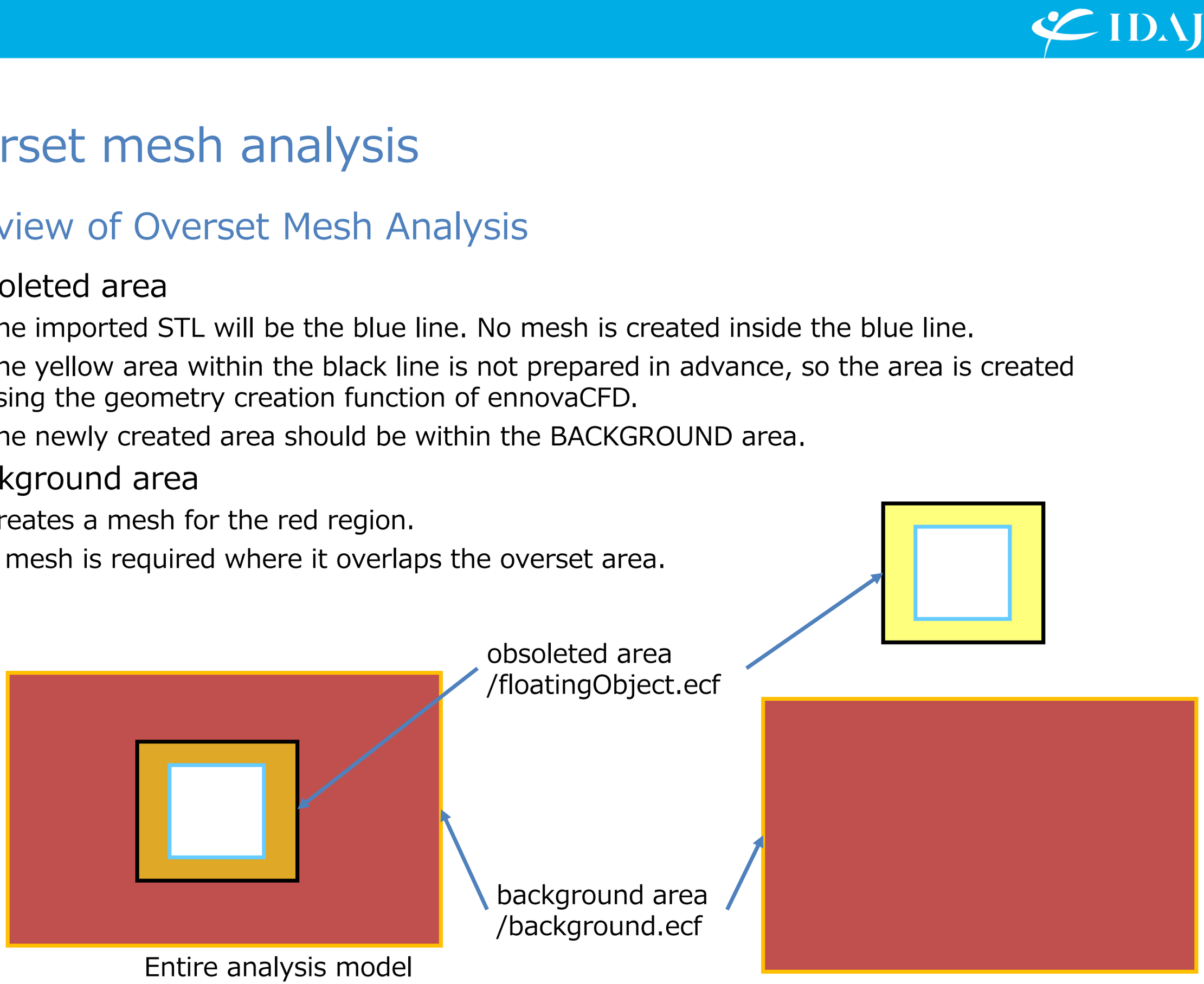
background area/background.ecf



Overset mesh analysis

Overview of Overset Mesh Analysis

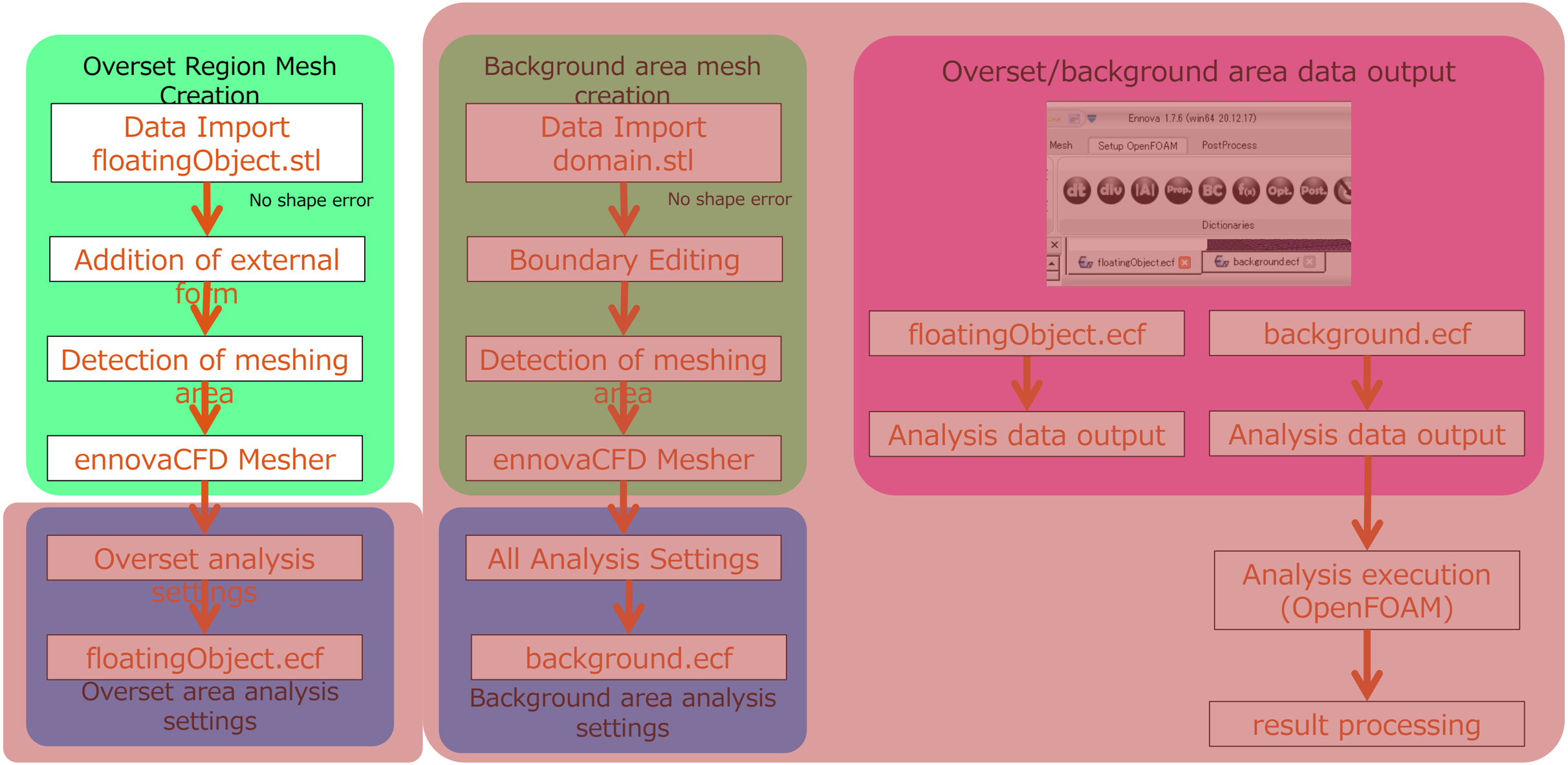
- obsoleted area
  - The imported STL will be the blue line. No mesh is created inside the blue line.
  - The yellow area within the black line is not prepared in advance, so the area is created using the geometry creation function of ennovaCFD.
  - The newly created area should be within the BACKGROUND area.
- background area
  - Creates a mesh for the red region.
  - A mesh is required where it overlaps the overset area.



Overset mesh analysis

Creating an Overset Region Mesh

- Create a mesh of the overset region from the prepared STL.



Overset mesh analysis

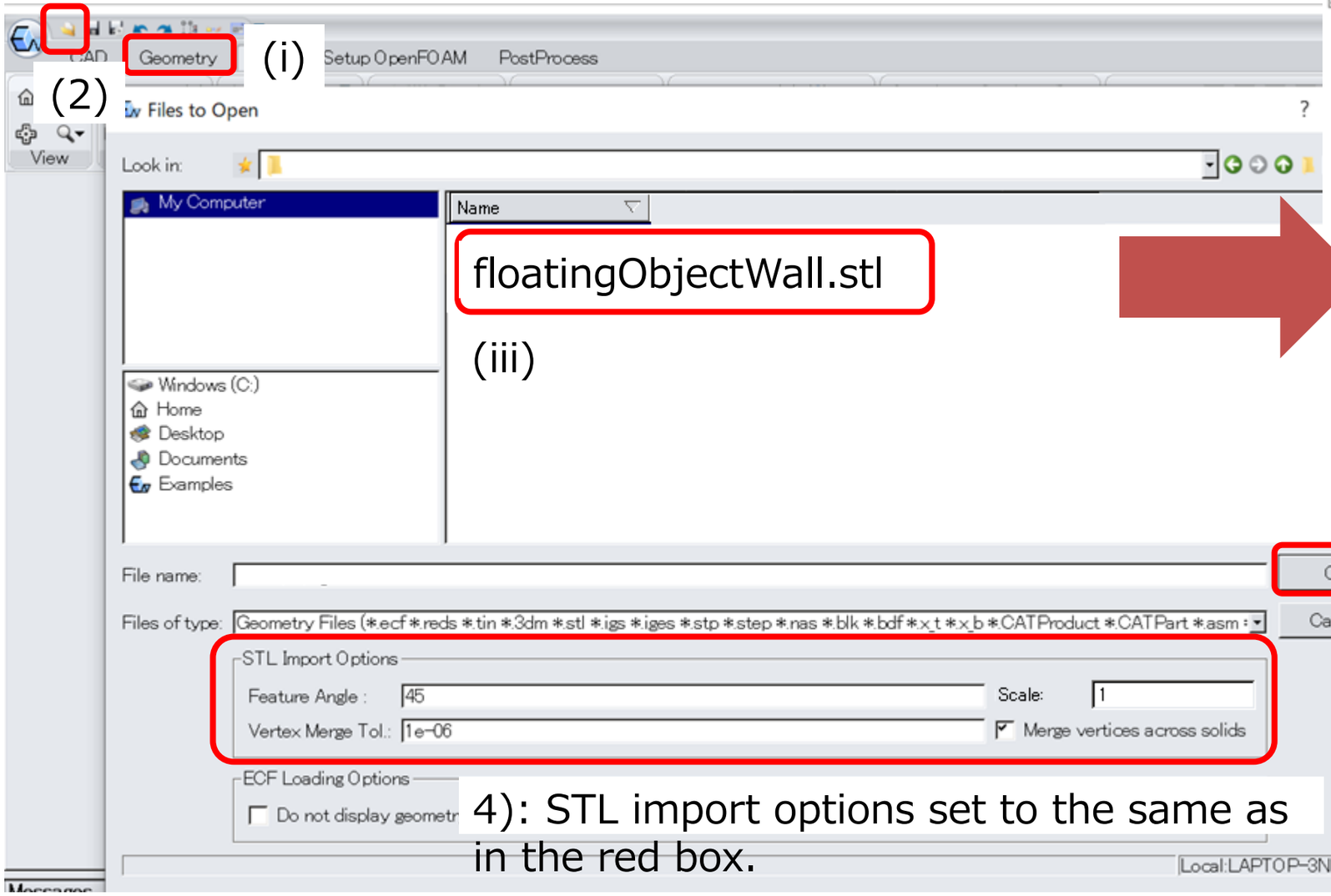
Launch ennovaCFD

- Start ennovaCFD.
  - If you are using a Windows-PC, there is a startup icon in the installation folder.
  - If you are using Linux-PC, start ennova with the command ennova in the installation folder.
  - If a license error occurs, please contact your license administrator or our support.

Overset mesh analysis

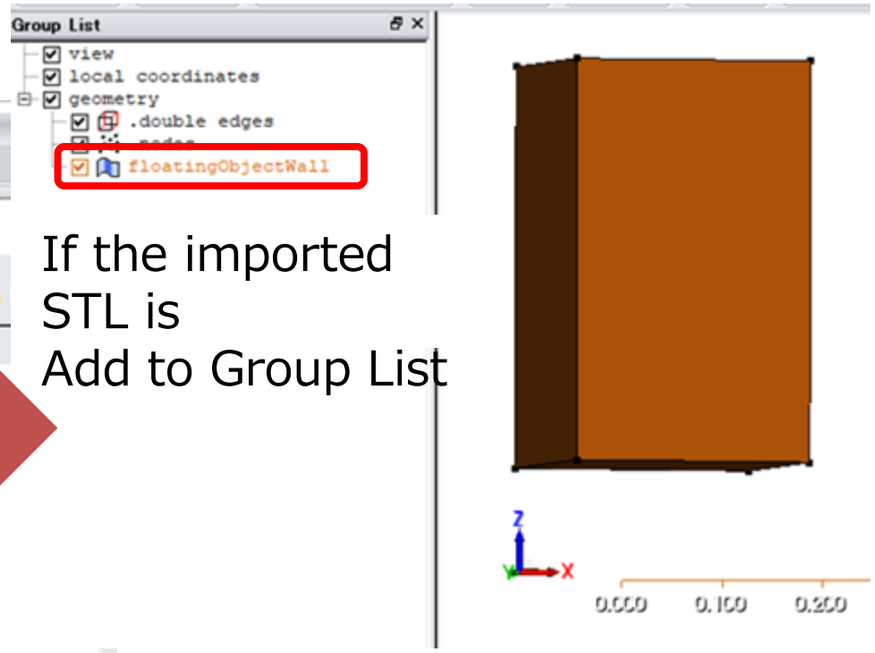
Shape data import procedure (overset area)

- Select a file to import.
- Select "Geometry" for the function tab.



5)

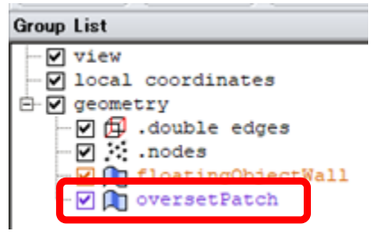
4): STL import options set to the same as in the red box.



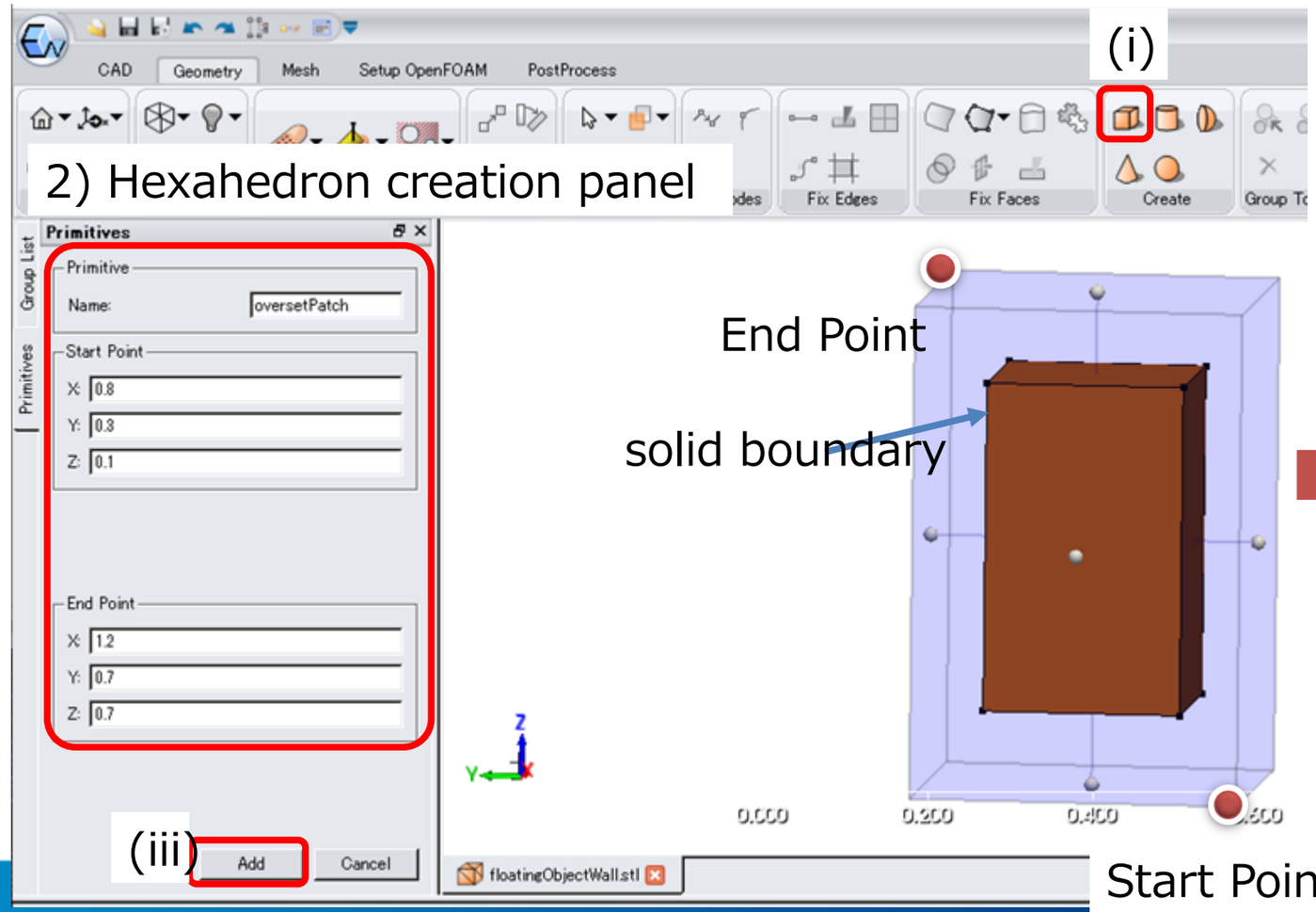
Overset mesh analysis

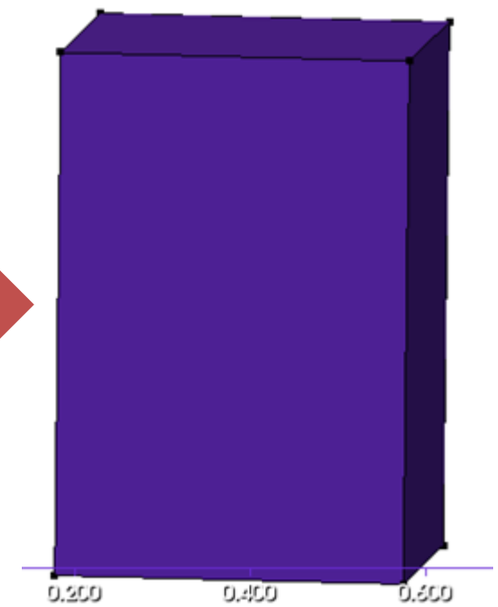
Creating the external shape of the overset area

- The overset region is created here because it requires an analytical space outside the solid boundary.
  - Open the panel to create the hexahedron with the Create Boundary function and enter a name, start point, and end point.
  - Primitive Panel
    - Name: oversetPatch
    - Start Point
    - X=0.8
    - Y=0.3
    - Z=0.1
    - End Point
    - X=1.2
    - Y=0.7
    - Z=0.7



Add shape to tree

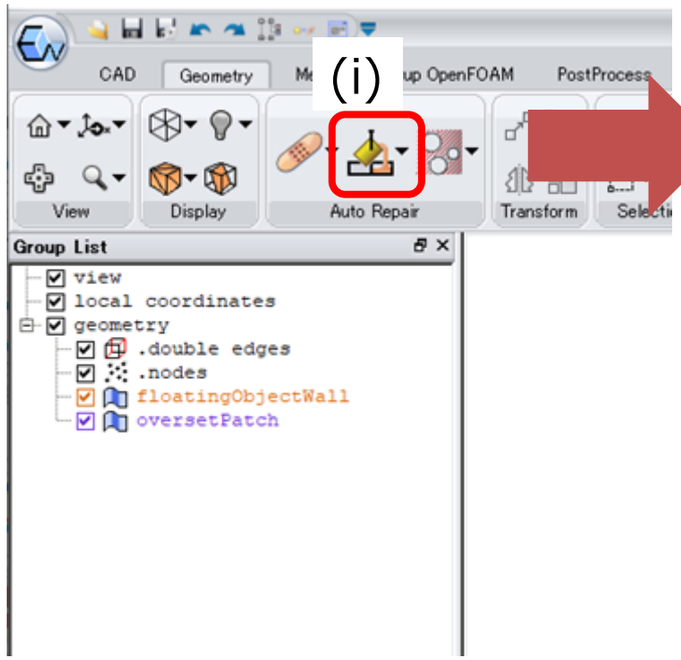


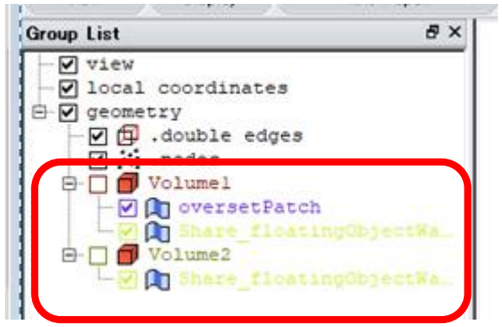


Overset mesh analysis

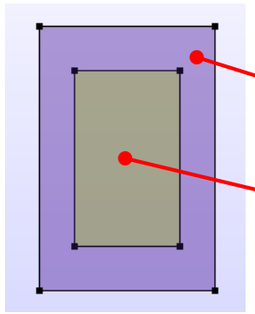
Analysis Area Creation

- The volume of the closed space where the mesh is generated is extracted by [Find volumes].
- [Find volumes] automatically recognizes any closed space as an analysis area.
- Among the recognized analysis areas, the solid area is deleted because it is unnecessary.
- The geometry operation is finished.



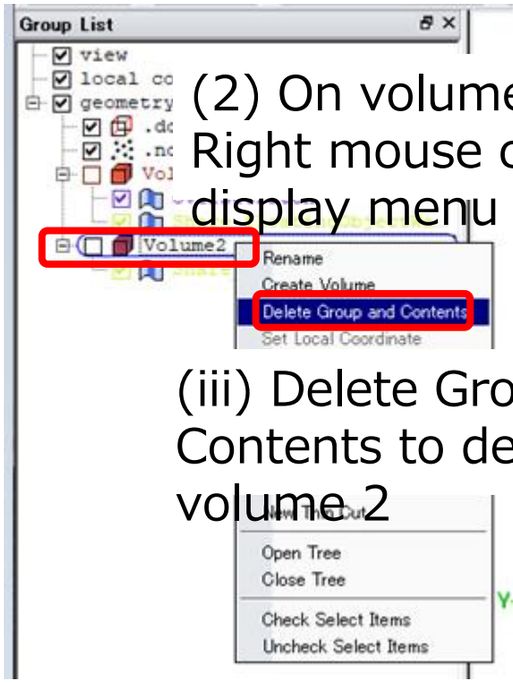


Extract two regions



Volume 1 (required)

Volume 2 (not required)



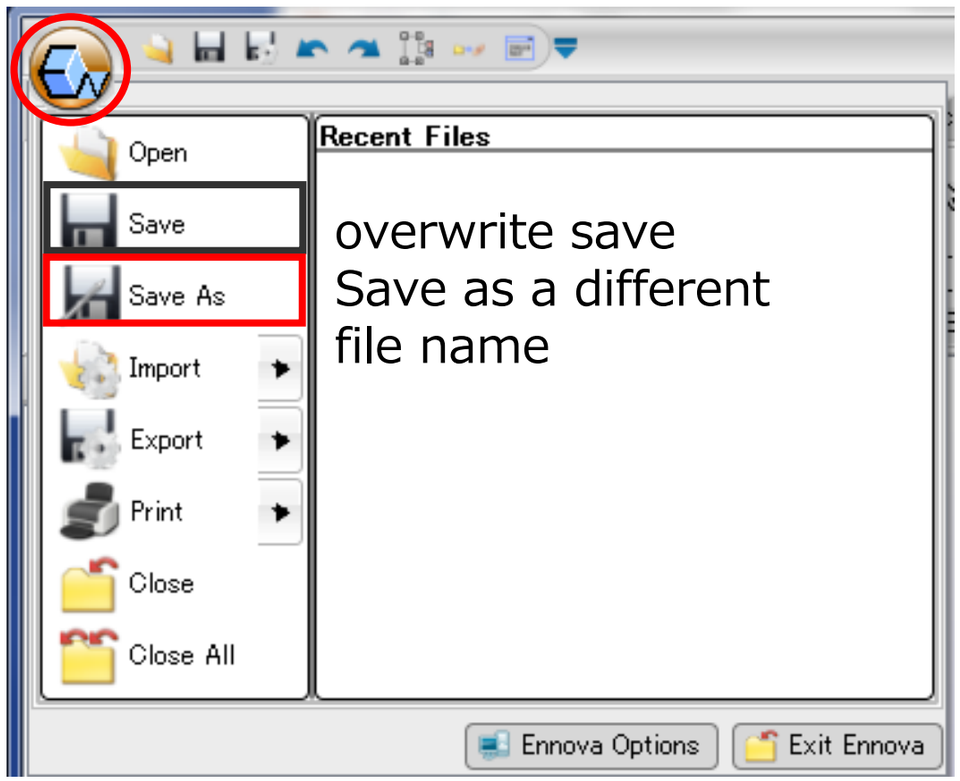
(2) On volume 2 Right mouse click to display menu

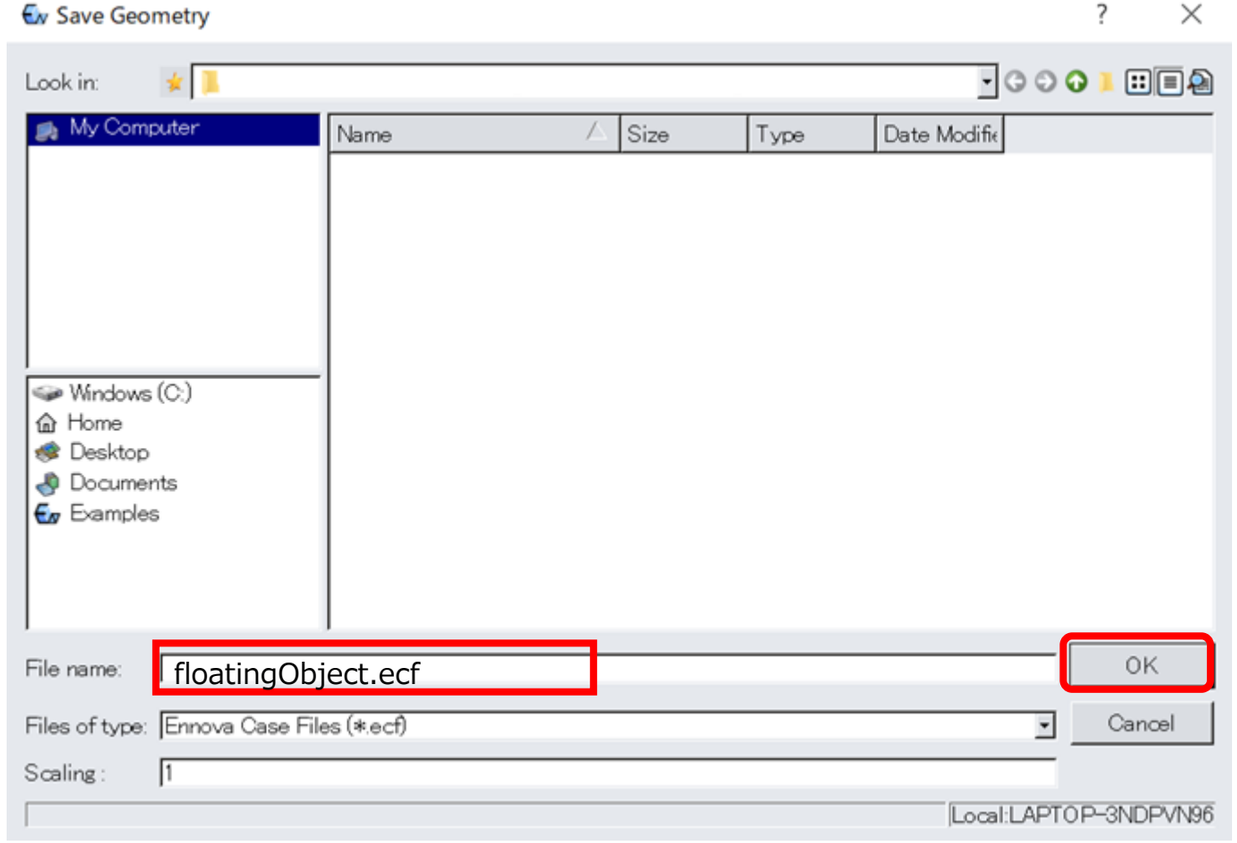
(iii) Delete Group and Contents to delete volume 2

Overset mesh analysis

Saving Files

- Save the results of your work to a file.
- ennovaCFD stores all information about the pre-process in the .ecf file.
- Here, we name the file floatingObject.ecf.



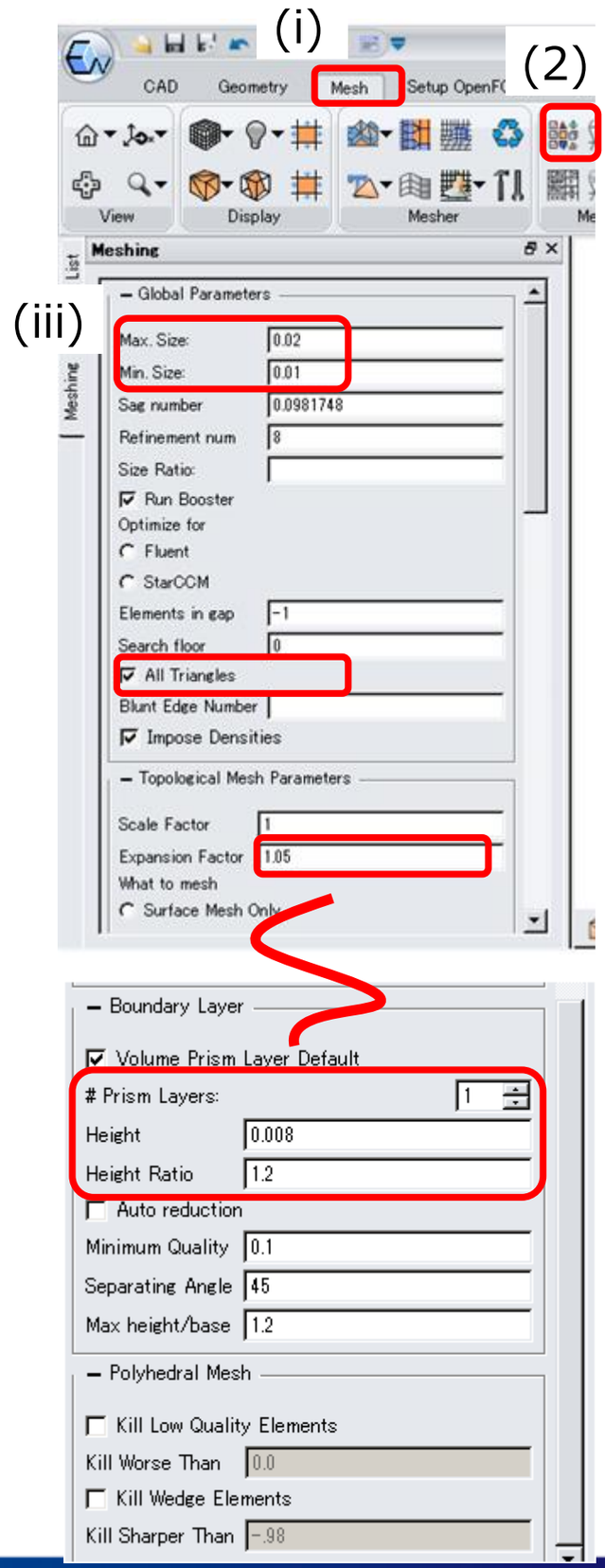


Overset mesh analysis

Mesh size setting

- Change to the Mesh tab for mesh creation.
- Sets the global mesh size to be applied to the entire model.
- Global Parameters
  - Maximum size 0.02
  - Minimum size 0.01
  - All Triangles Check on
- Topological Mesh Parameters
  - Expansion Factor 1.05
- Boundary Layer
  - Number of layers in layer mesh = 1
  - Initial height = 0.008
  - Layer growth rate = 1.2

No change as default settings are used except for the above



Overset mesh analysis

Set mesh size (Cont.)

