Murata Software

Femtet Seminar

Understanding Fluid Analysis

202009

Table of Contents

Murata Software

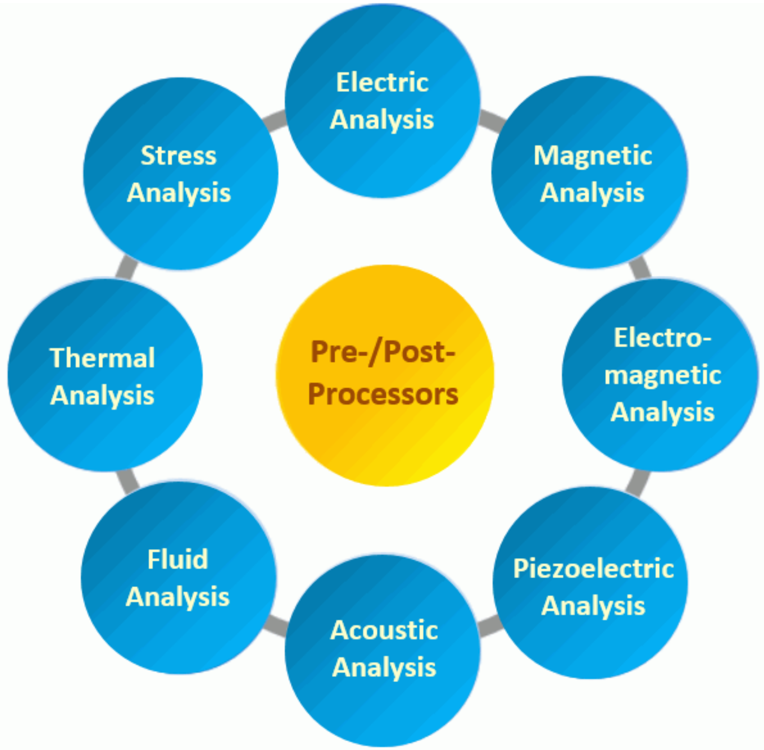
Basic

Advanced

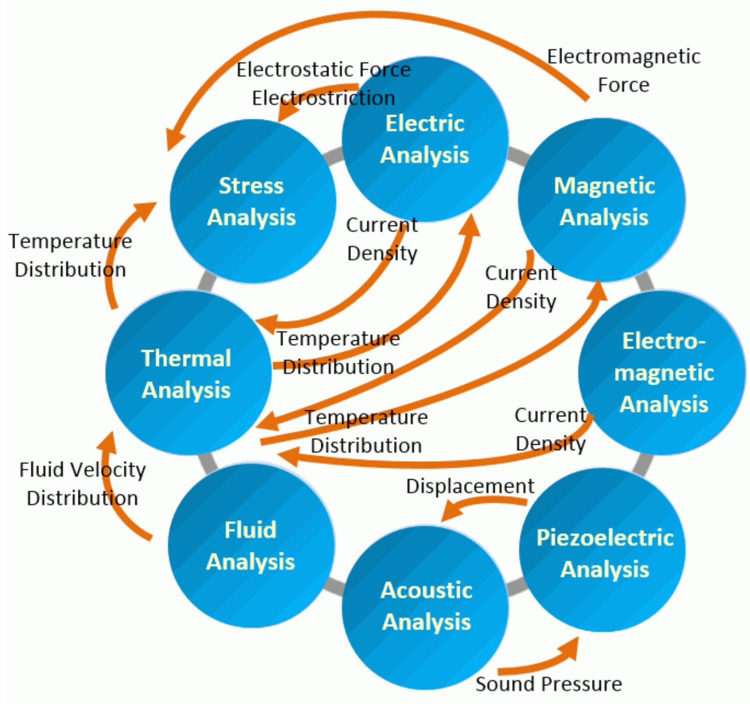
Appendix

1. Overview of Fluid Analysis & Fluid-Thermal Analysis

Functionalities



Solvers



Multiphysics

Fluid Analysis

Analysis Type

・Steady-state
・Transient analysis

Analysis Domain

・2D
・3D

Calculation Method

・Laminar Flow
・Turbulent Flow
(Realizable K-ε Model)

Preconditions

・Incompressible Flow
Density stays unchanged
Inflow and outflow must be set together
Flow velocity is less than a third of sound speed

Air ～100m/s
Water ～500m/s

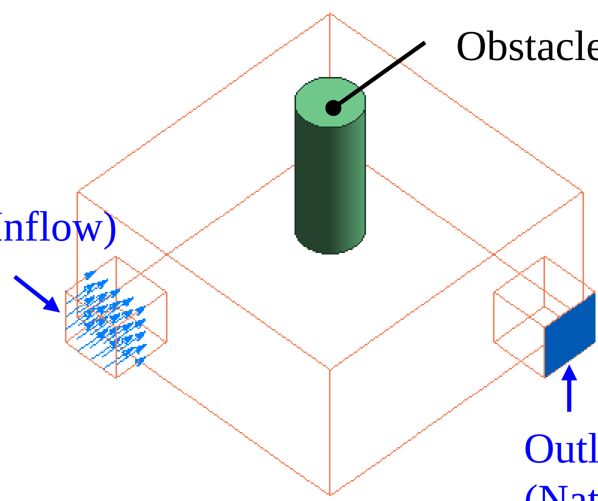
・Single-phase Flow
Single type of flow only
(Multiple flows are possible if there are separate paths)

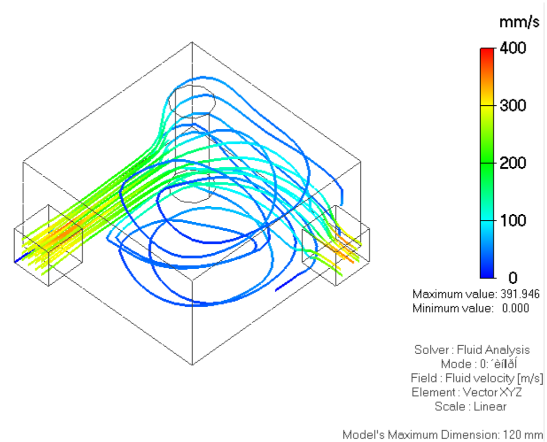
*Analysis with buoyancy taken into account is not supported by Femtet version 2019.0 or before

Analysis Example

Murata Software

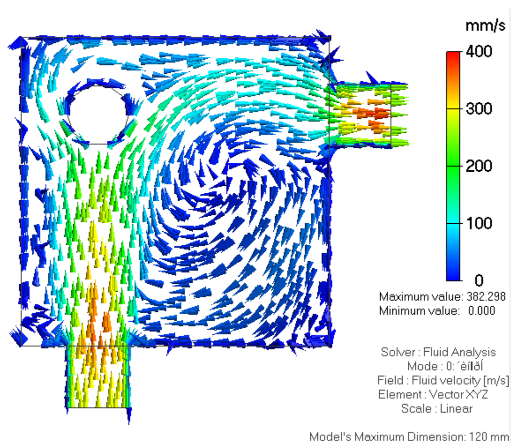
Flow in the flow path with an obstacle (see Tutorial)





Streamlines

The force on the solid (obstacle) and the pressure loss between the inlet and the outlet can be solved as well.



Vector Diagram

Analysis Type for Fluid

- Steady-state Analysis
- Transient Analysis

Analysis Type for Heat Conduction

- Steady-state Analysis
- Transient Analysis

Steady-state analysis is applied to the fluid, and then thermal analysis (steady-state or transient) is executed.

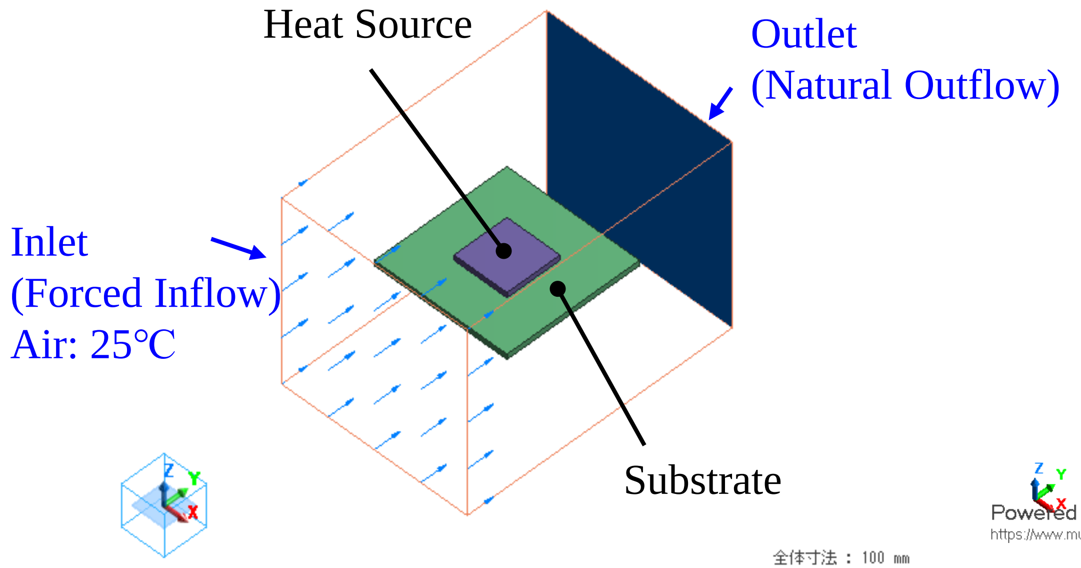
Taken into account are the effects of:

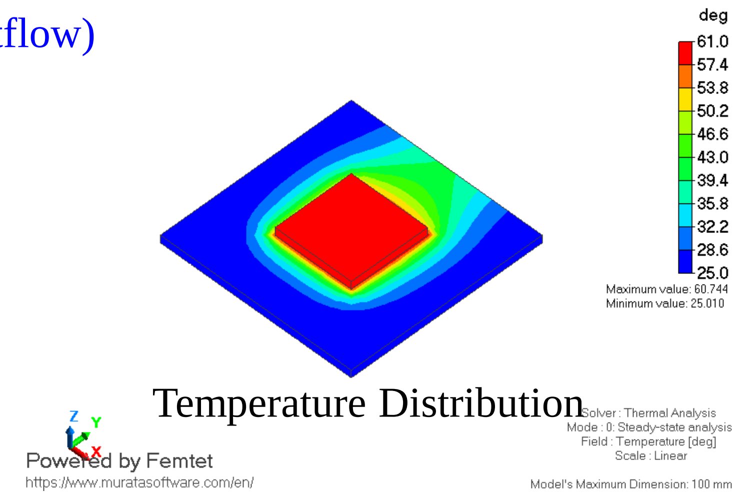
- Convection
- Heat transfer by turbulent flow

Femtet version 2019.0 or before does not support the analysis that takes into account the buoyancy or the temperature dependency of fluid material.

 Murata Software

Cooling of IC by Forced Convection (Fluid-Thermal Analysis Example 3)



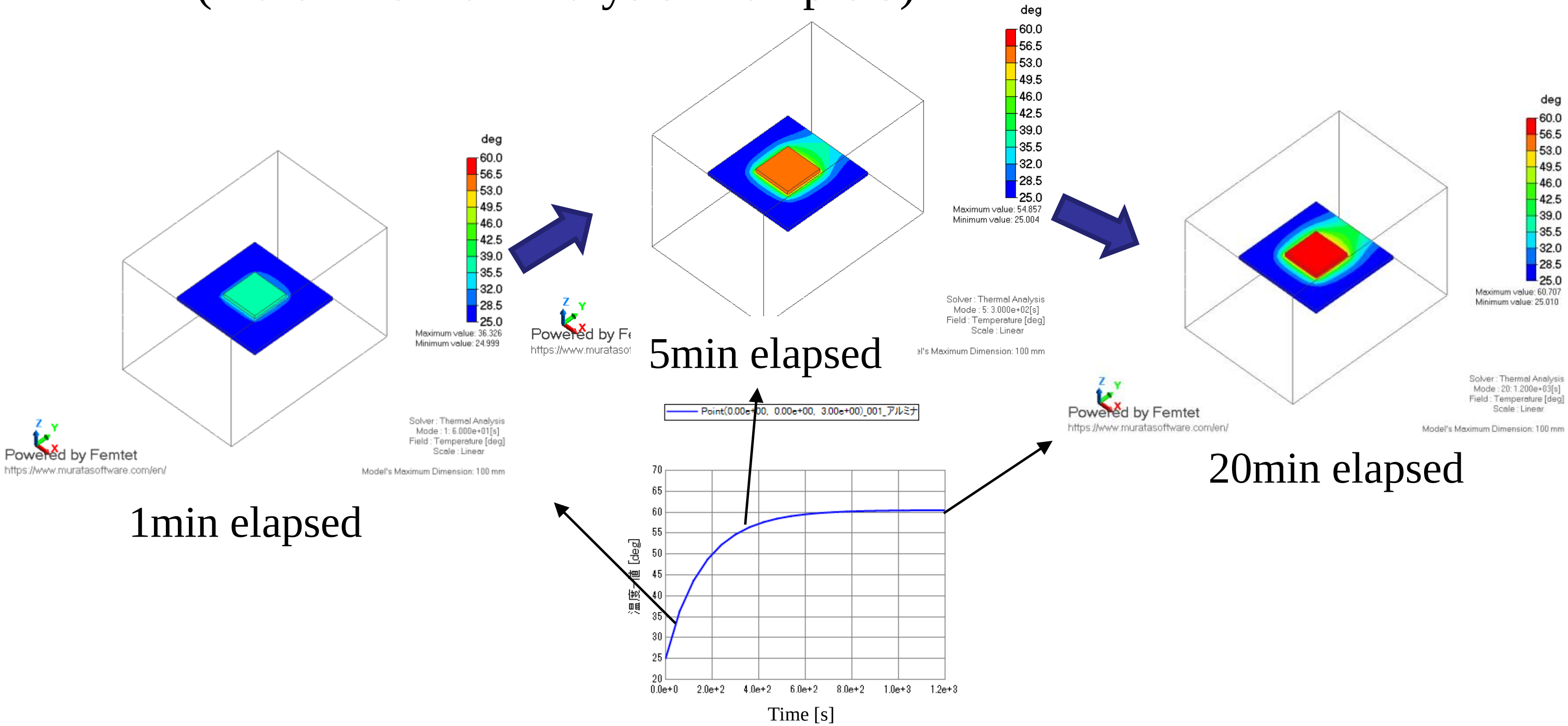


The heat is carried in the downstream by the forced convection.
The temperature distribution of the fluid is calculated as well.

In the simple fluid analysis, the temperature of the fluid is not calculated.
(see appendix 7-4)

Murata Software

Transient Analysis of Cooling of IC by Forced Convection
(Fluid-Thermal Analysis Example 3)



Temperature over the Heat Source

The state of flow is constant. Temperature rise is calculated from the start.
The reached temperature will be the same as that of the steady-state analysis.

2. Analysis Setting

Analysis Setting Flow

Basic Flow Common to All Solvers

①Analysis Condition

Solver setting, etc.

②Body Attribute / Material Property

Body attribute setup.
Material setup from the material DB or direct input.

③Boundary Condition

Setting body surface state as a boundary condition.

④Mesh Size

Space accuracy setting.

Analysis Setting Example

Murata Software

<Basic Flow of Settings>

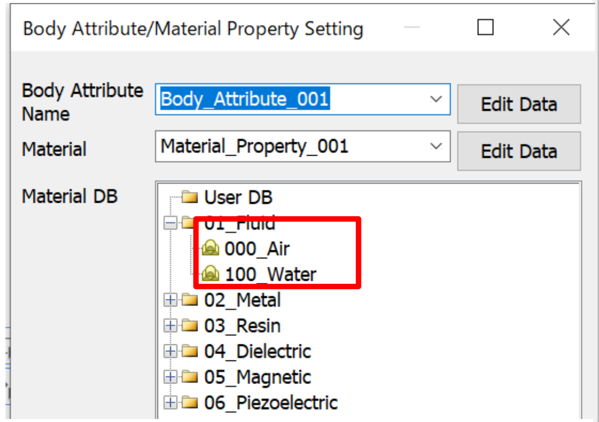
①<u>Analysis Condition</u>

Select fluid analysis.

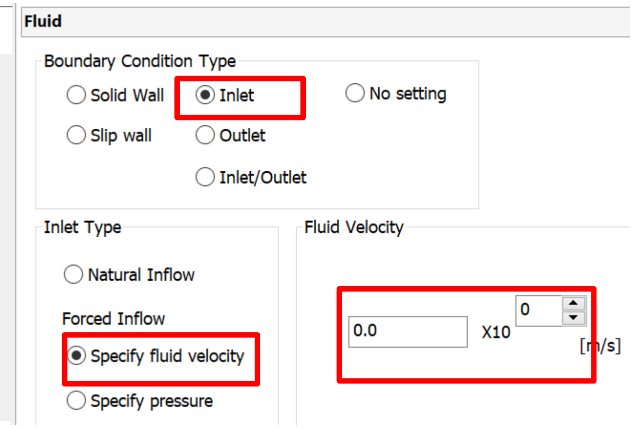
②<u>Body Attribute & Material Property</u>

Type only name for body attribute.
Select air or water from the material DB.

②Select air or water

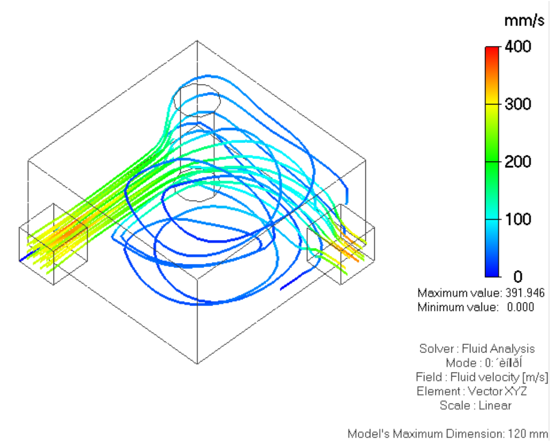


③Set Inlet



③<u>Boundary Condition</u>

[Inlet]: Set forced inflow (flow velocity)
[Outlet]: Set natural outflow

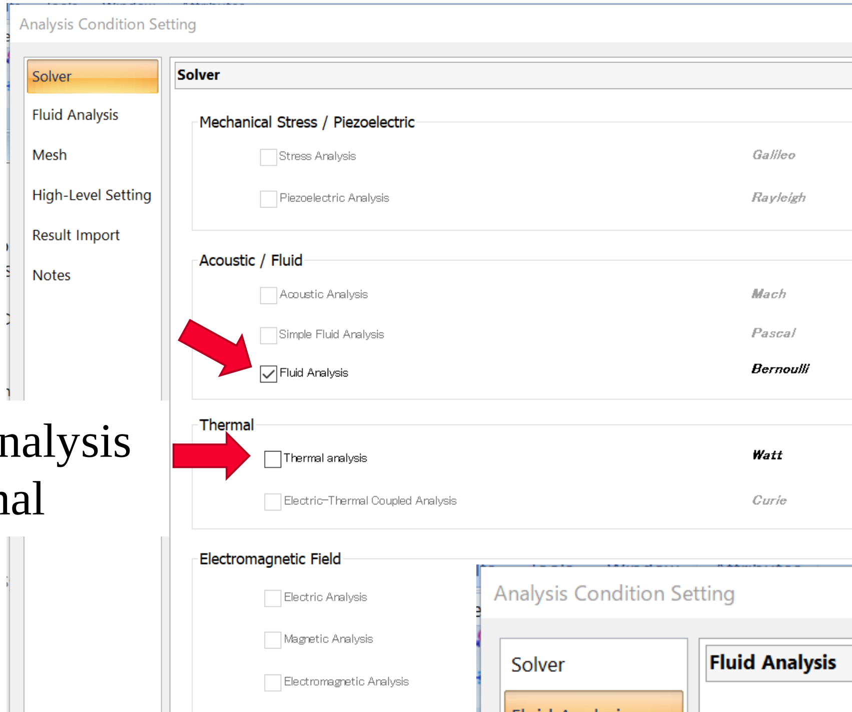


④<u>Mesh Size</u>

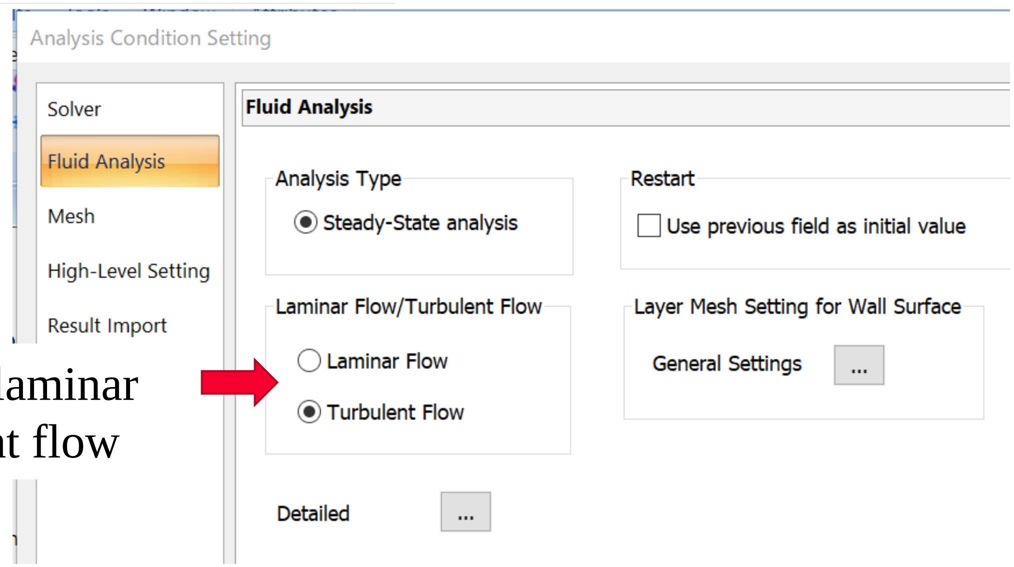
The setting above is for the problem in Tutorial

2-1. Analysis Condition

Analysis Condition



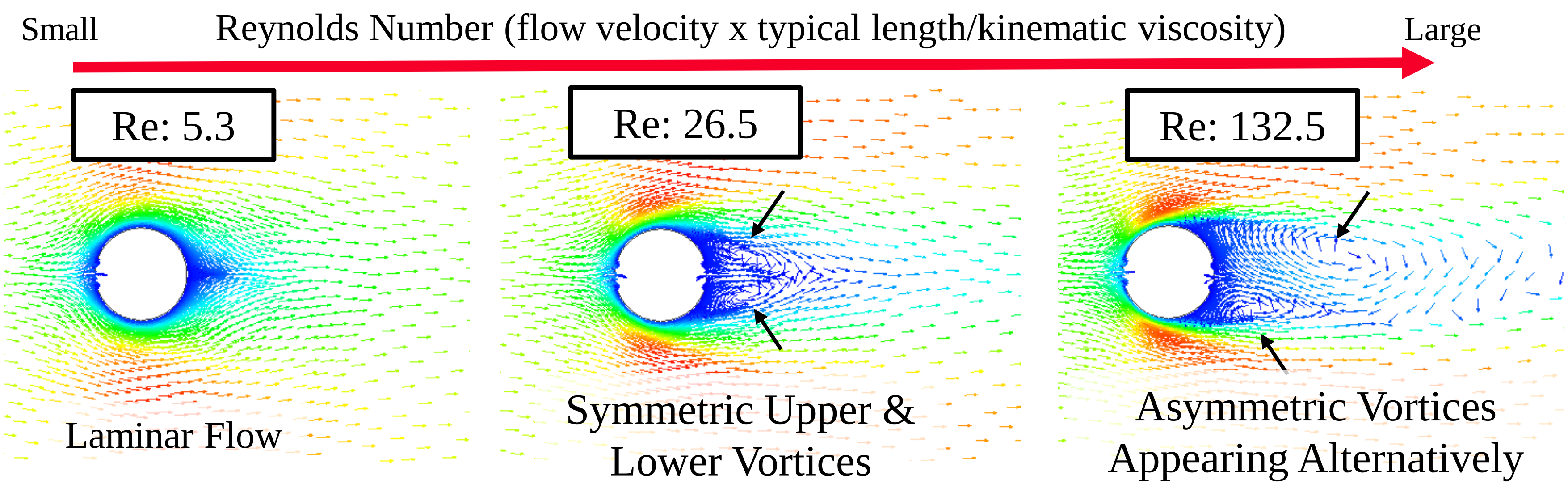
Coupled analysis with thermal



Option of laminar or turbulent flow

Laminar and Turbulent Flow

Murata Software



Characteristics of Turbulent Flow

- Small vortices intersect
- The flow changes irregularly

Problems in Turbulent Flow Calculation

1. For accurate calculation, the meshes need to be fine enough to represent the small vortices.
2. Due to the high irregularity, the steady-state analysis cannot calculate the flow.

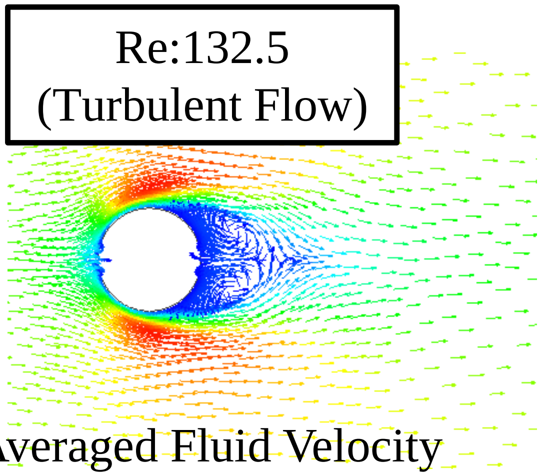
Turbulent Flow Analysis

Murata Software

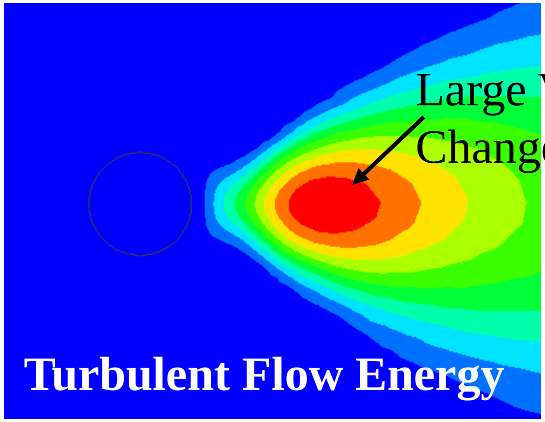
To solve the problems on the preceding page, the irregularity of the turbulent flow is averaged temporally and spatially for calculation. Various turbulent flow models are proposed.

Problem 1. Finer meshes needed to represent the small vortices
⇒ The effect of the small vortices is considered in the analysis model.
The meshes can be relatively coarse.

Problem 2. Steady-state cannot calculate the flow due to its high irregularity
⇒ Temporal averaging makes steady-state analysis possible in some cases.
(If the irregularity is very strong, steady-state analysis may not work.)



Averaged Fluid Velocity



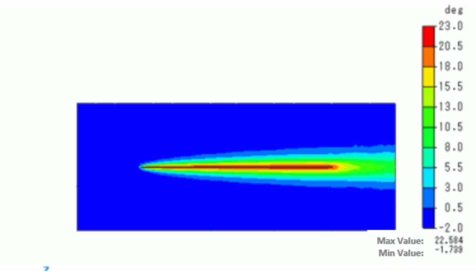
Options of Laminar or Turbulent Flow

Basically, choose the turbulent flow analysis.
If it is evident that the analysis model is laminar flow, select the laminar flow analysis, and save the calculation time.

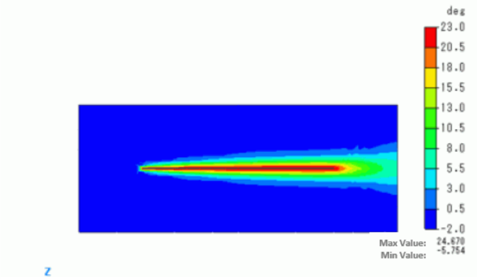
The disadvantages of the laminar flow analysis are;

- If the meshes are coarse, the analysis results tend to be inaccurate.
  - -The vortices smaller than the mesh size are not taken into account in the analysis.
  - -Approximation by the wall function is not executed.
- Due to an attempt to accurately calculate the irregularity by the small vortices, the calculation may not converge.

Fluid-Thermal Analysis Example 1
(The results of the laminar flow analysis are equivalent to those of the turbulent flow analysis)



Laminar Flow Analysis



Turbulent Flow Analysis

2-2. Body Attribute

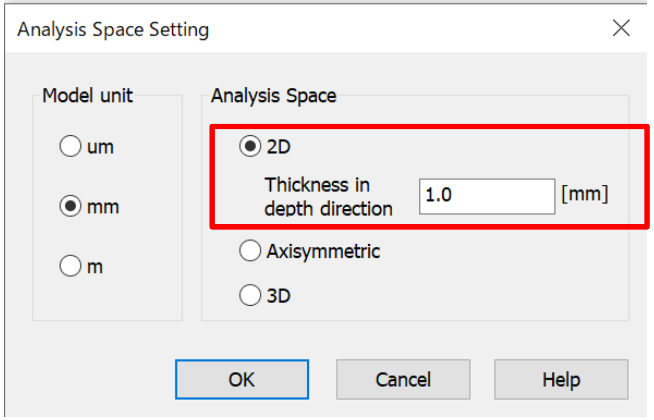
Body Attribute

Murata Software

Fluid Analysis

No settings are needed.
Enter body attribute name only.

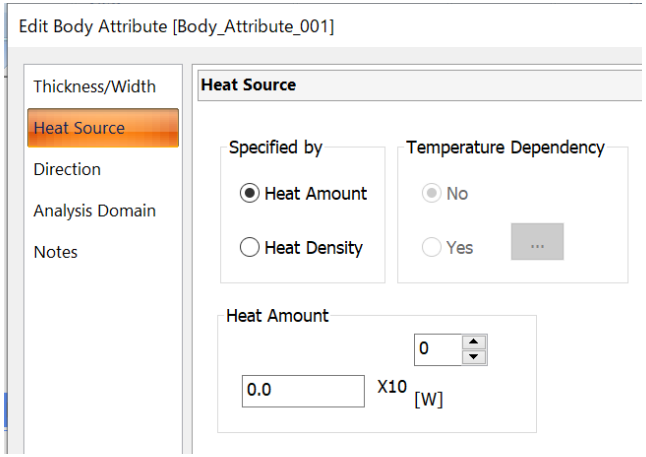
*In the 2D analysis, thickness in depth direction can be specified.
Usually, however, it is specified in the analysis space setting.



Fluid-Thermal Analysis

To analyze the solid, heat source and direction of the anisotropic material can be set.

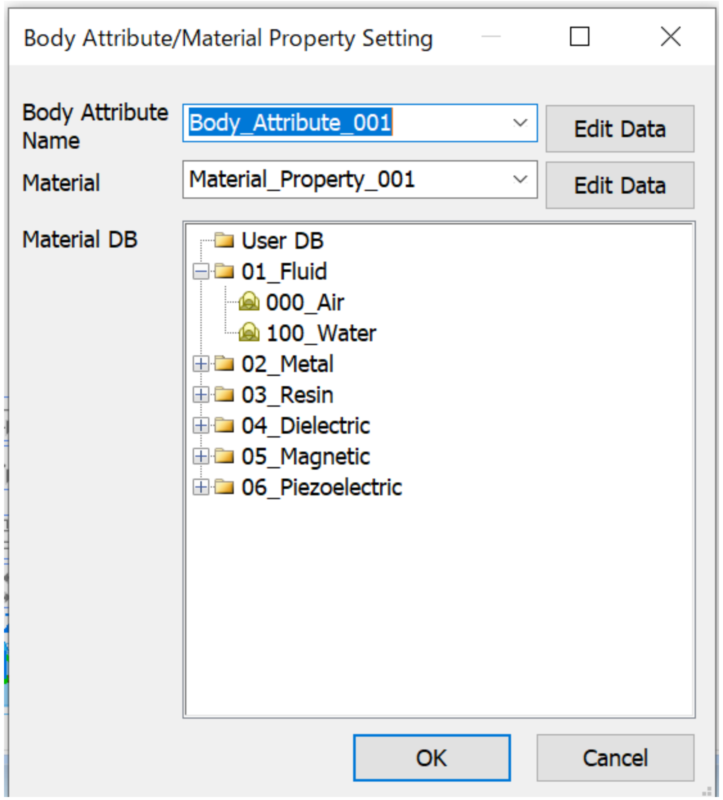
*Setting the heat source to the fluid is not taken into account in the analysis.



2-3. Material Property

Material Property

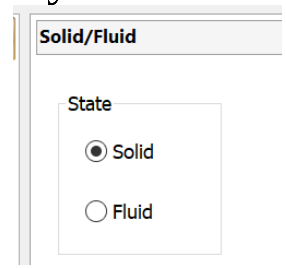
Murata Software



Air and water can be selected from the material DB.

Femtet version 2019.0 or before does not support the temperature dependency of the fluid material

①Specify either solid or fluid.



②Set material properties.

Fluid Analysis

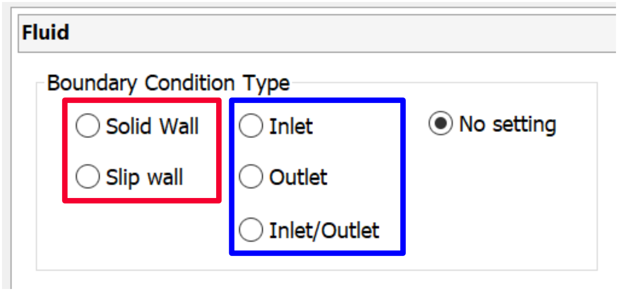
	Solid	Fluid
Density	-	Specify
Viscosity	-	Specify

Fluid-Thermal Analysis

	Solid	Fluid
Specific Heat	Transient analysis only	Specify
Density	Transient analysis only	Specify
Thermal Conductivity	Specify	Specify
Viscosity	-	Specify

2-4. Boundary Condition

Boundary Condition



Wall Boundary Condition: The fluid does not flow in or out

- Solid Wall: Boundary with the solid
- Slip Wall: Virtual wall without inflow/outflow of the fluid

Flow Boundary Condition: The fluid flows in and out

- Inlet: The fluid is known to flow in
- Outlet: The fluid is known to flow out
- Inlet/Outlet (Opening): The fluid is not known if it flows in or out

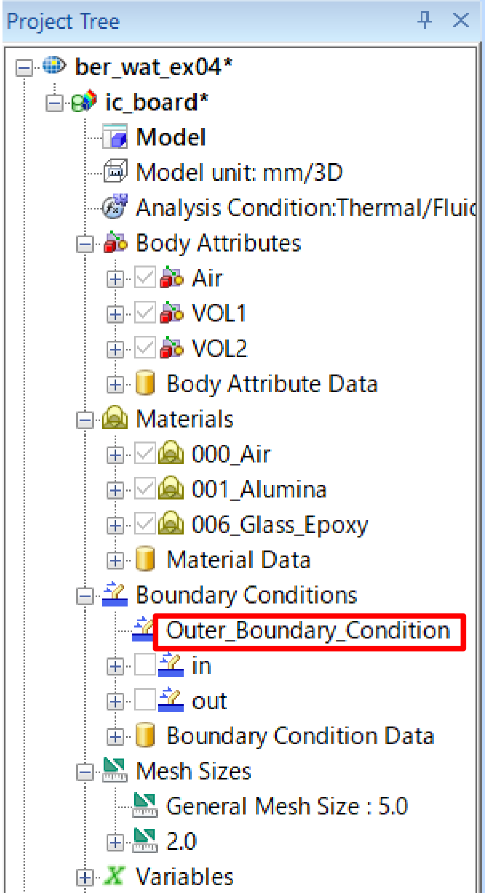
Boundary Condition for Fluid

Surrounding faces of the fluid must be set wall boundary condition or flow boundary condition.

⇒ For a boundary where any condition is not set,
boundary condition will be automatically set when meshing.

For a boundary with solid material ⇒ Solid wall

For other boundaries ⇒ Outer boundary condition



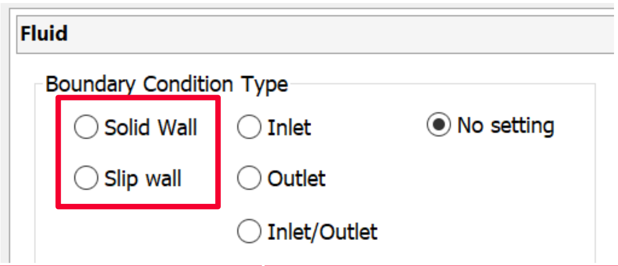
Usually, follow the steps below as the automatic setting above is available.

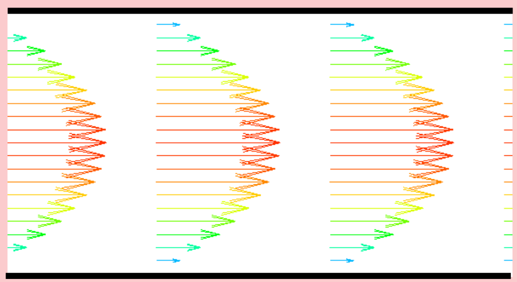
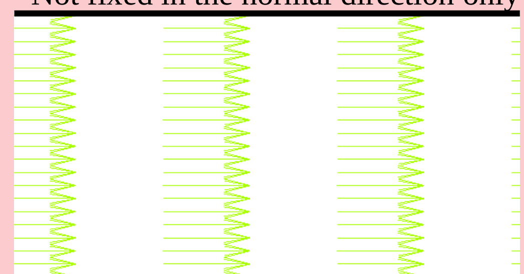
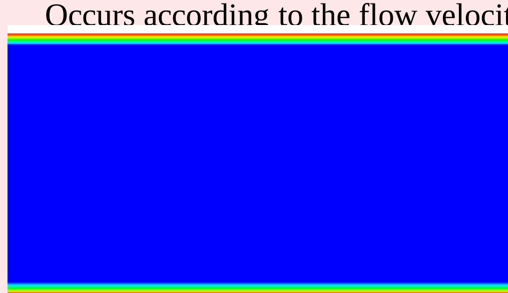
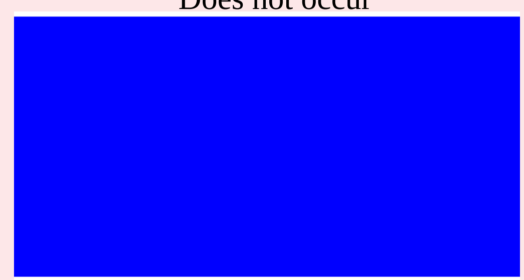
<u>Step 1</u>
Set the wall boundary condition for the outer boundary condition.
(Options of solid wall and slip wall)

* The solid wall is set by default

<u>Step 2</u>
Select the inflow and outflow faces and set the flow boundary condition.

Boundary Condition Type



	Solid Wall	Slip Wall
Flow Velocity on the Wall	Not Fixed 	Not fixed in the normal direction only 
Shear Stress on the Wall	Occurs according to the flow velocity 	Does not occur 
Used As	A boundary with solid (Automatically set for the boundary with a solid body)	A virtual wall with no inflow or outflow of the fluid Symmetric boundary condition

Internal & External Flow

Murata Software

Internal Flow

The flow in the domain surrounded by the solid.

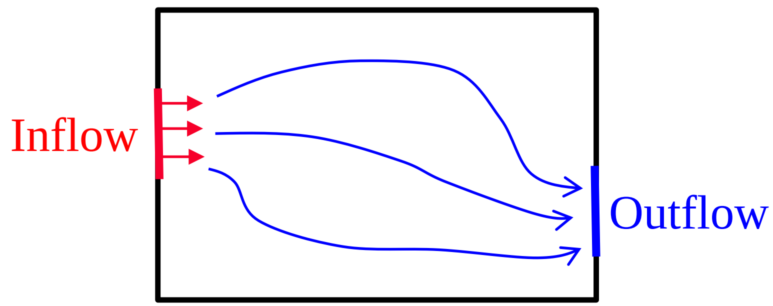
Example: Water flowing in the pipe

External Flow

The flow around the solid.