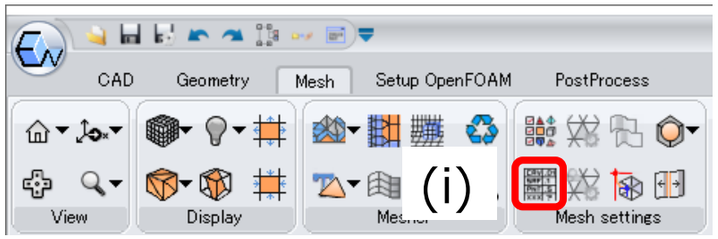
- Set the mesh size for each part.

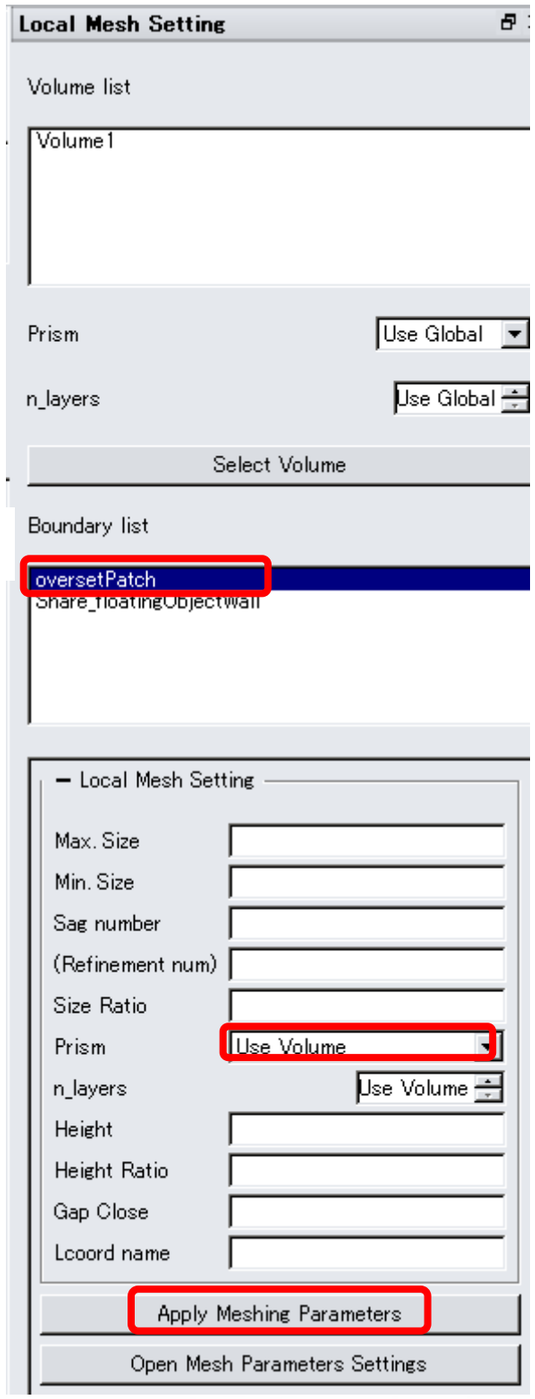
- Select oversetPatch
  - Prism :UseVolume
  - Click Apply Meshing Parameters to confirm.

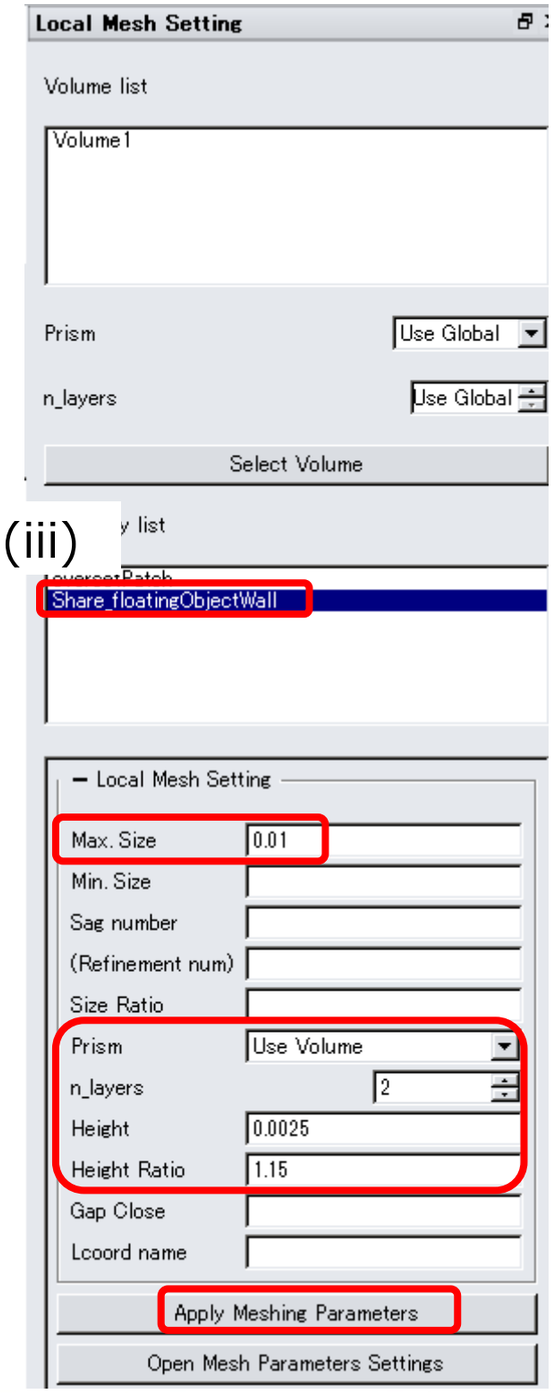
- Select Share_floatingObjectWall
  - Max Size : 0.01
  - Prism : UseVolume
  - n_layers : 2
  - Height : 0.0025
  - Height Ratio: 1.15
  - Click Apply Meshing Parameters to confirm.

Note that if you forget to click Apply Meshing Parameters, the settings will not be applied.

Other than the above, the default settings are used and will not be changed



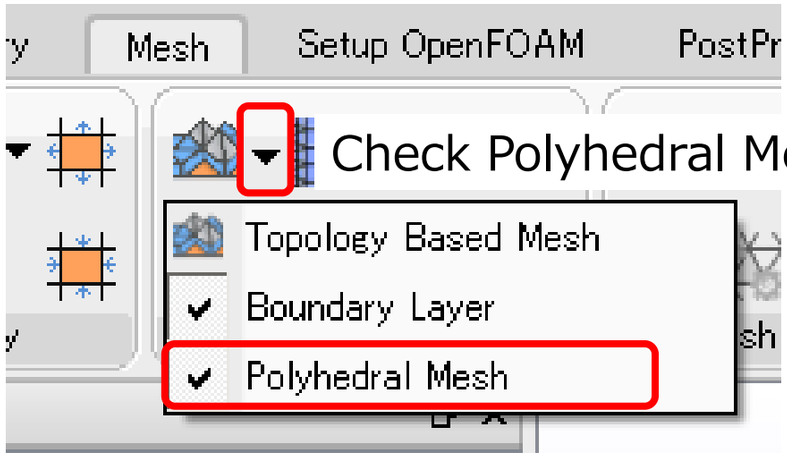




Overset mesh analysis

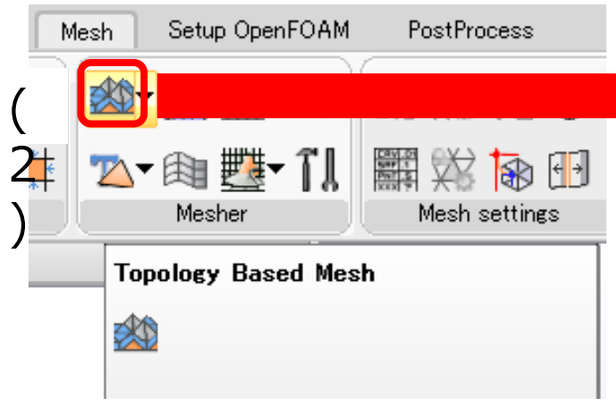
Mesher Execution

- Polyhedral mesh settings
  - Check the Polyhedral mesh flag.

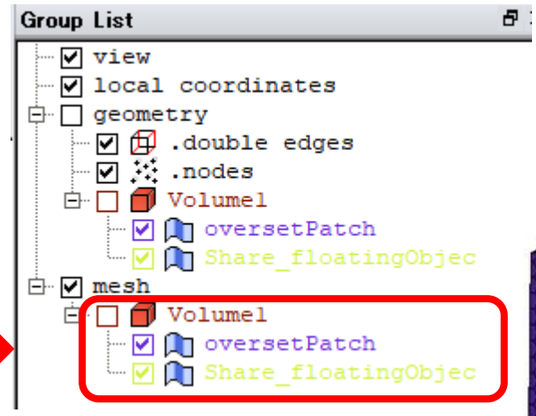


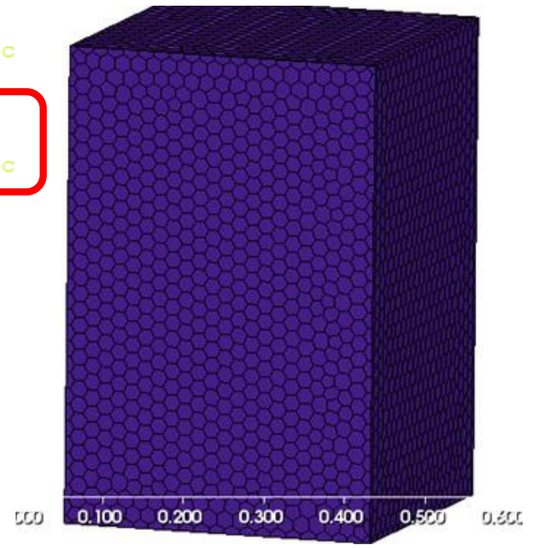
Check Polyhedral Mesh from ①:▼.

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



(2)



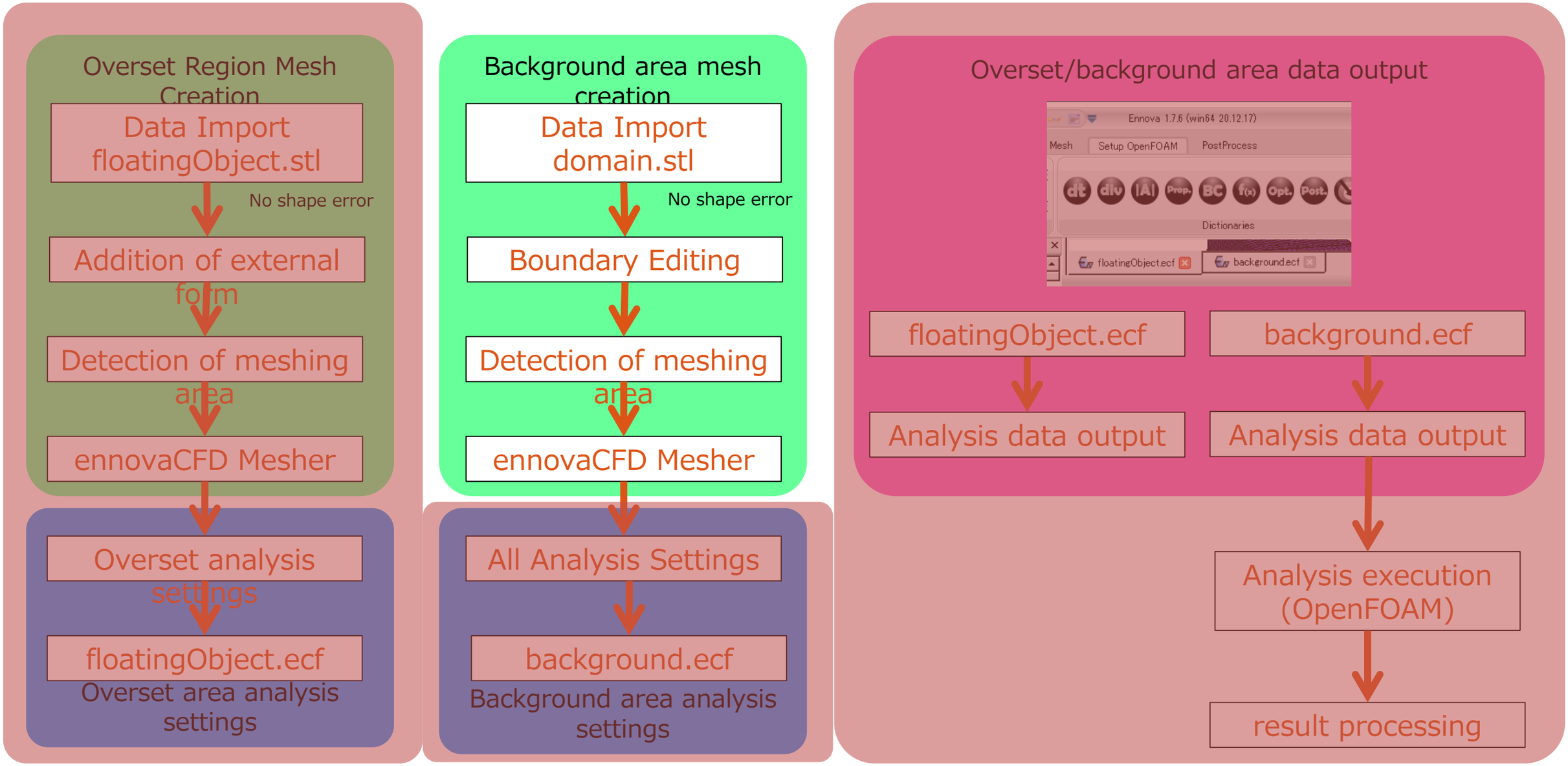


  - When mesh creation is complete, overwrite and save the ecf file.

Overset mesh analysis

Creating a background region mesh

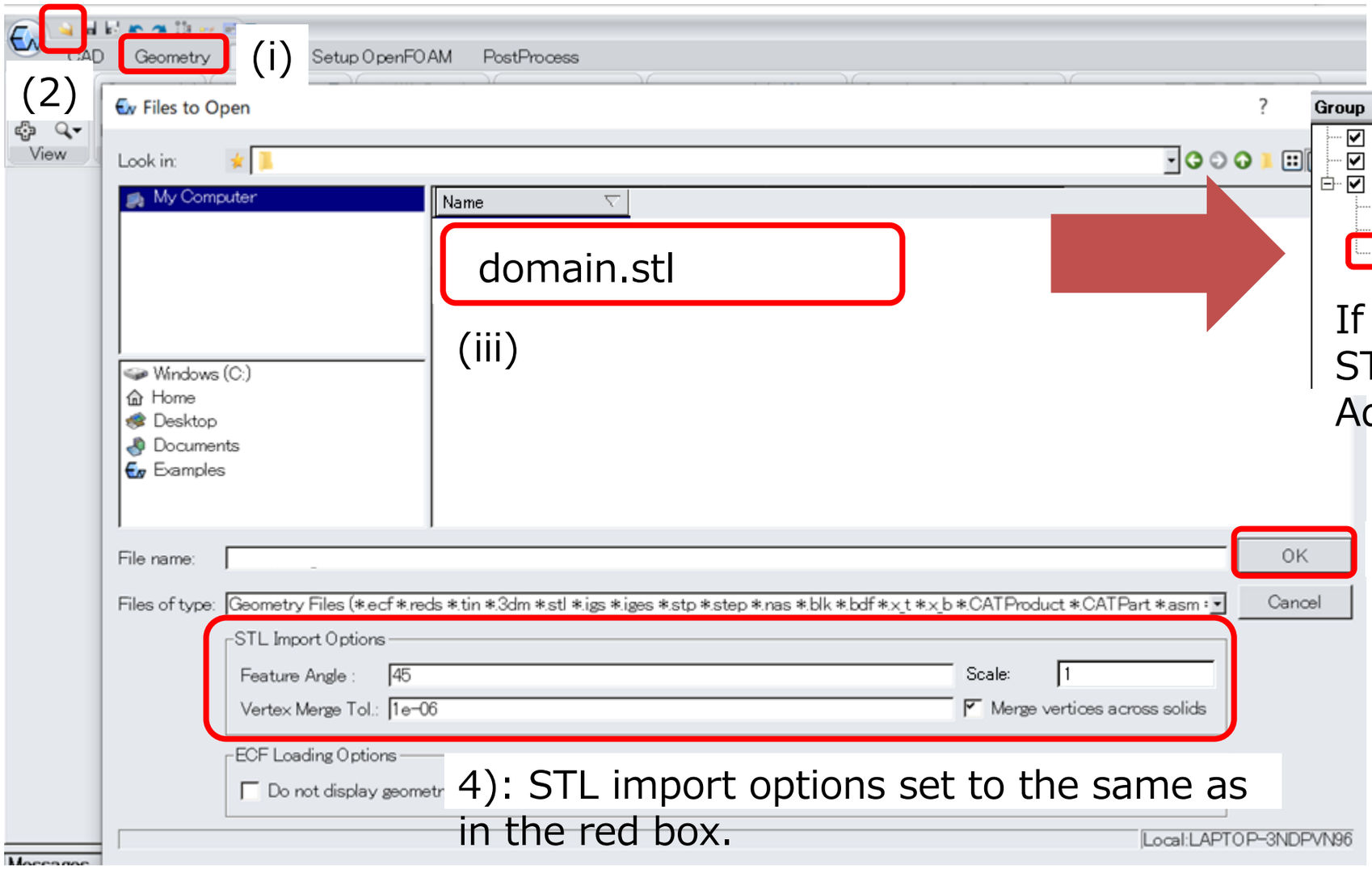
- Create a mesh for the background area from the prepared STL.



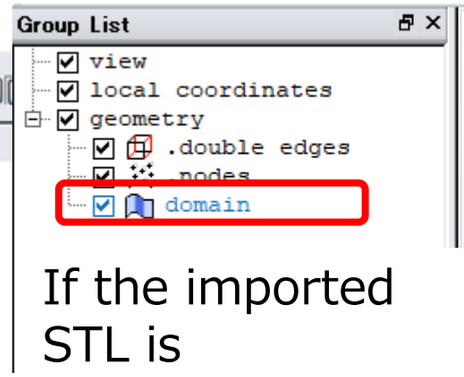
Overset mesh analysis

Shape data import procedure (background area)

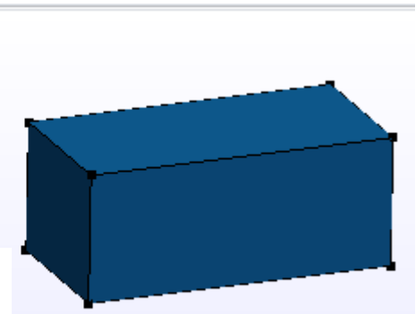
- Select a file to import.
- Select "Geometry" for the function tab.



4): STL import options set to the same as in the red box.



If the imported STL is
Add to Group List



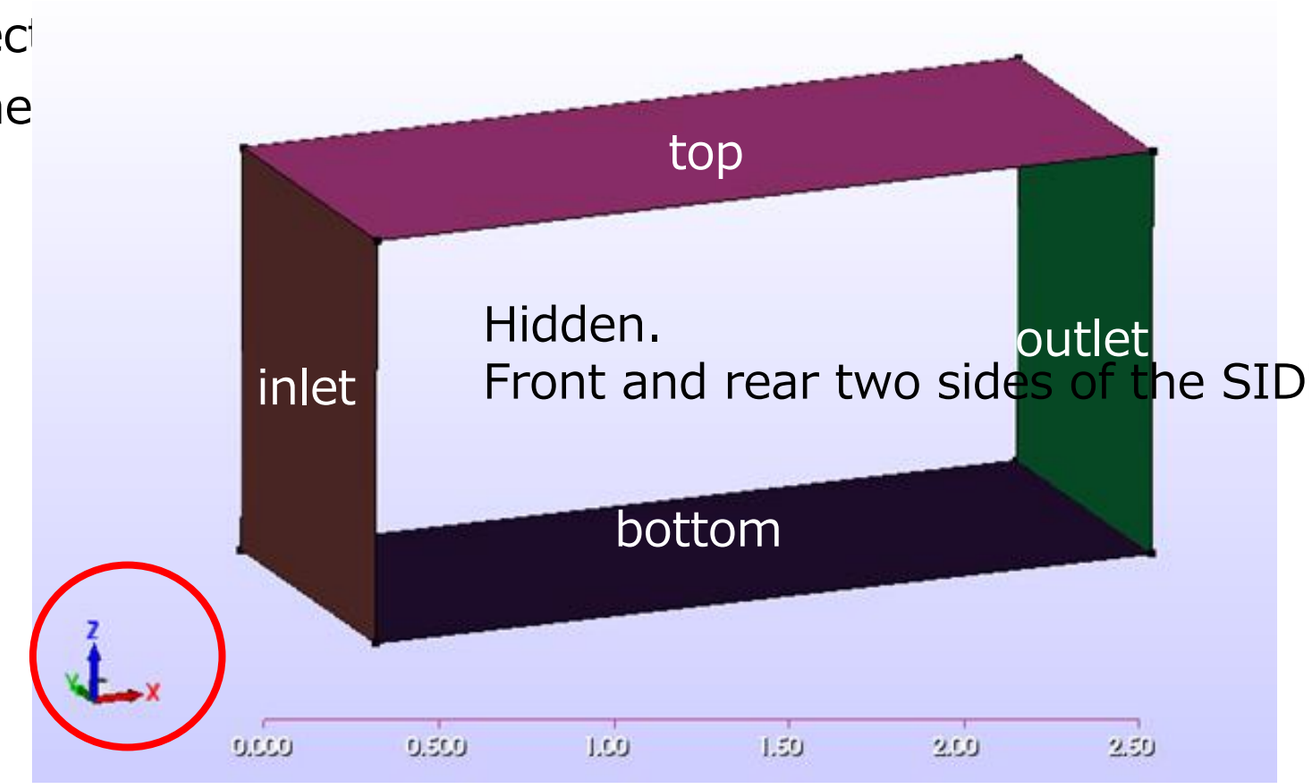
5)

Overset mesh analysis

Boundary segmentation of background area

- The boundaries of the read DOMAIN are divided as follows
  - Entrance boundary: inlet (surface in -X direction)
  - Exit boundary: outlet(+X-directional face)
  - Top surface boundary: top(+Z direction surface)
  - Bottom surface boundary: BOTTOM (surface in -Z direction)
  - Side boundary: side (Y-directional face)

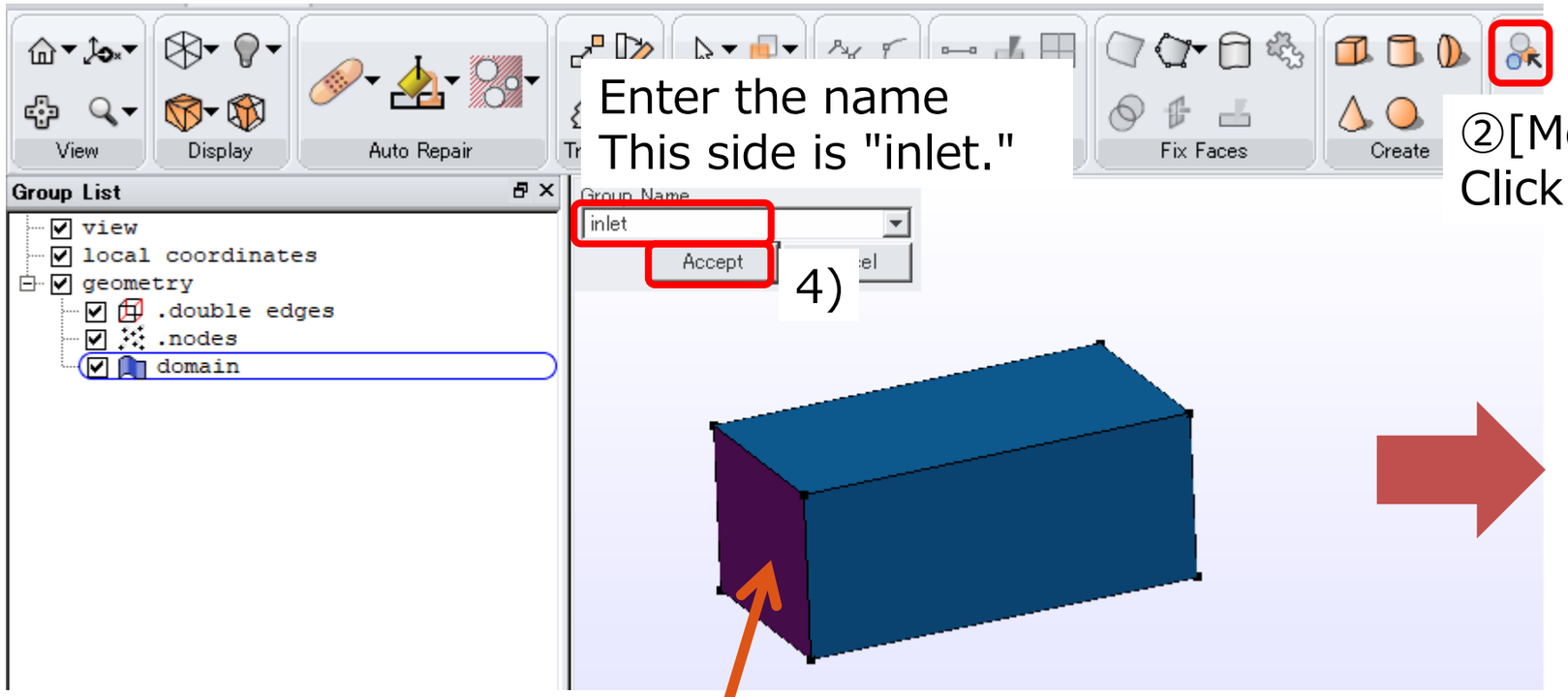
In this analysis, the -Z direction is the direct
Boundaries are referred to as vertical in the



Overset mesh analysis

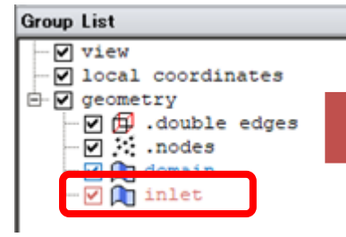
Boundary segmentation of background area

- Split the boundaries of the read DOMAIN.
  - Entrance Boundary inlet
  - Similar operations are used to create OUTLET, TOP, and BOTTOM.
  - The side is created by renaming the existing DOMAIN on the next page.

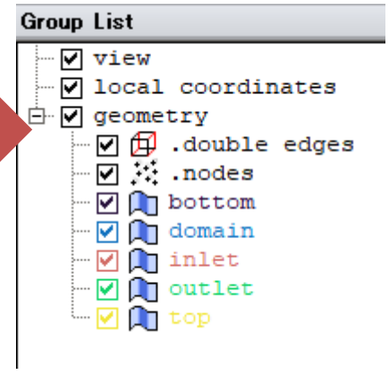


Enter the name
This side is "inlet."

②[Move to group].
Click on the icon



If the inlet is
Add to Group List

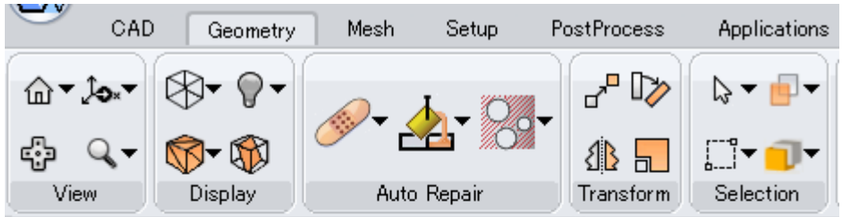


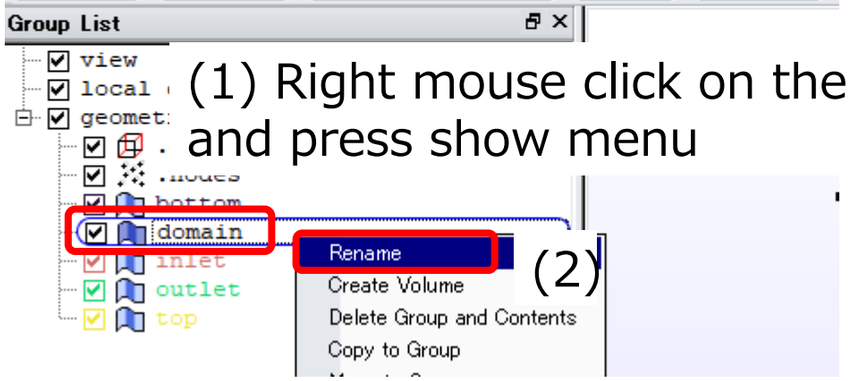
(1) Click on the surface to be set as the entrance boundary.
(When selected, changes to pink display)

Overset mesh analysis

Boundary segmentation of background area

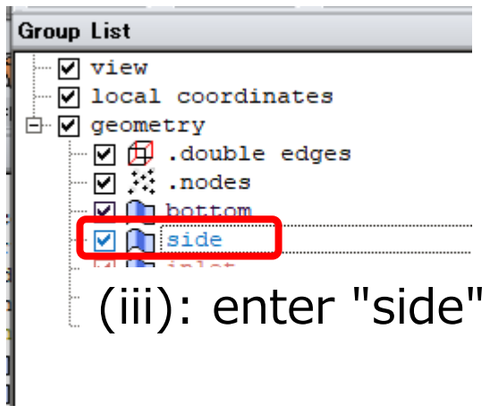
- Split the boundaries of the read DOMAIN.
  - Side boundary side





(1) Right mouse click on the DOMAIN and press show menu

(2)

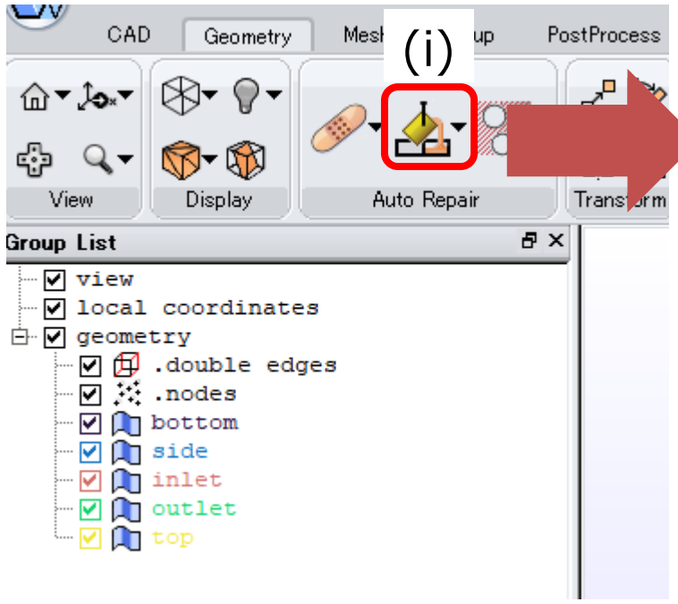


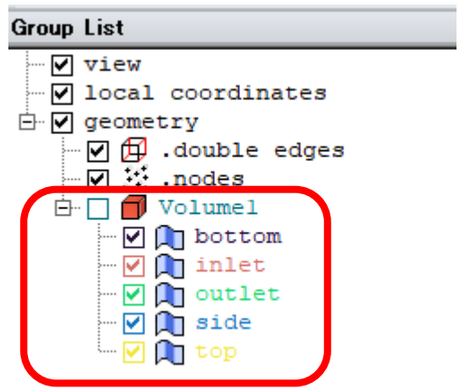
(iii): enter "side" and

Overset mesh analysis

Analysis Area Creation

- The volume of the closed space where the mesh is generated is extracted by [Find volumes].
- [Find volumes] automatically recognizes any closed space as an analysis area.
- The geometry operation is finished.
- Save the created data under the name background.ecf.