Example: Air around an airplane

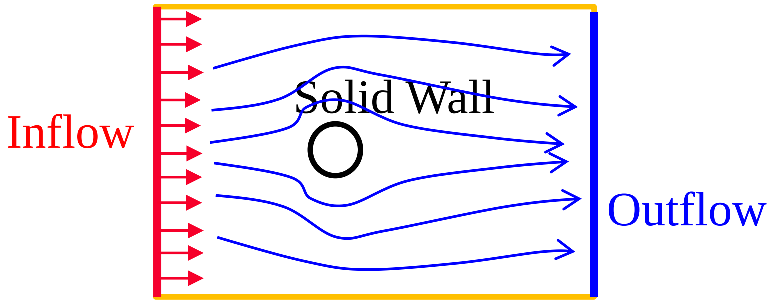
Solid Wall



Solid Wall

Internal Flow Boundary Condition

Virtual Wall (Slip Wall)



Virtual Wall (Slip Wall)

External Flow Boundary Condition

The proper boundary conditions are:

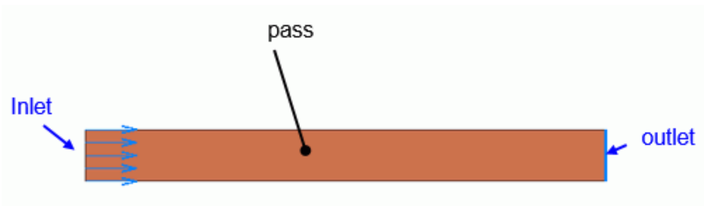
Internal flow: the solid wall except for the faces of inflow and outflow.

External flow: the slip wall except for the faces of inflow and outflow.

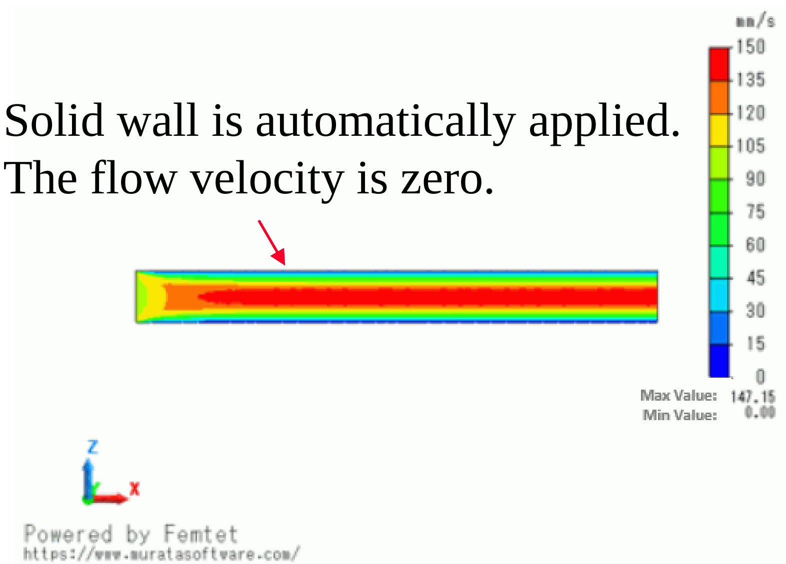
Outer Boundary Condition

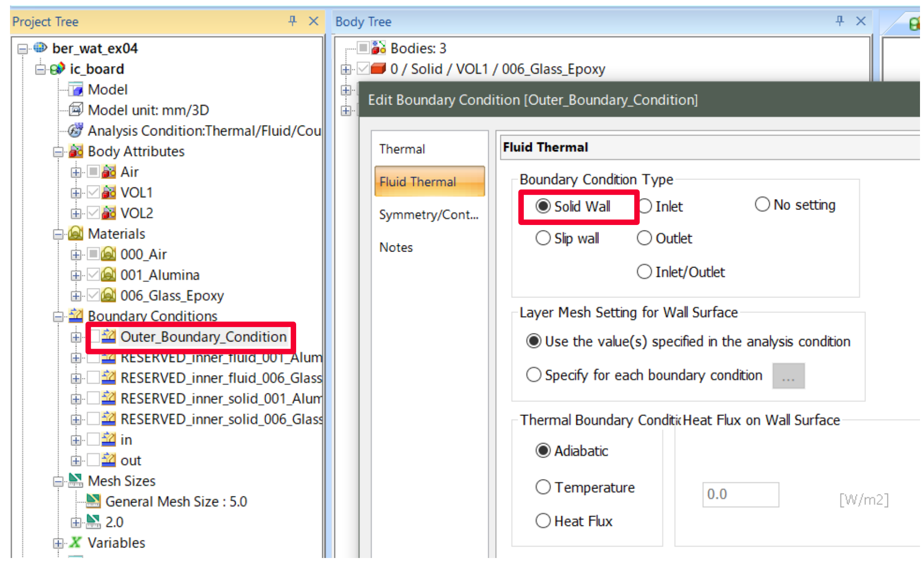
Internal Flow

Fluid Analysis Example 1:
Flow between Parallel Plates



Solid wall is automatically applied.
The flow velocity is zero.





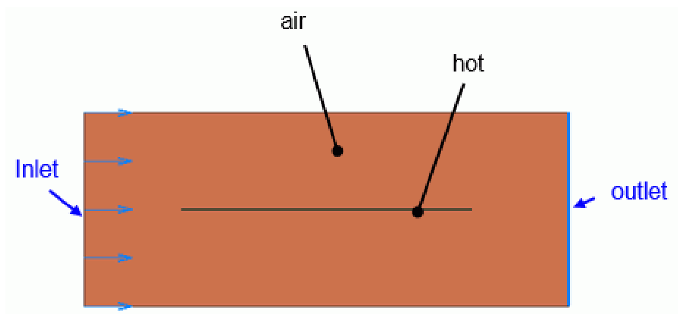
Solid wall is set for the outer boundary condition by default.
Set inlet or outlet only.

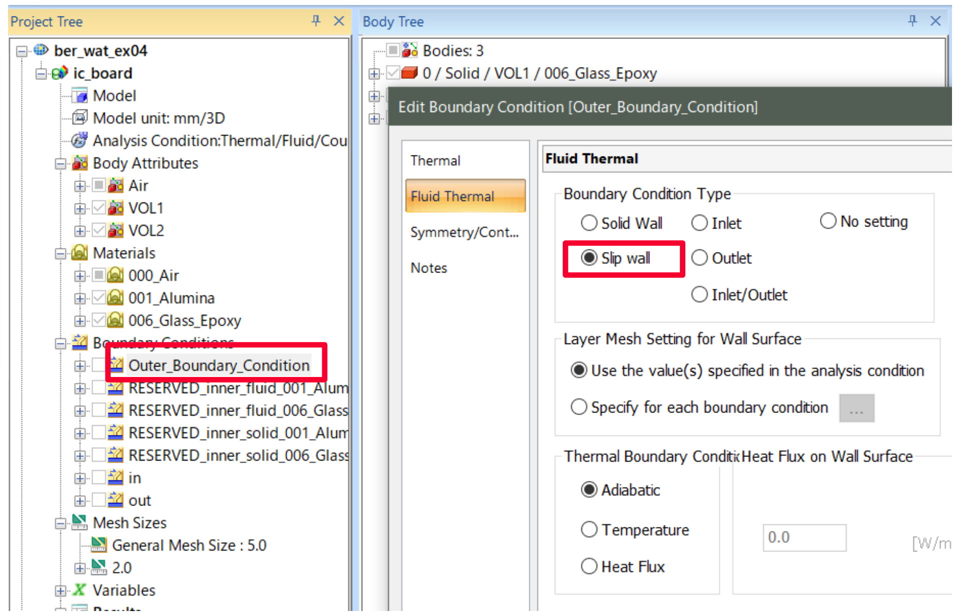
Outer Boundary Condition

Murata Software

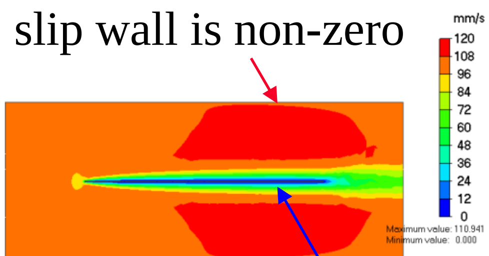
External Flow

Fluid-Thermal Analysis Example 1:
Flow between Parallel Plates





Flow velocity on the slip wall is non-zero



Select slip wall for the outer boundary condition

Powered by Femtet
https://www.muratasoftware.com/en/

Flow velocity on the solid surface is zero

Flow Boundary Condition

Murata Software

Forced Inflow

- Blow the wind inside from the inlet
- Apply force at the inlet
  (pressure difference from the exit?)

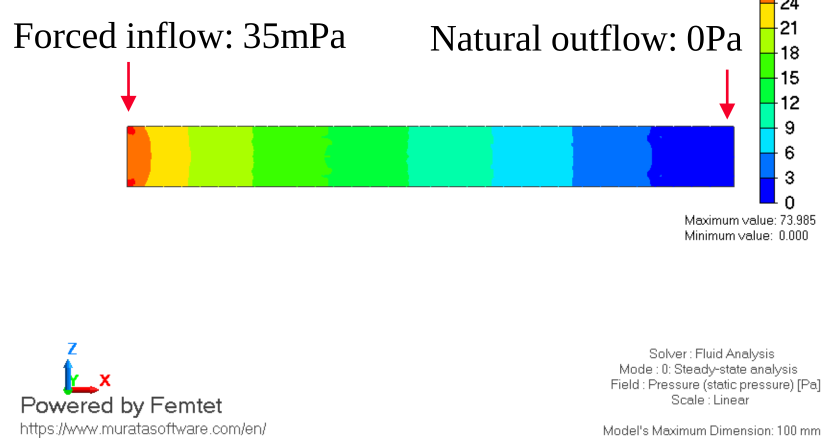
Forced Outflow

- Send the wind out of the outlet
  (ventilation)

Natural Inflow & Natural Outflow

The flow from and to the environment occurs according to the flow velocity and the pressure in the analysis domain where the ambient pressure is 0Pa.

Pressure Distribution



In Femtet, the pressure is the gauge pressure with reference to the environment pressure.

In the example above, the pressure at the inlet is interpreted to be higher than the environment pressure by 35mPa.

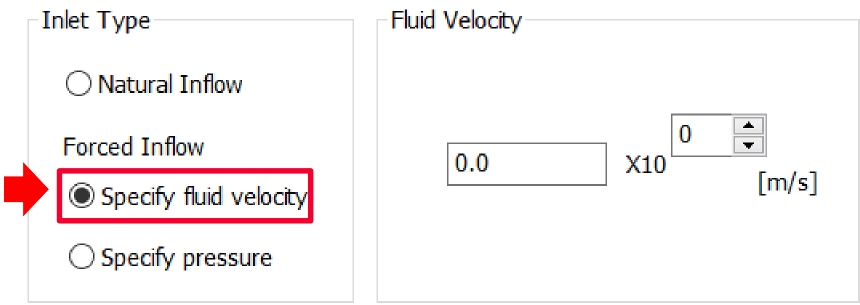
Flow Boundary Condition

Murata Software

Inflow (Inlet)

- Forced Inflow
  - Specify fluid velocity
  - Specify pressure
  - Fan
- Natural Inflow

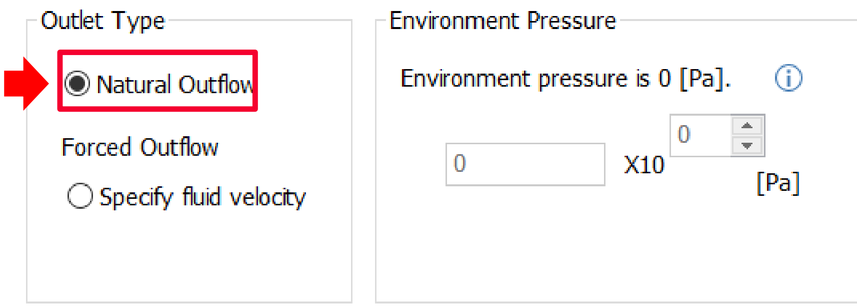
Default setting



Outflow (Outlet)

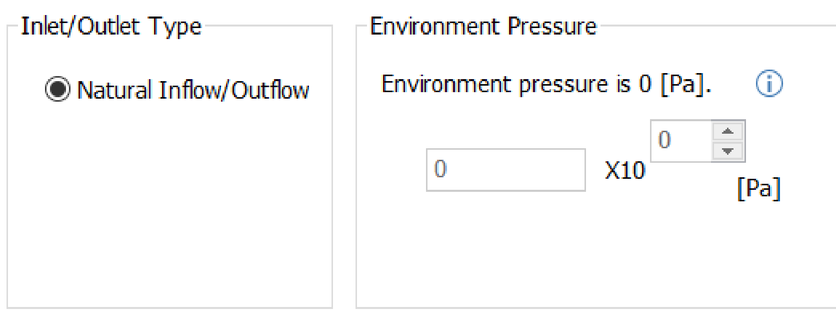
- Forced Outflow
  - Specify fluid velocity
  - Specify pressure
  - Fan
- Natural Outflow

Default setting



Inflow/Outflow (Opening)

- Natural Inflow/Outflow



Combination of Inflow and Outflow

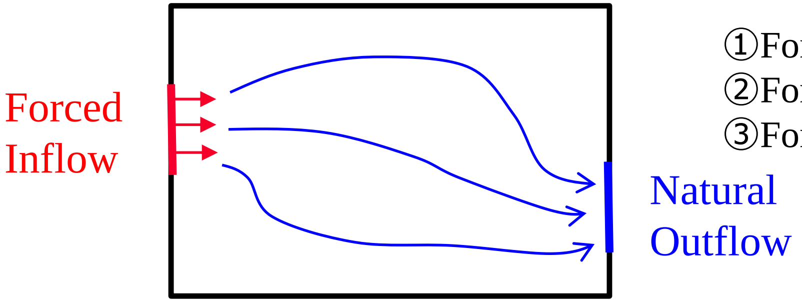
Murata Software

Specify inflow and outflow as a set

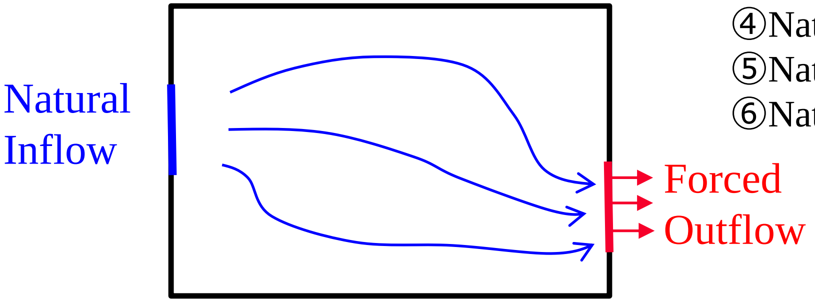
Essential setting as the flow is assumed to be incompressible.

Specify forced boundary and natural boundary as a set

Setting for stable calculation.



①Forced Inflow (flow velocity) - Natural Outflow
②Forced Inflow (pressure) - Natural Outflow
③Forced Inflow (fan) – Natural Outflow



④Natural Inflow - Forced Outflow (flow velocity)
⑤Natural Inflow - Forced Outflow (pressure)
⑥Natural Inflow - Forced Outflow (fan)

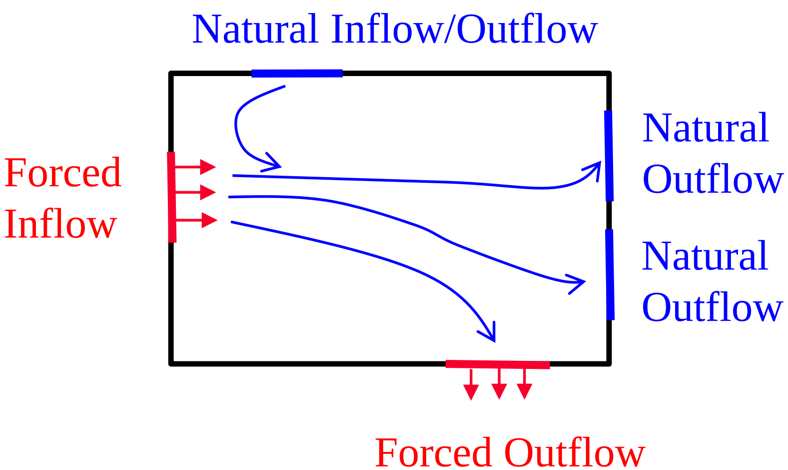
- Setting with forced inflow has higher likelihood of convergence.
- Specifying flow velocity gives higher likelihood of convergence.

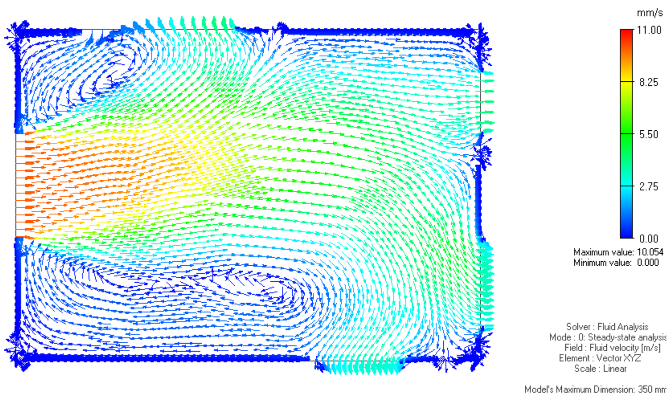
Multiple Inlets and Outlets

Murata Software

If multiple boundary conditions of inlets and outlets exist,
the setting will be as below.

- Set inflow and outflow at least one each.
- Set forced boundary and natural boundary at least one each.

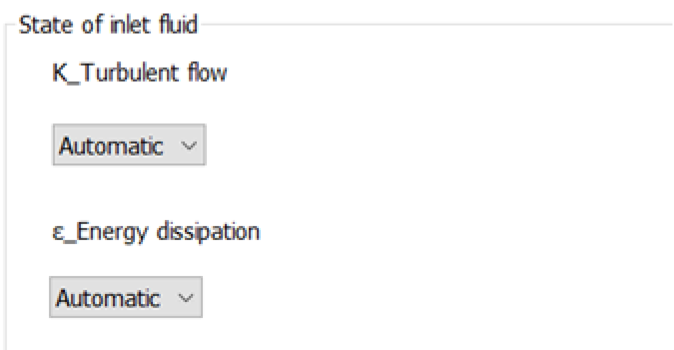




Turbulent Inflow Rate

Murata Software

Set the state of fluid if [inflow] or [inflow/outflow] is selected.
Specify the turbulent flow energy and the energy dissipation rate.



K_Turbulent Flow Energy
- Automatic calculation
- Turbulence intensity
- Direct entry

ε_Energy Dissipation Rate
- Automatic calculation
- Mixing length
- Hydraulic diameter
- Turbulent viscosity ratio
- Direct entry

<Typical Values for Inflow K and ε >

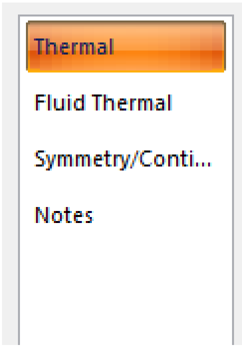
<u>Weak turbulent flow (external flow, etc.)</u>
・K turbulence intensity = 1[%]
・ε turbulent viscosity = 1

<u>Strong turbulent flow (internal flow, etc.)*</u>
・K turbulence intensity = 5[%]
・ε hydraulic diameter (cross section of the flow path is assumed to be circular)

Settings above are for automatic calculation.
*If a solid surrounds the inlet boundary, use the values for the strong turbulent flow.

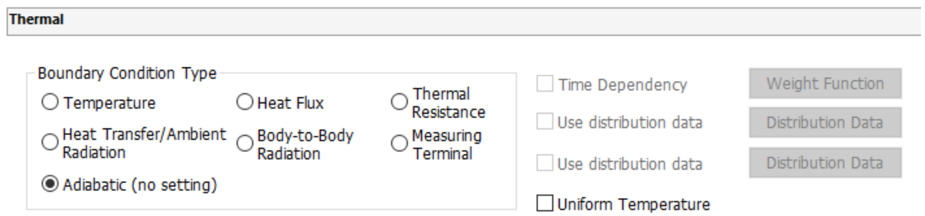
There are various ways for the setting.
If you are not sure, select the automatic calculation.

 Murata Software



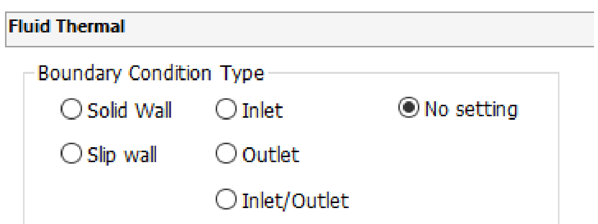
Thermal Tab

Boundary condition of the solid surface is set as in the thermal analysis.



Fluid Thermal Tab

Boundary condition of the fluid surface is set.



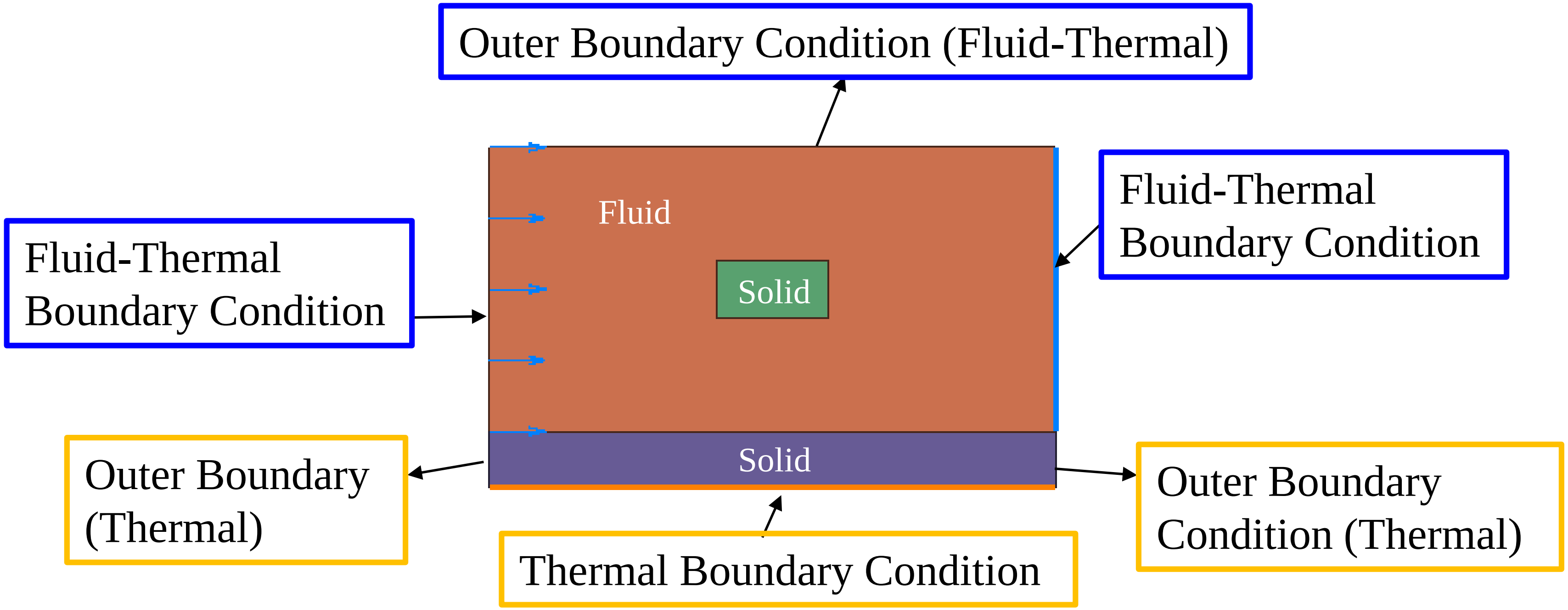
Condition Settings for Fluid

Wall Boundary Condition

- Adiabatic
- Temperature: specify value
- Heat flux: specify value

Inlet, Inlet/Outlet Boundary Condition

- Inflow temperature: specify value



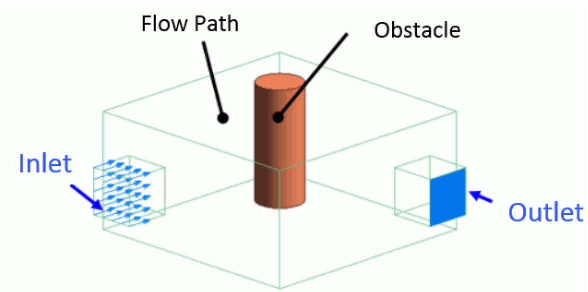
- For the fluid surface, [Fluid-Thermal] tab is used for setting.
  (If no condition is set on the fluid surface, the settings on the [Fluid-Thermal] tab in the outer boundary condition setting will be applied.)
- For the solid surface [Thermal] tab is used for setting.
  (If no condition is set on the solid surface, the settings on the [Thermal] tab in the outer boundary condition setting will be applied.)
- Basically, the setting on the boundary of solid and fluid is not required.
  Only if the radiation from the solid surface is taken into account, the radiation setting on the solid surface is required.

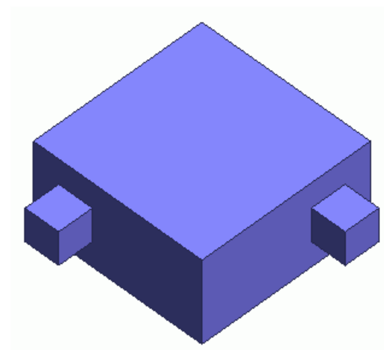
3. Modeling

Modeling

The fluid is defined by a single body.
If the fluid consists of multiple bodies, they must be united by the Boolean operation.
Note: If multiple fluids

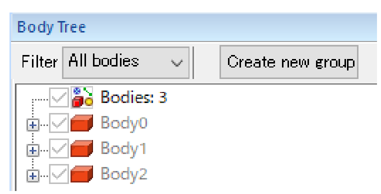
Tutorial Model



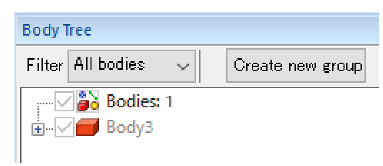


A flow path consisting of three boxes

Unit by Boolean operation



Before Unite

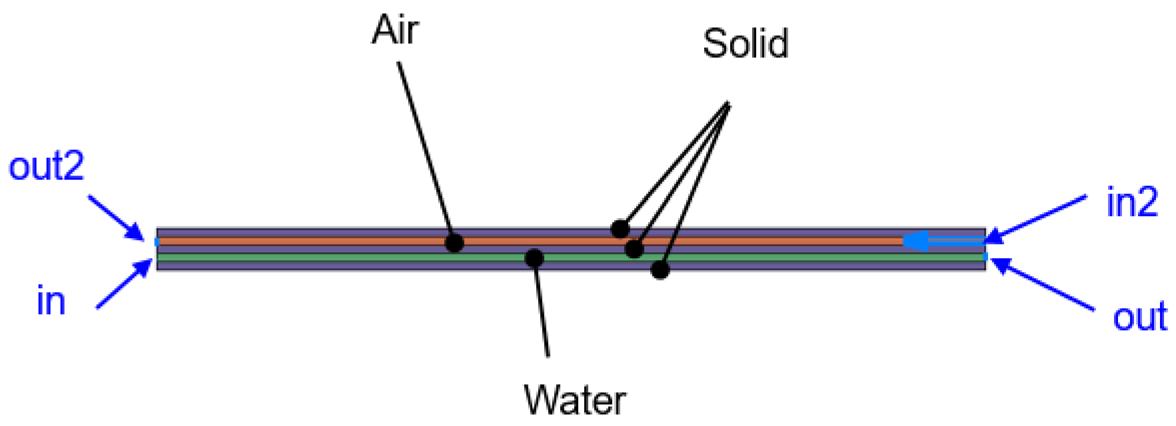


After Unite

Modeling

If the fluid bodies are adjacent to each other, an error will occur, but if they are not, there will be no problem.

Different kinds of fluids can be analyzed.

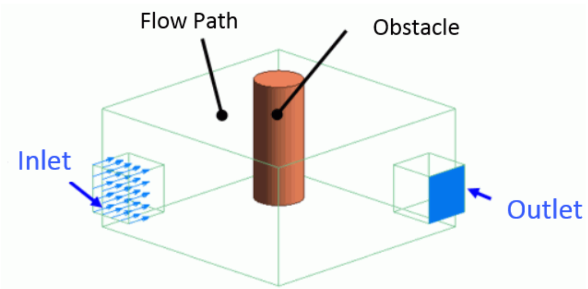


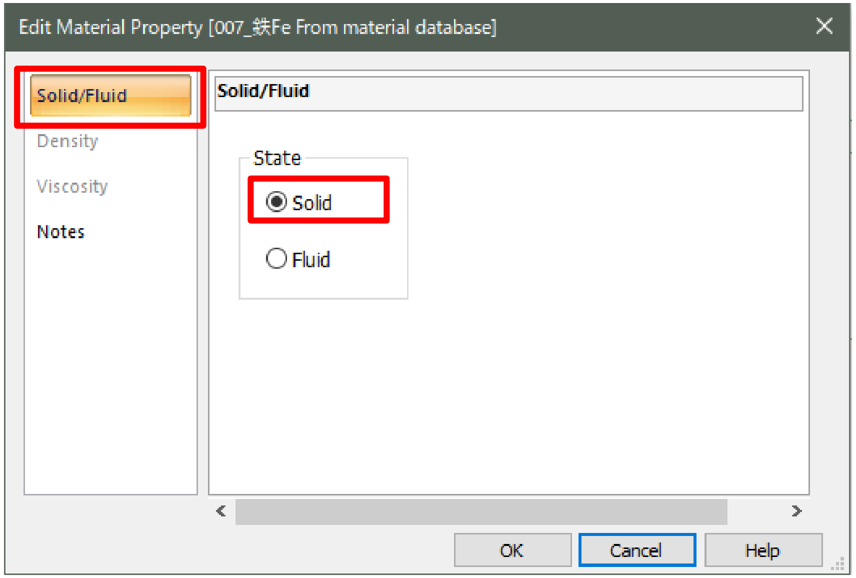
Heat transfer between water and air flowing in different flow paths

Modeling

A material in a solid state is considered to be an obstacle.

Tutorial Model





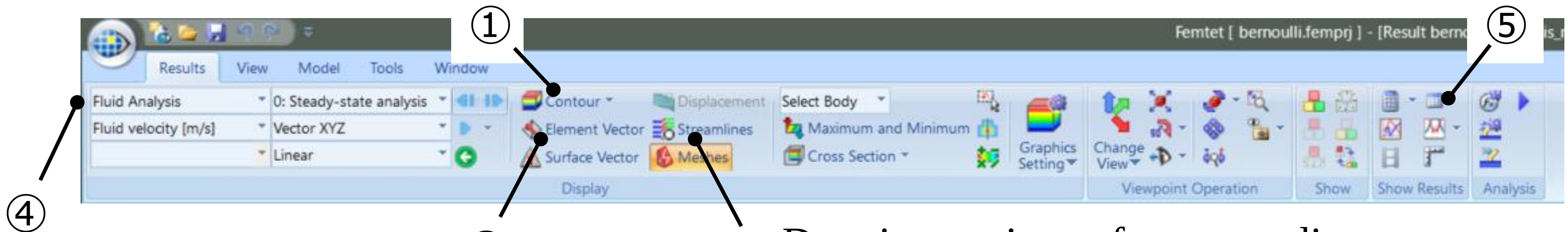
*Unless it is fluid-thermal analysis, any material can be selected from the material DB.

Boolean Subtract with fluid is not required

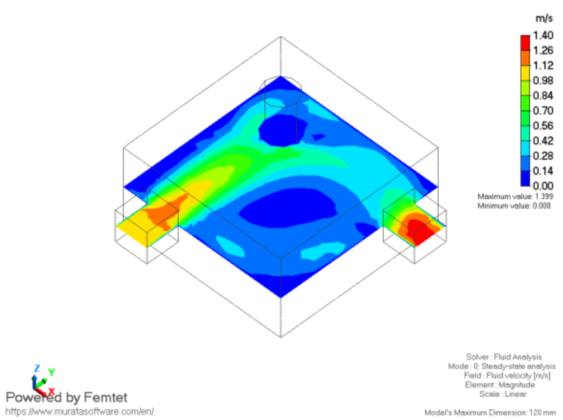
4. Results Display

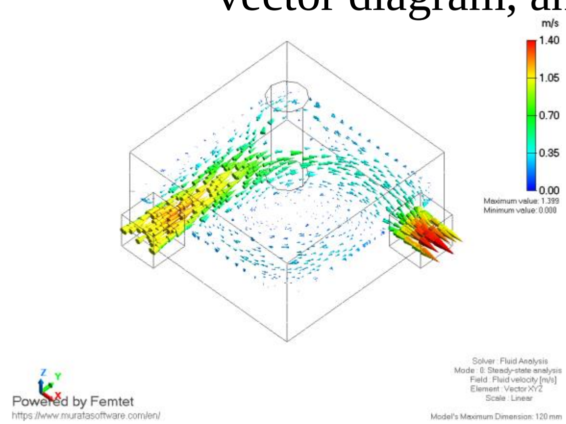
Displayable Results

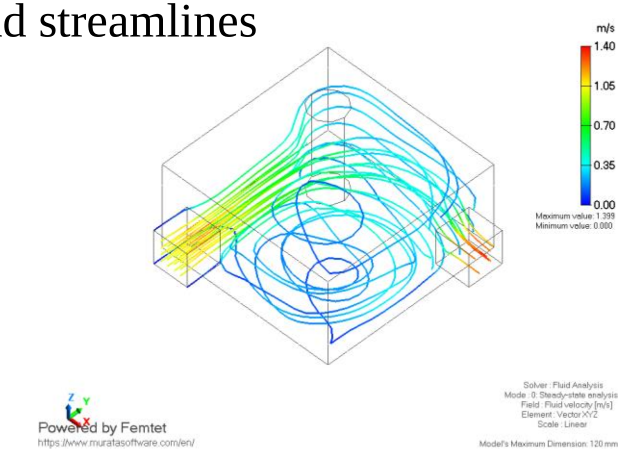
Murata Software



③ Drawing settings of contour, diagram, vector diagram, and streamlines





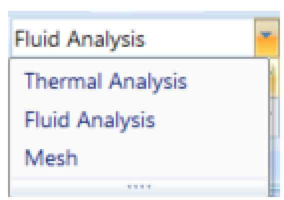


①Contour diagram of flow velocity

②Vector diagram of flow velocity

③Streamlines

④Solver selection (for fluid-thermal analysis)

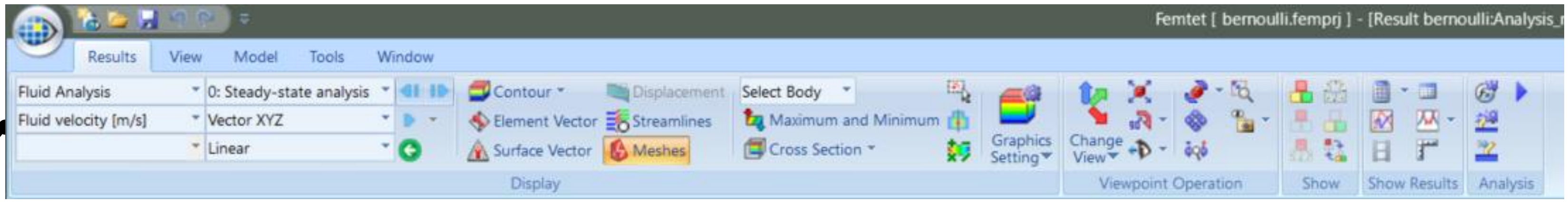


⑤Results table

Displayable Results Fields

Murata Software

Flow velocity vectors, pressure, temperature, and heat flux vectors, etc. are displayable.



Fields are switched here.

Quantities relating to fluid/solid boundary

Quantities relating to turbulent flow

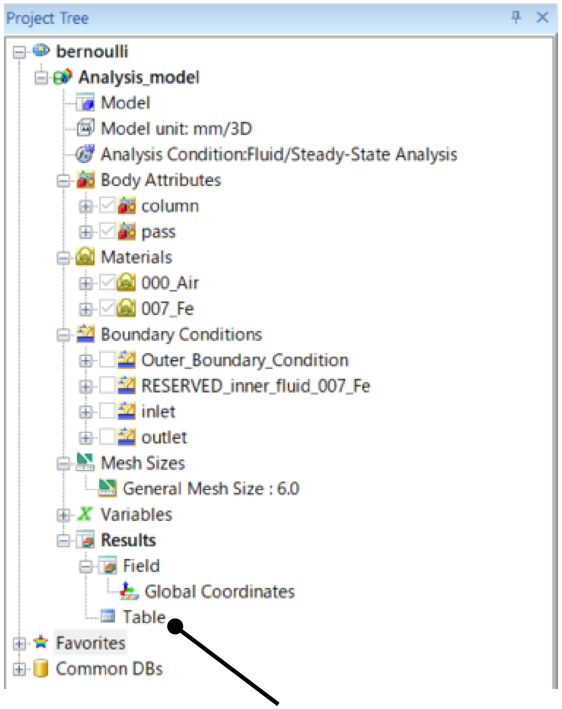
Quantities relating to mesh status

Quantities relating to turbulent flow

Quantities relating to fluid/solid boundary

Quantities relating to turbulent flow

Displayable Results Table



Results table display

	Display Condition	Notes
Force on Wall Face[N]	Boundary Condition ・Solid wall ・Slip wall	The force given to the wall face from the solid. Drag and lift can be calculated.
Volumetric Flow Rate[m3/s]	Boundary Condition ・Inlet ・Outlet ・Inlet/Outlet	Volumetric quantity per unit time flowing in and out through boundary.
Pressure Loss[Pa]	Boundary Condition ・Inlet-Outlet	Pressure loss between inlet and outlet.
y+ Distribution[%]	Boundary Condition ・Solid wall	The ratio of the y+ (dimensionless height of the 1st-layer mesh) in the five domains[%]
Mesh Height of1st Layer[m]	Boundary Condition ・Solid wall	・Average value ・Recommended value of the 1st layer mesh height
Convergence Judgment	-	Convergence/Non-convergence The number of iterations

y+ range and the height of the 1st layer mesh will be explained in detail in [6. Mesh Setting near Wall Surface]

Reynolds Number

Reynolds number is an indicator of turbulence state of the flow.
Where the forced inflow and forced outflow are set, the values are displayed, which are calculated based on the flow velocity, hydraulic diameter, and material property.

```
Bernoulli
<< Steady-state analysis: Turbulent Flow Model: Realizable K-epsilon Model: Double Layer Model>>
Advection Scheme
Fluid Velocity: 2nd-order Upwind Differencing Scheme
K: 1st-order Upwind Differencing Scheme
Epsilon: 1st-order Upwind Differencing Scheme

Forced inflow  Flow velocity specified: Boundary condition [inlet]
Indide/Outside: Internal Flow
Fluid velocity=5.000e+01 [m/s]
Hydraulic diameter (typical)=1.000e-02 [m]
Effective cross-sectional area=9.762e-06 [m2]
Cross-sectional area=1.000e-05 [m2]
Kinetic viscosity= 1.510e-05 [m2/s] Material property [000_空気]
Reynolds number:  33104
```

Hydraulic Diameter (2D)

$$l = L\ [m]$$

Hydraulic Diameter (3D)

$$l = \frac{4S}{L_{all}}\ [m]$$

where
L $[m]$: Edge length
L_{all} $[m]$: Perimeter length
$S[m]$: Area

* For the forced inflow (pressure specified), the pressure is converted to the flow velocity.

$$U_{max} = \sqrt{\frac{2P}{\rho}}\ [m/s]$$

Typical length can be given in various ways.
In the case of flows around a cylinder, its diameter is used, and conversion is required.