Extract one area

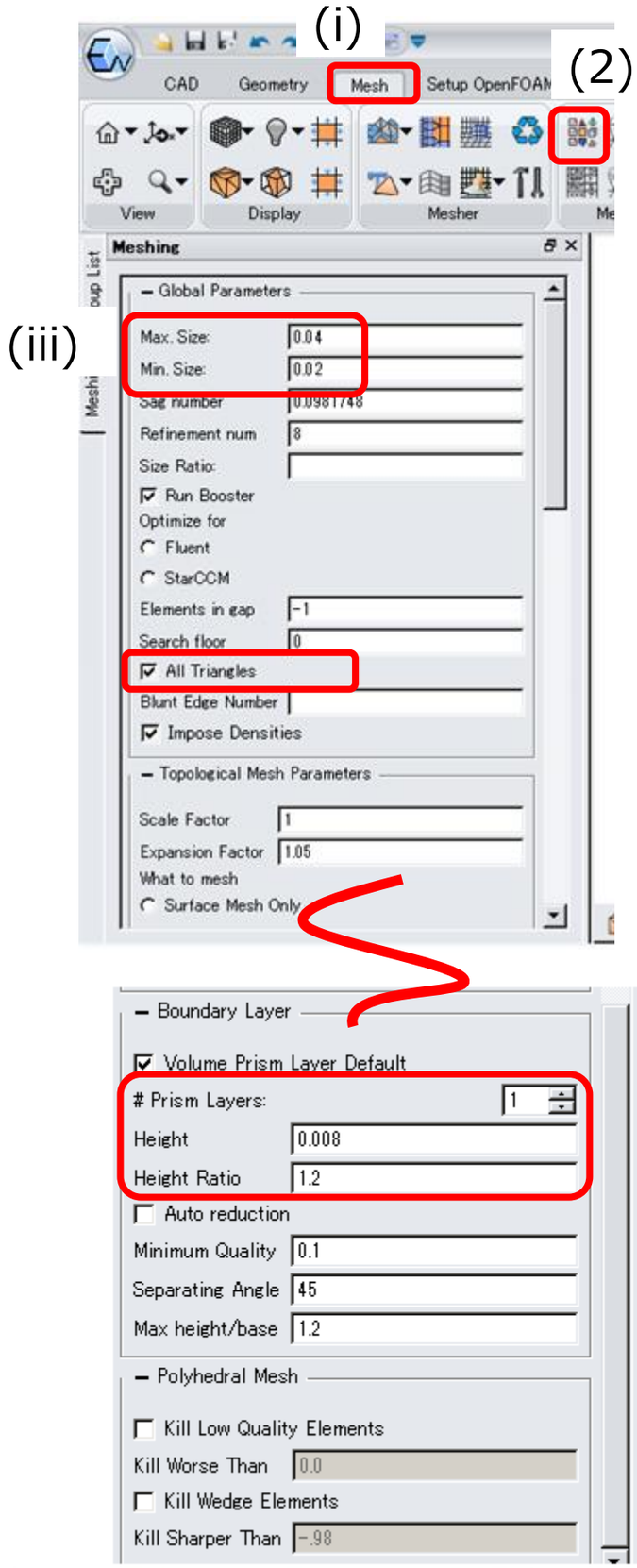
Overset mesh analysis

Mesh size setting

- Change to the Mesh tab for mesh creation.
- Sets the global mesh size to be applied to the entire model.
- Global Parameters
  - Maximum size 0.04
  - Minimum size 0.02
  - All Triangles Check on

- Boundary Layer
  - Number of layers in layer mesh = 1
  - Initial height = 0.008
  - Height Ratio: 1.2

No change as default settings are used except for the above



Overset mesh analysis

Set mesh size (Cont.)

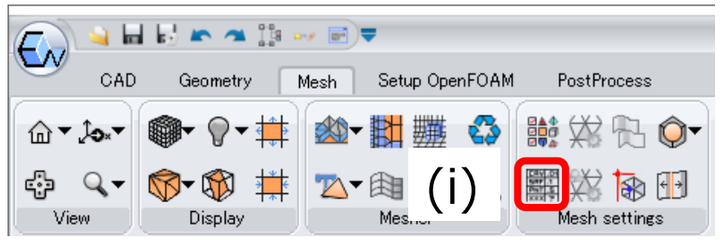
- Set the mesh size for each part.

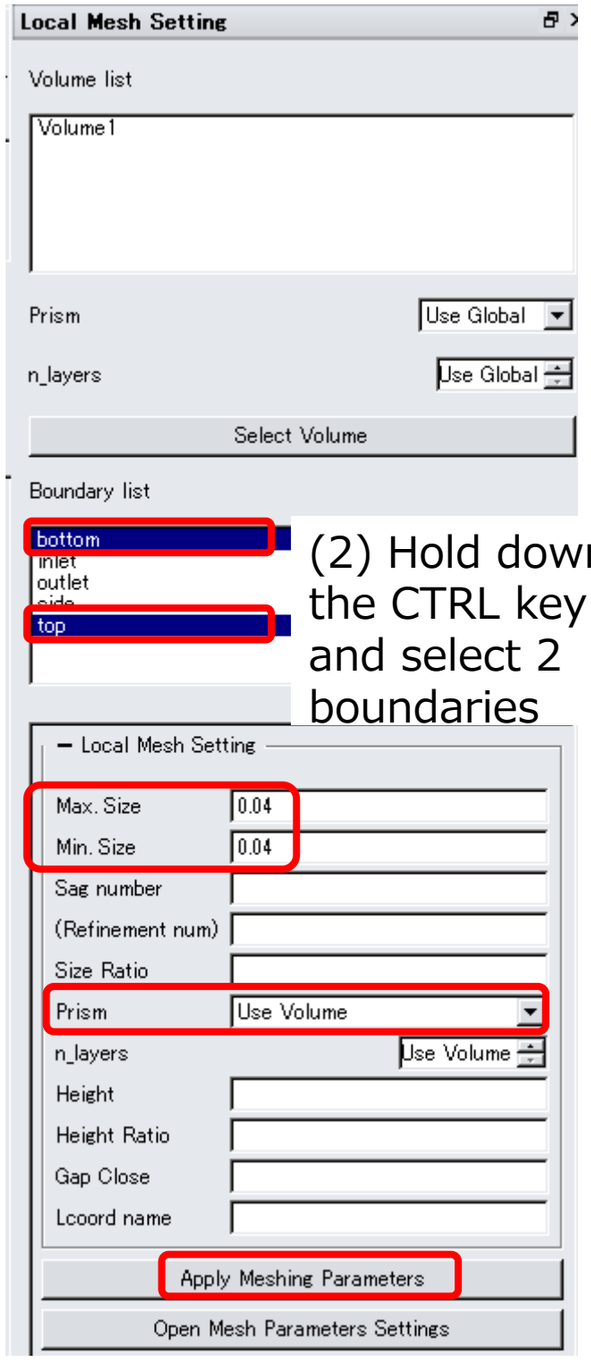
- Select bottom and top
  - Max. Size :0.04
  - Min. Size :0.04
  - Prism :UseVolume
  - Click Apply Meshing Parameters to confirm.

  Note that if you forget to click Apply Meshing Parameters, the settings will not be reflected.

  No change as default settings are used except for the above

- No change in INLET, OUTLET, SIDE



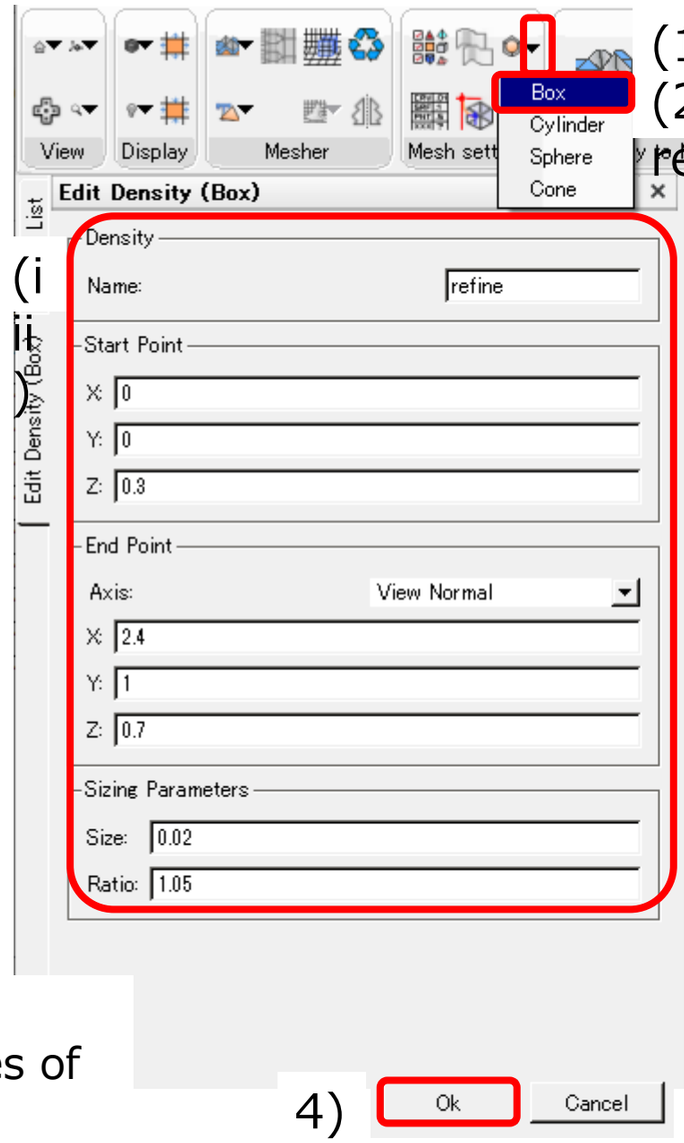


Overset mesh analysis

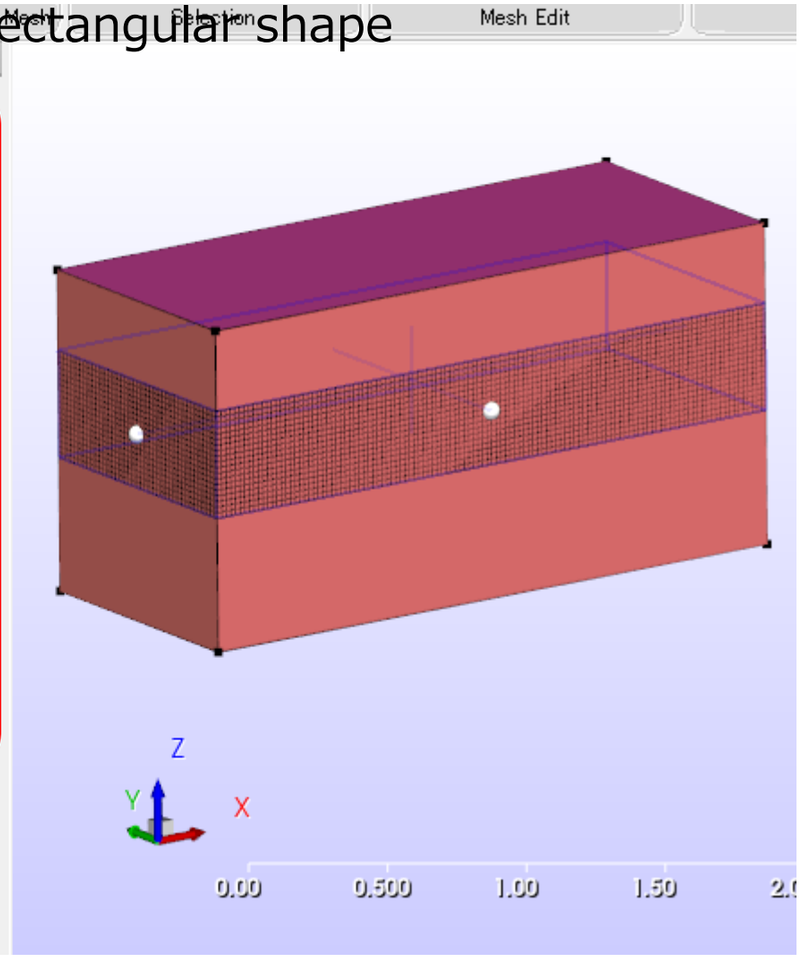
The location and cell size of the subdivision area, Visual confirmation.

Set mesh size (Cont.)

- To locally reduce the mesh size near the water surface, set the subdivision area.
- Density
  - Name : refine
- Start Point
  - X=0
  - Y=0
  - Z=0.3
- end point
  - X=2.4
  - Y=1
  - Z=0.7
- Size : 0.02
- Ratio : 1.05



(1) Pull-down menu display
(2) Select subdivision area [Box] of rectangular shape

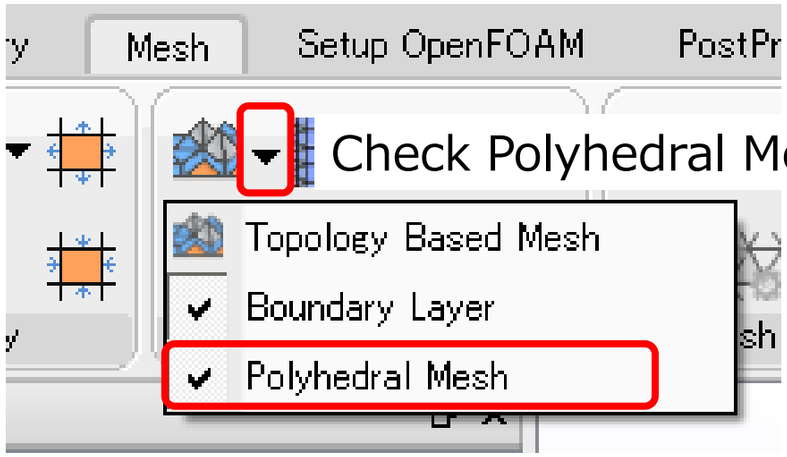


[Name]: Name of subdivision area
[Start Point],[End Point]: Coordinates of two points at the vertex
[Size]: Cell size
[Ratio]: cell size expansion ratio

Overset mesh analysis

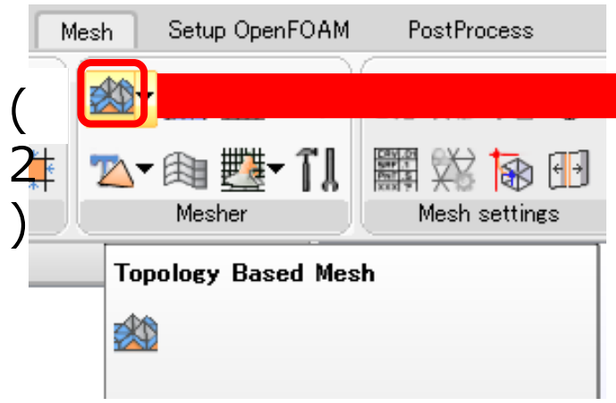
Mesher Execution

- Polyhedral mesh settings
  - Check the Polyhedral mesh flag.

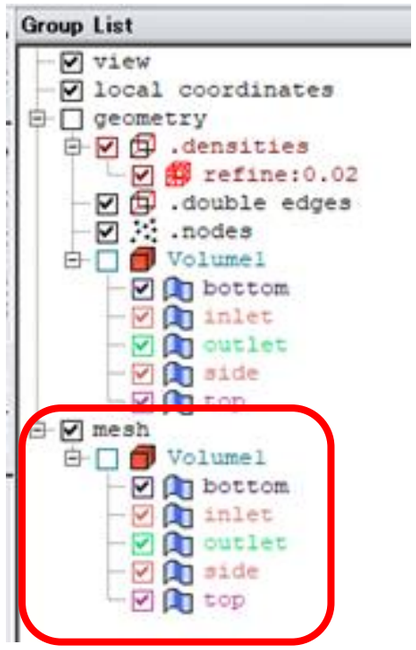


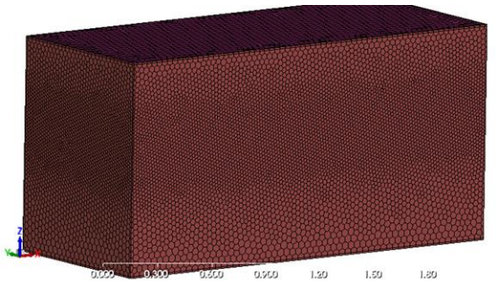
Check Polyhedral Mesh from ①:▼.

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



(2)





  - When mesh creation is complete, overwrite and save the background.ecf file.

Overset mesh analysis

Analysis settings for the overset region

- The analysis settings are applied to the mesh data of the created overset area.

Overset Region Mesh Creation

Data Import floatingObject.stl

No shape error

Addition of external form

Detection of meshing area

ennovaCFD Mesher

Overset analysis settings

floatingObject.ecf

Overset area analysis settings

Background area mesh creation

Data Import domain.stl

No shape error

Boundary Editing

Detection of meshing area

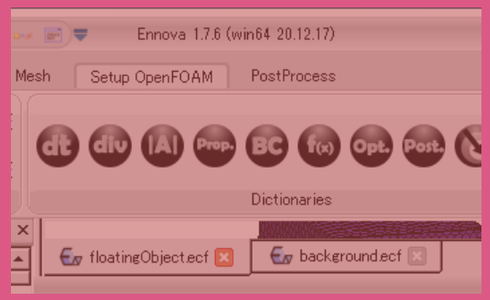
ennovaCFD Mesher

All Analysis Settings

background.ecf

Background area analysis settings

Overset/background area data output



floatingObject.ecf

Analysis data output

background.ecf

Analysis data output

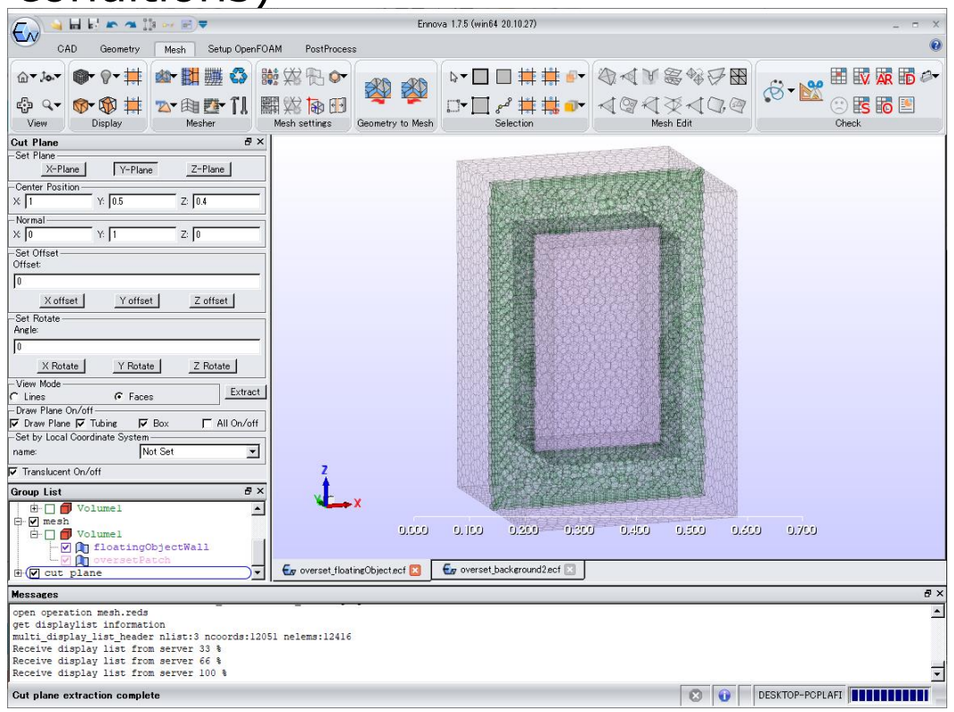
Analysis execution (OpenFOAM)

result processing

Overset mesh analysis

Setting ofverset area

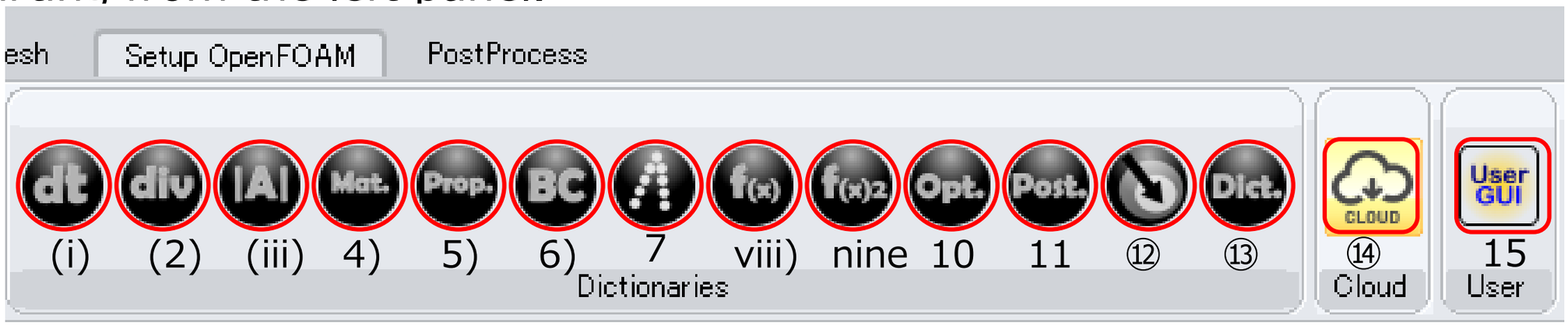
- The first step is to set up the analysis of the model in the overset region.
- Settings that overlap between the overset area and the background area are used for the background area, so only the following settings should be made in the overset area.
  - Specifies that the model is a model of the overset region
  - Rigid body motion setup (6-DOF motion, defined rotational motion)
  - Volume setup (required before setting boundary conditions)
  - Boundary condition setting



Overset mesh analysis

Setup OpenFOAM Tab Description

- OpenFOAM calculation settings are made in the Setup OpenFOAM tab. The menu is roughly categorized by OpenFOAM Dictionary file. Work is performed in order, starting from the left panel.



(1) controlDict panel
Solver, version used
Domain partitioning method for parallel computation
Time increments and frequency of output of results
Turbulence model
Settings such as

② fvSchemes panel
Discrete scheme for each term Configuration of

(3) fvSolution panel
Linear solver
Relaxation coefficient
Settings such as

4) Materials Panel
Material Properties
Configuration of

(5) Constant panel
Initial conditions
Physical Properties
CellZone
Settings such as

(6) caseProperties panel
Boundary conditions
Configuration of

(7) Lagrangian panel
Particle setting

8) Functions panel
. function
Configuration of

(9) functions2 panel
Function2
Configuration of

⑩ fvOptions panel
fvOptions
Configuration of

(11) postProcessing panel
CaseDicts
Configuration of

⑫Optimisation panel
Optimisation settings

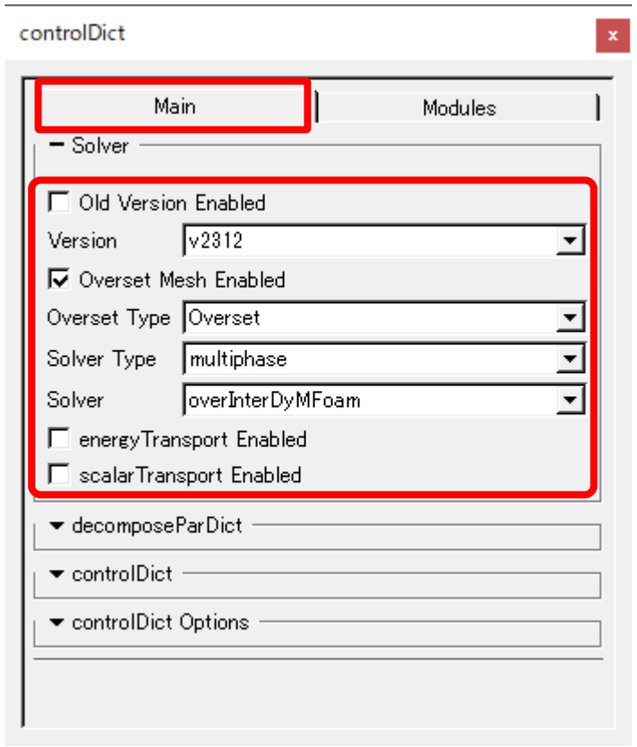
⑬ writeDict panel
Output a set of OpenFOAM files

⑭ cloud panel
Computational execution by cloud servers

(15) UserGUI panel
GUI added by user

Overset mesh analysis

Setting of overset area | controlDict panel



- Set up the solver version to be used and basic settings for calculations.
- Change the settings in red text.
- solver
  - Version ：OpenFOAM version
  - Overset Mesh Enabled ：Check ON
  - Overset Type ：Overset

    Select Overset because the Overset area is being set up
  - Solver Type*2 ： multiphase
  - Solver*3 ： overInterDyMFoam(unsteady)
  - energyTransport Enabled：checked off
  - scalarTransport Enabled：checked off

Settings other than those listed above are not required.

*1: Select the version of OpenFOAM to be used in the calculation. Some functions may not be available depending on the version.

*2: Select the solver type.

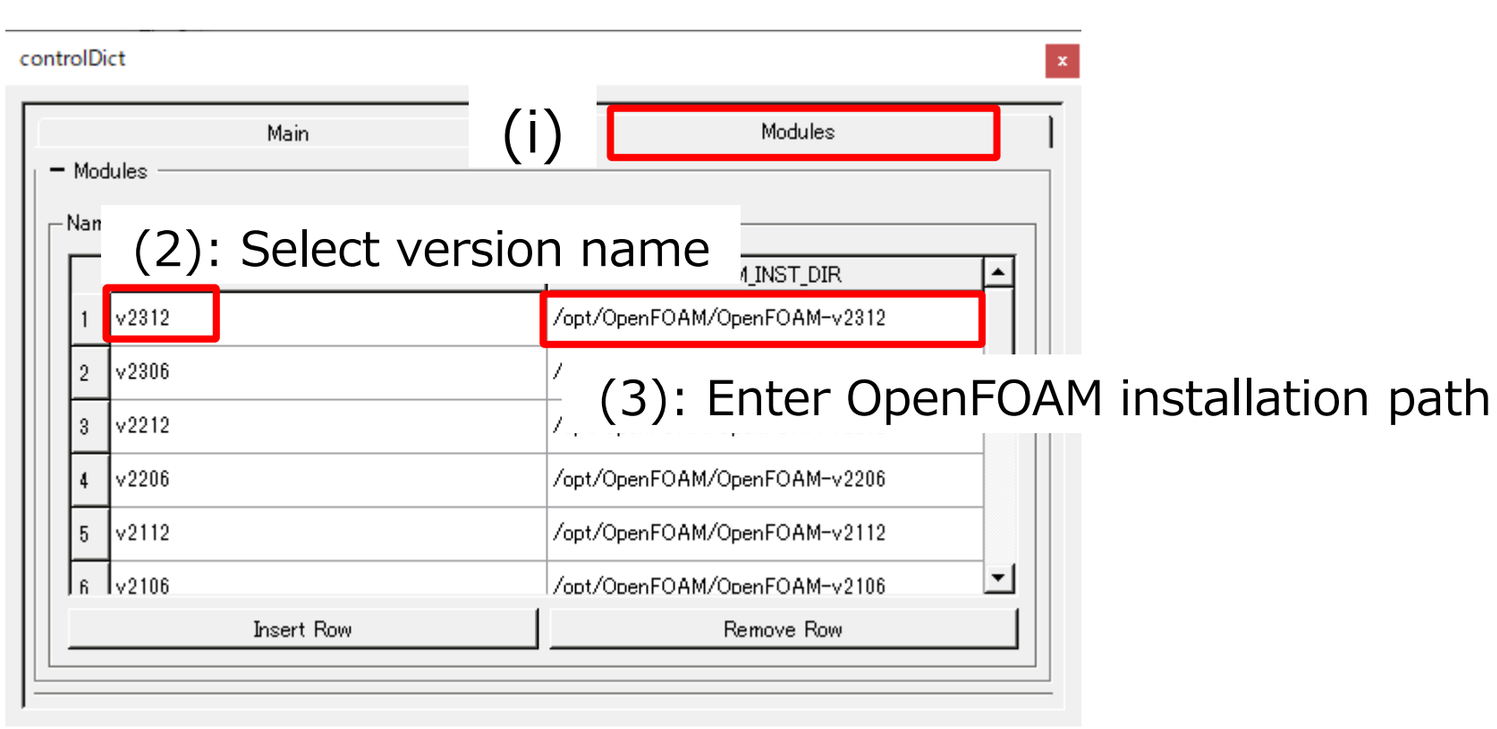
*3: Select the solver name to be used in the calculation.

The calculation method and solver name should be selected appropriately based on the analyst's understanding of the type of calculation to be performed.

Overset mesh analysis

controlDict panel > Modules tab

- Enter the installation location of the solver program.
- OpenFOAM solvers can be run directly from ennovaCFD by setting up OpenFOAM in a Linux or Windows 10 environment with WSL installed, and setting the version name and OpenFOAM installation folder in this panel.
- See pages 66 and 67 for how to perform the calculations.

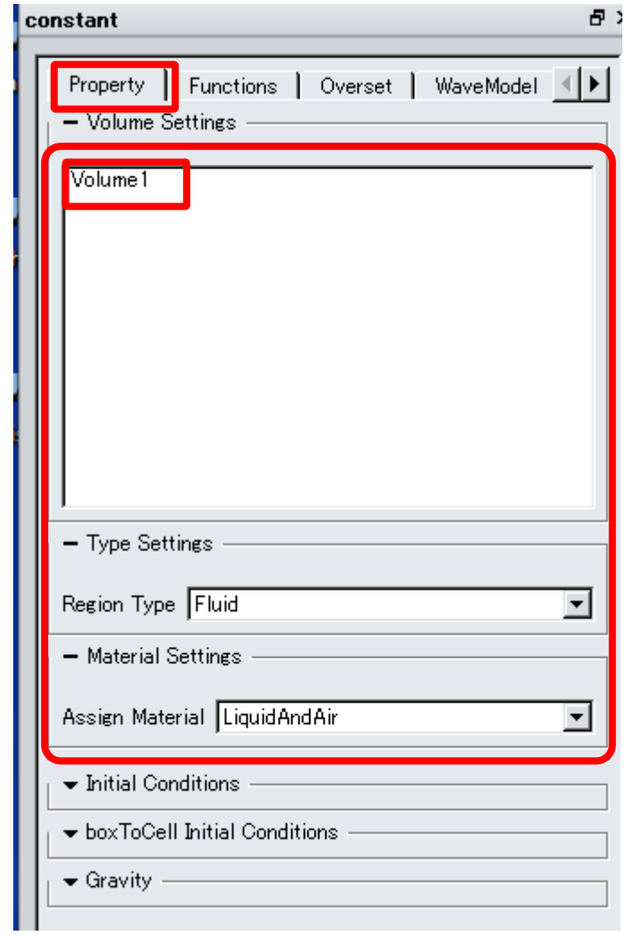


Overset mesh analysis

constant panel > Property tab

- Set the physical properties.
- Volume Settings
  - Area name : Volume1
- Type Settings
  - Region Type　: Fluid
- Material Settings
  - Assign Material　: LiquidAndAir

No configuration changes are required.