5. How to Cope with Non-convergence

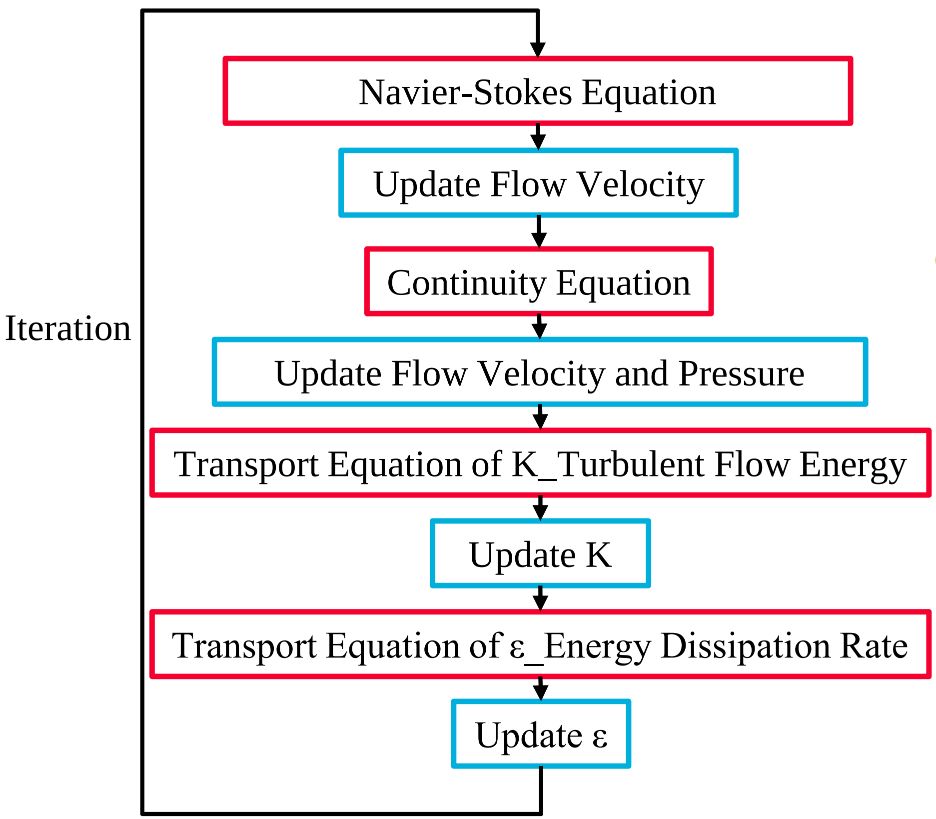
For the details, refer to
Home > Technical Note > Fluid Analysis/Fluid-Thermal Analysis >
If fluid Analysis/Fluid-Thermal Analysis Does Not Converge

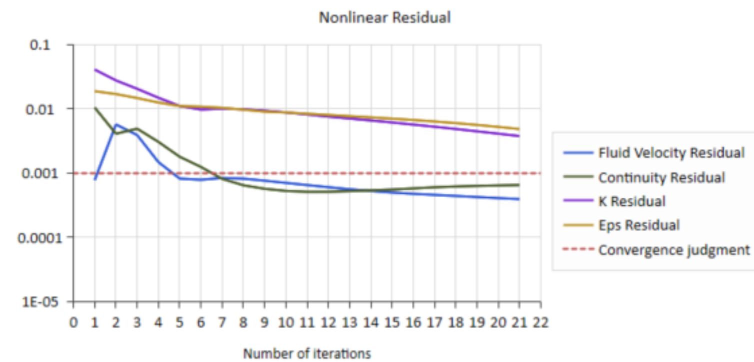
5-1. Iterative Calculations

Iterative Calculations

Murata Software

In the fluid analysis, the same calculation is repeated to reach the right state.
In one iteration, flow velocity, pressure, K, and ε are solved in this sequence.
The calculation is repeated until the residual in each equation becomes lower than certain value.





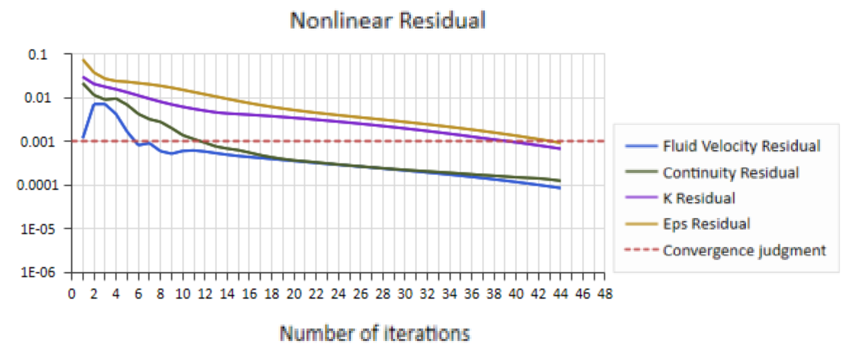
Residual in	Equation
Flow Velocity	Navier-Stokes Equation
Continuity	Continuity Equation
K	Transport Equation of K
ε	Transport Equation of ε

Residual is an indicator of difference between the solution of equation and the actual results of calculation. Smaller residual indicates the calculation results are closer to the solution of equation.

Convergence and Non-convergence

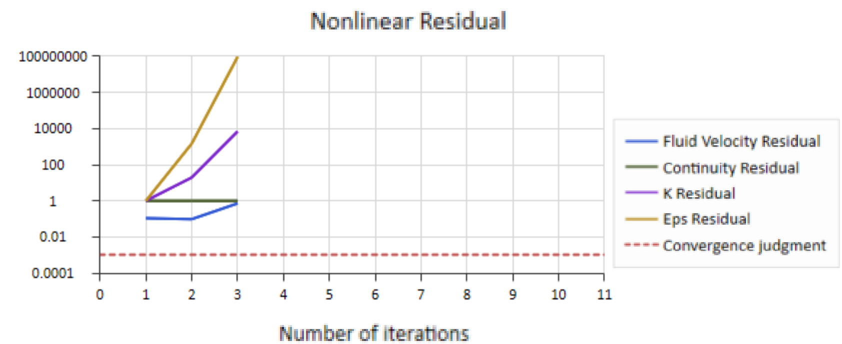
①Convergence

Residuals are small enough.
The results can be said right.



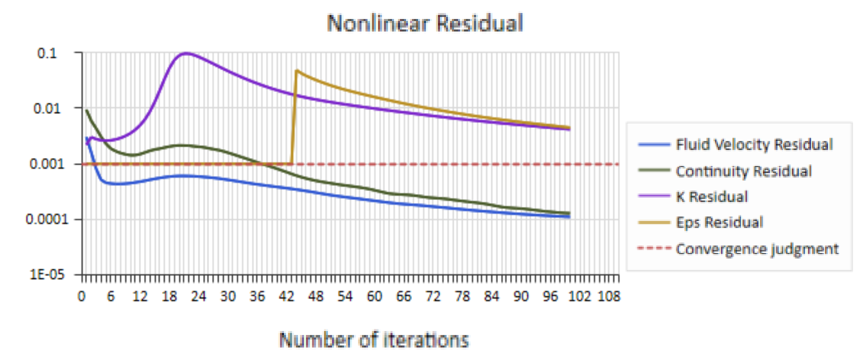
②Divergence

Residuals become larger while calculations are iterated.
Doesn't make sense.



③Non-convergence

Residuals do not become small and accuracy is low.
The results are not far wrong.

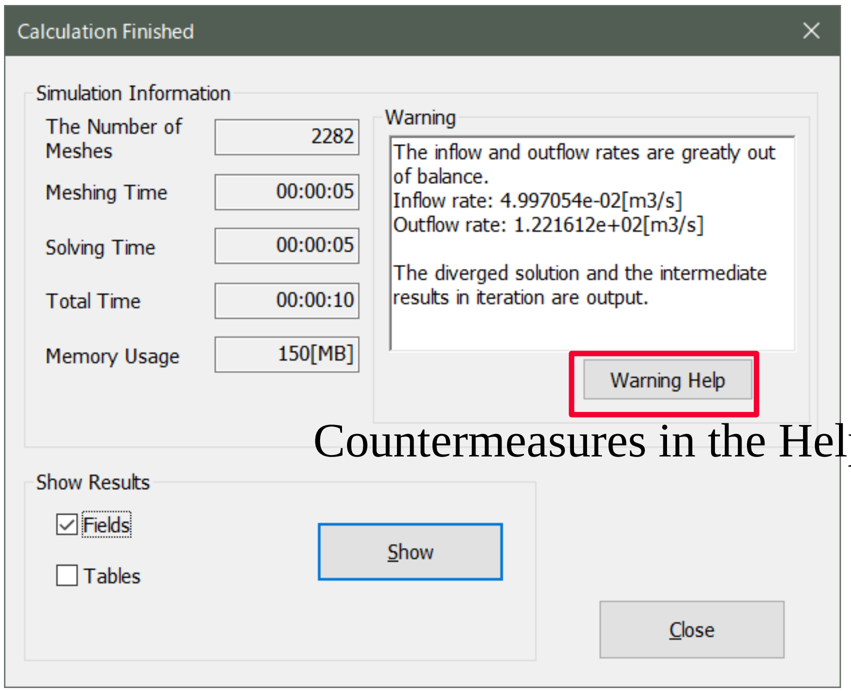


5-2. Divergence

If Calculation Diverged

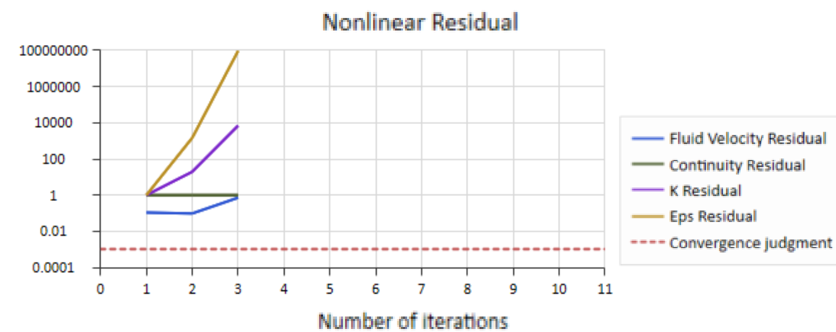
Murata Software

A warning will appear if the calculation diverged.
Press the Warning Help for countermeasures.



Countermeasures in the Help

Divergence



Countermeasures for Divergence

①Mesh Modification

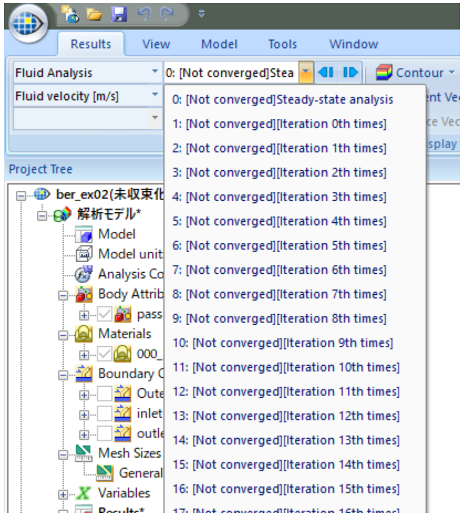
The divergence may be caused by;
- Meshes of poor quality
- Aspect ratio of the layer mesh (mesh size/mesh height)

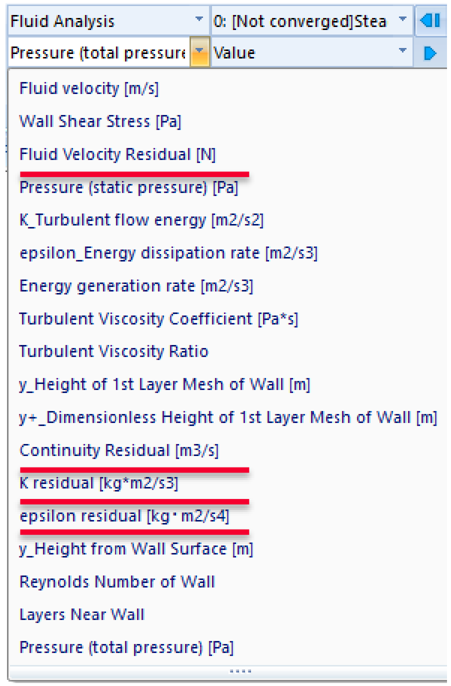
The possible solutions are:
- Make the mesh size smaller.
- Change the mesh size and remove the meshes of poor quality.
- Make the higher layer meshes.

②Analysis Condition Change

In [Fluid Analysis tab] > [Detailed Setting],
- Set the smaller relaxation coefficient.
  *Divergence can be prevented although the calculation time becomes longer.
- Try 1st-order upwind differencing scheme.
  *Divergence can be prevented but the accuracy may be deteriorated.

Identify Where Did Not Converge



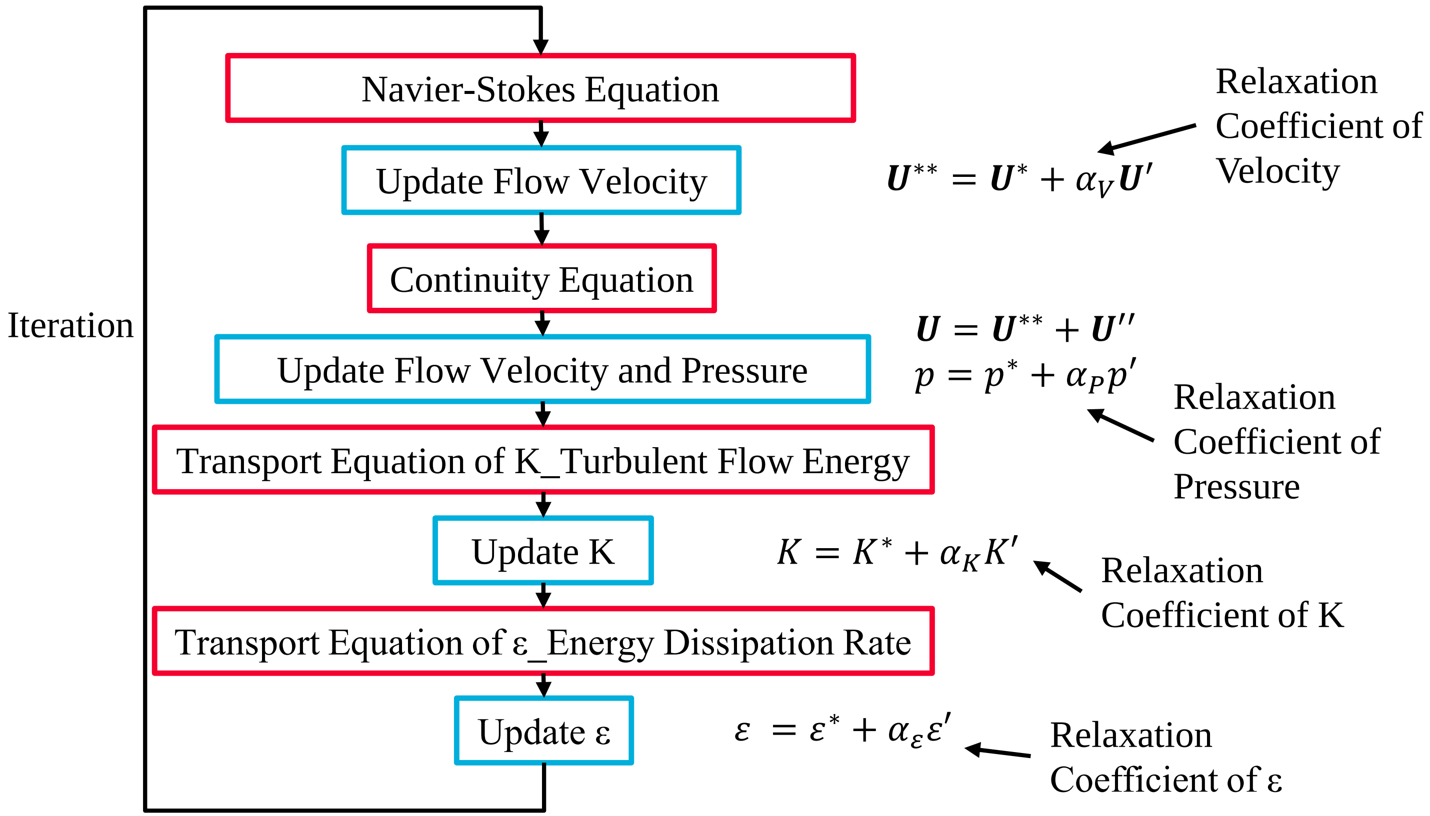


In the case of divergence or non-convergence, the intermediate results can be viewed.
The displayable results are flow velocity residual, continuity residual, K residual, and ε residual.

The cause of the non-convergence can be identified.
Where the residual values are extremely large, if the mesh quality is poor, it must be improved.

Relaxation Coefficient

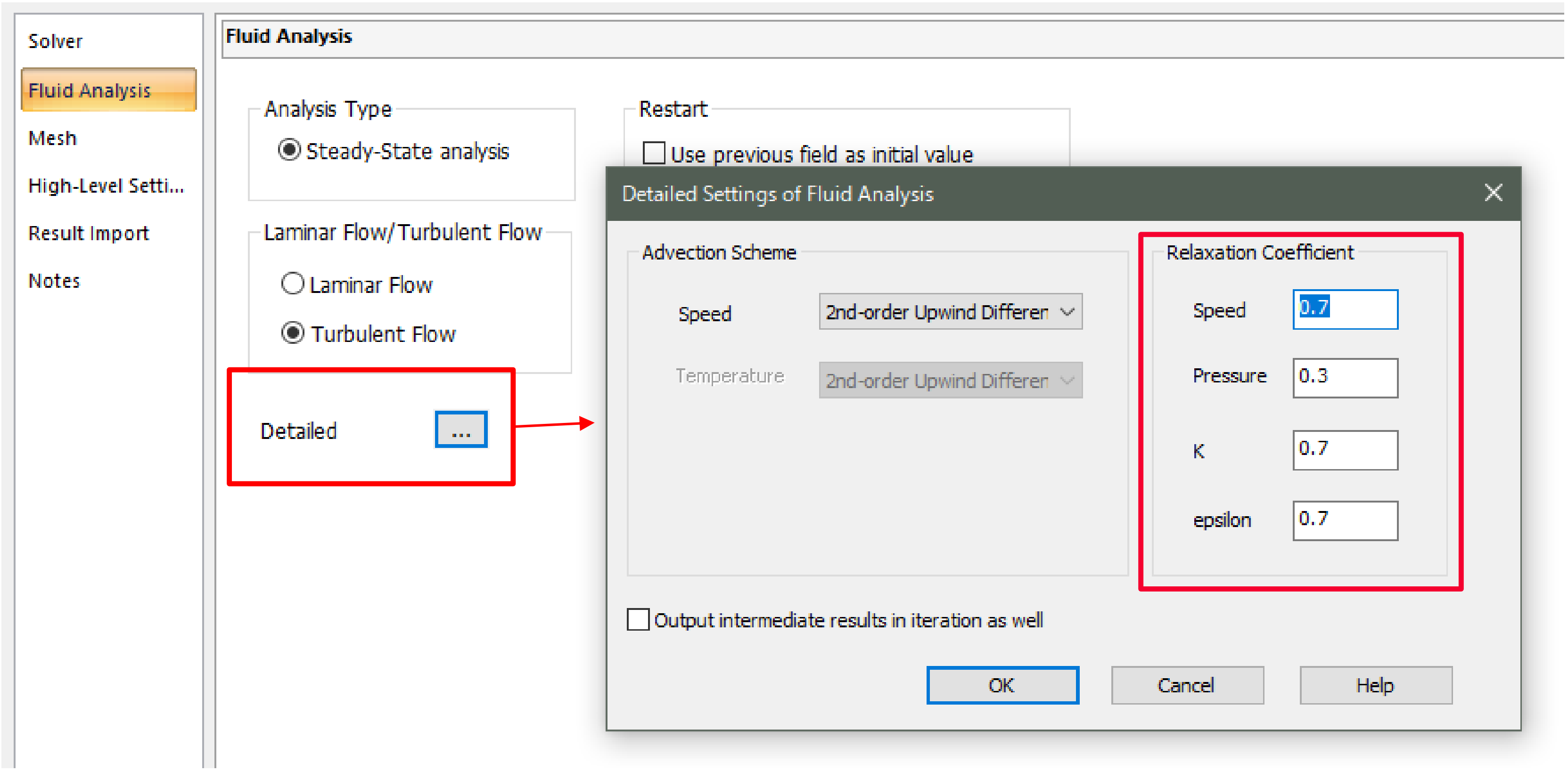
Murata Software



Updating quantity is set smaller for each iteration in order to prevent the divergence. (relaxation coefficient <1)
Smaller the relaxation coefficient, less likely the divergence. However, it increases the iterations and takes longer time for analysis.

Setting Relaxation Coefficient

Murata Software

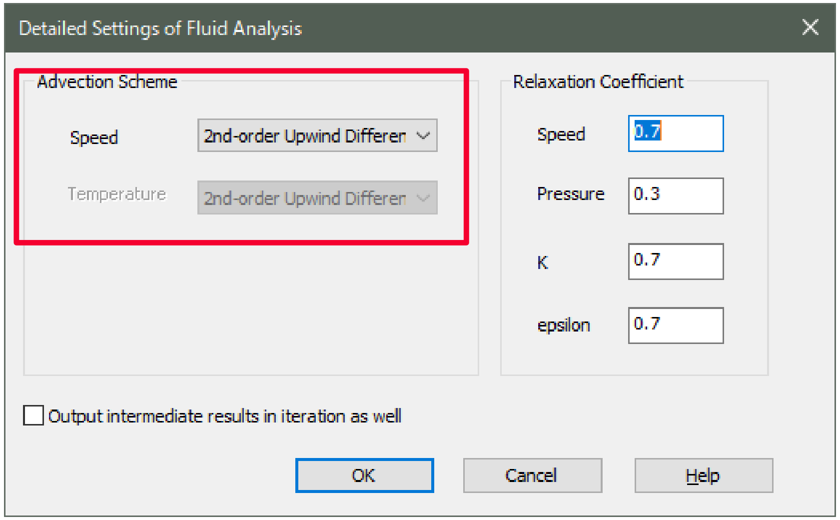


The cause of divergence is often found in the updating of speed and pressure.
At first, it is recommended to reduce the relaxation coefficients of speed and pressure.

Advection Scheme

Murata Software

When calculating the advection in the fluid analysis, it is known that the results by the upwind differencing scheme is closer to the analysis results of the material of high viscosity due to the numerical viscosity.



1st-order Upwind Differencing Scheme

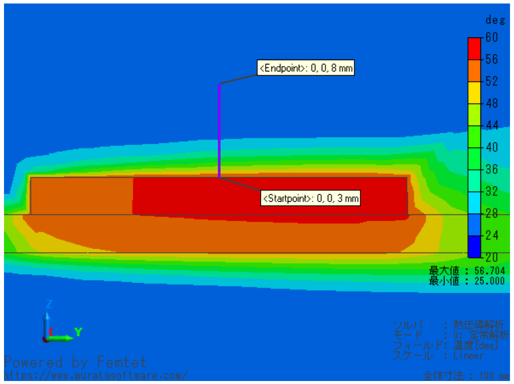
The numerical viscosity is large and the accuracy is poor.
The flow velocity difference and temperature difference are smaller than actual.
Convergence is good.

2nd-order Upwind Differencing Scheme (set by default)

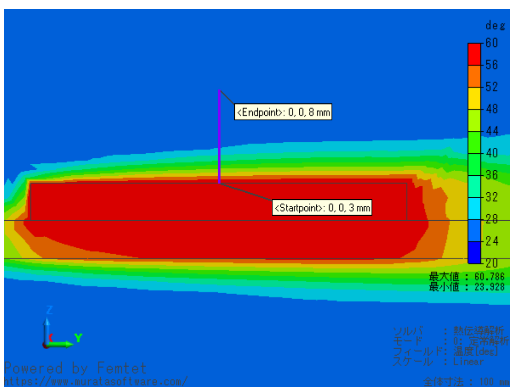
The numerical viscosity is small and the accuracy is good.
Convergence is poor (too many iterations and divergence in some cases).
Unusual flow velocity or temperature may appear locally.

Upwind Differencing Methods

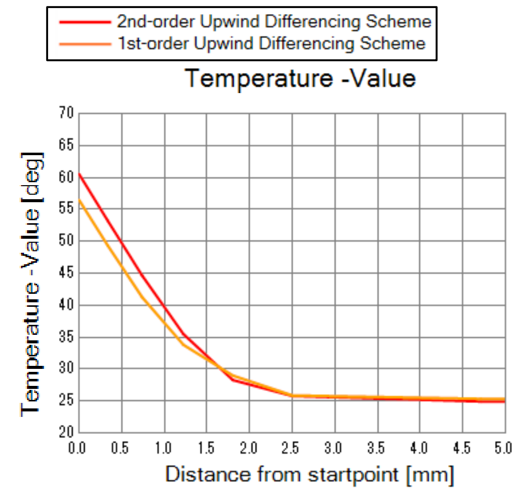
Fluid-Thermal Analysis Example 3
with two different advection calculation methods for temperatures



1st-order Upwind Differencing Method



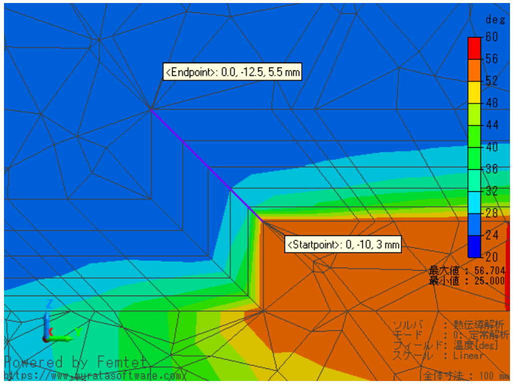
2nd-order Upwind Differencing Method



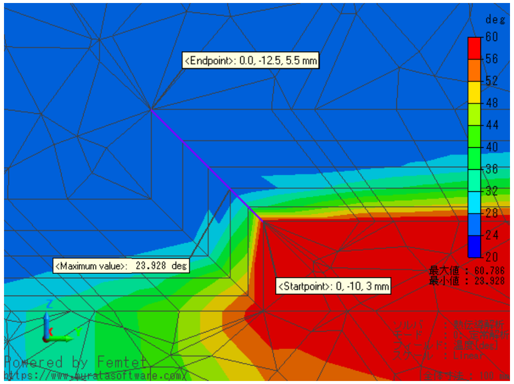
1st-order upwind differencing method tends to show the lower temperatures of the heat source.

Upwind Differencing Methods

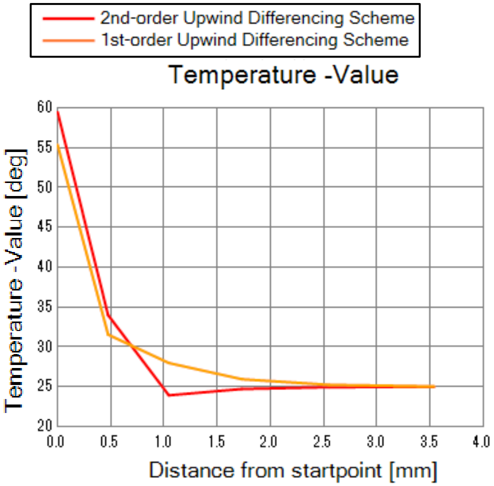
Fluid-Thermal Analysis Example 3
with two different advection calculation methods for temperatures



1st-order Upwind Differencing Method



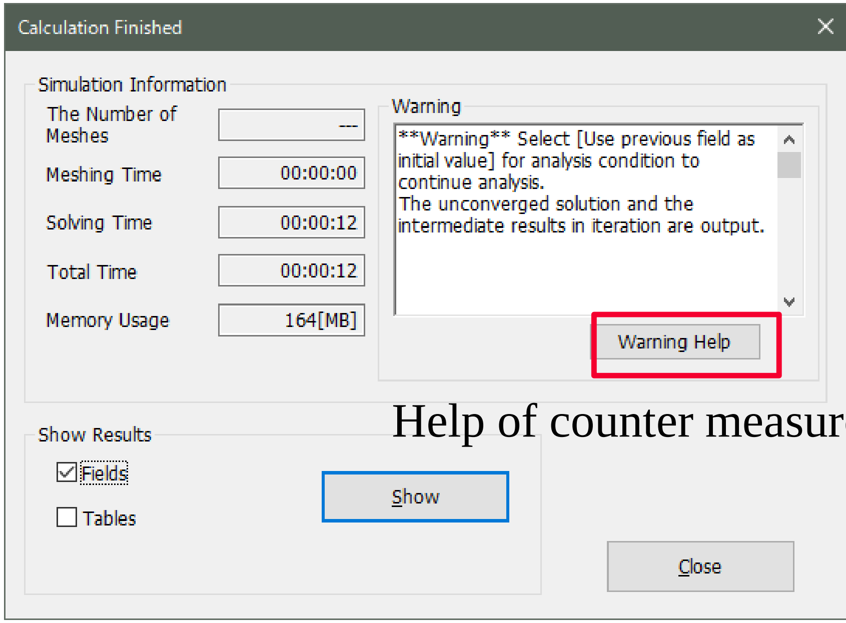
2nd-order Upwind Differencing Method



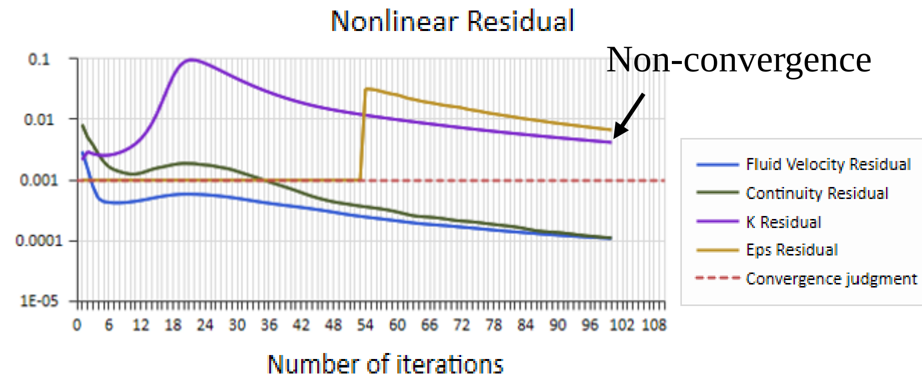
Since the calculation is for the heat source and the air of 25°C, the temperature should not be below 25°C. In the 2nd-order differencing method, a local undershoot occurs.

5-3. Non-convergence

Non-convergence



Help of counter measure

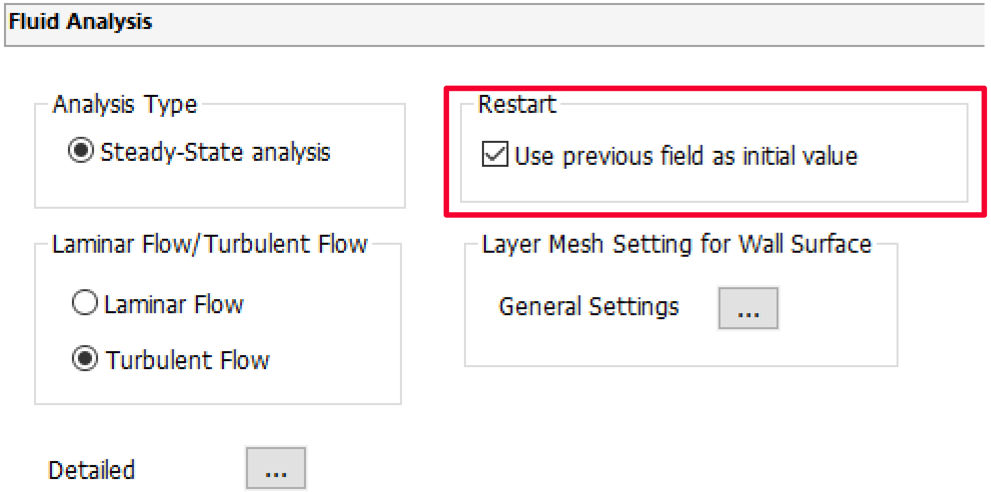


The message above will appear if the calculation did not converge.

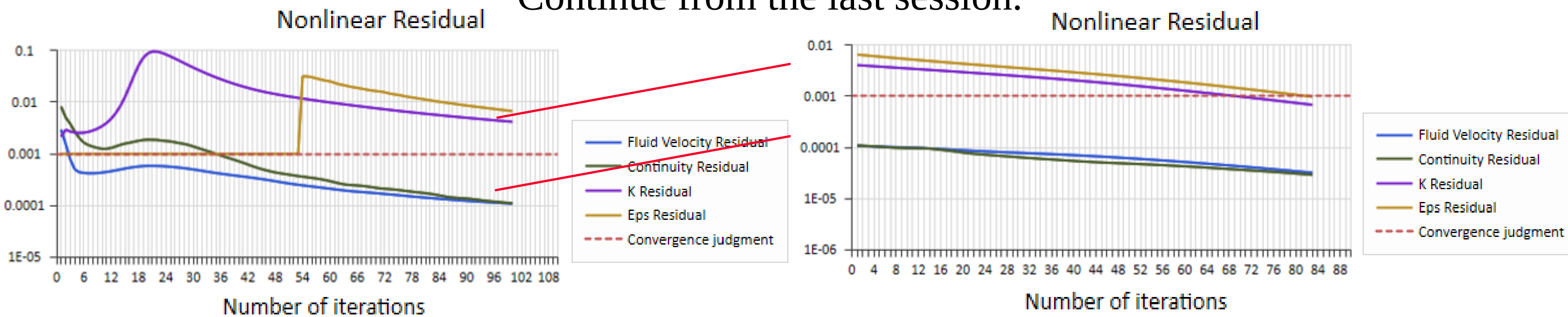
On the [Fluid analysis/Fluid-Thermal Analysis] tab, select [Use previous field as initial value].
Also, refer to the Help for the countermeasure.

Restart

An option for [Use previous field as initial value]



Continue from the last session.

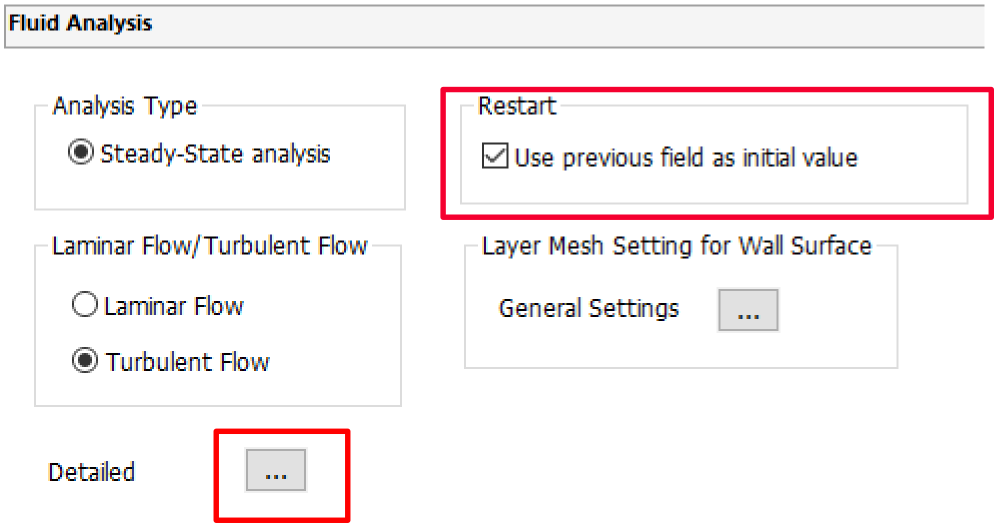


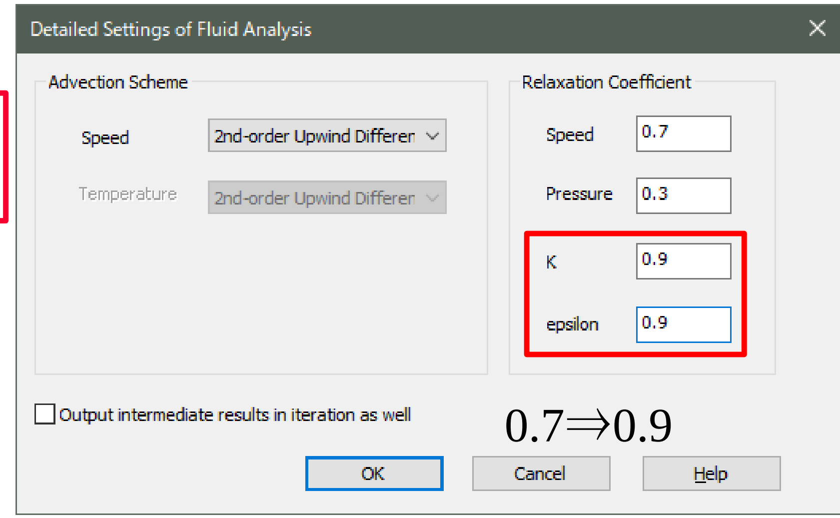
The analysis continues by reading the results of the last session.
The relaxation coefficient and the convergence judgement can be modified.

Modify Relaxation Coefficient

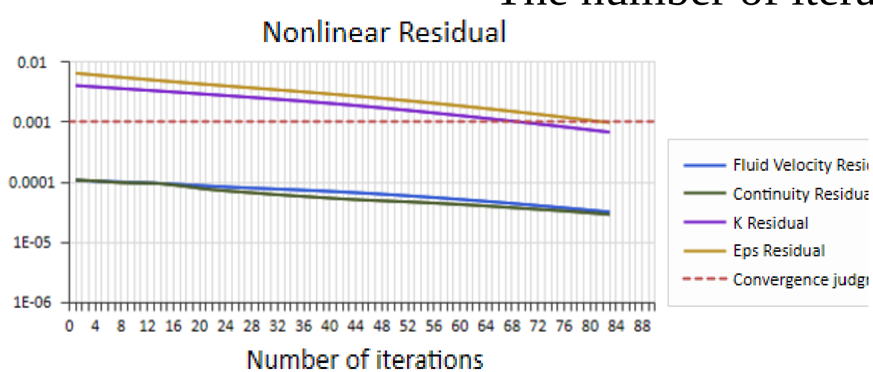
Murata Software

An option for [Use previous field as initial value] with modified relaxation coefficient

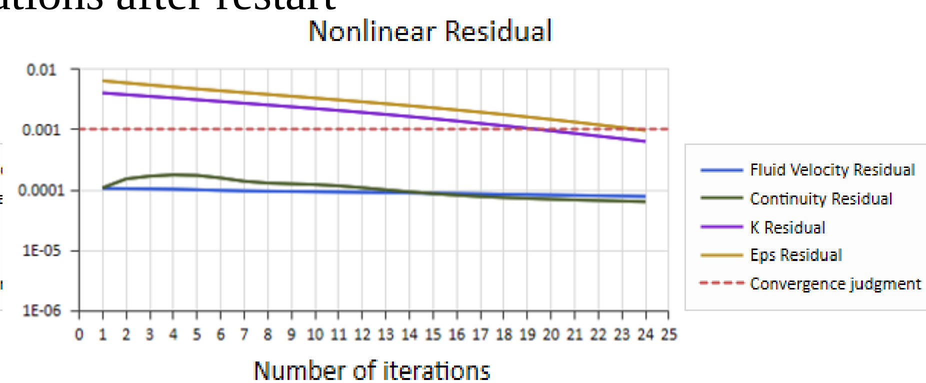




The number of iterations after restart



83 times with default setting



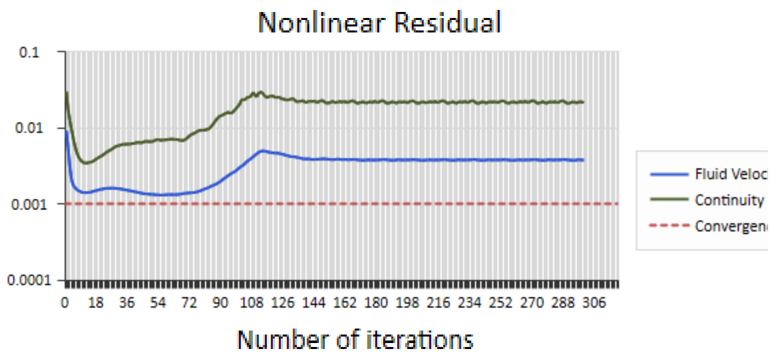
24 times with modified coefficient

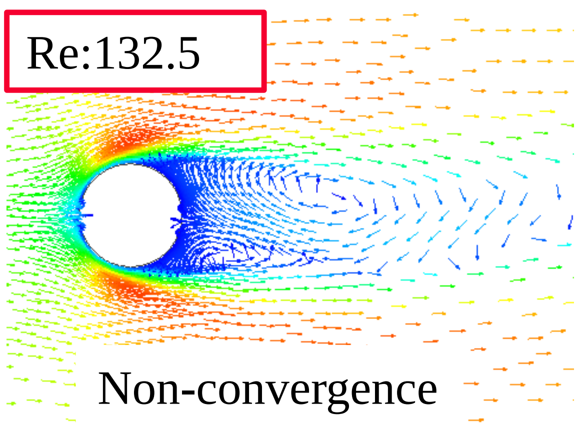
Other Countermeasures

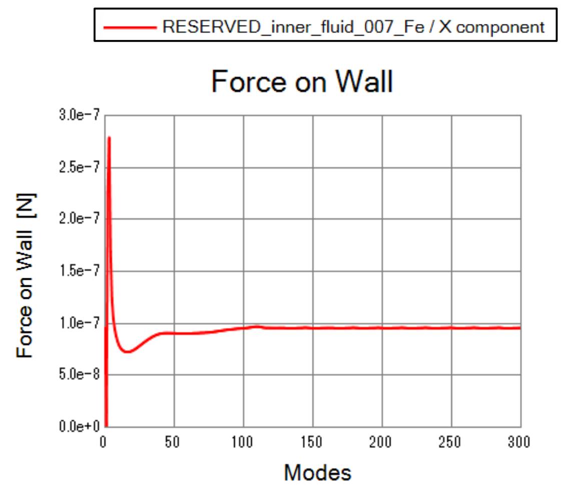
①Use the non-converged results as the final results.