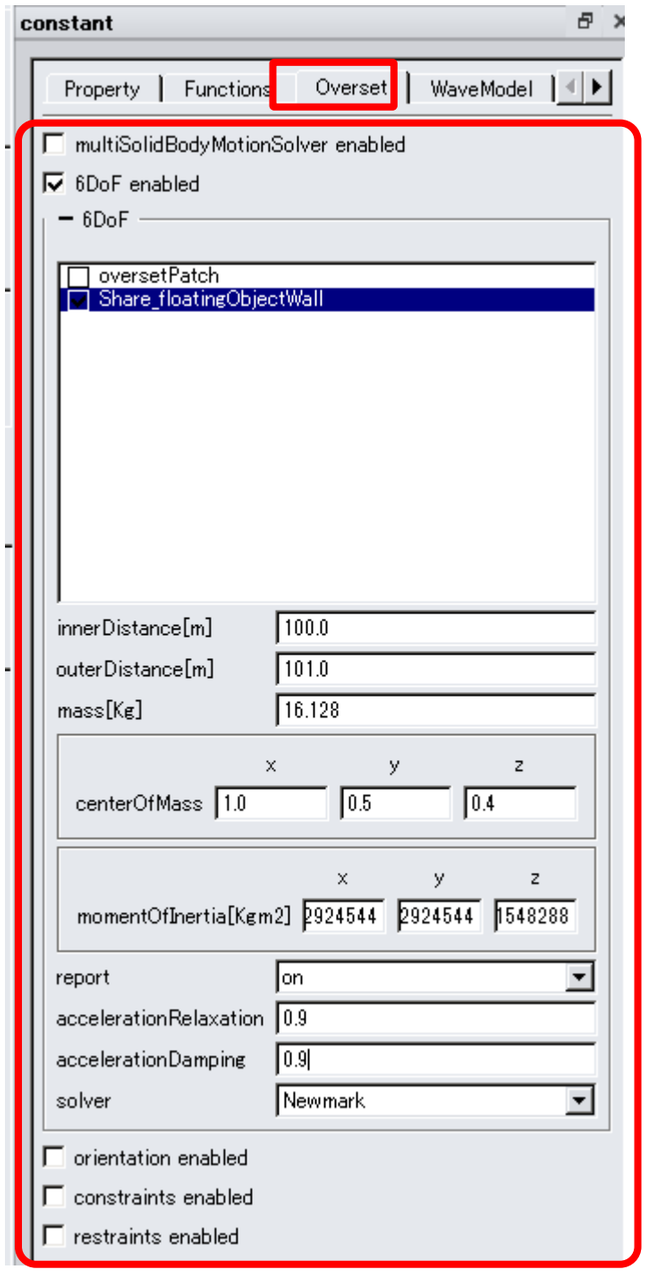


Overset mesh analysis

constant panel > Overset tab

- The 6-DOF motion is configured in the [Overset] tab.
- 6DoF enabled : checked on
- 6DoF
  - oversetPatch : checked off
  - Share_flotingObjectWall : Check ON
  - innerDistance[m] : 100.0
  - outerDistance[m] : 101.0
  - mass[Kg]: 16.128
  - centerOfMass(center of mass): x=1.0, y=0.5, z=0.4
  - momentOfInertia[kgm2] : x=0.2924544
    Principal moment of inertia: y=0.2924544
    z=0.1548288
  - report : on
  - accelerationRelaxation : 0.9
  - accelerationRelaxation : 0.9
  - solver : Newmark

Use defaults for all other settings except those above; see next page for settings for restraints (mooring cables)

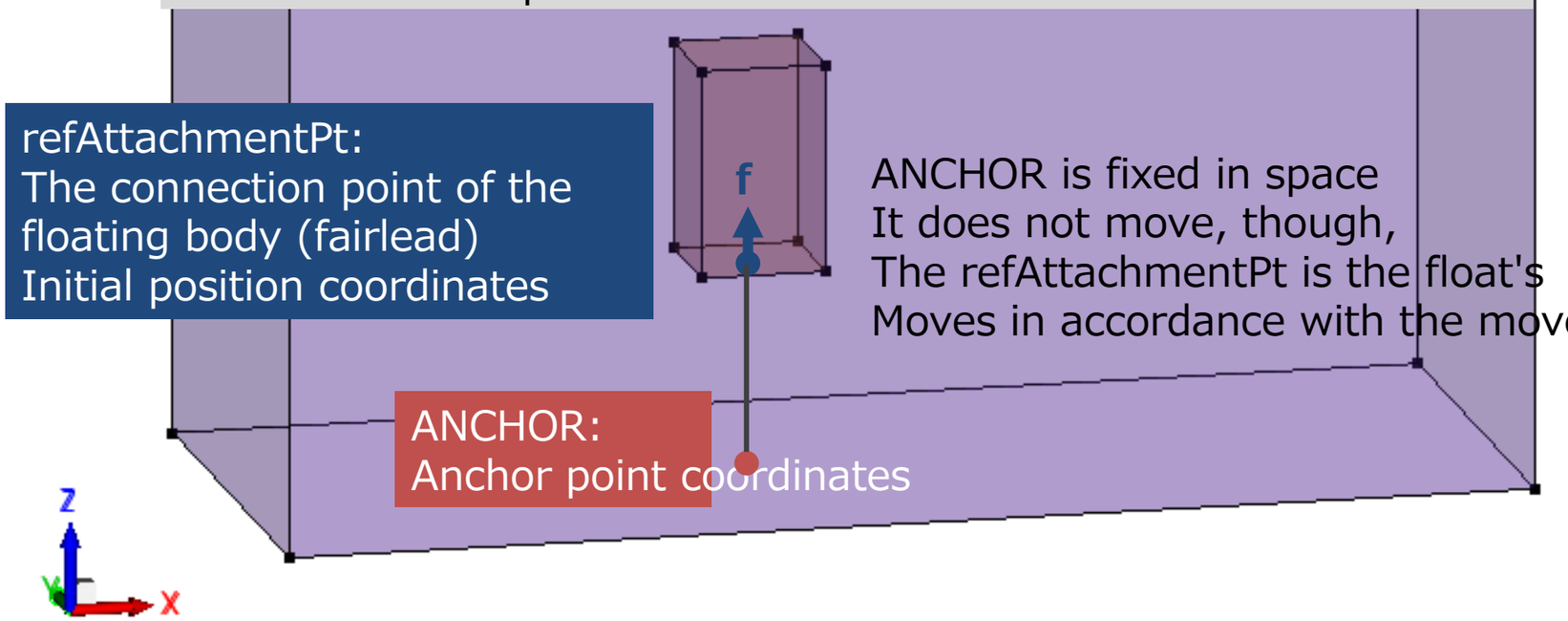


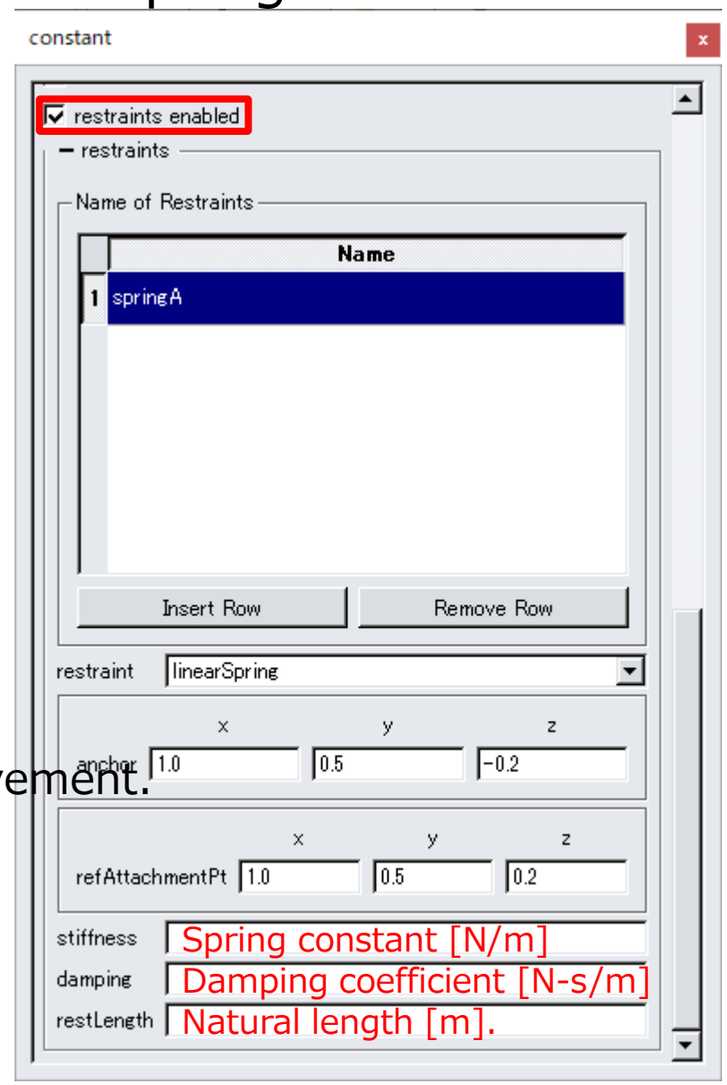
Tips] Modeling of mooring force acting on a floating body

The restraints model is used to model mooring forces. It is not used in this calculation.

- The linearSpring model, one of the restraints that can be configured from within the GUI, models the mooring force **F in terms of** spring force and damping by dampers.

F = spring force + damping force in linearSpring model
- Spring force = -stiffness × (R - restLength)**r**
- Damping force = -damping×(**r** v)**r**
- R: Linear distance between anchor and current refAttachmentPt
- r**:** Unit vector in the direction connecting the anchor and the current refAttachmentPt
- **v**: Movement speed of refAttachmentPt





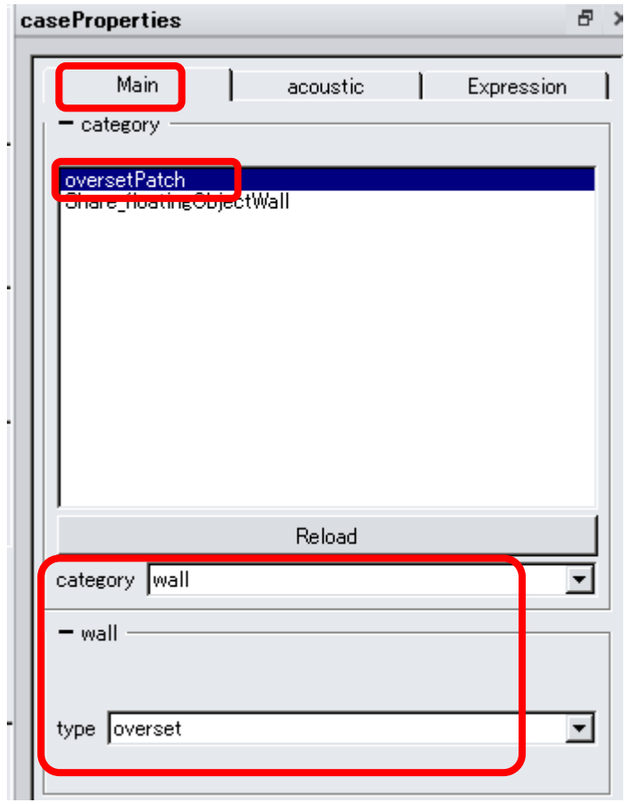
A more practical model of a catenary mooring cable is being developed as ennovaCFD ADD-ONS.

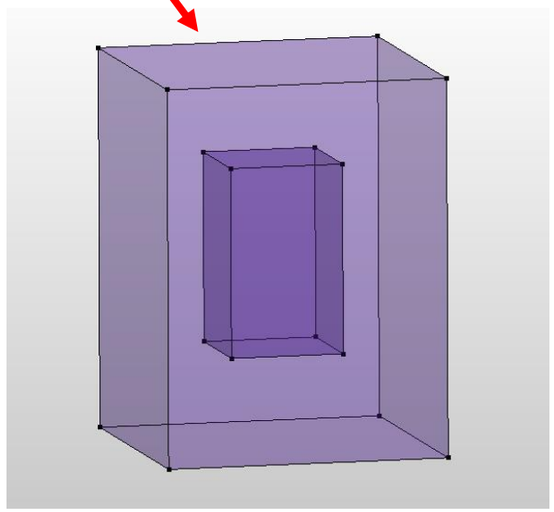
Overset mesh analysis

Boundary Conditions panel > Main tab

- OversetPatch sets the Overset wall.

- category
  - Boundaries : oversetPatch
  - category : wall
- Wall
  - type : overset



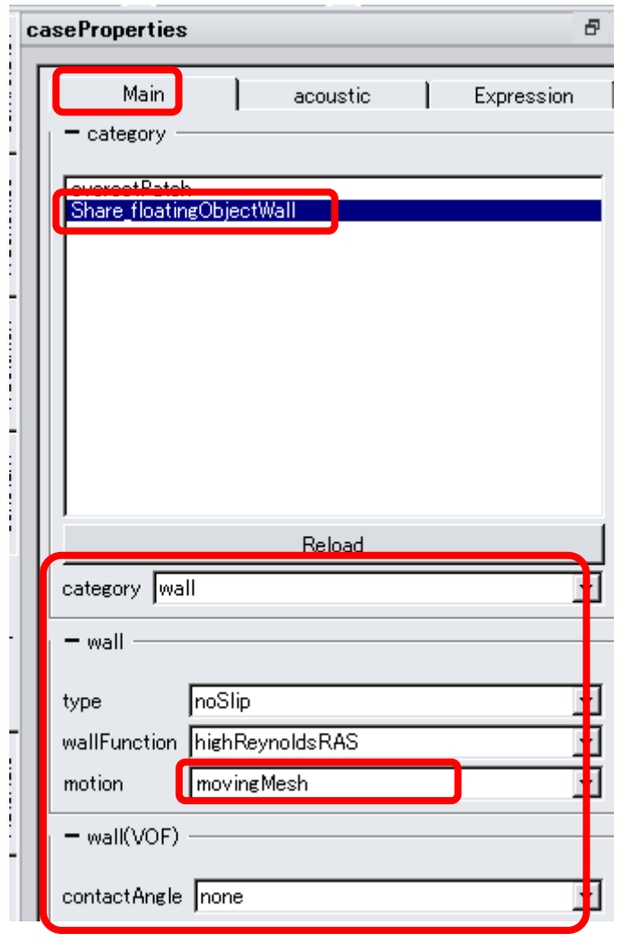


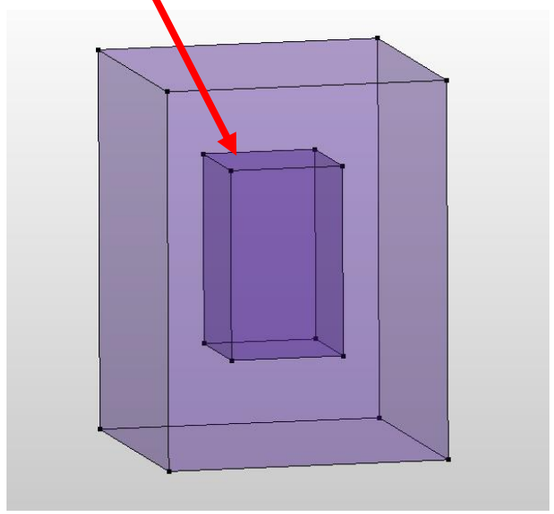
Overset mesh analysis

Boundary Conditions panel > Main tab

- share_flottingObjectWall sets the wall to be moved.

- category
  - Boundaries : share_flotingObjectWall
  - category : wall
- Wall
  - type : noSlip
  - wallFunction : highReynoldsRAS
  - motion : movingMesh
- Wall(VOF)
  - conatactAngle : none

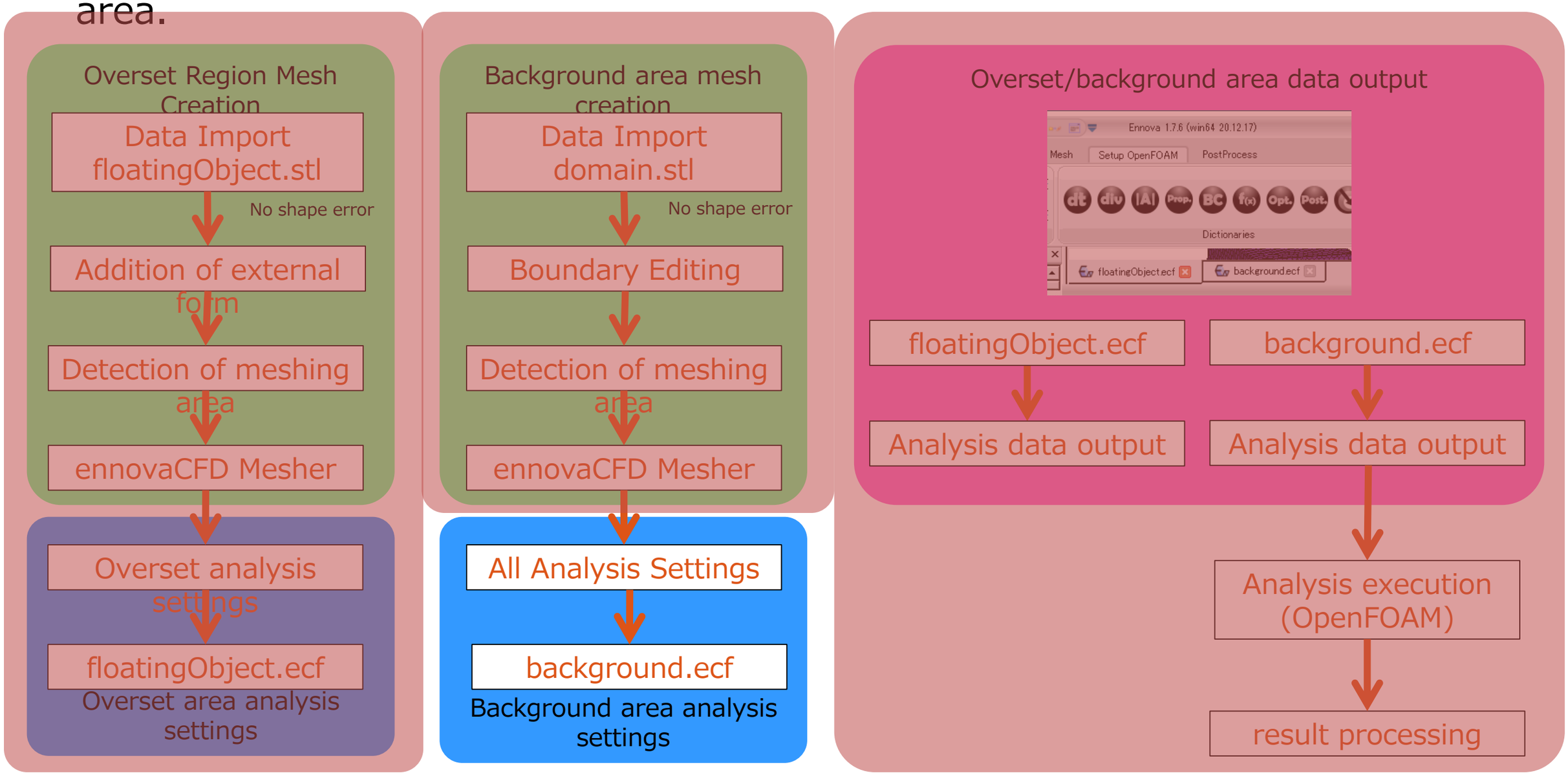




Overset mesh analysis

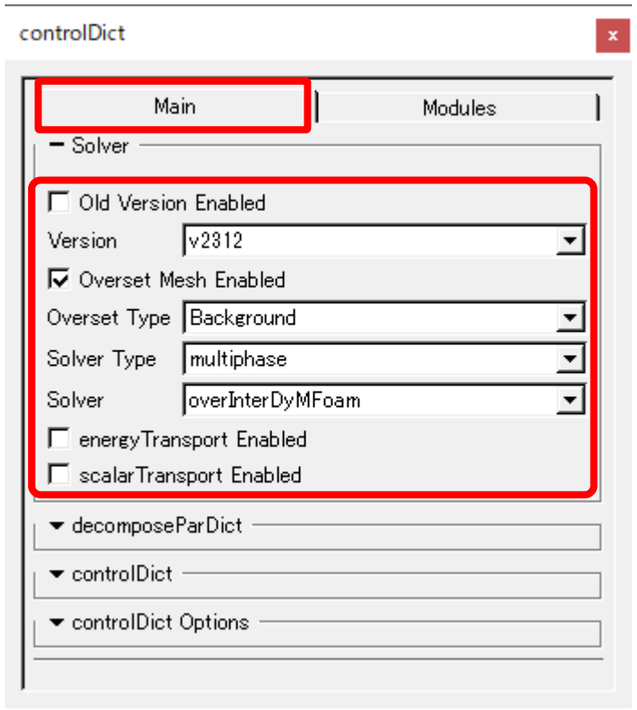
Background area analysis settings

- The analysis settings are applied to the mesh data of the created background area.



Overset mesh analysis

controlDict panel > Main tab



- Set up the solver version to be used and basic settings for calculations.
- Change the settings in red text.
- solver
  - Version ：OpenFOAM version
  - Overset Mesh Enabled ：Check ON
  - Overset Type ：Background
    Select Background because the background area is set.
  - Solver Type*2 ： multiphase
  - Solver*3 ： overInterDyMFoam(unsteady)
  - energyTransport Enabled：checked off
  - scalarTransport Enabled：checked off

*1: Select the version of OpenFOAM to be used in the calculation. Some functions may not be available depending on the version.

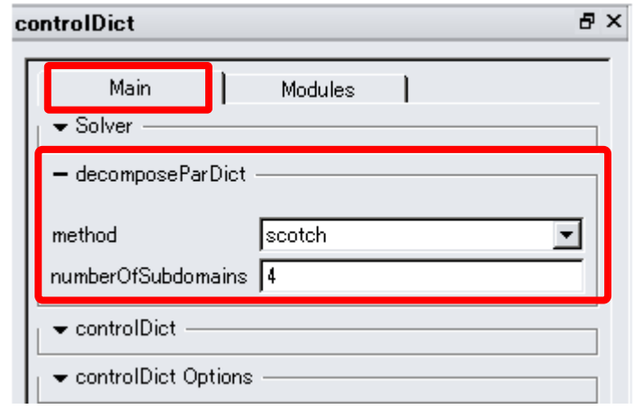
*2: Select the solver type.

*3: Select the solver name to be used in the calculation.
The calculation method and solver name should be selected appropriately based on the analyst's understanding of the type of calculation to be performed.

Continued on next page

Overset mesh analysis

controlDict panel > Main tab



- Set up parallel processing.

- decomposeParDict
  - Method (parallel partitioning method): Scotch

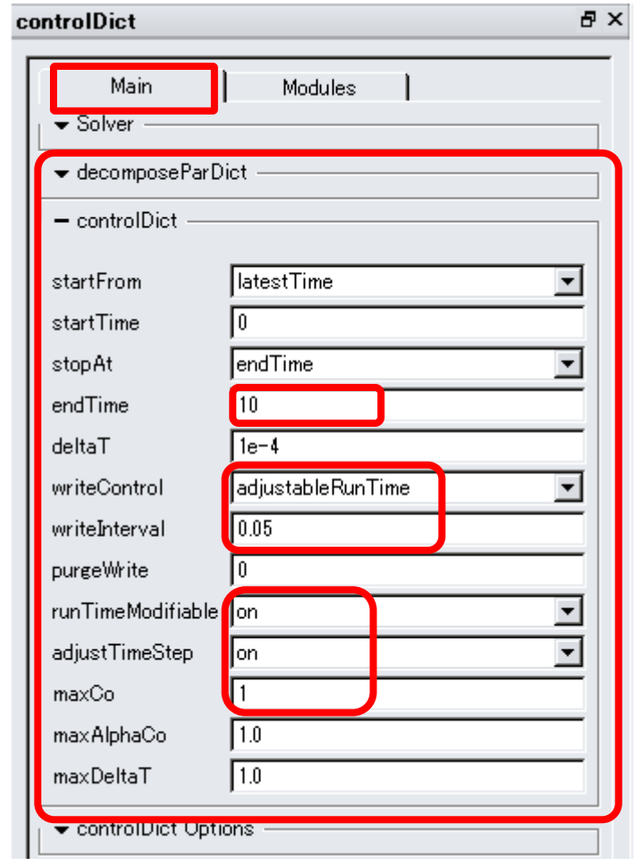
    How to divide the　mesh so that the numberof meshes is approximately the same for each CPU
  - HPC Num (number of area divisions): 4

    Specify the same number of CPUs　available for calculations

Continued on next page

Overset mesh analysis

controlDict panel > Main tab



- Set the iterations, frequency of output of results, etc.

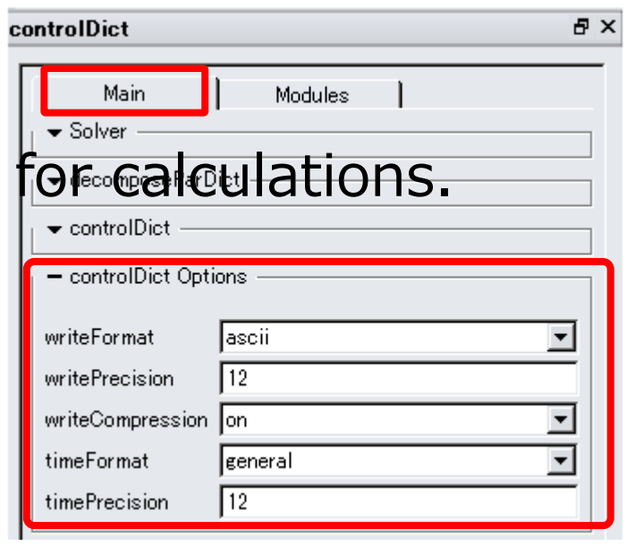
- controlDict
  - Start Form (calculation start method): latestTime
  - startTime (calculation start iteration): 0
  - stopAt : endTime
  - endTime (calculation end time): 10
  - deltaT (time increments): 1e-4
    For non-stationary calculations, adjust time to the phenomenon
  - writeControl(Result output method): adjustableRunTime
  - writeInterval(Result output frequency): 0.05
    If Run Time is selected, results are saved at the time entered
  - purgeWrite(Number of results saved): 0
    If the specified number of times is exceeded, old results are deleted and all "0" results are stored.
  - runTimeModifiable : on
    on reads controlDict for each iteration (reflected when settings are changed)
  - adjustTimeStep : on
  - maxCo (maximum coulomb number): 1.0
  - macxAlphaCo (max. number of alpha coolants): 1
  - maxDeltaT (maximum time increment): 1

Continued on next page

Overset mesh analysis

controlDict panel > Main tab

- Configure the solver version to be used and basic settings for calculations.



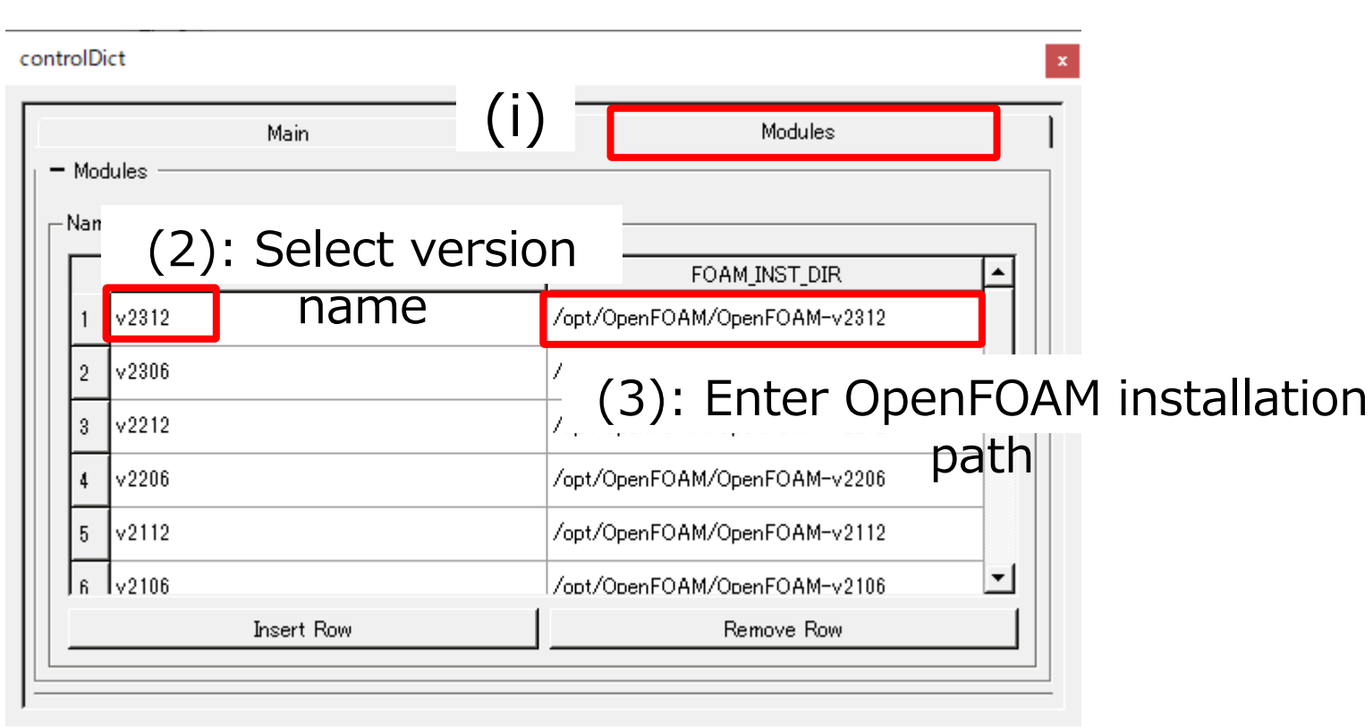
- contorlDict Options
  - writeFormat : ascii
  - writePrecision : 12
  - writeCompression : on
  - timeFormat : general
  - timePrecision : 12

Continued on next page

Overset mesh analysis

controlDict panel > Modules tab

- Enter the installation location of the solver program.
- OpenFOAM solvers can be run directly from ennovaCFD by setting up OpenFOAM in a Linux or Windows 10 environment with WSL installed, and setting the version name and OpenFOAM installation folder in this panel.
- See pages 66 and 67 for instructions on how to perform this.

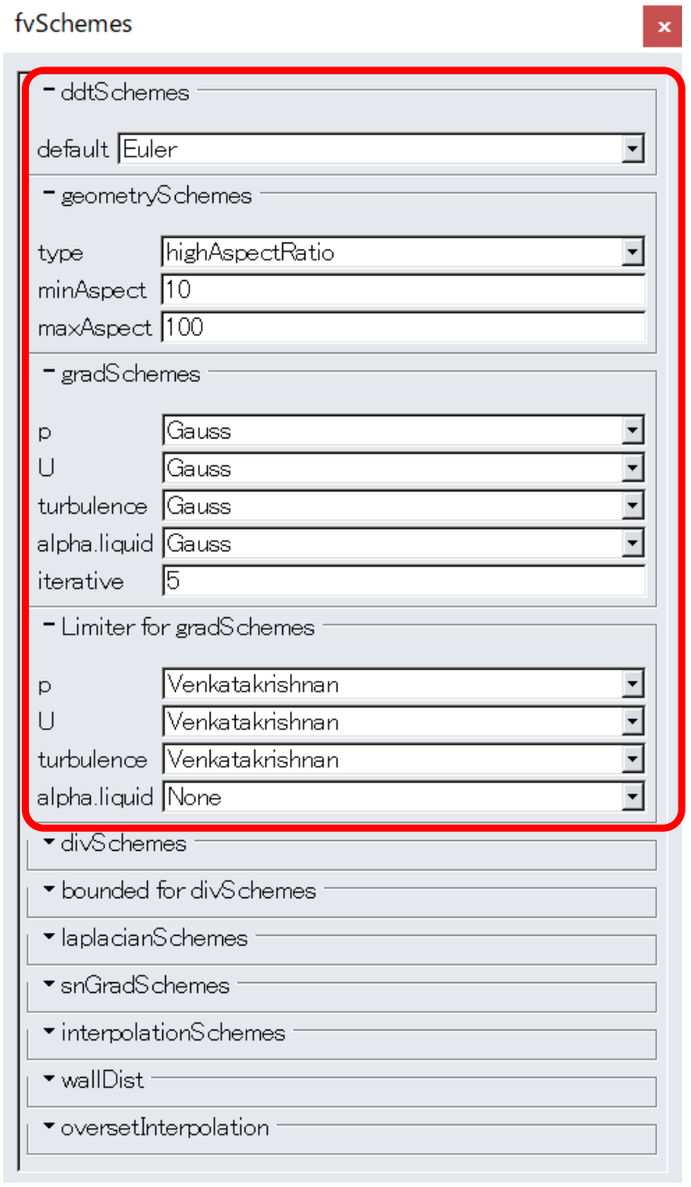


Overset mesh analysis

fvSchemes panel

- Sets the discrete scheme to be used for fluid calculations.
- Change the settings in red text.

- ddtSchemes(time term scheme)
  - default : Euler
- geometrySchemes (gradient term schemes)
  - type : highAspectRatio
  - minAspect : 10
  - maxAspect : 100
- gradSchemes (gradient term schemes)
  - p : Gauss
  - U : Gauss
  - turbulence : Gauss
  - alpha.liquid : Gauss
- Limiter for gradSchemes
  - p : Venkatakrishnan
  - U : Venkatakrishnan
  - turbulence : Venkatakrishnan
  - alpha.liquid : None

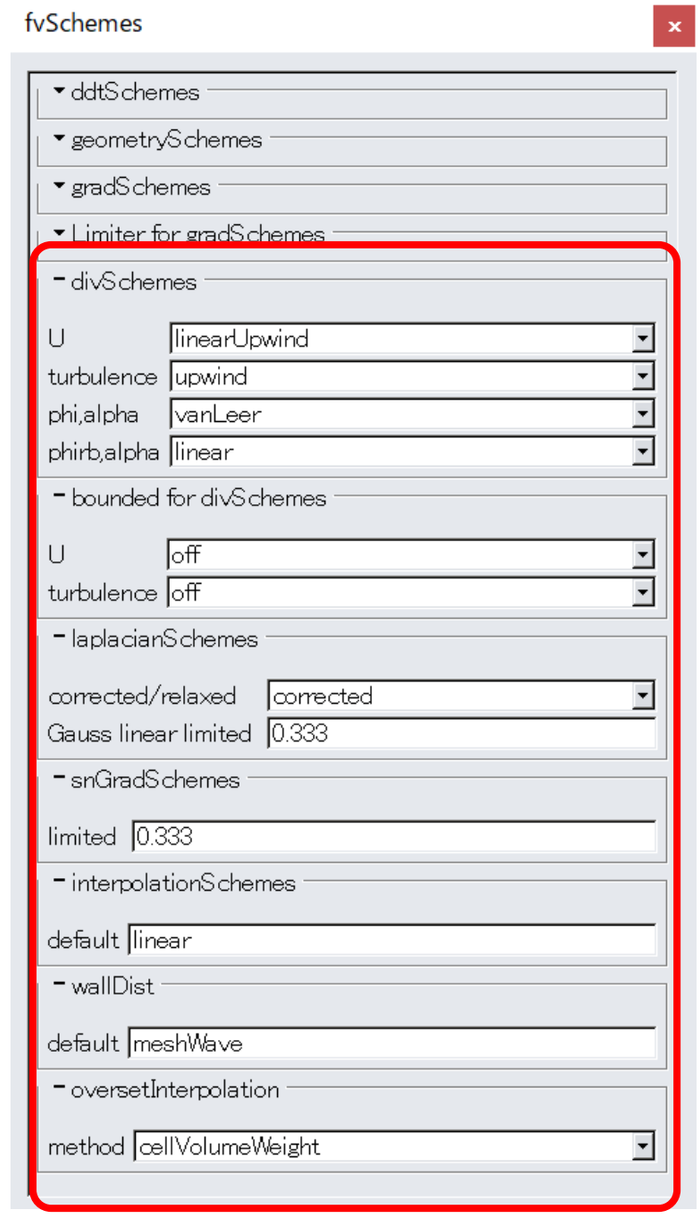


Overset mesh analysis

fvSchemes panel (Cont.)