-> The accuracy will be deteriorated but the results can be examined.

If there is no change in the evaluating quantity with respect to the number of iterations, it is interpreted that sufficient accuracy is obtained in some cases.







Iterations and Force on the Wall

Force on the wall converges to a certain value

Other Countermeasures

②Set larger convergence judgment for the nonlinear analysis setting on the High-level setting tab, and restart.
-> The accuracy will be deteriorated but the results can be examined.

③For the physical property that takes time to converge, set larger relaxation coefficient in the detailed setting of the [Fluid Analysis] tab, and restart.
-> The number of iterations can be reduced but divergence may occur.

④Select 1st-order upwind differencing method in the detailed setting of the [Fluid Analysis] tab.
-> The number of iterations can be reduced but the accuracy will be deteriorated.

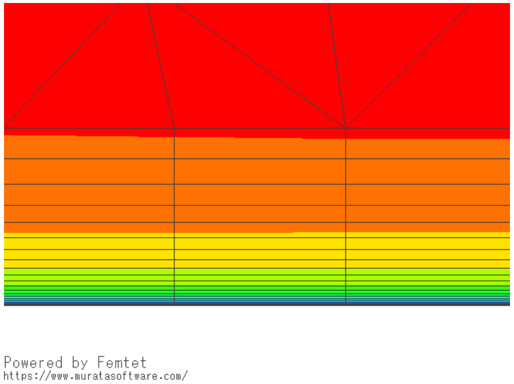
6. Mesh Setting near Wall Face

For the details, refer to
Home > Technical Note > Fluid Analysis/Fluid-Thermal Analysis > Meshing Setup near the Wall Face

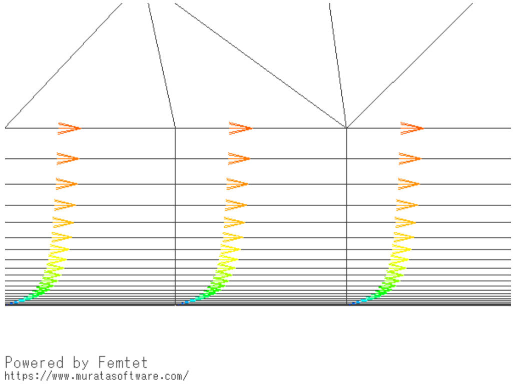
6-1. Wall Function

Flow Velocity Distribution near Solid Wall

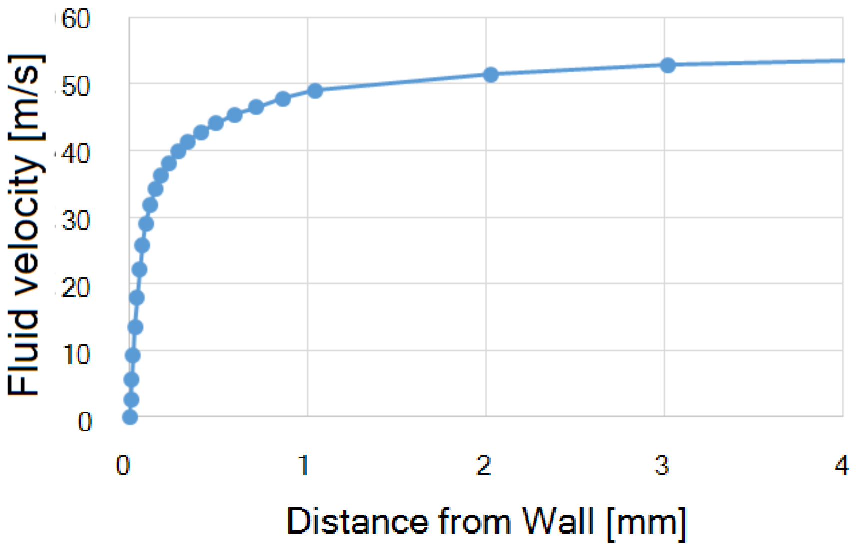
Example



Contour Diagram



Vector Diagram



Characteristics of the flow velocity distribution near the solid wall;

- The flow velocity vectors run in the wall surface direction (The fluid flows along the wall)
- The flow velocity distribution changes according to the distance from the wall

Wall Function

Wall function is a relationship of u^+ and y^+ where y^+ is a dimensionless height and u^+ is a dimensionless flow velocity.

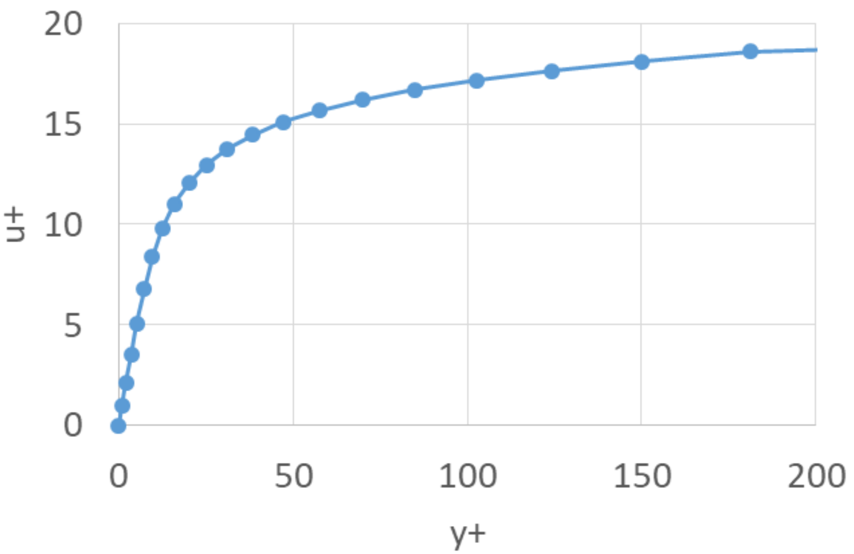
Dimensionless Height

$$y^+ = \frac{u_w y}{\nu}$$

Dimensionless Flow Velocity

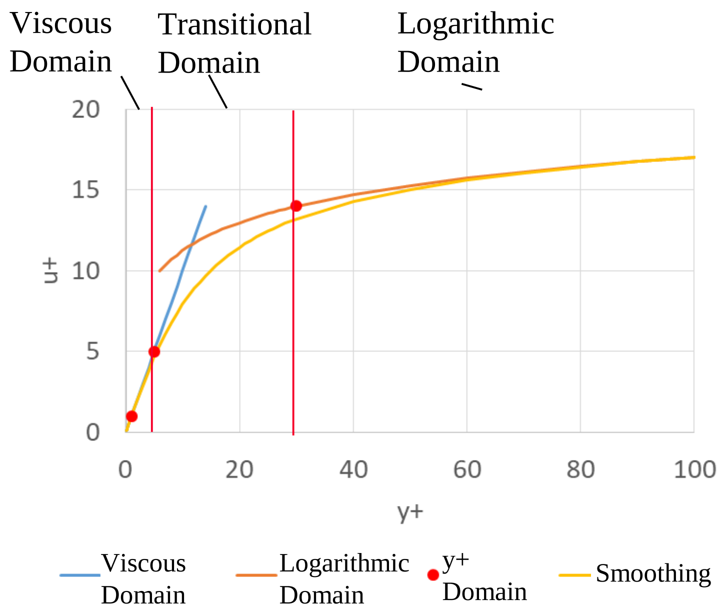
$$u^+ = \frac{u}{u_w}$$

Variable	Note
$y[m]$	Distance from wall
$u_w\ [m/s]$	Friction velocity
$\nu\ [m^2/s]$	Kinematic viscosity
$u[m/s]$	Flow velocity

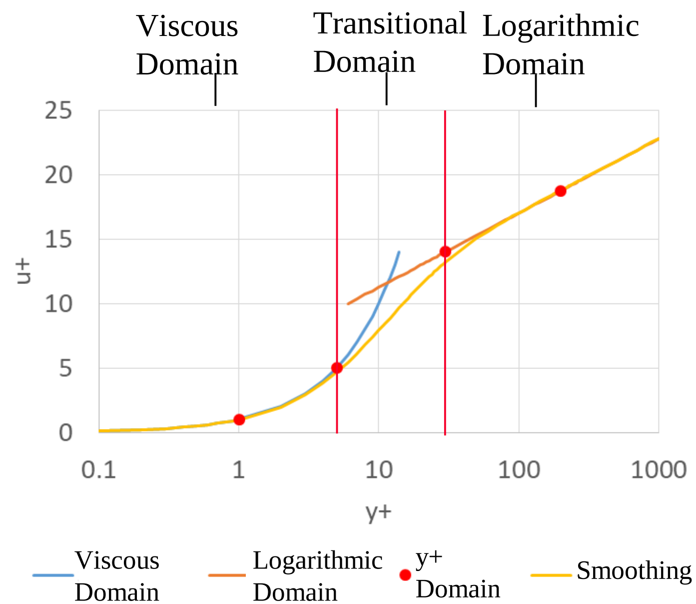


Wall Function

Nature of Wall Function



Linear Plot



Logarithmic Plot

Variable	**Note**
$u^+ = y^+$	Dimensionless velocity (viscous domain) $y^+ \leq 5$
$u^+ = \frac{1}{\kappa}\ln(Ey^+)$	Dimensionless velocity (logarithmic domain) $30 \leq y^+ \leq 200$

Constant	**Note**
$E = 9.0$	Model constant
$\kappa = 4.0$	Model constant (Karman constant)

$y^+ < 5$ is viscous domain. y^+ and u^+ are proportional.

$y^+ > 30$ is logarithmic domain. Logarithm of y^+ and u^+ are proportional.

Temperature-Wall Function

Murata Software

There is a regularity between the height from the wall and the temperature where y^+ is a dimensionless height and T^+ is a dimensionless temperature.

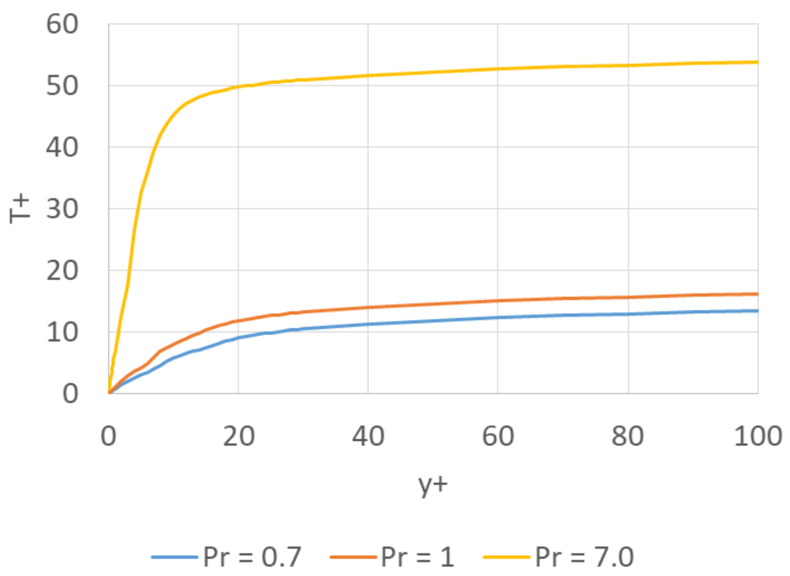
Dimensionless Height

$$y^+ = \frac{u_w y}{\nu}$$

Dimensionless Temperature

$$T^+ = \frac{\rho C_p u_w \Delta T}{q_w}$$

Variable	Note
$\Delta T[deg]$	Temperature difference from the wall
$q_w[W/m^2]$	Heat flux on wall surface
$\rho\ [kg/m^3]$	Density
$C_p\ [J/kg/deg]$	Specific heat

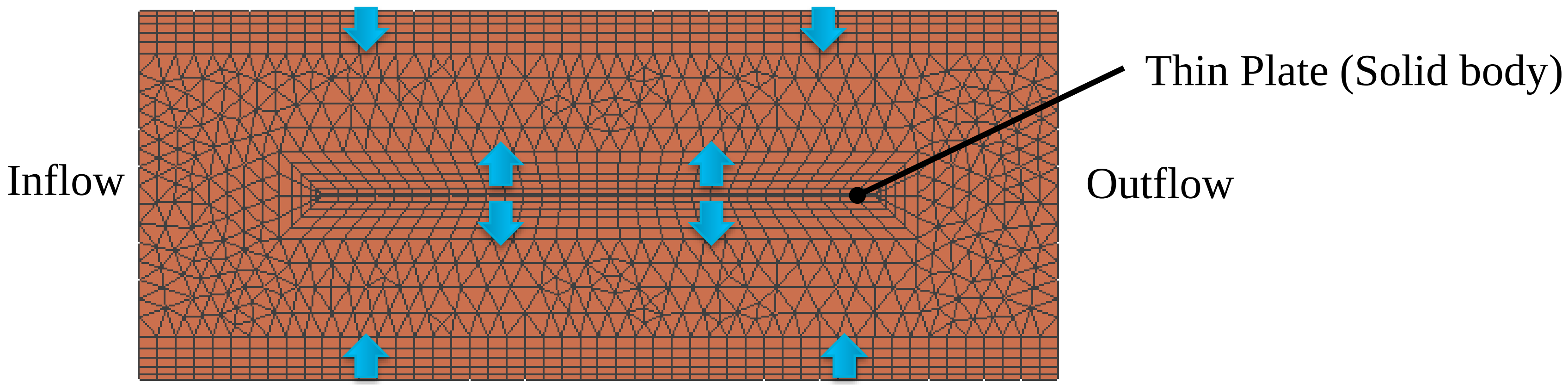


The relationship of dimensionless height and dimensionless temperature is determined by the Prandtl number Pr of material.

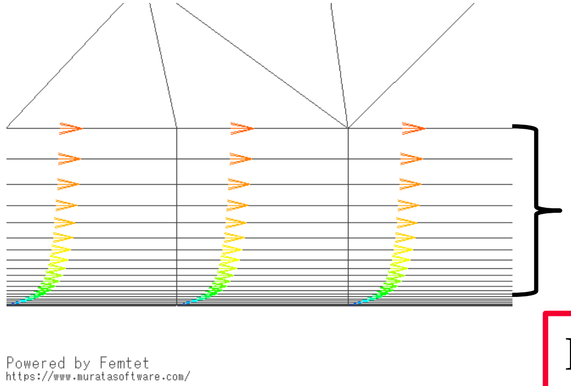
Air: $Pr \sim 0.7$
Water: $Pr \sim 7$

6-2. Layer Mesh Setting

Layer Mesh



Meshes are layered from the wall face.
Quadrangular mesh in 2D and triangular prism in 3D are used.



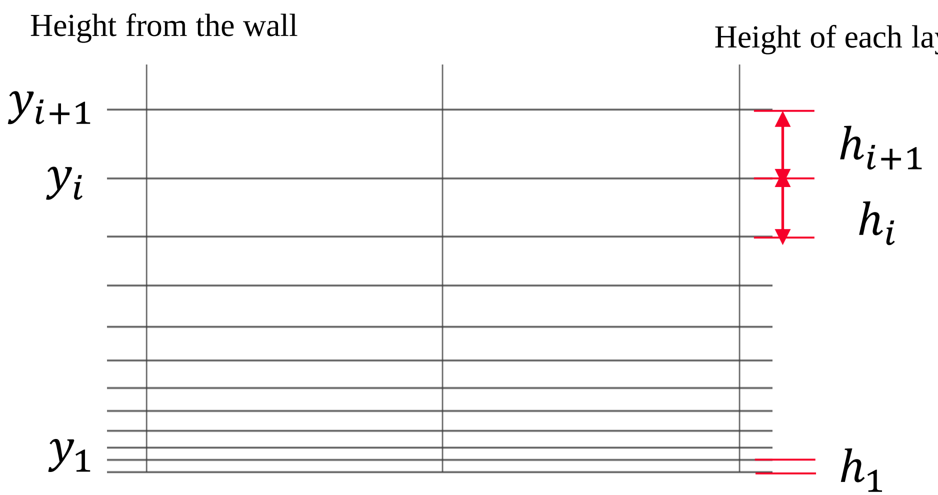
Vectors of flow velocity
on the wall surface

The flow velocity near the solid wall changes greater than the velocity in the flowing direction. The meshes that are finer in the height direction are suitable.

Layer meshes are automatically created on the face where solid wall or slip wall is set.
The automatic creation can be deselected.

Layer Mesh Setting

In Femtet, the setting items are;
- Mesh height of the 1st layer,
- Growth rate
- The number of layers



h_1: mesh height of the 1st layer

$r = \dfrac{h_{i+1}}{h_i}$: growth rate

n: number of layers

$h_i = rh_{i-1} = r^{i-1}h_1$: height of the ith layer

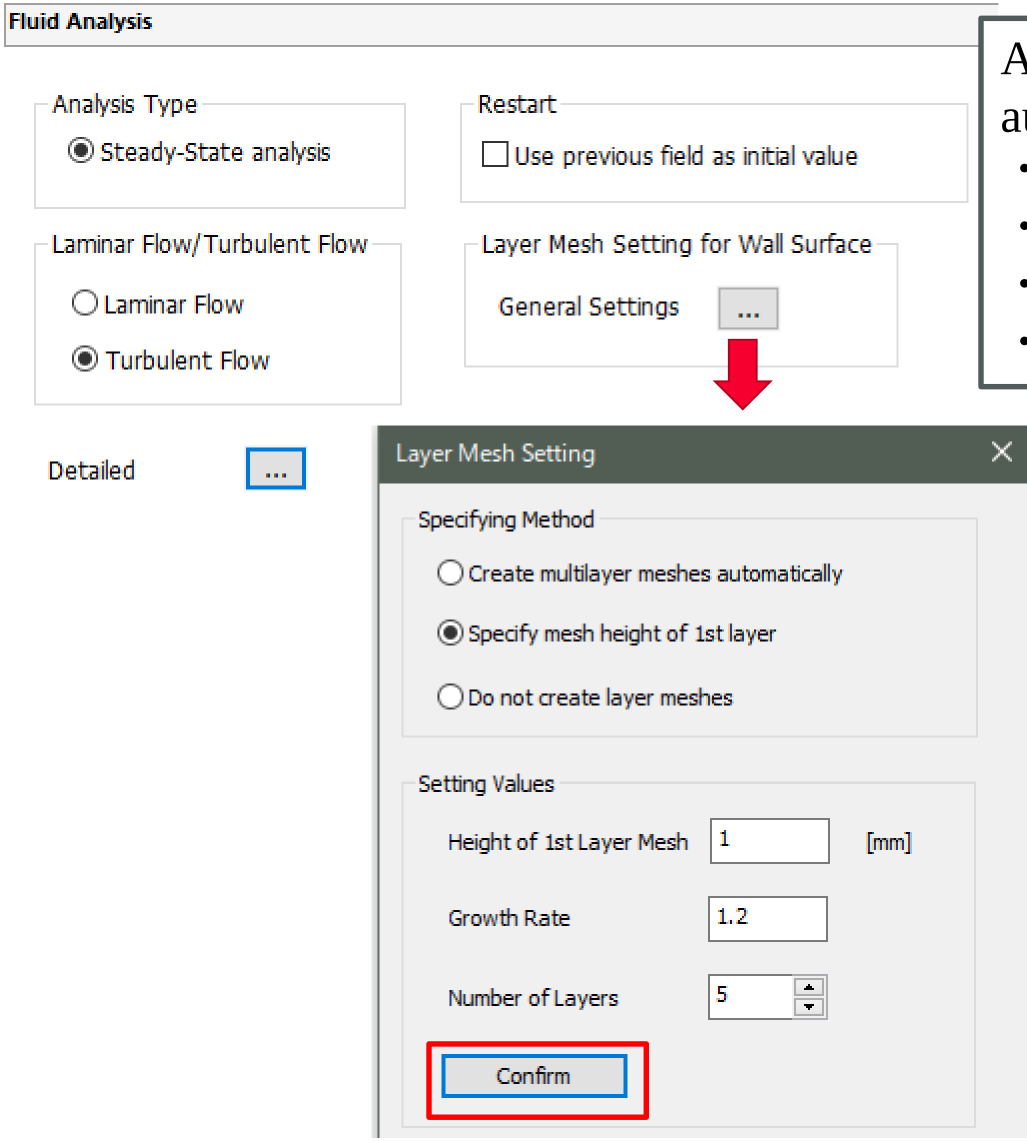
$$y_i = \sum_{j=1}^{i} h_j$$

$$= \frac{r^i - 1}{r - 1} y_1 \text{: height from the wall of the ith layer}$$

General setting can be done in the analysis condition setting.
Individual setting can be done in the boundary condition setting.