- Sets the discrete scheme to be used for fluid calculations.

- divSchemes
  - U : linearUpwind
  - turbulence : upwind
  - phi.alpha : vanLeer
  - phirb.alpha : linear
- bounded for divSchemes
  - U : off
  - turbulence : off
- laplacianSchemes
  - corrected/relaxed : corrected
  - Gauss liner limited corrected : 0.333
- snGradSchemes
  - limited : 0.333
- interpolationSchemes
  - default : linear
- wallDist
  - default : meshWave
- OversetInterpolation (Overset interpolation scheme)
  - method : cellVolumeWeight

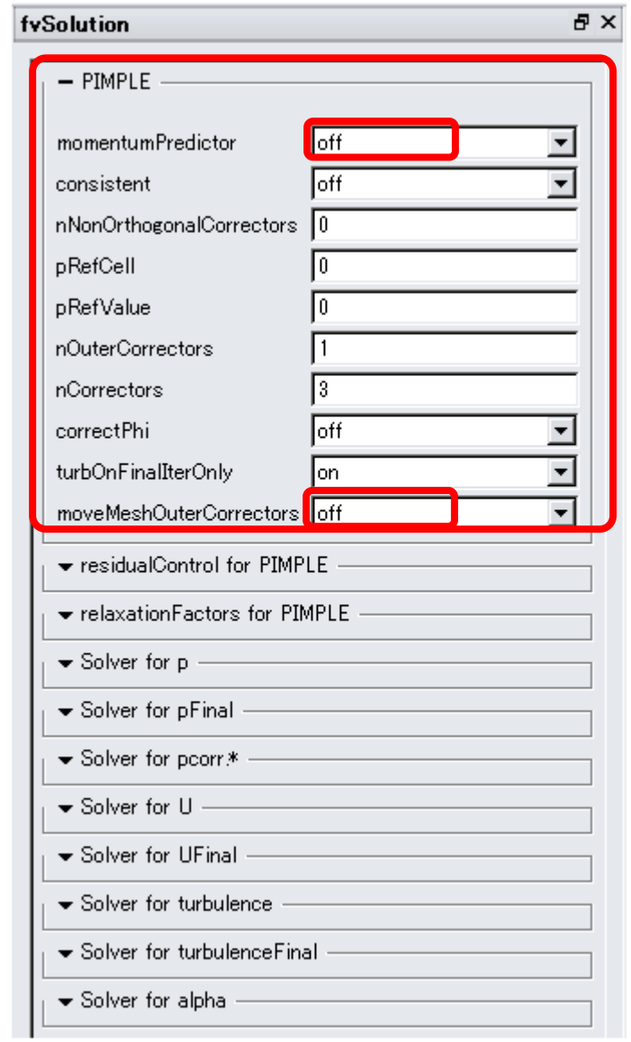


Overset mesh analysis

fvSolution Panel

- Set up the PIMPLE algorithm and linear solver for each variable.

- PIMPLE.
  - momnentumPredictor : off
  - consistent : off
  - nNonOrthogonalCorrectors : 0
  - pRefCell : 0
  - pRefValue : 0
  - nOuterCorrectors : 1
  - nCorrectors : 3
  - correctPhi : off
  - turbOnFinalInterOnly : on
  - moveMeshOuterCorrectors : off

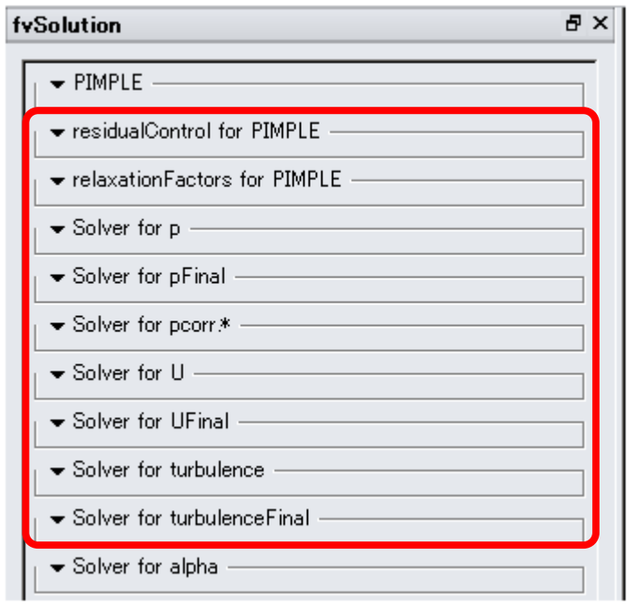


Continued on next page

Overset mesh analysis

fvSolution panel (Cont.)

- Set up a linear solver for each variable.
- The following items are not changed because they use default settings.



- residualContorl for PIMPLE
- residualFactors for PIMPLE
- Solver for p
- Solver for pFinal
- Solver for U
- Solver for UFinal
- Solver for turbulence
- Solver for turbulenceFinal

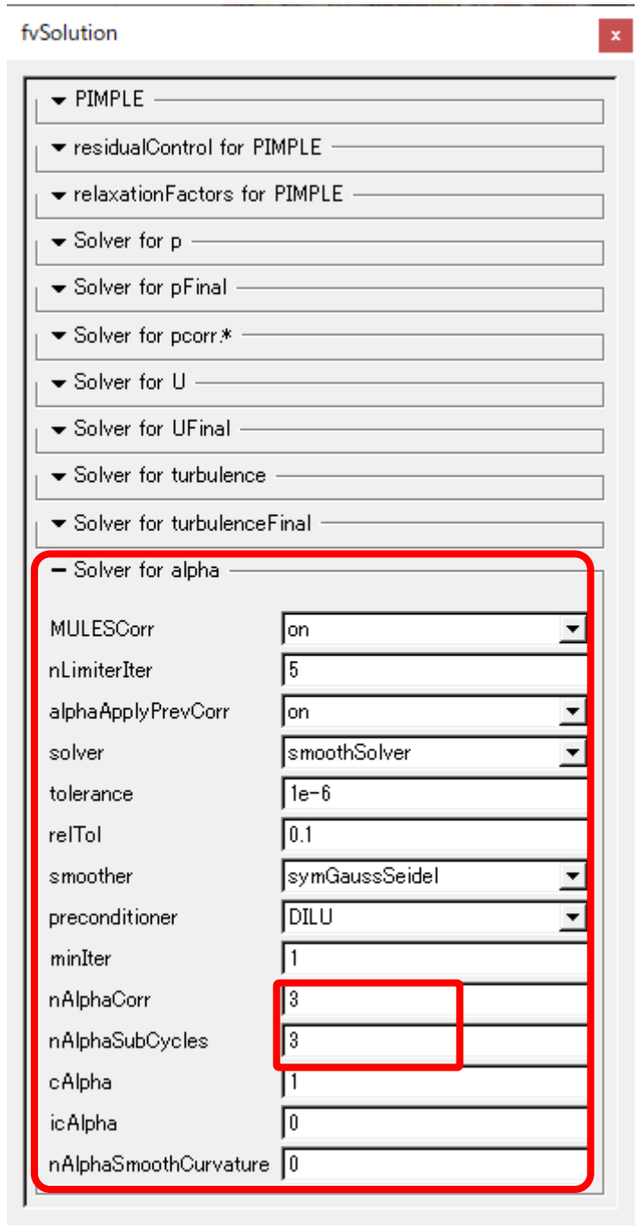
Continued on the next page.

Overset mesh analysis

fvSolution panel (Cont.)

- Set up a linear solver for each variable.
- Change the settings in red text.

- Solver for alpha
  - MULESCorr : on
  - nLimiterIter : 5
  - alphaApplyPrevCorr : on
  - solver : smoothSolver
  - tolerance : 1e-6
  - relTol : 0
  - smoother : symGaussSeidel
  - preconditioner : DILU
  - minIter : 1
  - nAlphaCorr : 3
  - nAlphaSubCycles : 3
  - cAlpha : 1
  - icAlpha : 0
  - nAlphaSmoothCurvature : 0



Overset mesh analysis

Physical properties used in the analysis

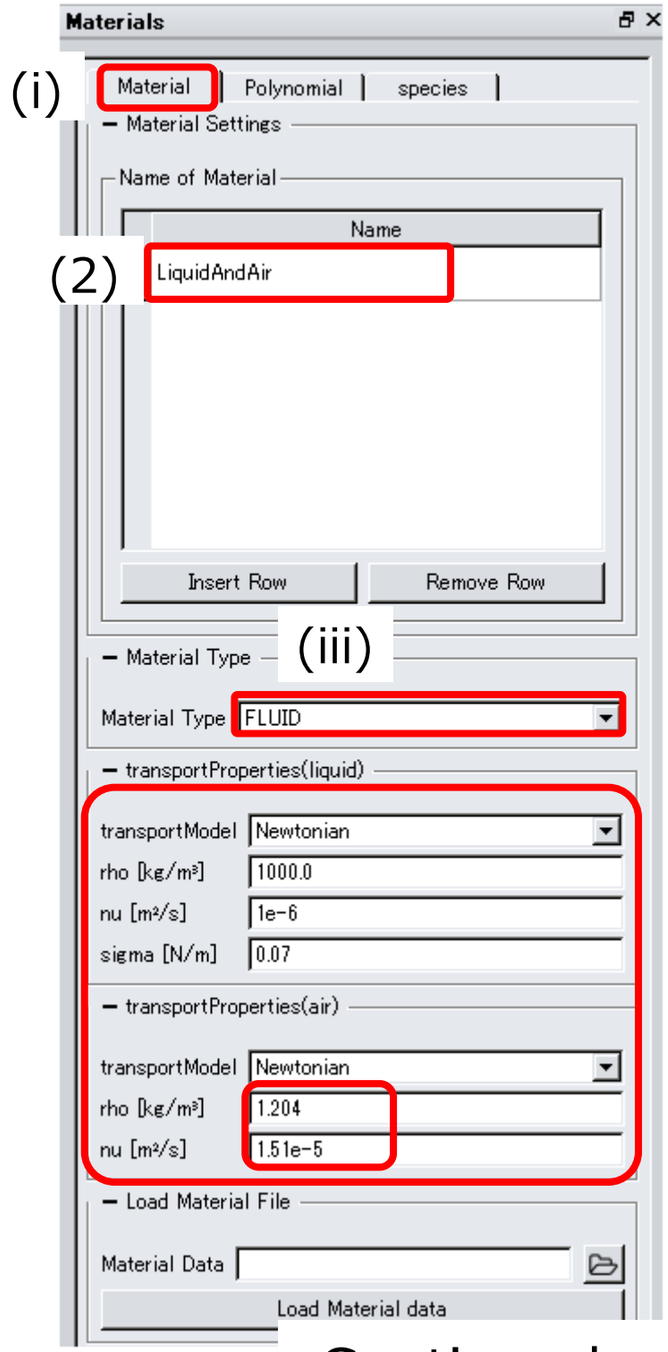
- Since this analysis is incompressible, kinematic viscosity is used for viscosity.

Liquid: water		**remarks**
density	1000 [kg/m ^{3}]	Equivalent to water 20°C
kinematic viscosity	1.0e-6[m^2 /s]	Equivalent to water 20°C (Kinematic viscosity: viscosity/density)
Gas: Air		**remarks**
density	1.204[kg/m ^{3}]	Equivalent to 20°C air
kinematic viscosity	1.51e-5[m^2 /s]	Equivalent to 20°C air (Kinematic viscosity: viscosity/density)

Overset mesh analysis

Constant panel > Material tab

- Set physical properties.
- Material Settings
  - Name of Material : LiquidAndAir
  - Material Type : FLUID
- tranportProerties(liquid)/liquid
  - transportModel : Newtonian
  - rho[kg/m^3]: 1000.0
  - nu[m^2 /s]: 1e-6
  - sigma[N/m] : 0.07
- tranportProerties(air)/gas
  - transportModel : Newtonian
  - rho[kg/m^3]: 1.204
  - nu[m^2 /s]: 1.51e-5



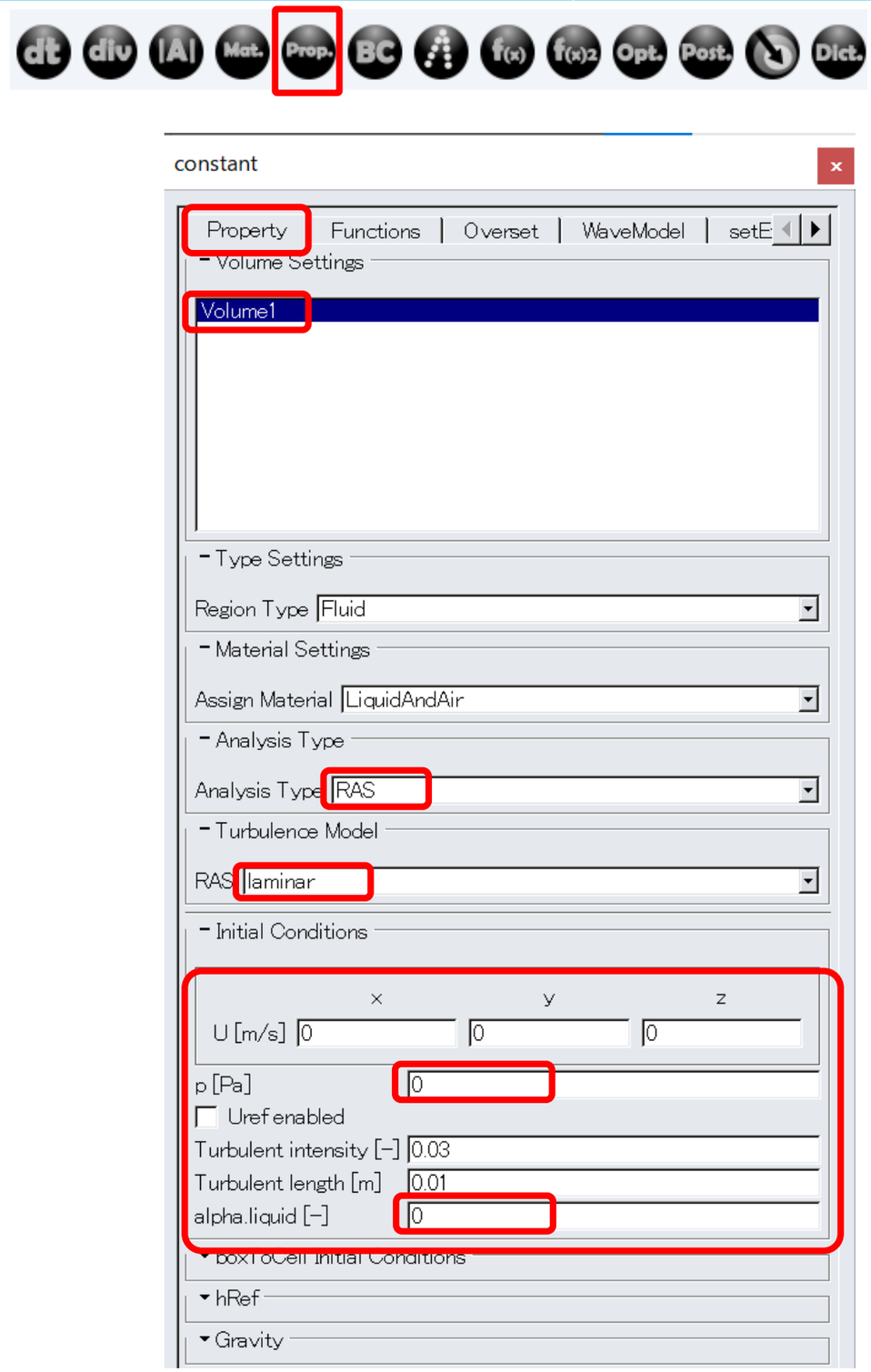
Continued on next page

Overset mesh analysis

constant panel > Property tab

- Set physical properties.
- Volume Settings
  - Area name : Volume1
- Type Settings
  - Region Type : Fluid
- Material Settings
  - Assign Material : LiquidAndAir
- Analysis Type
  - Analysis Type : RAS
- Turbulence Model
  - RAS : laminar

  Select Laminar Flow Calculation
- Initianl Conditions
  - U[m/s] : 0 0 0
  - p[Pa] : 0
  - Turbulent intensity[-] : 0.03
  - Turbulent length[m] : 0.01
  - alpha liquid[-] : 0 (specifies 100% air in the area)



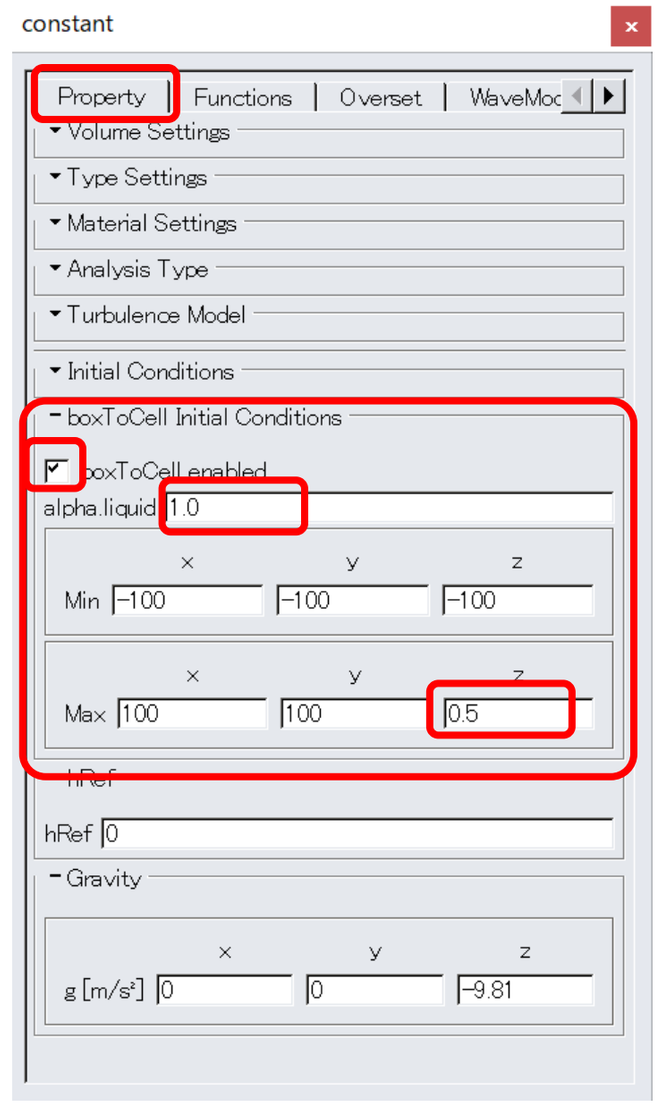
Continued on next page

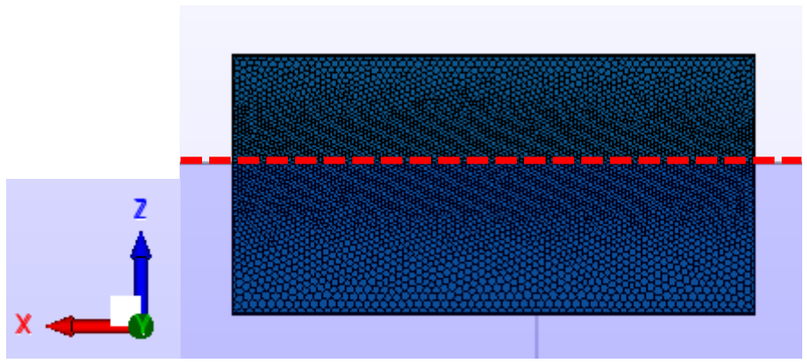
Overset mesh analysis

constant panel > Property tab

- Set physical properties.
- boxToCell Initianl Conditions(Initial conditions by rectangle)

  Specify the area where the liquid phase exists at the initial time as a rectangle.
  - boxToCell enabled : checked on
  - alpha.liquid : 1.0 (water)
  - Min : x=-100, y=-100, z=-100
  - Max : x=100, y=100, z=0.5
- Gravity (direction of gravity)
  - g[m^2 /s]: 0 0 -9.81



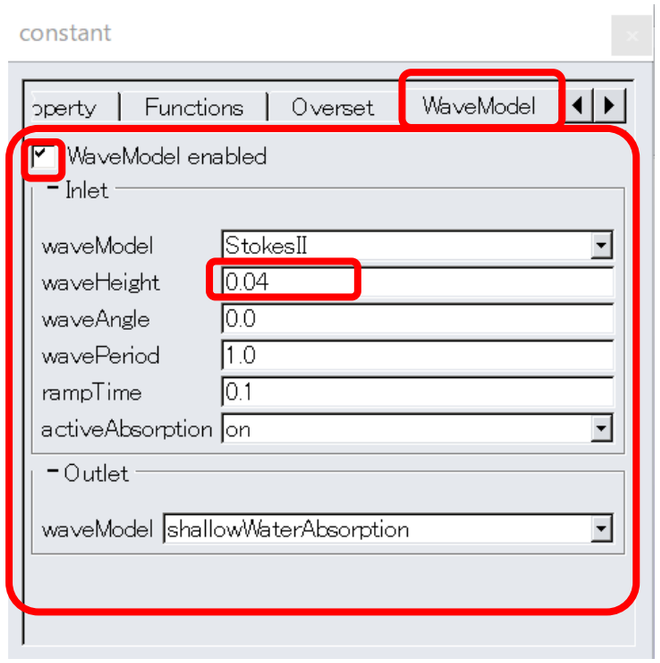


Z=0.5

Overset mesh analysis

constant panel > WaveModel tab

- Set up the wave model.
- WaveModel enabled : checked on
- Inlet
  - waveModel ： StokesII
  - waveHeight ：0.04
  - waveAngle ：0.0
  - wavePeriod ： 1.0
  - rampTime ：0.1
  - activeAbsorption ：on
- Outlet
  - waveModel ：shallowWaterAbsorption

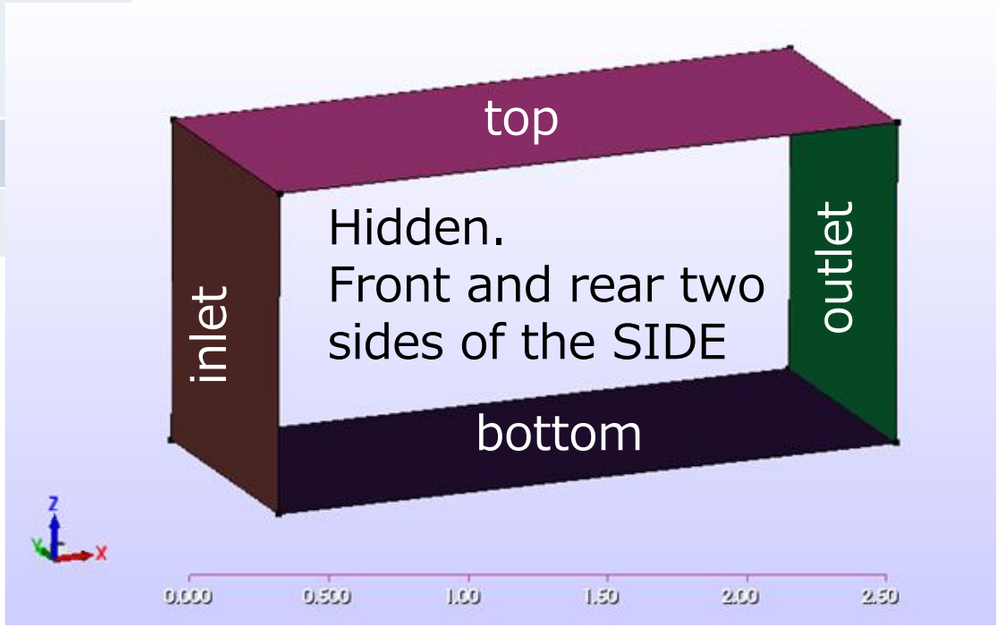


Overset mesh analysis

Boundary Conditions panel

- The boundary conditions to be set are summarized below.
  - For solvers that solve for multiple pressures, the unit of pressure is [Pa].

part	Group Name	boundary condition
entrance requirement	inlet	speed-determining entrance boundary Air flows into the analysis area at 0 m/s
Exit Conditions	outlet top	Pressure-defined outlet boundary p=0[Pa]. Air is discharged from the outlet
wall	bottom	Wall of Resistance
symmetry	side	symmetry boundary

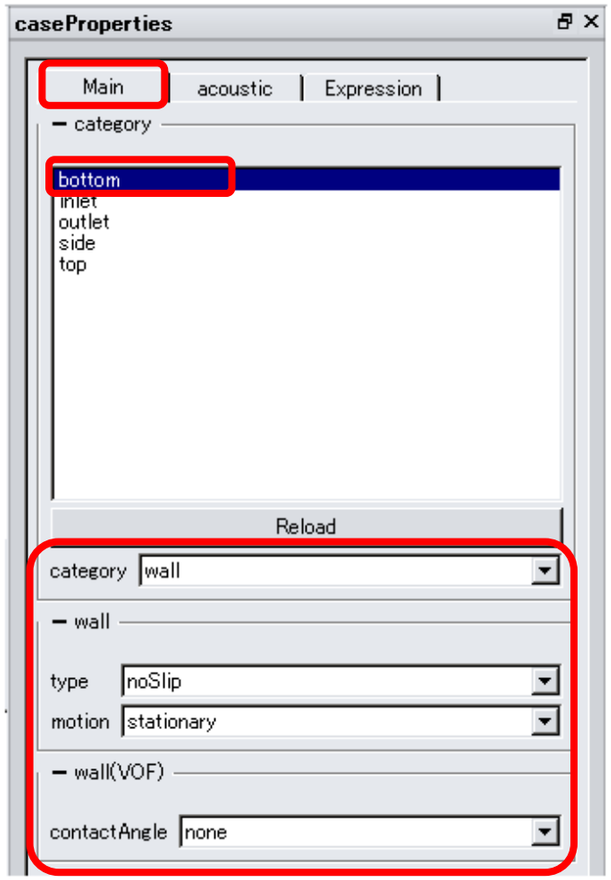


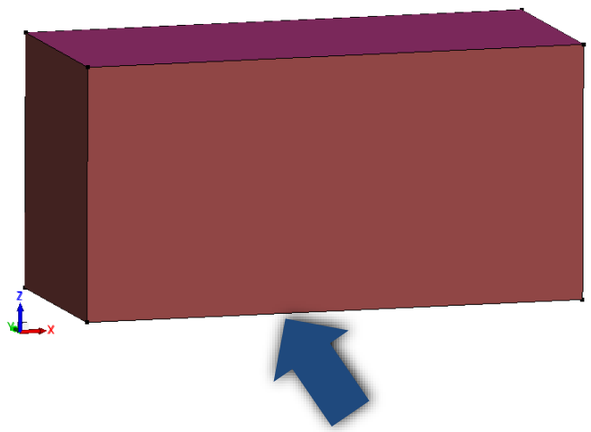
Overset mesh analysis

Boundary Conditions panel > Main tab

- BOTTOM sets the wall.

- category
  - Boundaries ：bottom
  - category：wall
- Wall
  - type：noSlip
  - motion：stationary
- Wall(VOF)
  - conatactAngle ：none



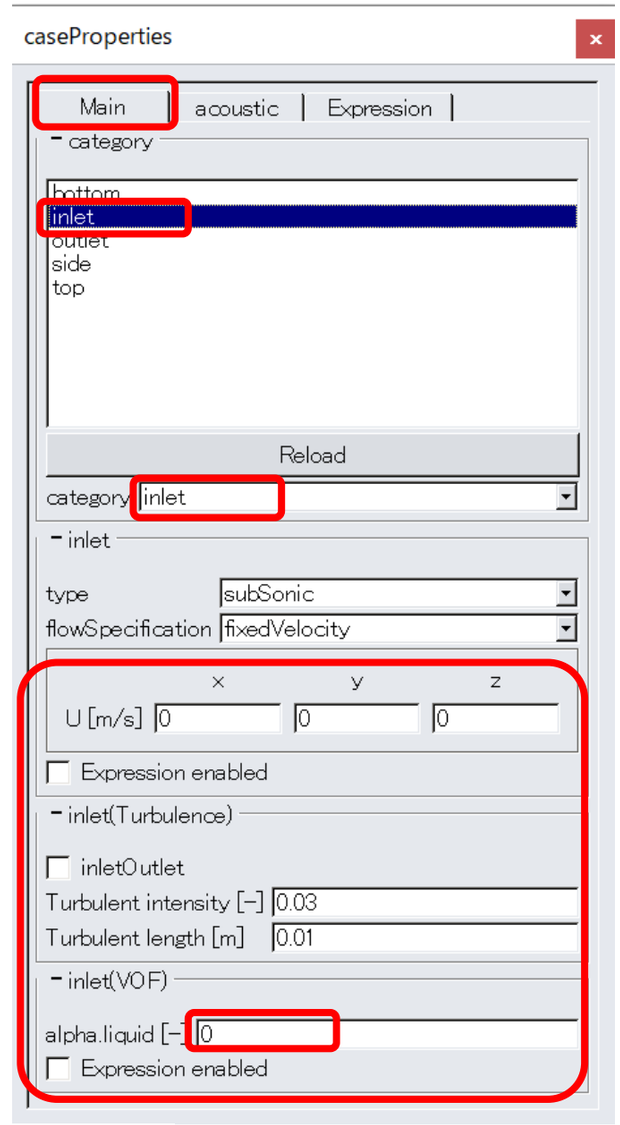


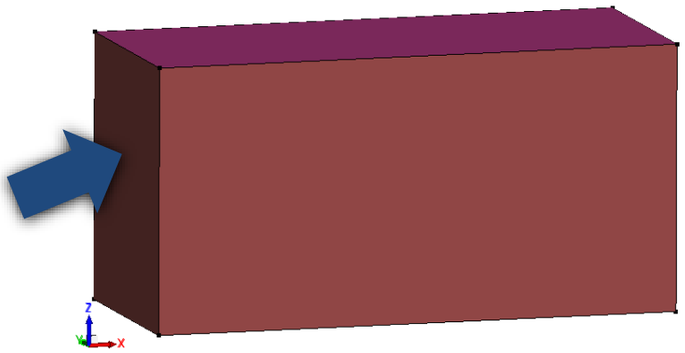
Overset mesh analysis

Boundary Conditions panel > Main tab

- INLET sets the inlet speed condition.

- category
  - Boundary Name : inlet
  - category : inlet
- inlet
  - type : subSonic
  - returnFlow : fixedVelocity
  - U[m/s] : 0 0 0
- inlet(Turbulence)
  - inletOutlet : checked off
  - Turbulent intensity[-]: 0.03
  - Turbulent length[m] (length scale): 0.01
- inlet(VOF)
  - alphaliquid[-] inflow physical property (air) : 0



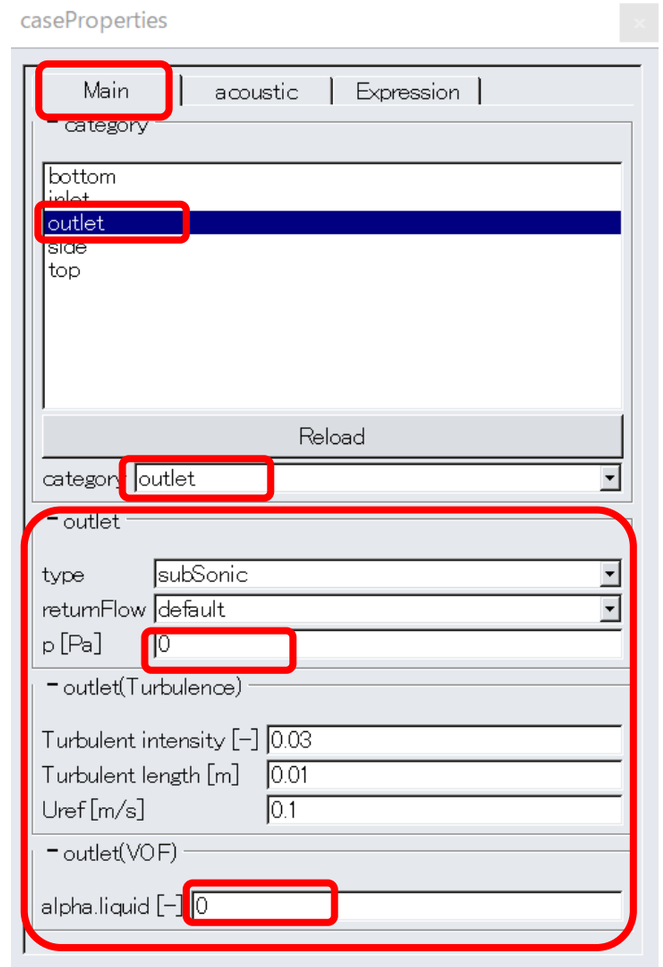


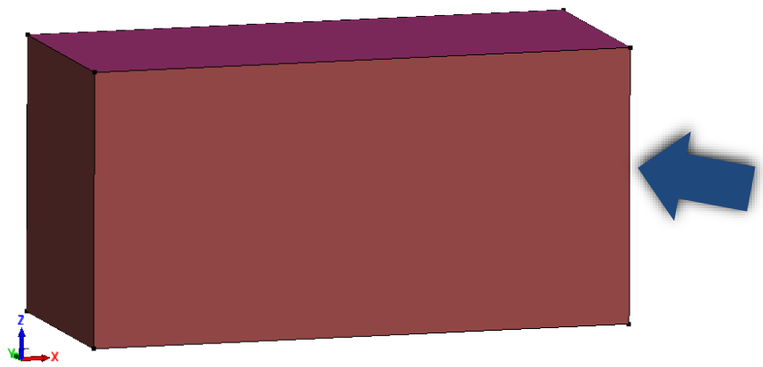
Overset mesh analysis

Boundary Conditions panel > Main tab

- OUTLET sets the outlet pressure condition.

- category
  - Boundary name: OUTLET
  - category : outlet
- outlet
  - type : subSonic
  - returnFlow : default
  - p[Pa] : 0
- outlet(Turbulence)
  - Turbulent intensity[-]: 0.03
  - Turbulent length[m] (length scale): 0.01
  - Uref[m/s] (reference speed): 0.1
- outlet(VOF)
  - alphaliquid[-] inflow physical property (air)



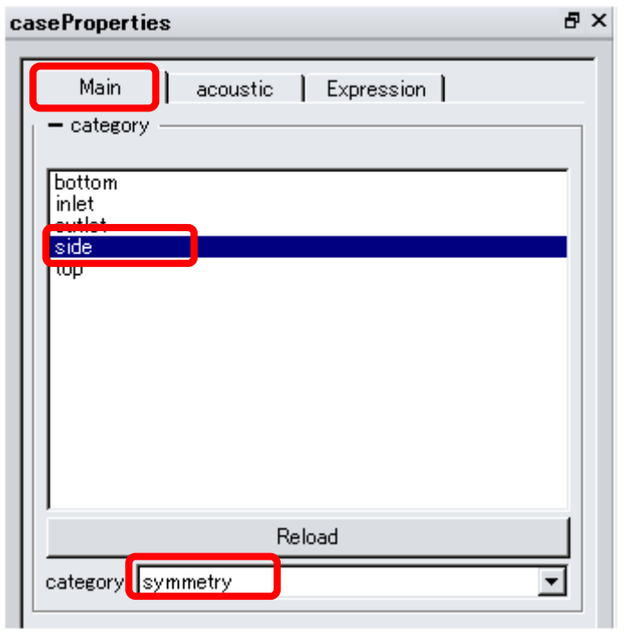


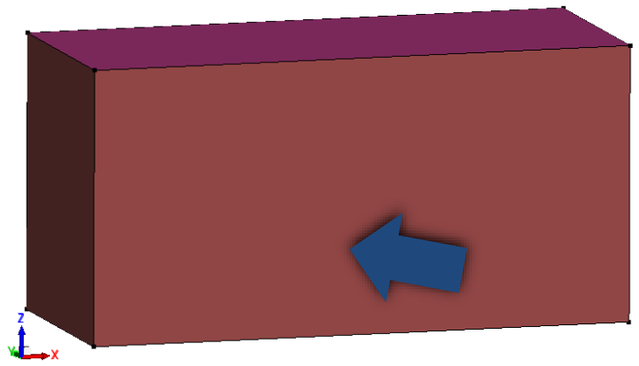
Overset mesh analysis

Boundary Conditions panel > Main tab

- side sets the symmetry condition.

- category
  - Boundary Name : side
  - category : symmetry



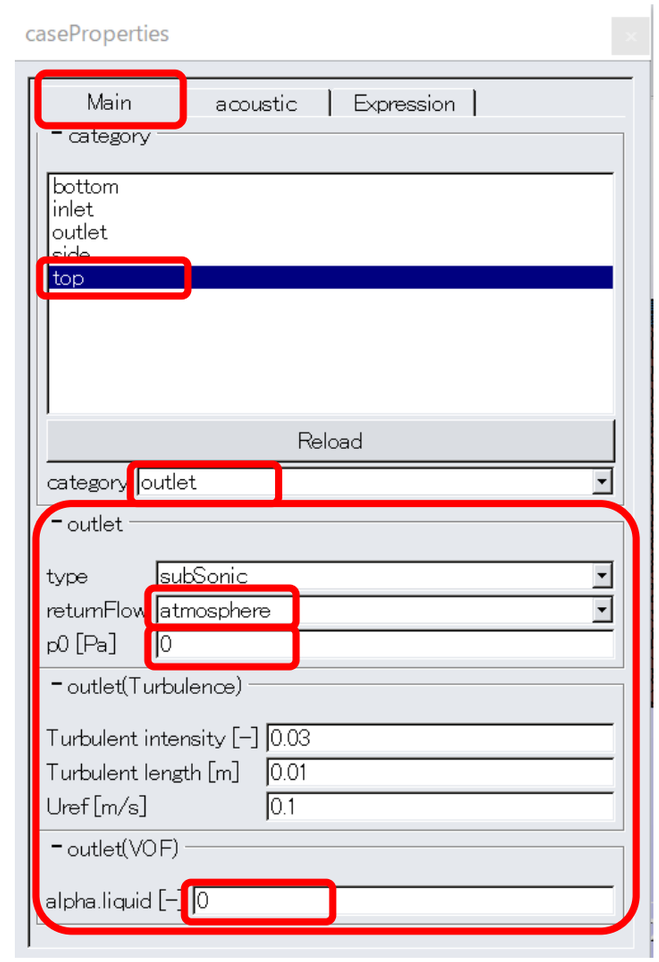


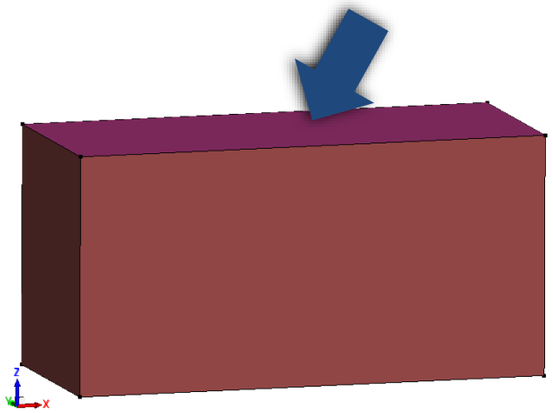
Overset mesh analysis

Boundary Conditions panel > Main tab

- top sets the conditions for atmospheric release.

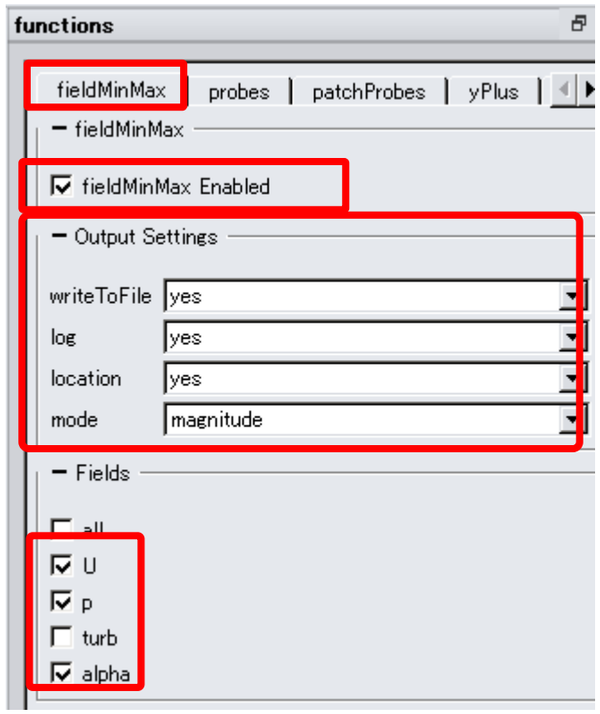
- category
  - Boundary Name : top
  - category : outlet
- outlet
  - type : subSonic
  - returnFlow : atmosphere
  - p[Pa] : 0
- outlet(Turbulence)
  - Turbulent intensity[-]: 0.03
  - Turbulent length[m] (length scale): 0.01
  - Uref[m/s] (reference speed): 0.1
- outlet(VOF)
  - alphaliquid[-] inflow physical property (air) :





Overset mesh analysis

function panel > fieldMinMax tab



- Sets a record of the maximum/minimum values in the calculation area during the calculation.

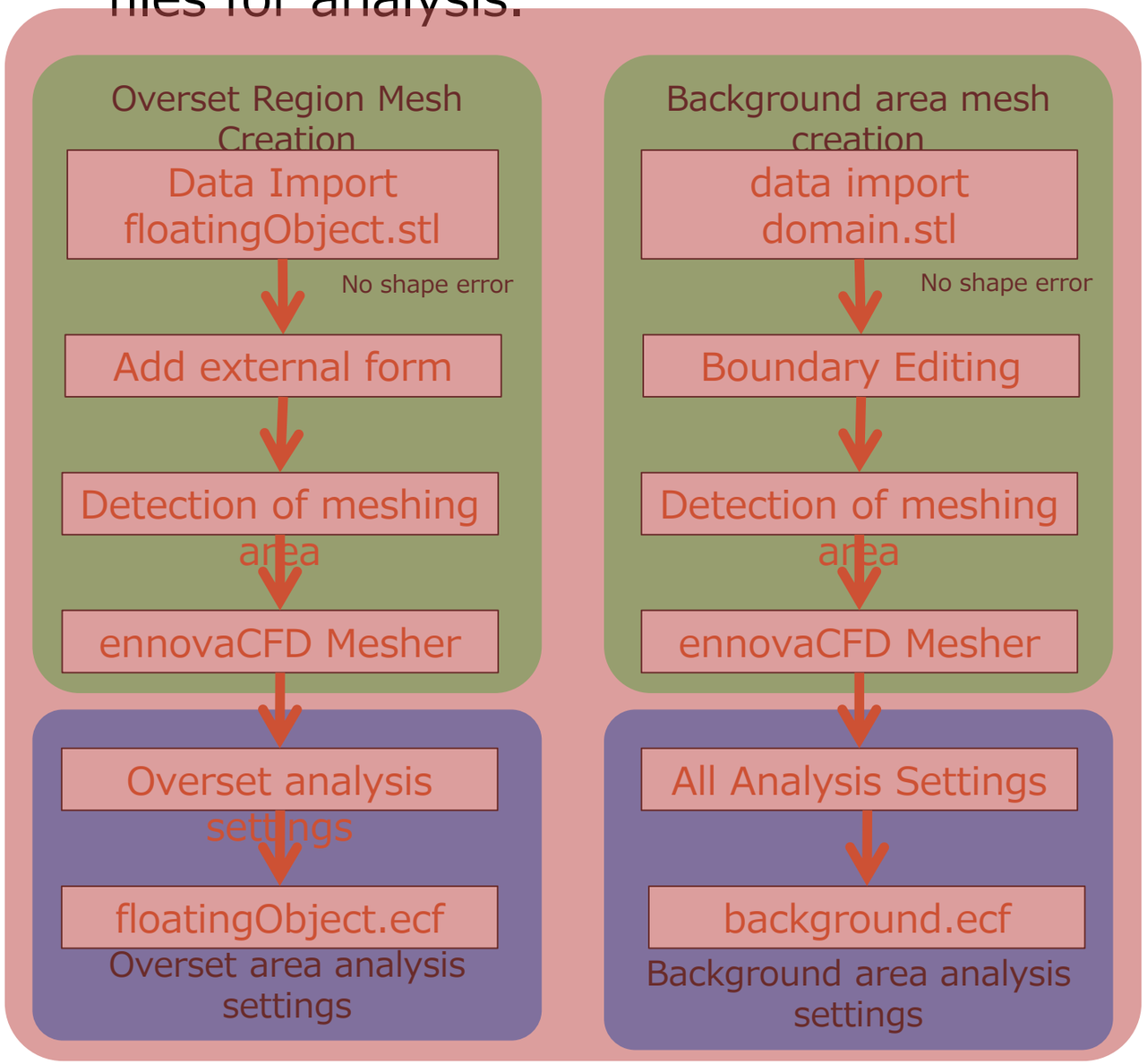
- filedMinMax Enabled : checked
- Output Settings
  - writeToFile : yes
  - log : yes
  - location : yes
  - mode : magnitude
- Fields.
  - U : Check on
  - p : Check on
  - alpha : check on

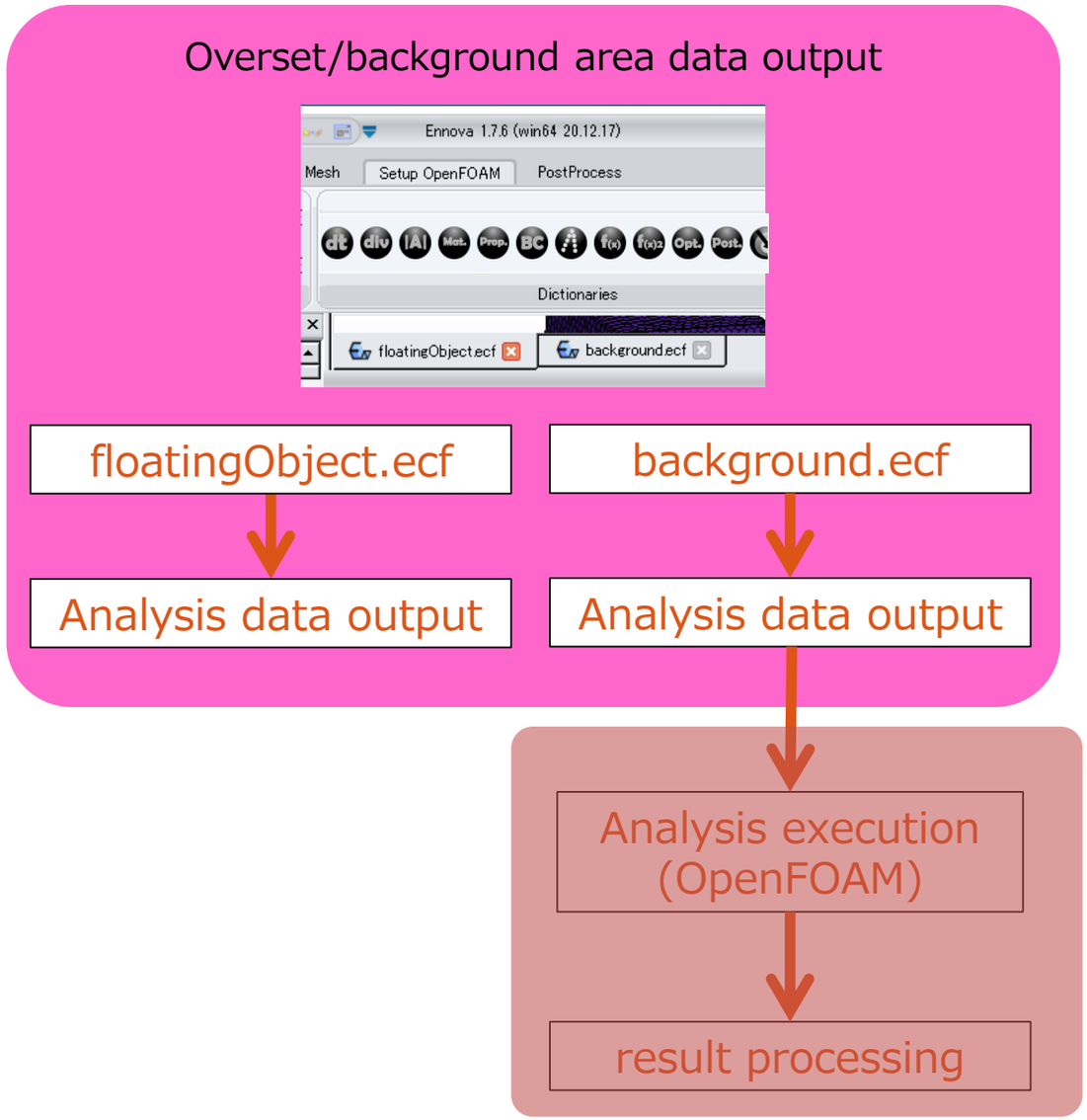
    Check the physical quantity to be acquired individually, since the all command may not be available for some OpenFOAM versions.

Overset mesh analysis

Overset/background area data output

- The data in the created OVERSET and BACKGROND areas are used to output files for analysis.

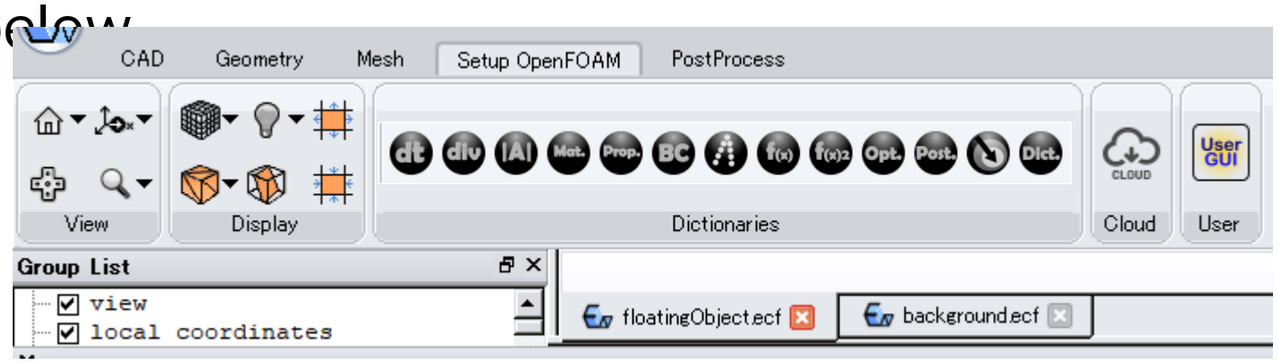




Overset mesh analysis

Overset/background area data output

- With the flotingObject.ecf file in the coverset area open, load the background.ecf file in the background area.
- There should be two tabs on the GUI of ennovaCFD as shown in the figure below.

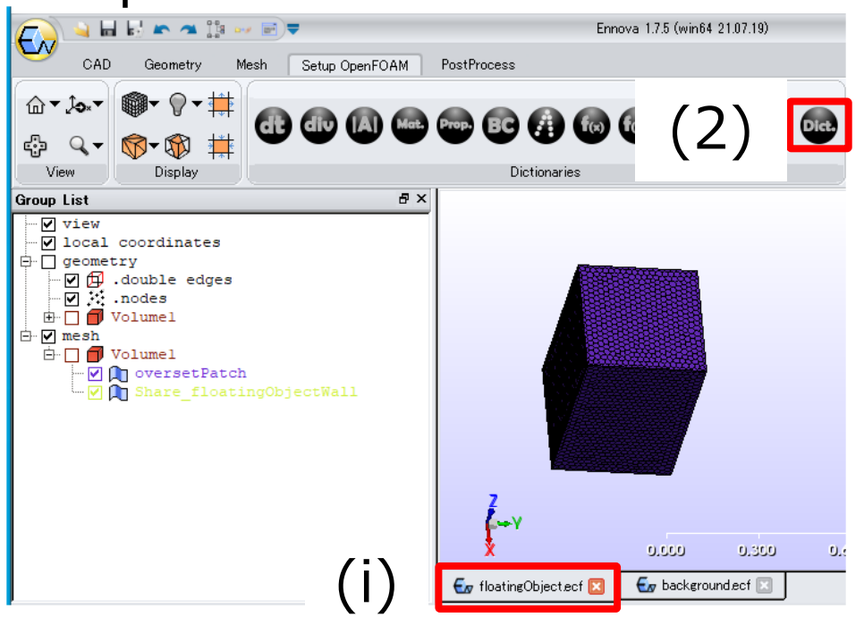


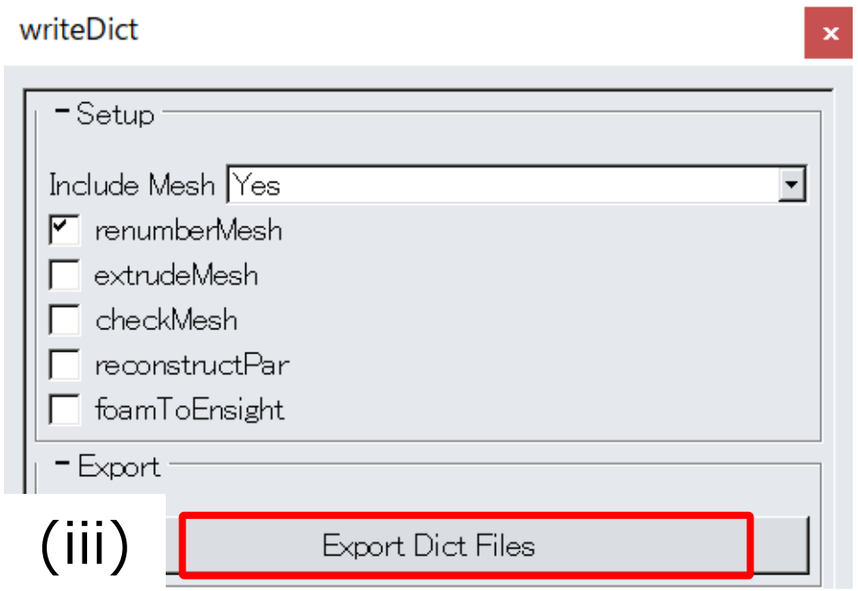
- Here, the mesh created in the verset and background areas in the ennovaCFD GUI and the analysis settings you have configured are exported to a specified folder.
- When running an analysis from ennovaCFD, do not exit ennovaCFD after exporting the file, as the file export and calculation are performed in succession.

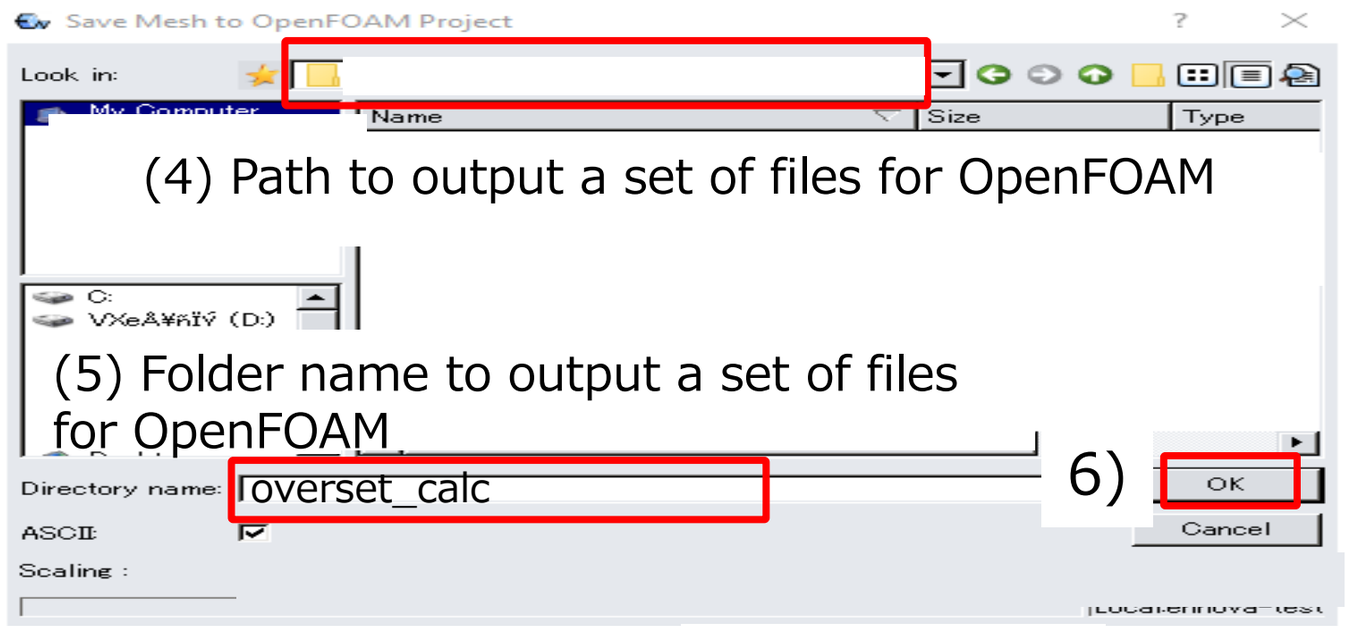
Overset mesh analysis

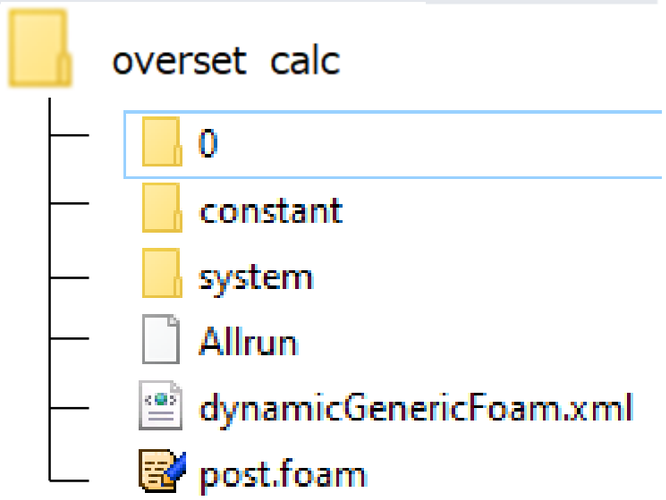
Output files for OpenFOAM (writeDict panel)

- Output from tabs in the overset area (floatingObject.ecf).







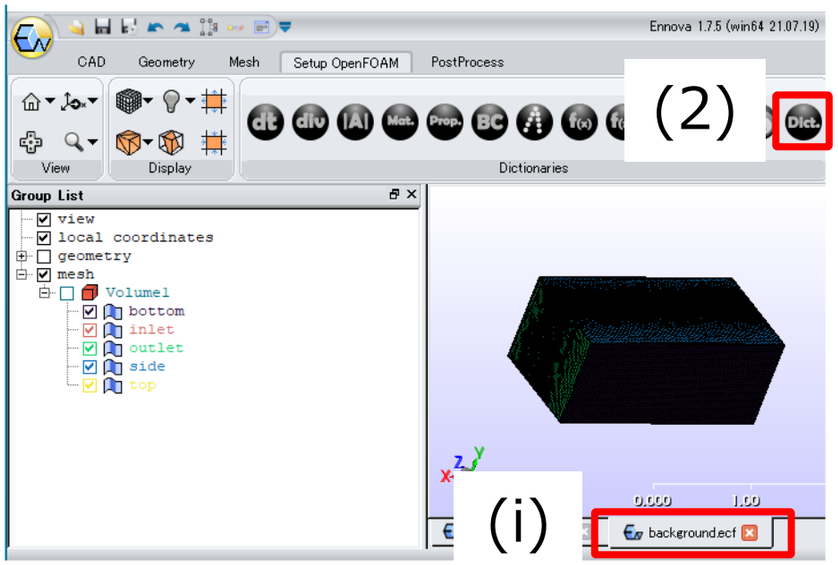


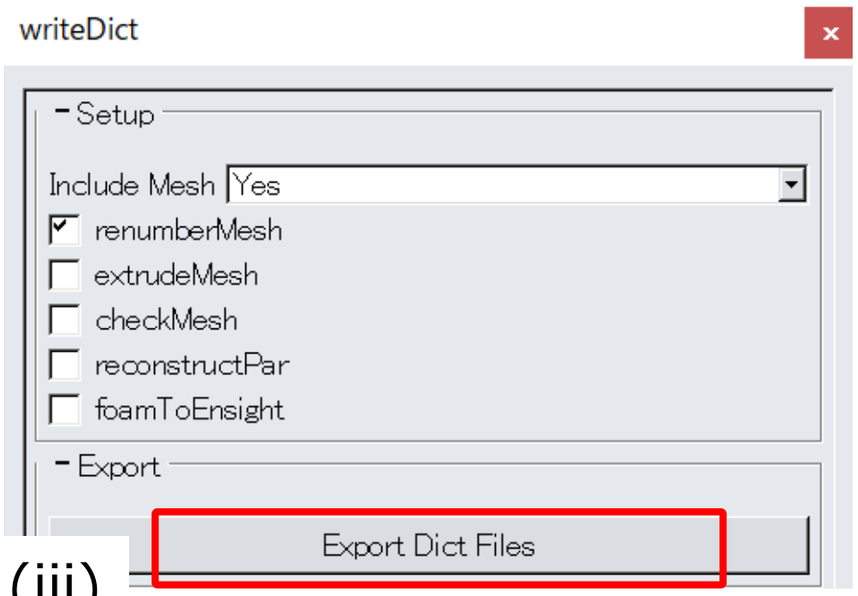
A set of output files for OpenFOAM

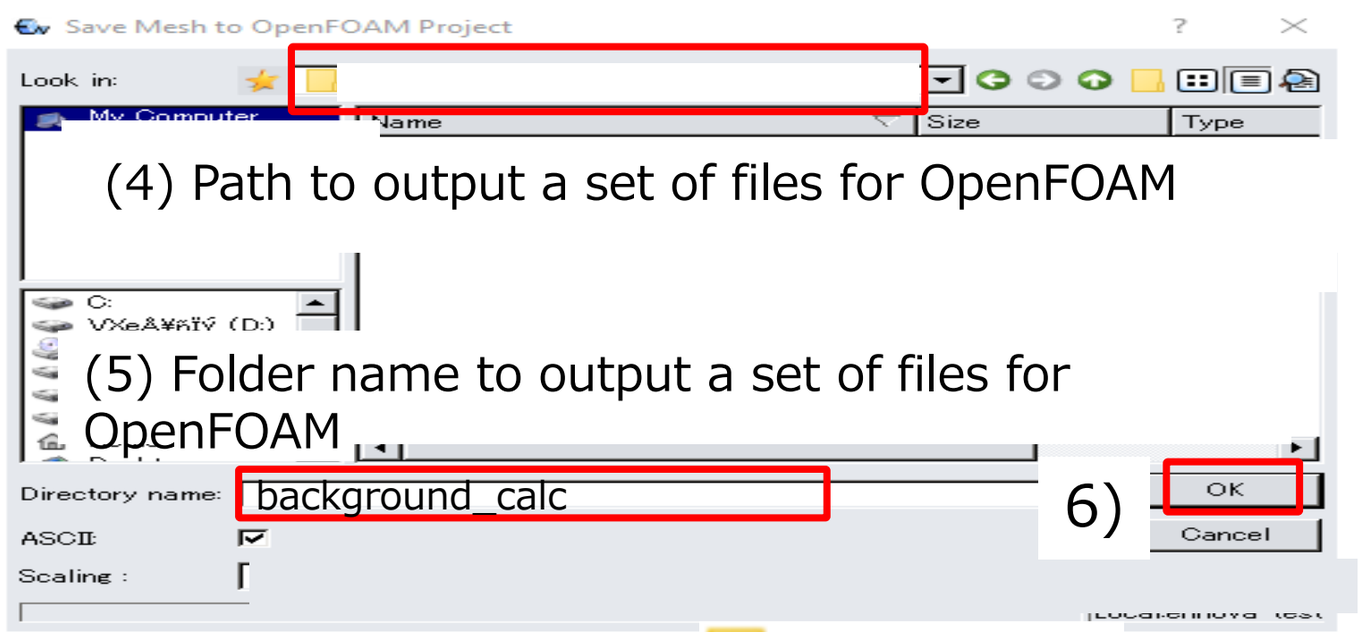
Overset mesh analysis

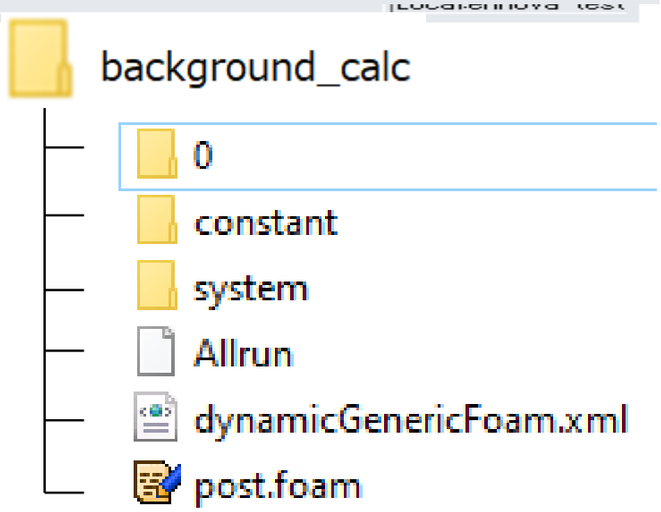
Output files for OpenFOAM (writeDict panel)

- Change to a tab in the backgroud area and output.







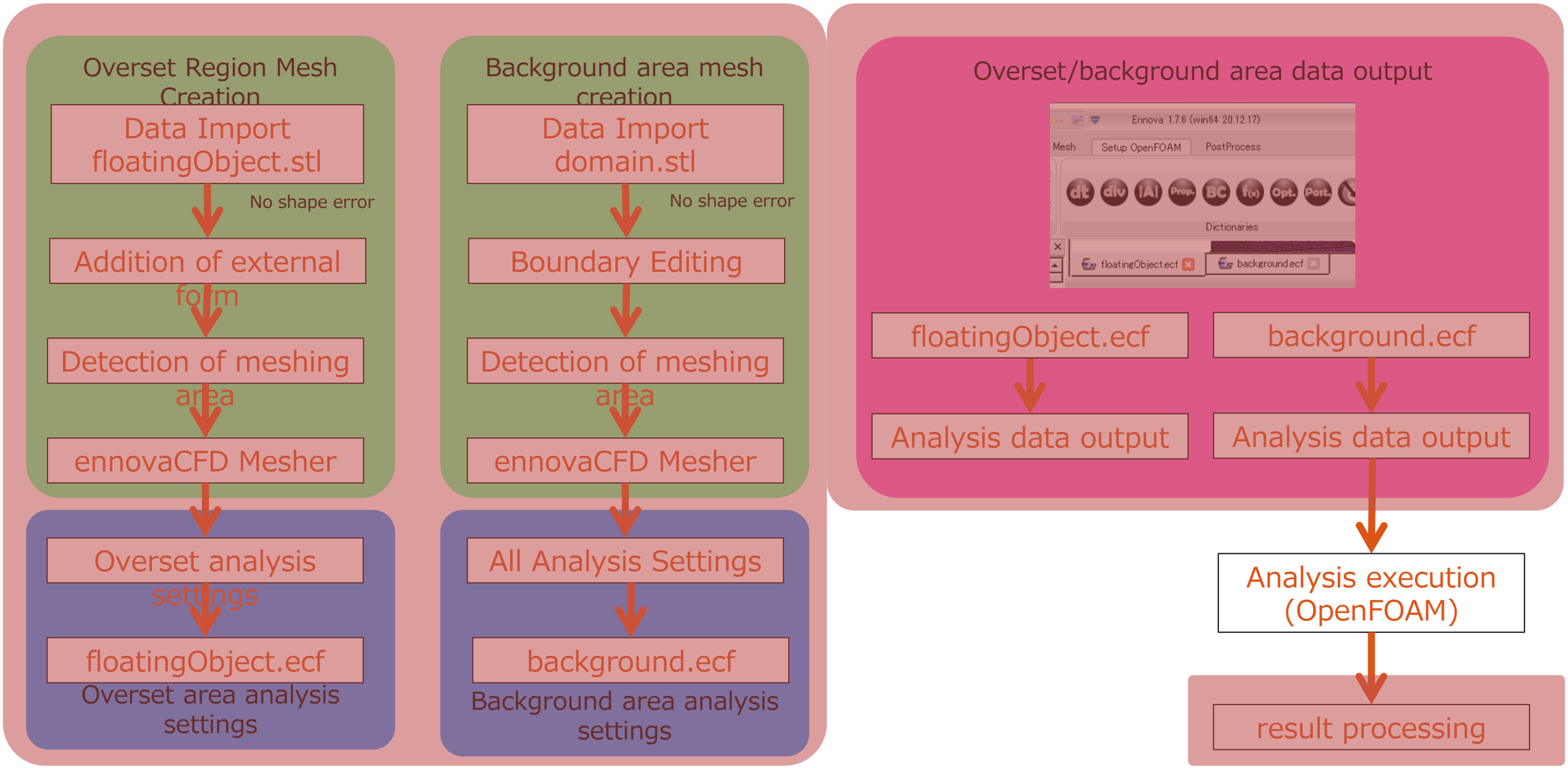


A set of output files for OpenFOAM

Overset mesh analysis

Analysis Execution Method

- Analysis execution (OpenFOAM)



Overset mesh analysis

Analysis Execution Method

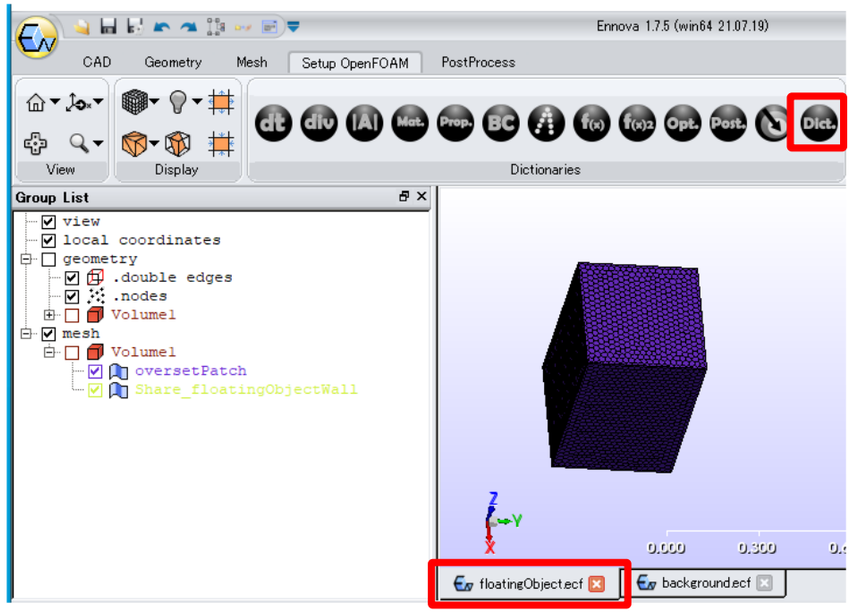
- To perform a calculation, follow these steps
  1. OpenFOAM environment variable settings (Please complete the settings in advance. If you want to execute directly from ennovaCFD, the settings on P31 and P42 are required.)
  2. Execution of overset area
     - Preparations necessary for overset analysis are made.
  3. Execution of background area
     - The following processes are performed
     - Merging the mesh of the overset area into the mesh of the background area The mesh data under constant/polyMesh/ is modified after the calculation is performed.
       - Before allrun: only the mesh of the background area
       - After allrun: mesh of background area + mesh of overset area
     - Perform Overset mesh calculations

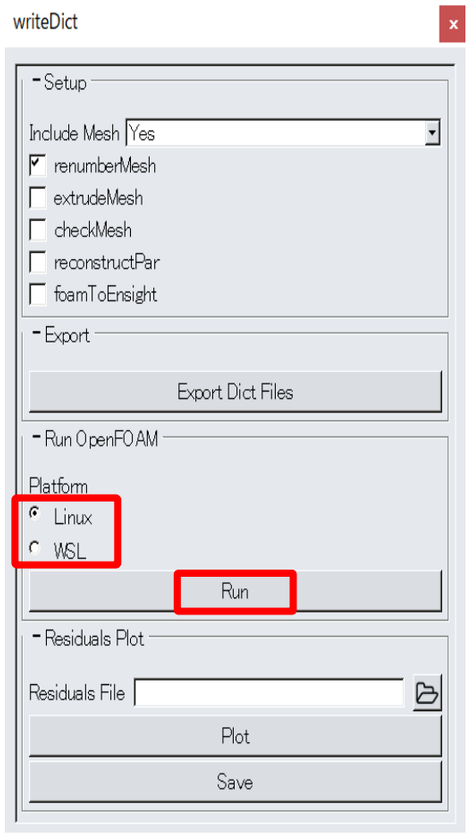
- The main case of the BACKGROUND area is where the calculation is performed and the resulting data is saved.

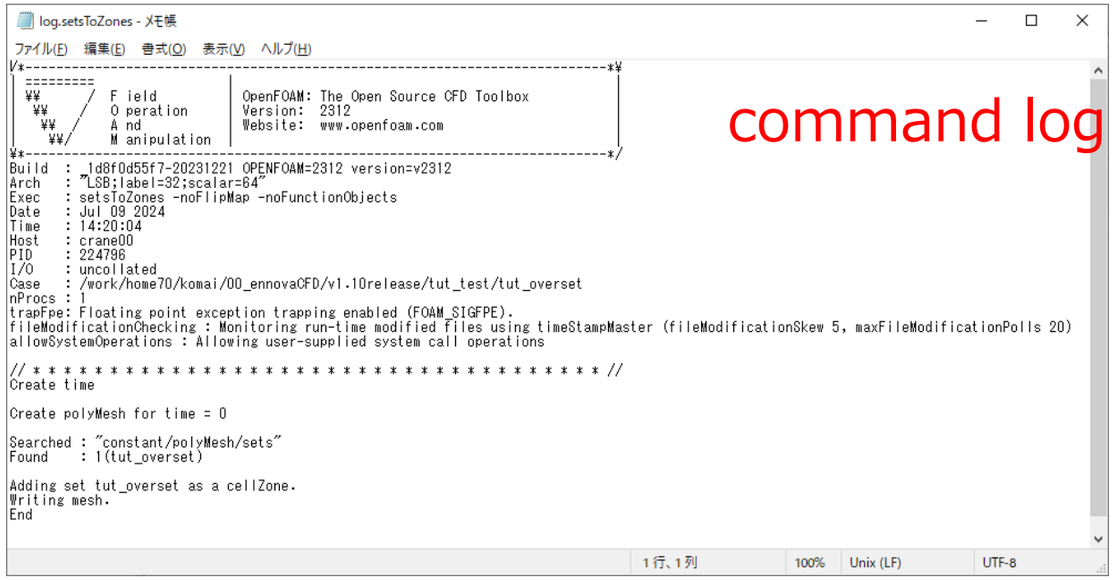
Overset mesh analysis

ennovaCFD-Run calculations from GUI (writeDict panel)

- If ennovaCFD and OpenFOAM are installed in the same environment, calculations can be performed from the GUI.





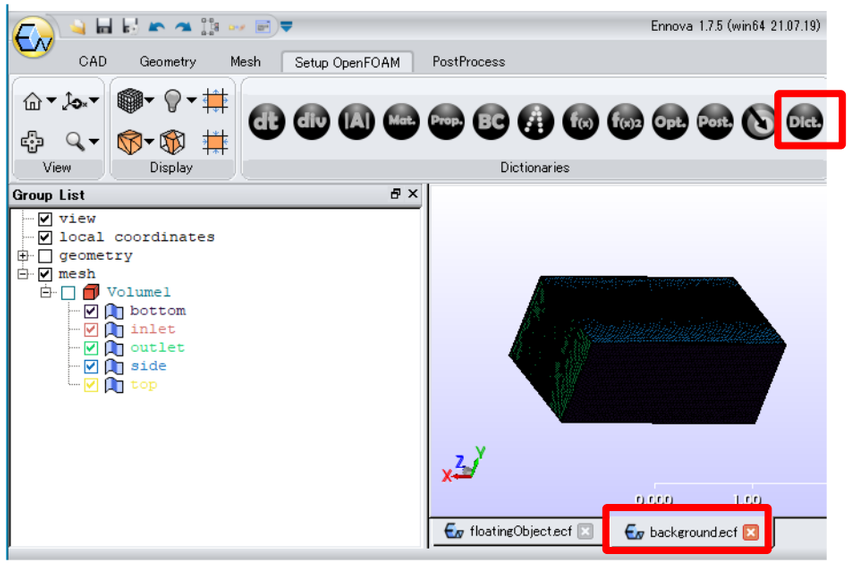


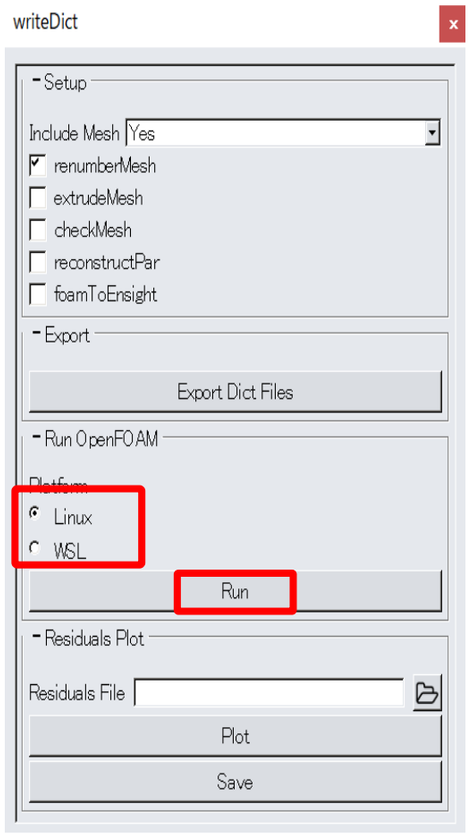
command log

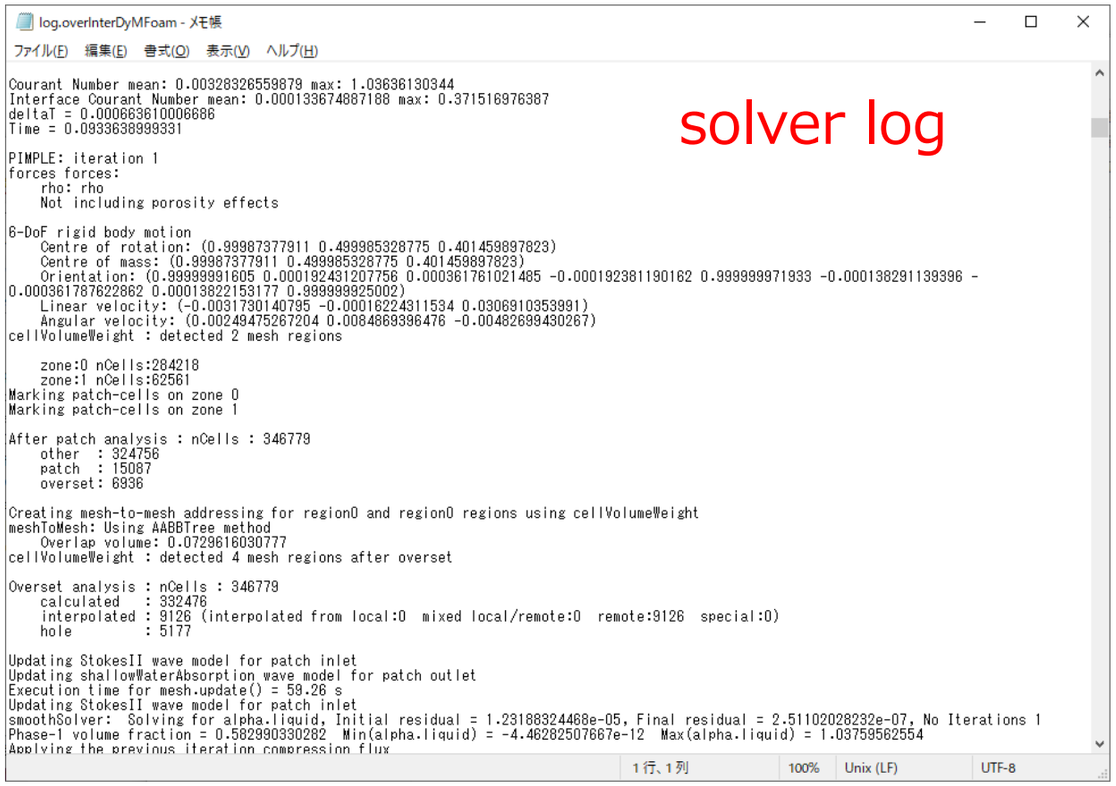
Overset mesh analysis

ennovaCFD-Run calculations from GUI (writeDict panel)

- Then change to the tab in the background area and run the calculation with Run.





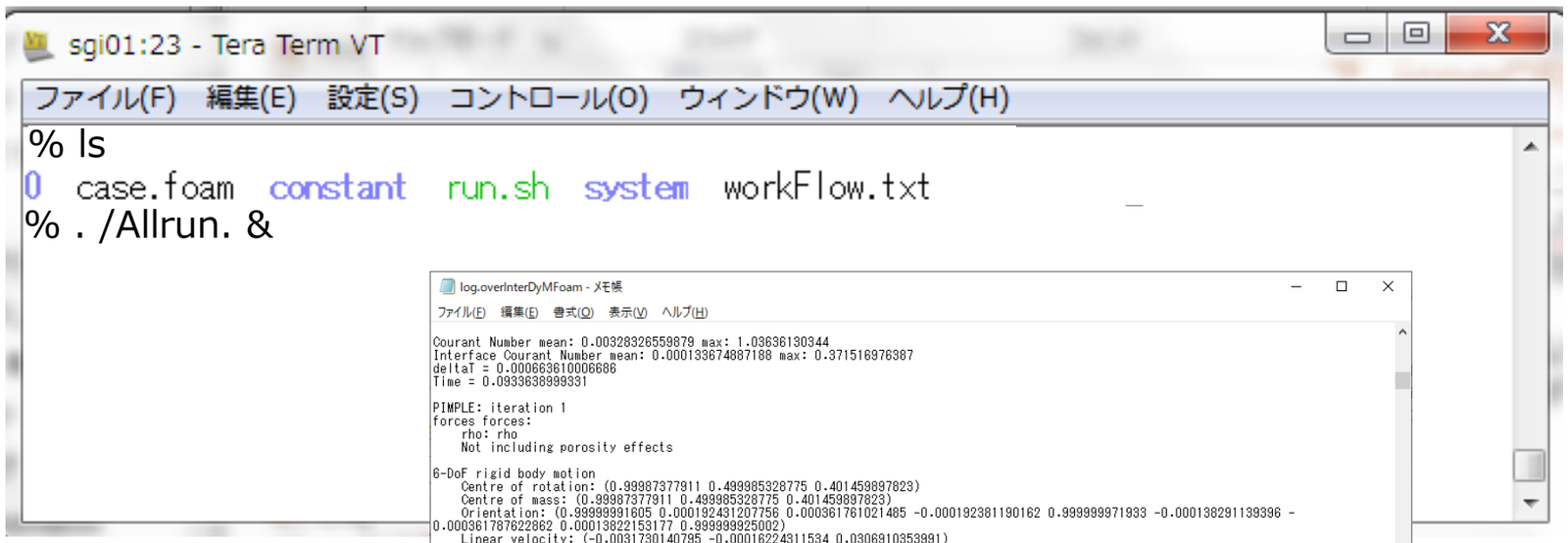


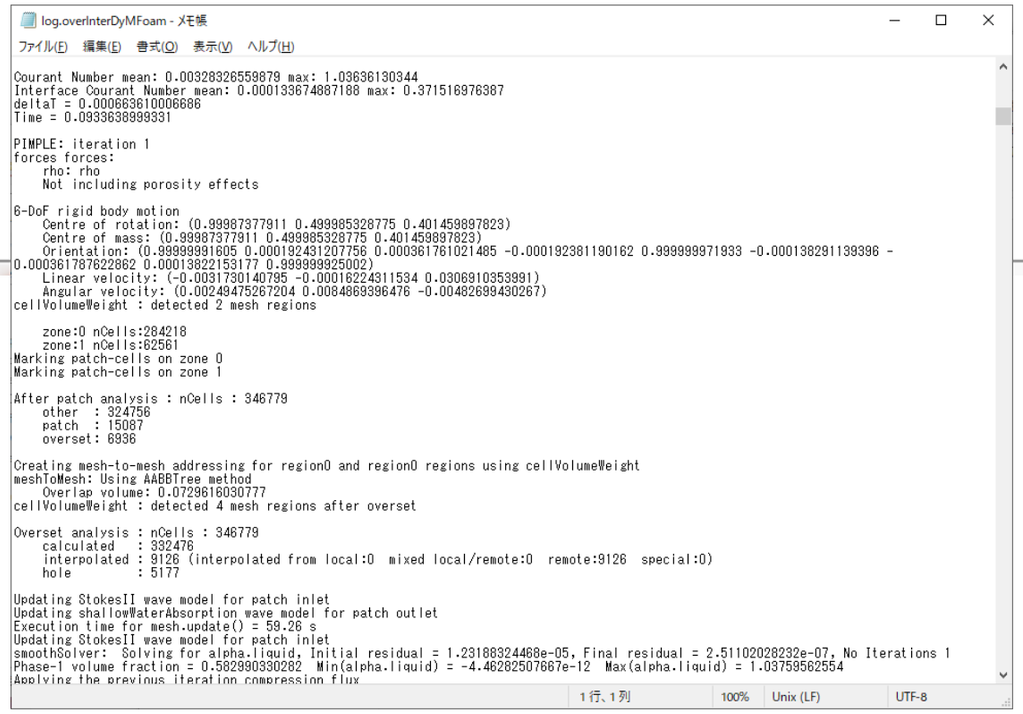
solver log

Overset mesh analysis

How to run the OpenFOAM solver from the command line

- Copy the OpenFOAM file set to an environment where OpenFOAM can run (e.g., Linux server), and first run an allrun on the "overset_calc" folder. Once that is completed, run Allrun on the background_calc folder.



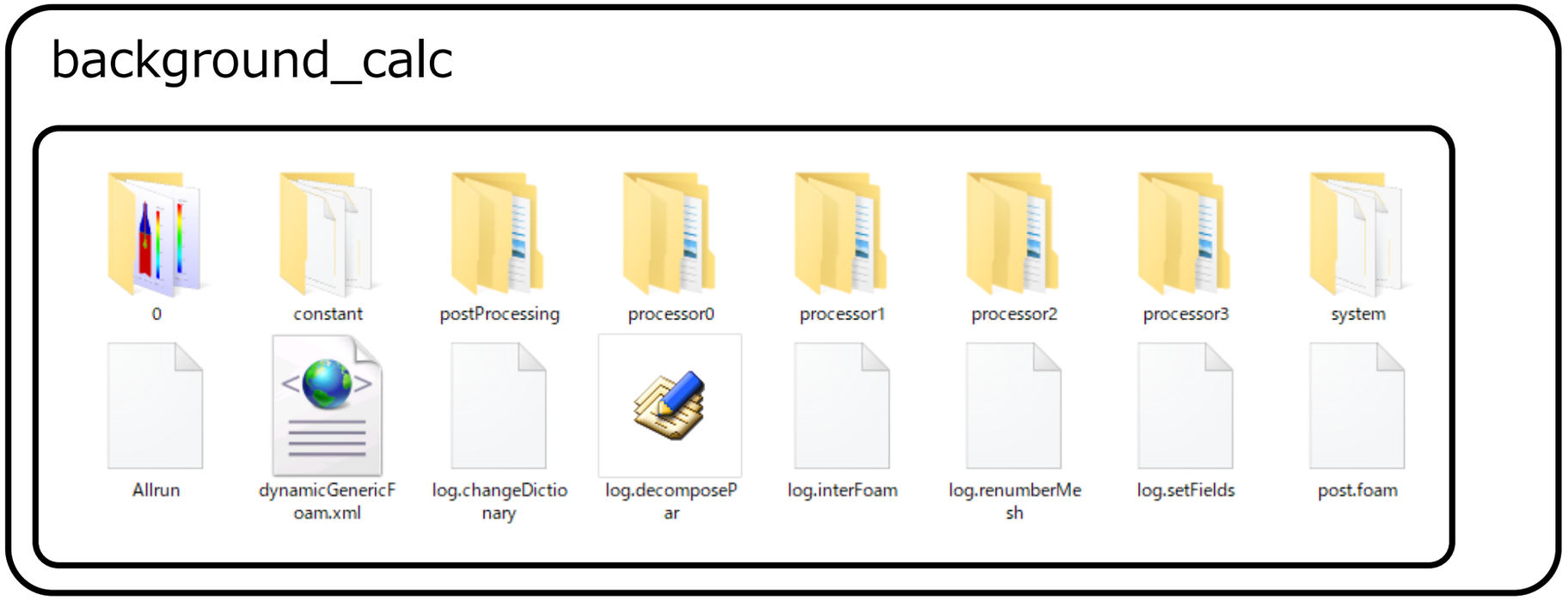


solver log

Overset mesh analysis

Calculation Results

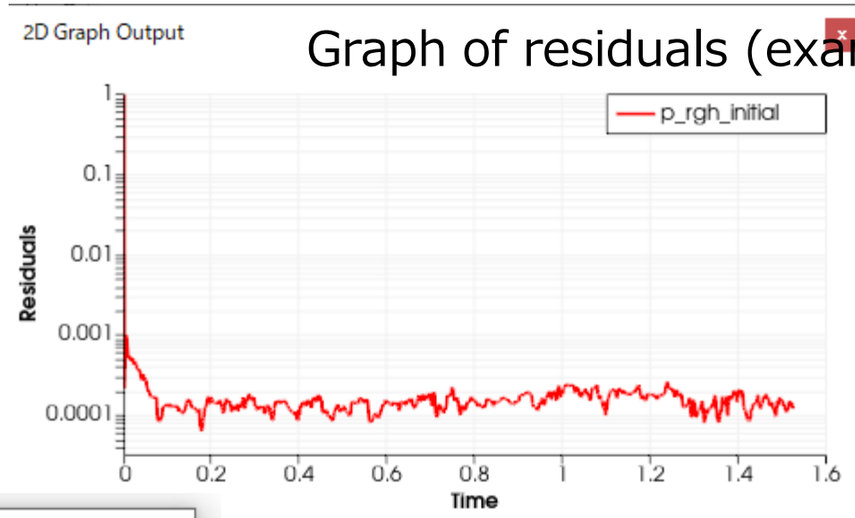
- The background_calc folder, which has been calculated by 4CPU, looks like the following.
- The folder overset_calc is not used for results processing, as it only contains preparatory information for this calculation.



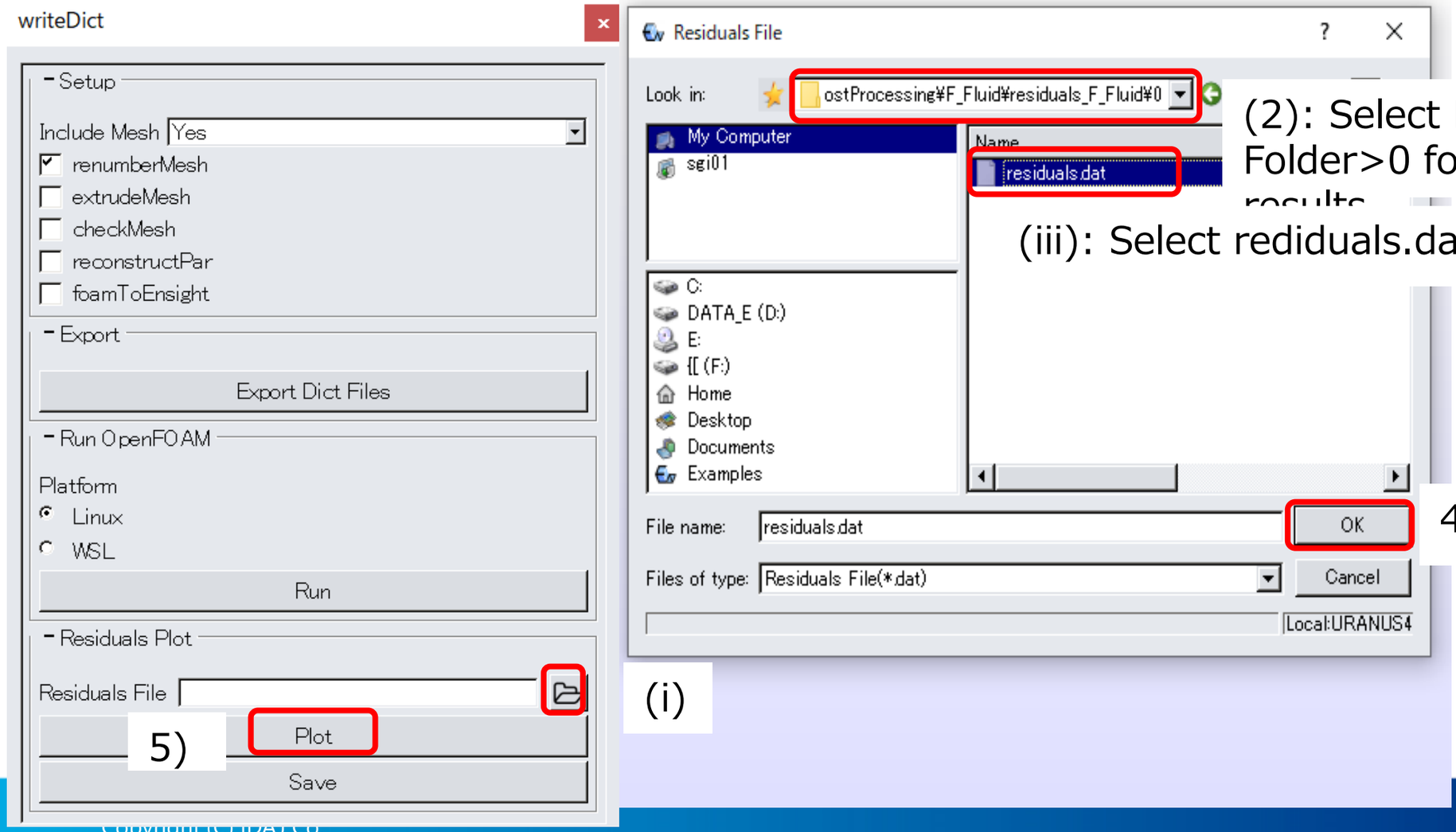
Overset mesh analysis

Result processing: Display of residual graph

- Use rediduals.dat in PostProcessing in the background_calc folder.



Graph of residuals (examp

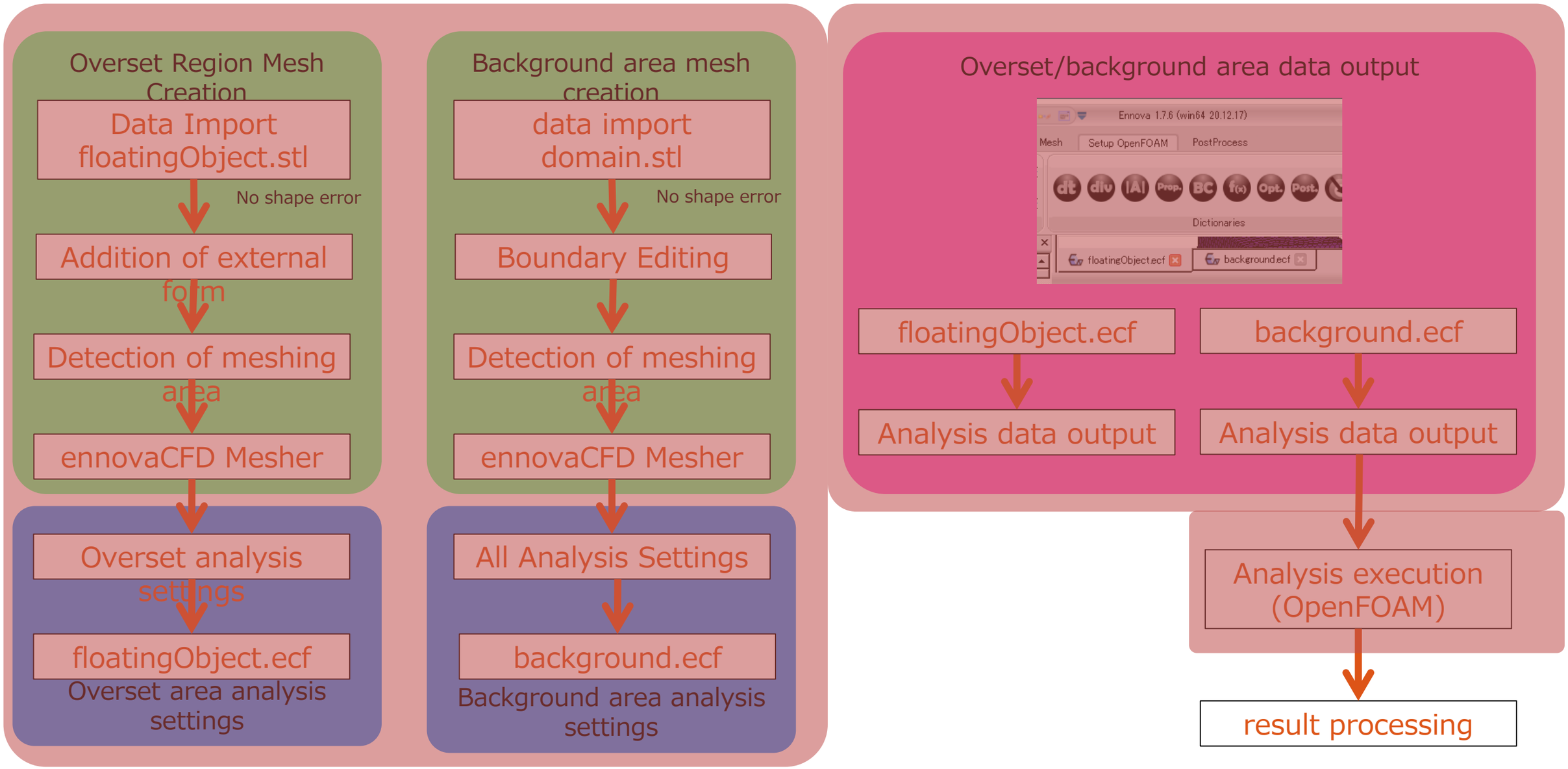


(2): Select PostProcessing>Area Folder>0 folder in the analysis results

Overset mesh analysis

result processing

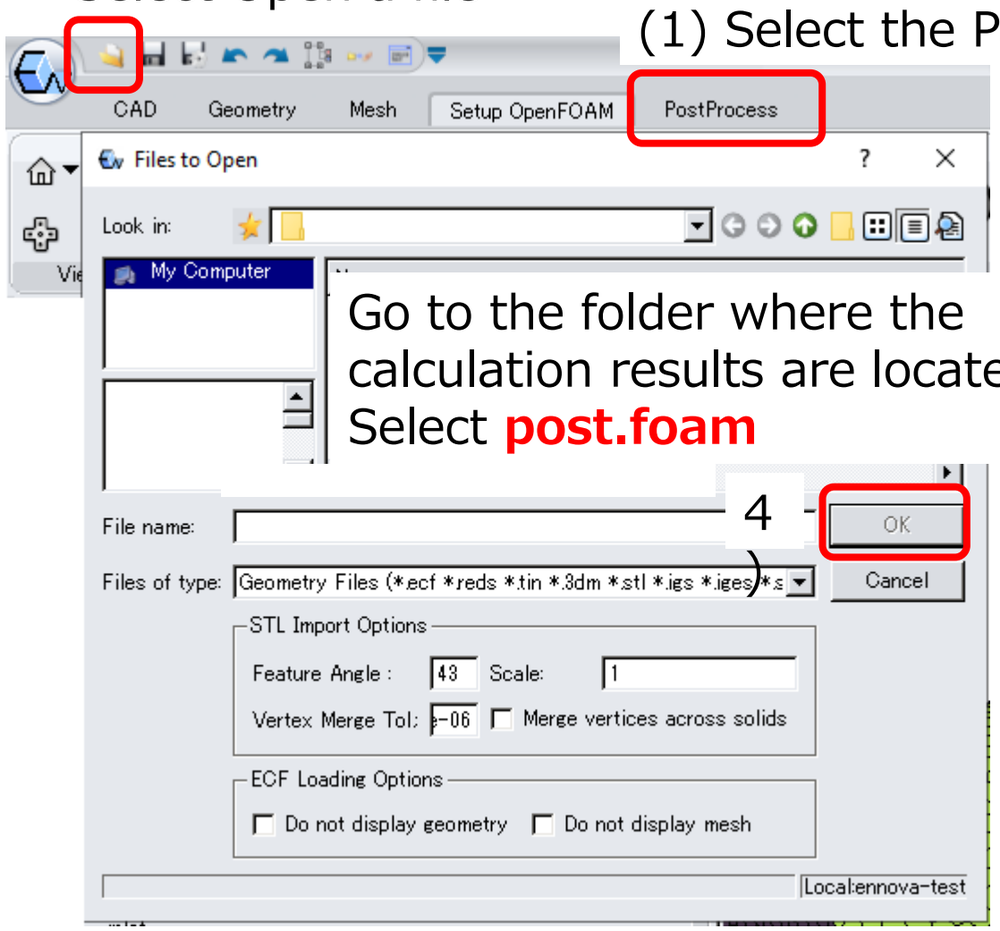
- Result processing is performed from the calculation results.

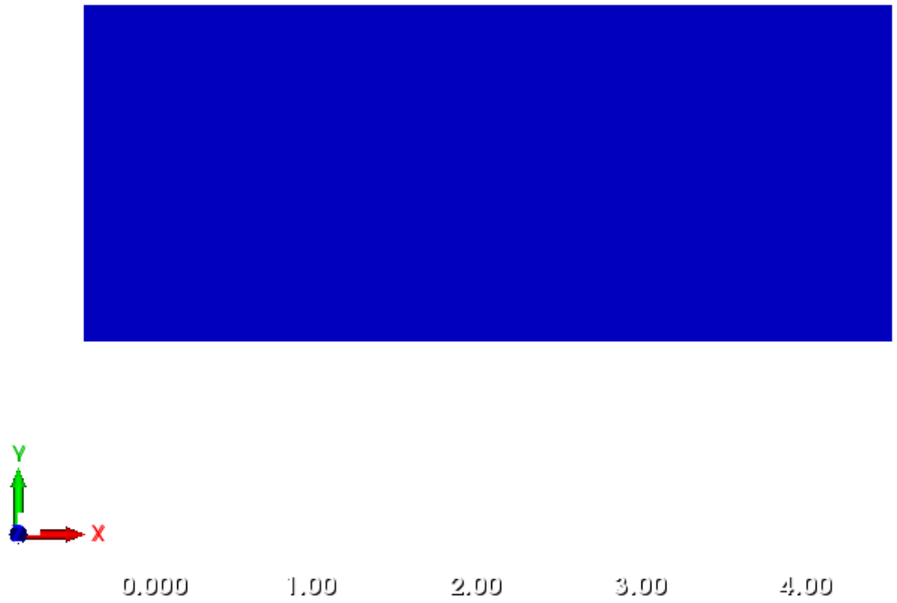


Overset mesh analysis

Result processing: data loading

- After the calculation is completed, the calculation result data is loaded.
- The result data to be read is post.foam, located in the background_calc folder.

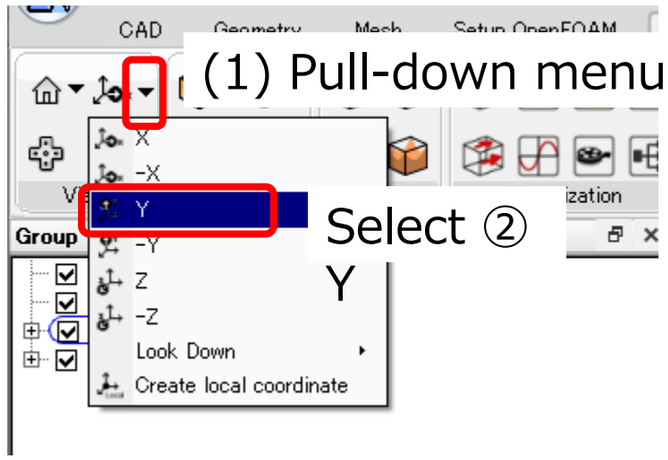


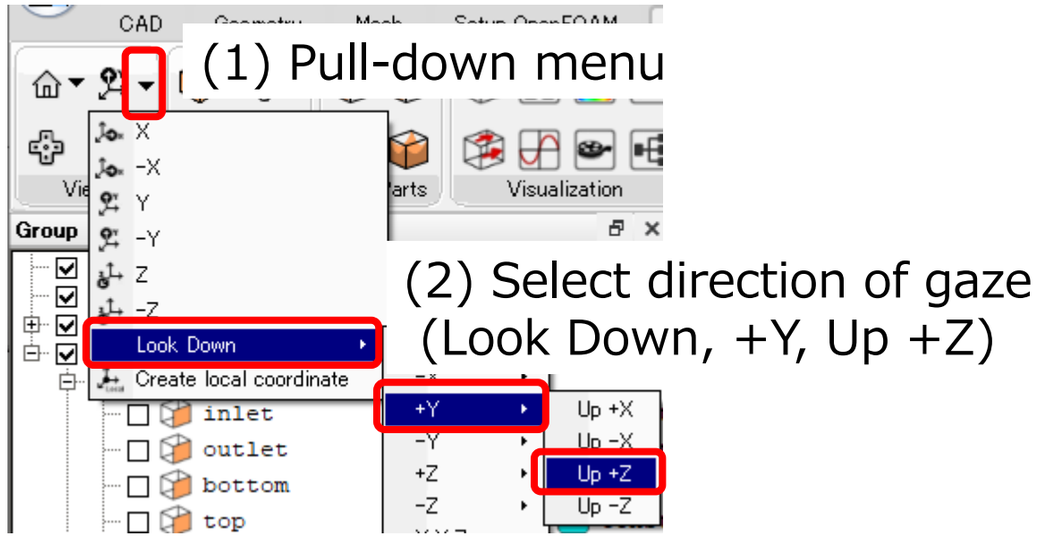


Overset mesh analysis

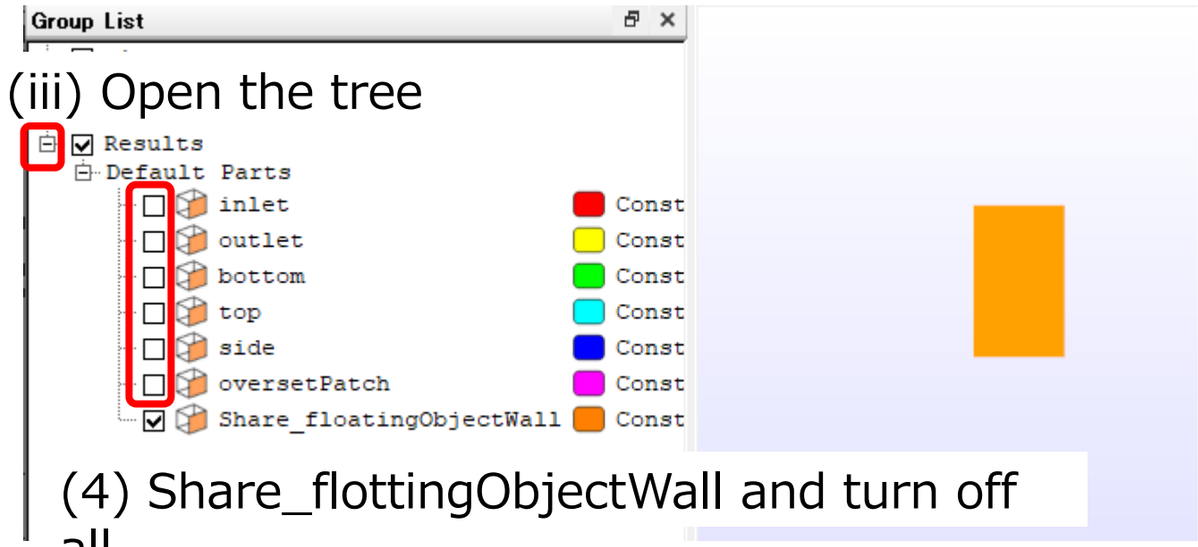
Result processing: display of alpha1

- Change the display method to Y direction.





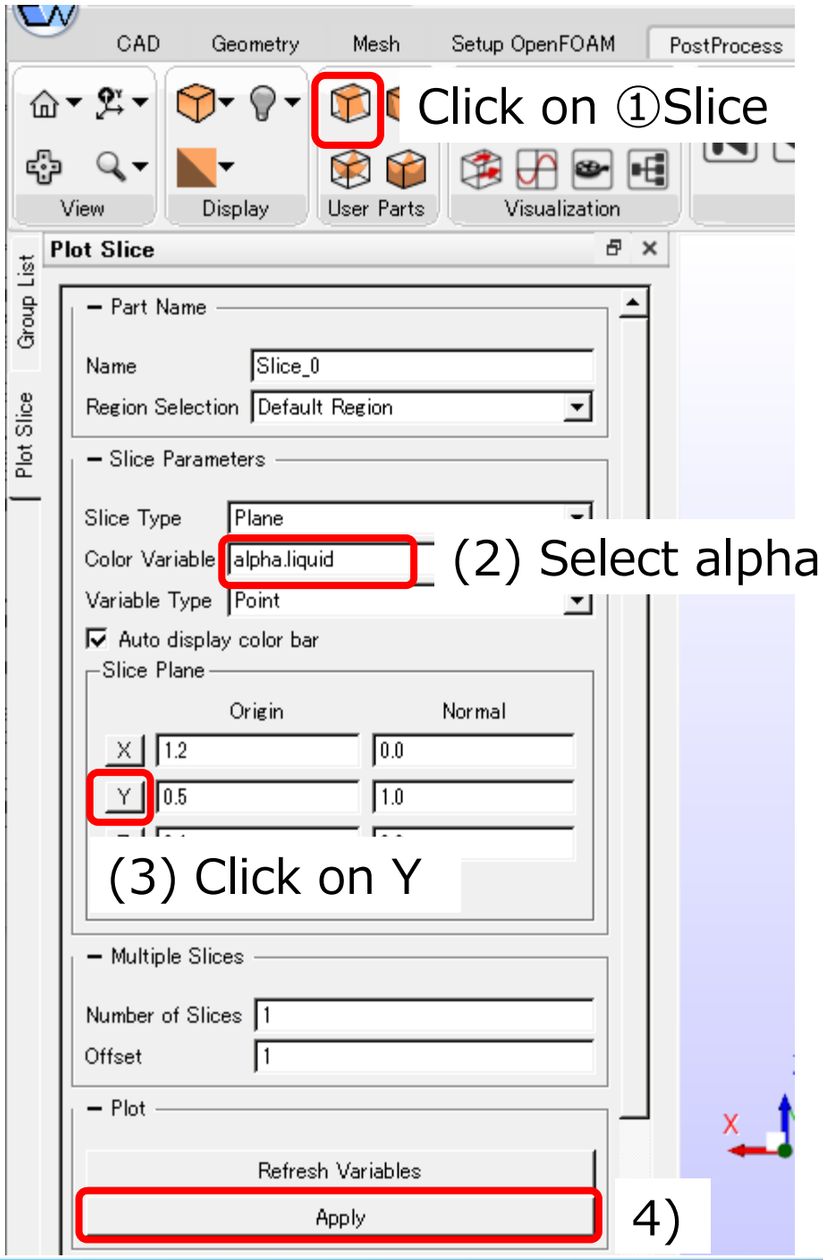
- Open the Result tree of the Group List and turn off everything except for the floatingObjectWall.

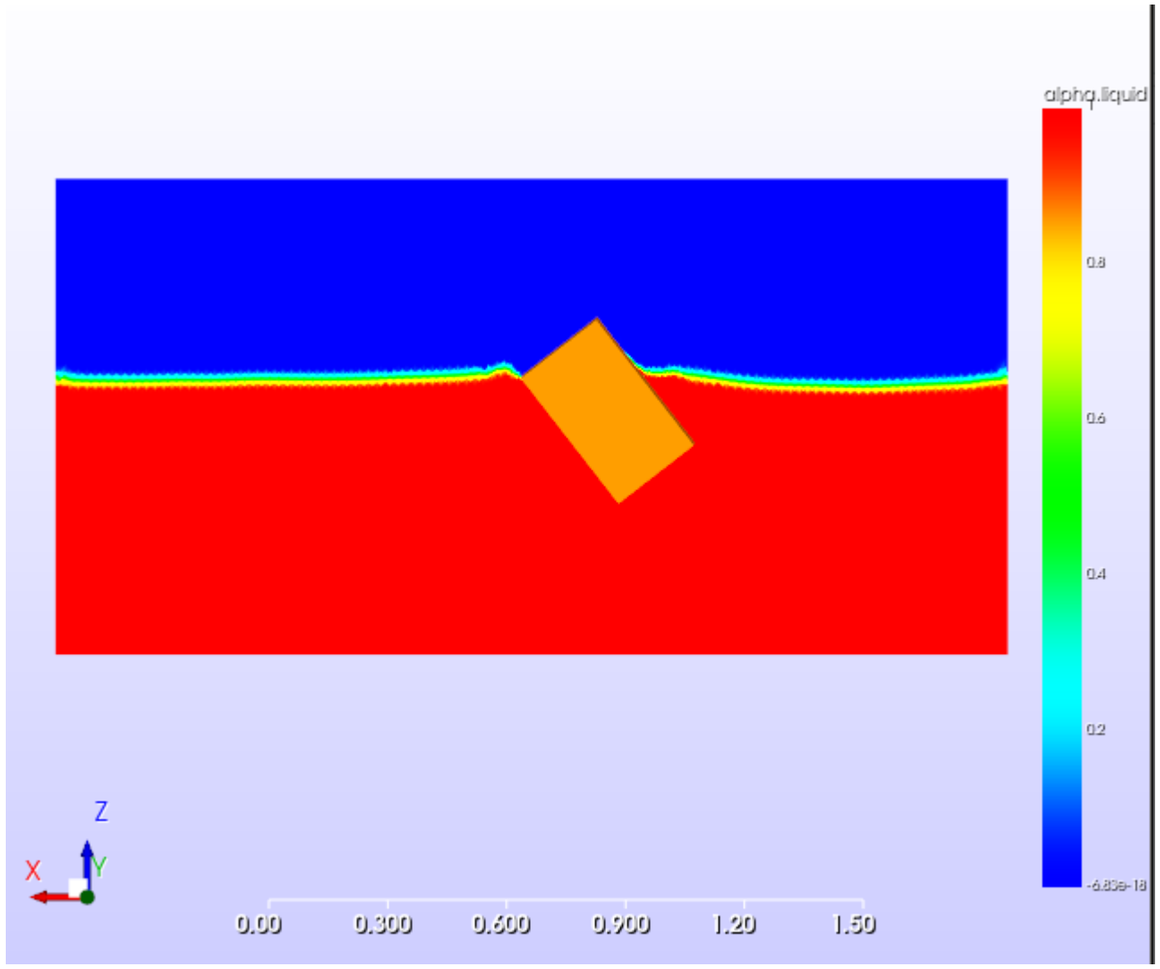


Overset mesh analysis

Result processing: display of alpha1

- Creates a Y=0.5 cross section and displays the volume fraction of water.

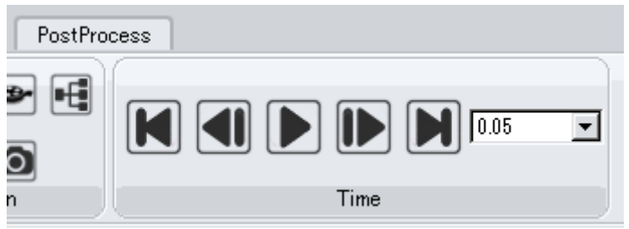


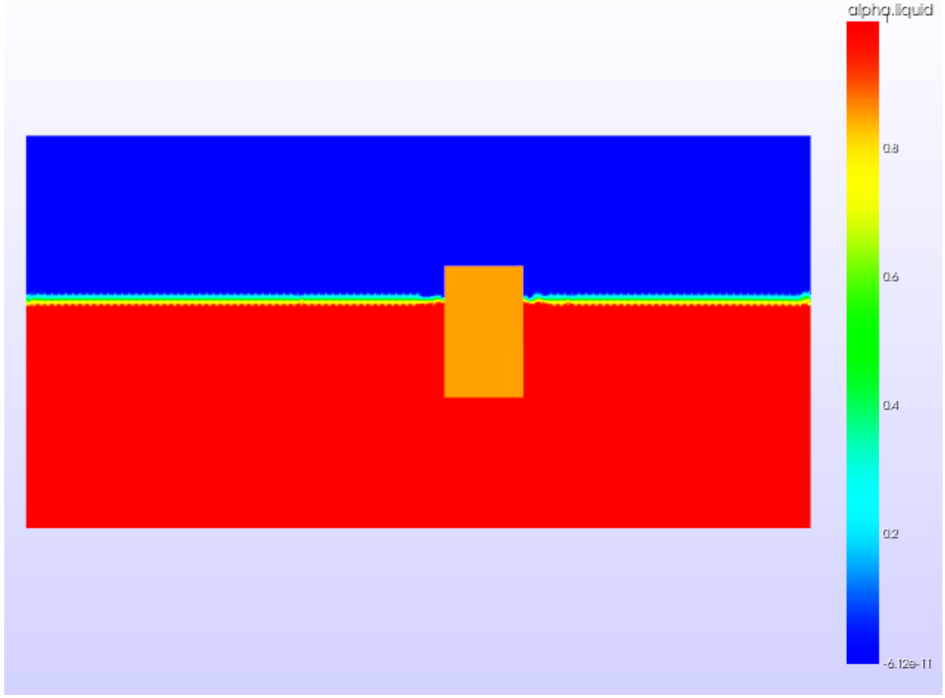


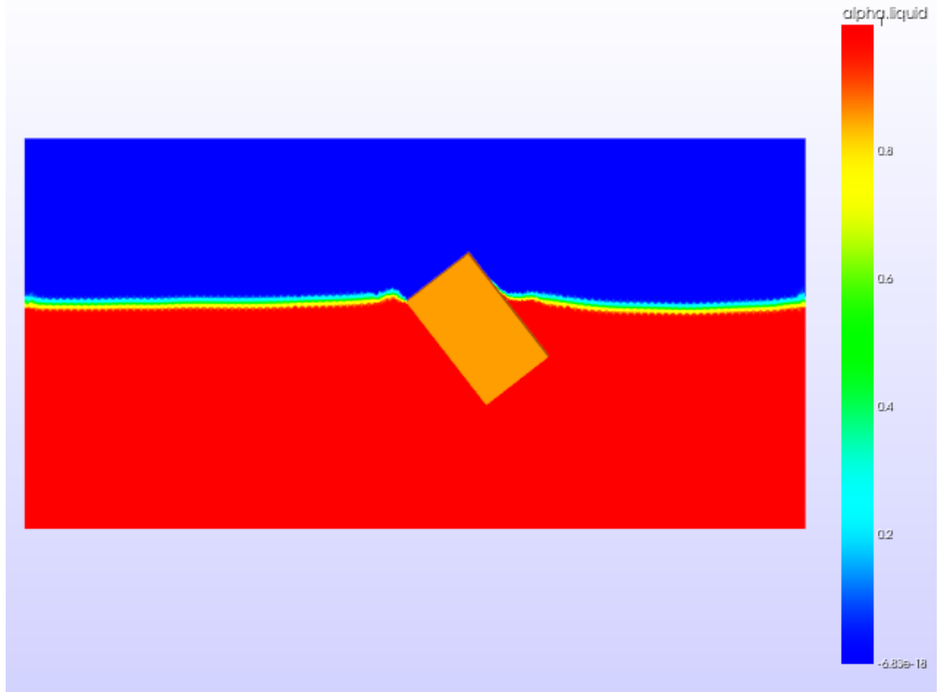
Overset mesh analysis

Result processing: display of alpha1

- Change Time to display results by hour.







0.05 sec.

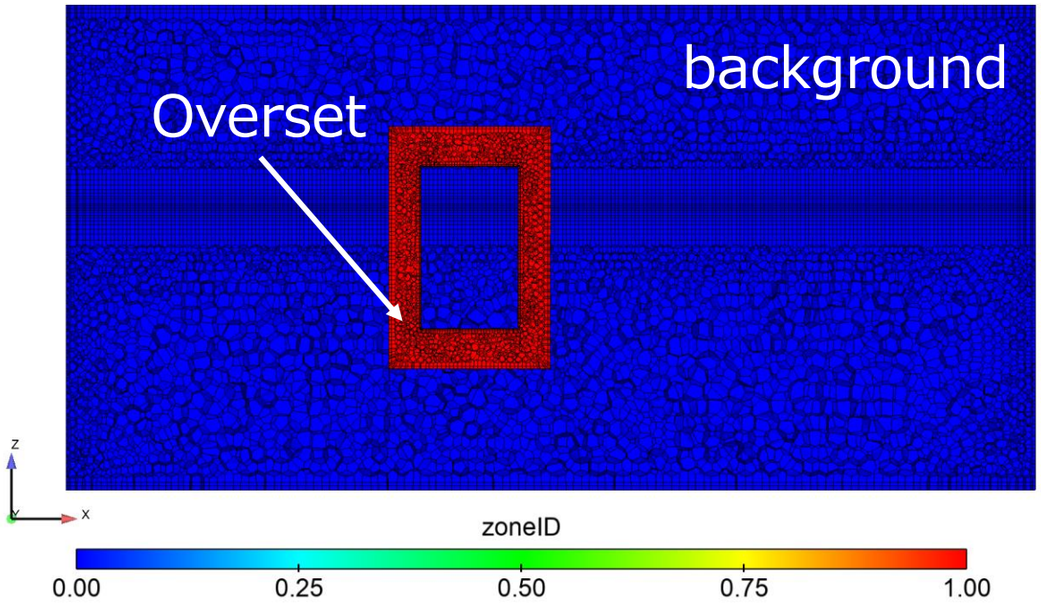
0.15 sec.

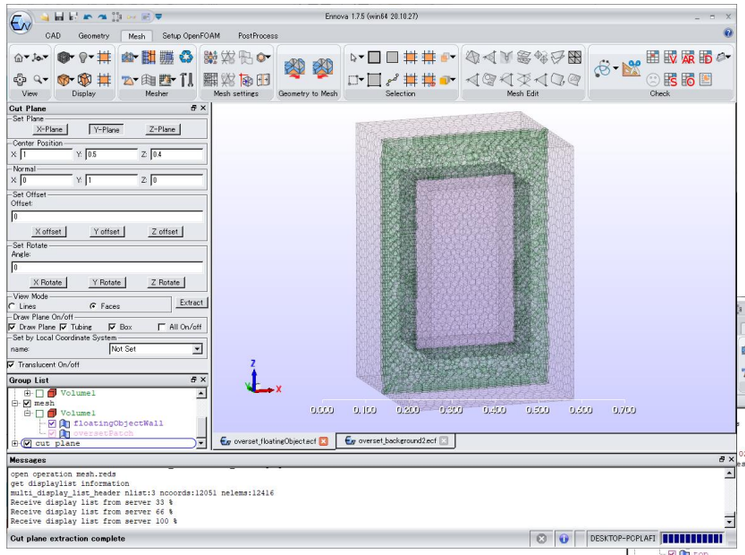
- precautions

Overset mesh analysis

Notes on Overset mesh calculation settings

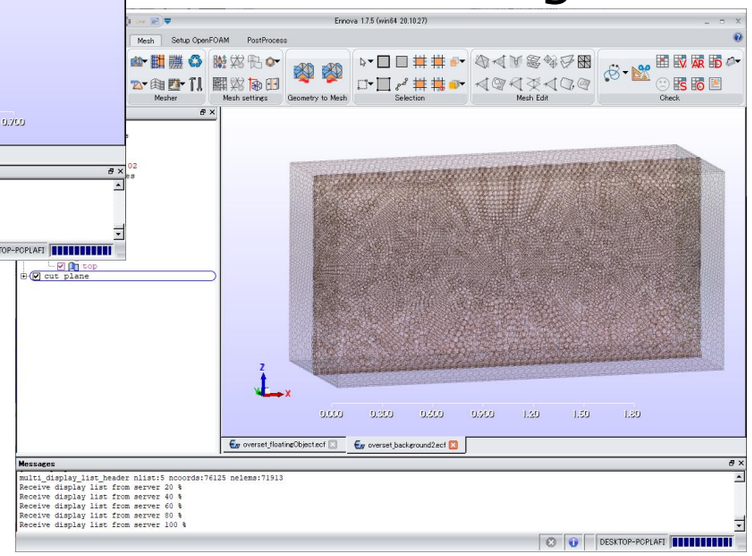
- When using ennovaCFD for OpenFOAM to set up an Overset mesh calculation, a model (ecf file) is created for each of the overset and background regions.
- When outputting an OpenFOAM case from an ecf file in the background region, the ecf file of the model in the overset region must also be loaded into the GUI.
- There is no limit to the number of models in the overset area, and any number of models can be configured from the GUI.





Overset

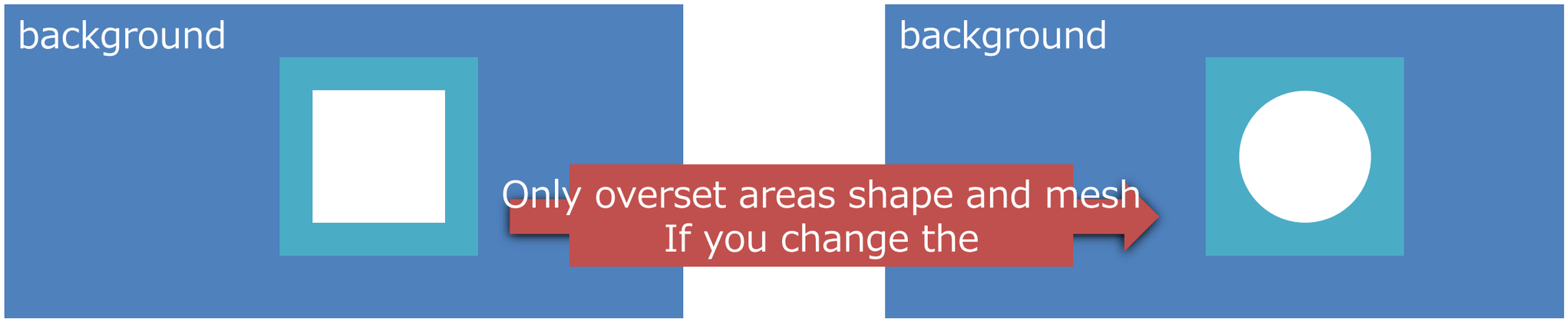
background



Overset mesh analysis

Procedure 1 for changing the model of the overset area

- To perform calculations without changing the background area settings, but by changing the shape of the offset area, follow the steps below.

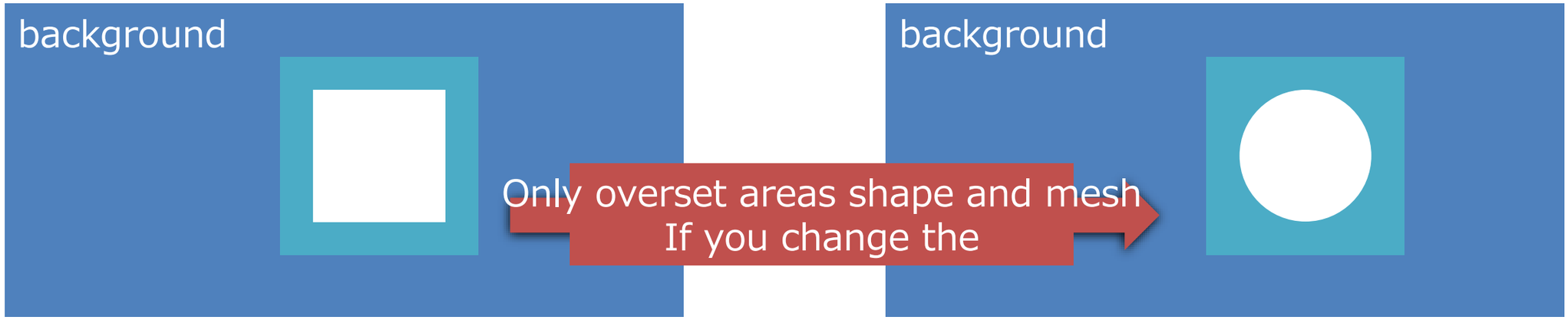


1. Only the overset area is re-generated with the geometry and mesh.
2. Set up the analysis of the overset region and output the case of the overset region.
3. With the model of the modified overset area loaded, open the original background area ecf file and output the case of the background area.
4. Execute the Allrun script in the order of the OVERSET area and BACKGROUND area (see p. 62~).

Overset mesh analysis

Step 2 when the model of the overset area is changed.

- To perform calculations without changing the background area settings, but by changing the shape of the offset area, follow the steps below.



If there is still a case in the background before the allrun is run, this data can be used to create a new case below.

The procedure described above can also be used.

1. Only the overset area is re-generated with the geometry and mesh.
2. The analysis settings for the overset area are configured, and the case of the overset area is output. The directory name for the output should be the same as the original Overset case name (floatingObject in the case of this procedure).
3. Execute the Allrun script in the order of the OVERSET area and BACKGROUND area.

Thank you for your kind attention.
We are looking forward to doing business with you.

Contact Us

- web: https://www.idaj.co.jp/
- e-mail: info@idaj.co.jp