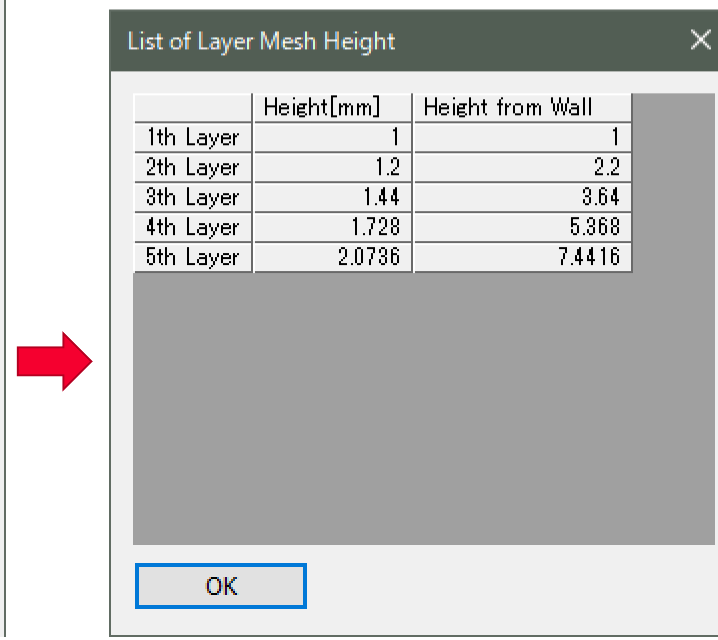
General Setting

Murata Software

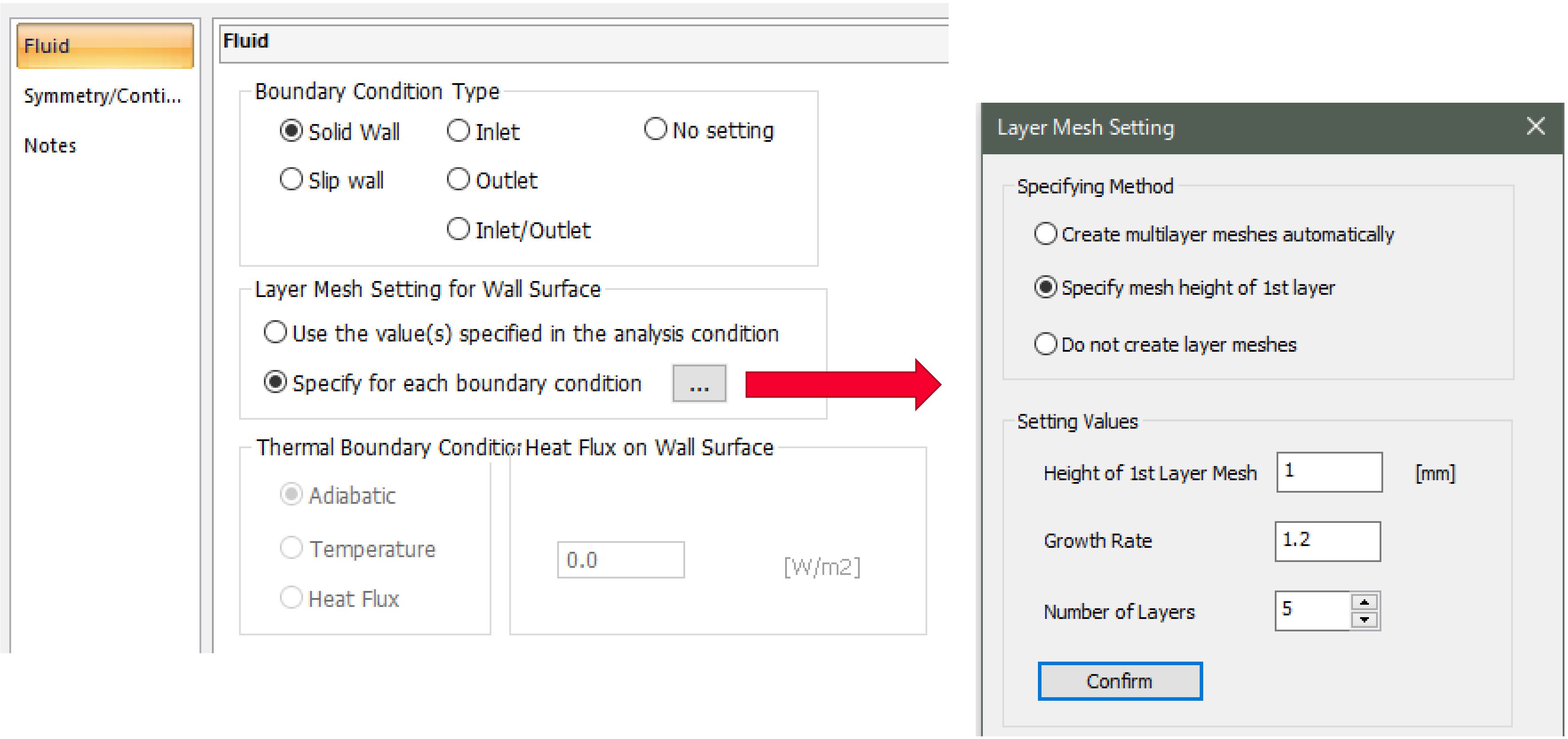


Automatic: Layer meshes are automatically created based on;

- flow velocity
- material property
- shape of inflow/outflow faces
- laminar or turbulent flow

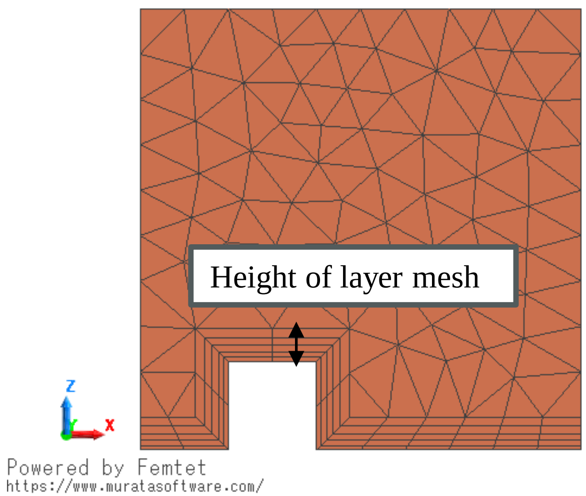


Setting for Each Boundary Condition



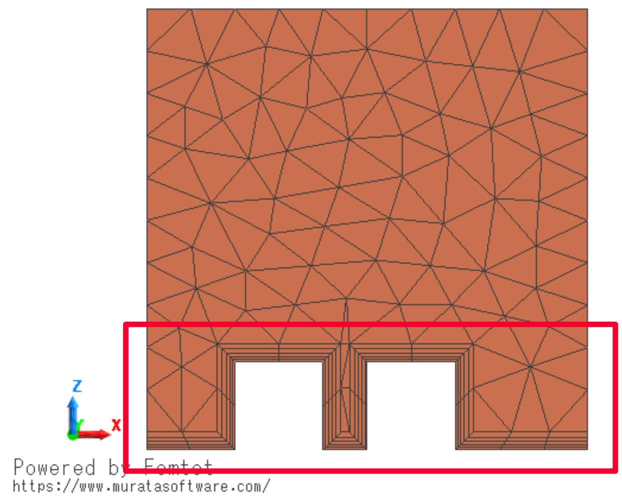
The setting is done in the general setting of the analysis condition by default.
By selecting [Specify for each boundary condition], individual setting is possible.

If there exist narrow areas, the meshes may not be created with specified height.
The thickness will be automatically adjusted.



Specified Mesh Height

Narrow areas exist



The connected walls are aligned to a certain height

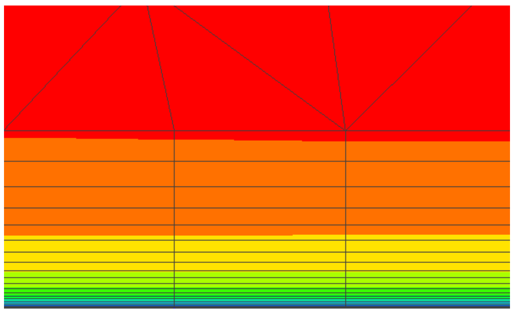
6-3. Optimum Layer Mesh Setting

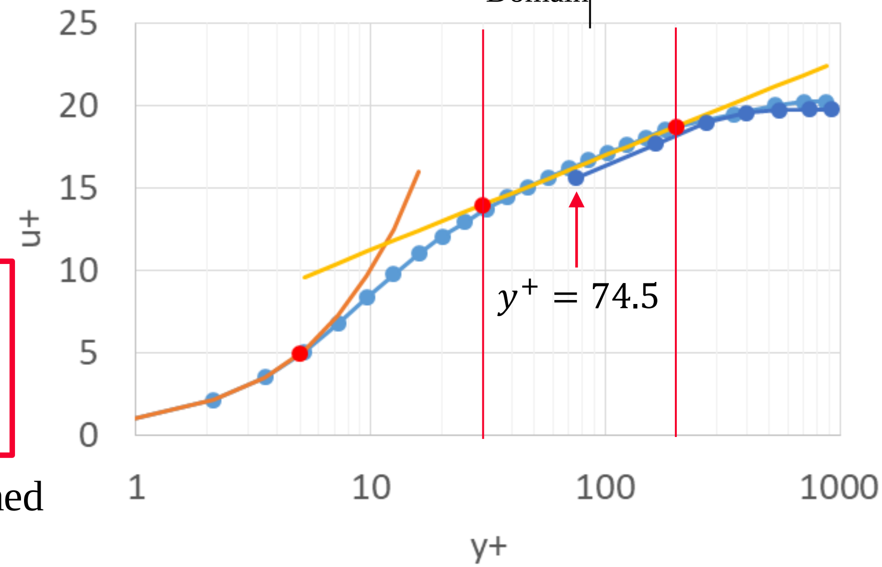
Two Points on Optimum Layer Mesh

Murata Software

①Meshes must be fine enough so that the flow velocity distribution near the wall face is smooth.

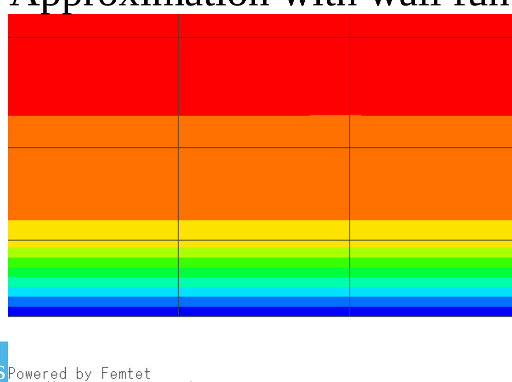
➡ Accurate calculation near the wall face





②1st layer must be placed in the Logarithmic domain of the wall function. (turbulent flow analysis only)

➡ Approximation with wall function performed



$y^+ = 74.5$

Optimum Layer Mesh Setting

Murata Software

①Meshes must be fine enough so that the flow velocity distribution near the wall face is smooth.

- 1st layer is $y^+ = 1 \sim 5$
- Recommended layer mesh domain is up to $y^+ = 200$ where the fluctuation is large. Table below is for your reference.

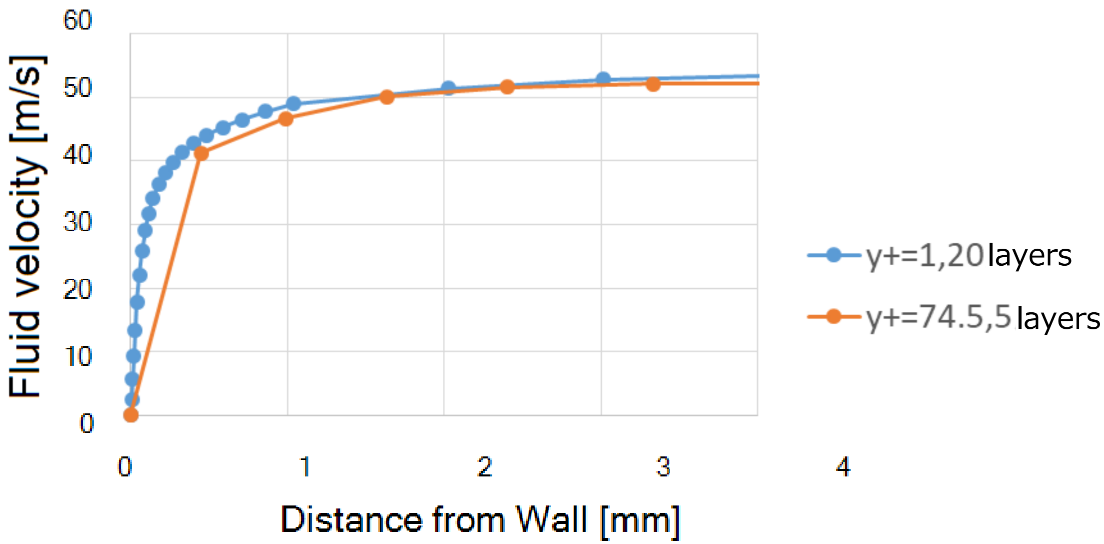
$$y_n^+ = \frac{r^n - 1}{r - 1} y_1^+$$

Target of 1st Layer	Example Setting	Note
$y^+ = 1$	Growth rate: 1.2 Number of layers: 20	When 1st layer is $y^+ = 1$, Layer mesh domain is up to $y^+ = 186$
$y^+ = 1$	Growth rate: 1.5 Number of layers: 12	When 1st layer is $y^+ = 1$, Layer mesh domain is up to $y^+ = 257$
$y^+ = 5$	Growth rate: 1.2 Number of layers: 12	When 1st layer is $y^+ = 5$, Layer mesh domain is up to $y^+ = 197$
$y^+ = 5$	Growth rate: 1.5 Number of layers: 8	When 1st layer is $y^+ = 5$, Layer mesh domain is up to $y^+ = 246$

②1st layer must be placed in the logarithmic domain of the wall function. (turbulent flow analysis only)

- y^+ of the 1st layer must be in the logarithmic domain ($30 < y^+ < 200$).

Comparison of Optimum Meshes ① and ②



Flows between Parallel Plates(Example 2)

For the simple flows, two kinds of optimum meshes ① and ②result in almost the same flow velocity distributions.

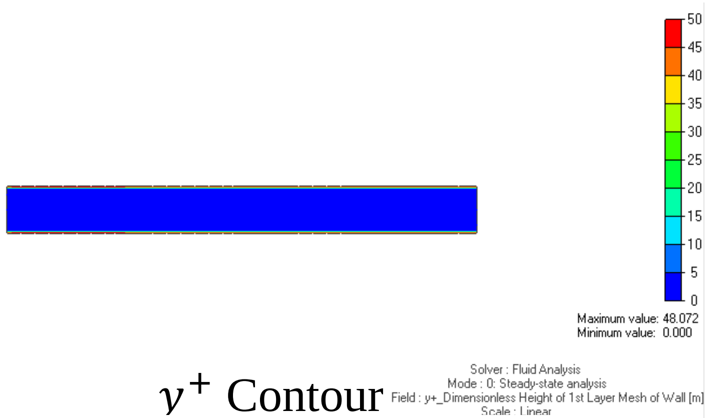
If complicated flows occur near the wall surface, meshing method ② may be less accurate. Selection of ① or ② depends on the degree of accuracy you want.

6-4. Verification of Meshes near the Wall Face

How to Verify Proper Height

Murata Software

To verify if the mesh height is proper, look at the output of y^+ (Mesh height of the 1st layer of the wall face).
You can check in the contour diagram and the numerical result table.



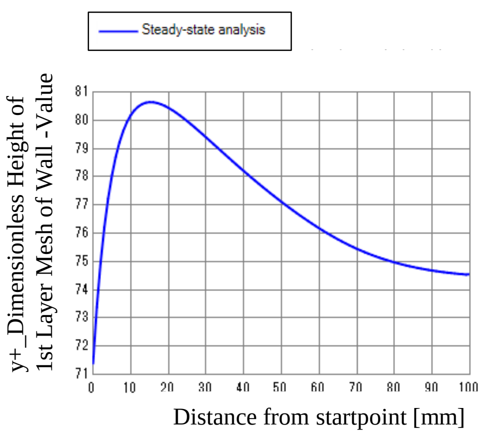
y^+ Contour

Table

Convergence status | FEM Info | Force on Wall [N] | Volumetric Flow Rate [m3/s] | y+ distribution [%] | Height

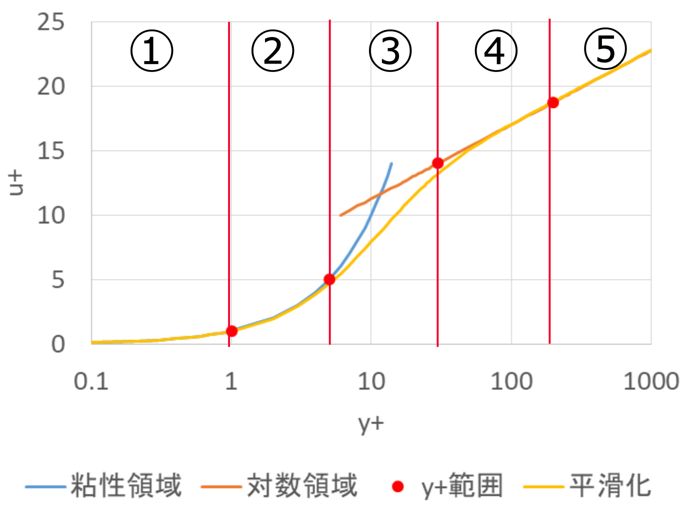
	y+<1	1<=y+<5	5<=y+<30	30<=y+<200	200<y+
Outer_Boundary_Condition	54	45	0	0	0

Distribution of y^+



Plot of Edges

y+ Distribution



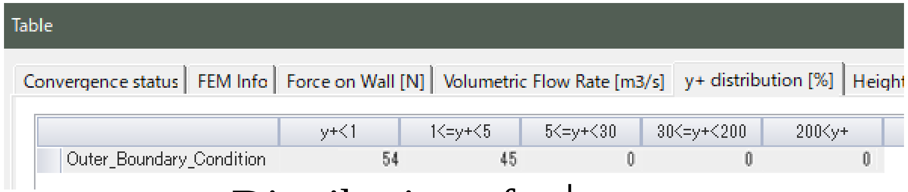
①$y^+ < 1$

②$1 \le y^+ < 5$

③$5 \le y^+ < 30$

④$30 \le y^+ < 200$

⑤$200 \le y^+$



	y+<1	1<=y+<5	5<=y+<30	30<=y+<200	200<y+
Outer_Boundary_Condition	54	45	0	0	0

Distribution of y^+

The ratio of five domains are calculated with sum of y^+ of all nodes on the boundaries.

When setting fine meshes near the wall face, if most of y^+ are in the domains ① and ②, the mesh setting is considered appropriate.
Please note however that it cannot be judged if the number of layers and growth rate are proper.
When placing the 1st layer of the wall face in the logarithmic domain, if most of y^+ are in the domain ④, the mesh setting is considered appropriate.

Recommended Height of the 1st Layer Mesh

In order to set $y^+ < 1$、 $y^+ < 5$、 $y^+ < 200$, recommended values are shown in the table.
After performing the analysis, execute the analysis again with recommended values.
The higher accuracy can be obtained.

Table

Convergence status | FEM Info | Force on Wall [N] | Volumetric Flow Rate [m3/s] | y+ distribution [%] | Height of 1st Layer Mesh [mm]

	Average Value	Recommendation (y+<1)	Recommendation (y+<5)	Recommendation (y+<200)
Outer_Boundary_Condition	0.448	5.555e-3	2.777e-2	1.111

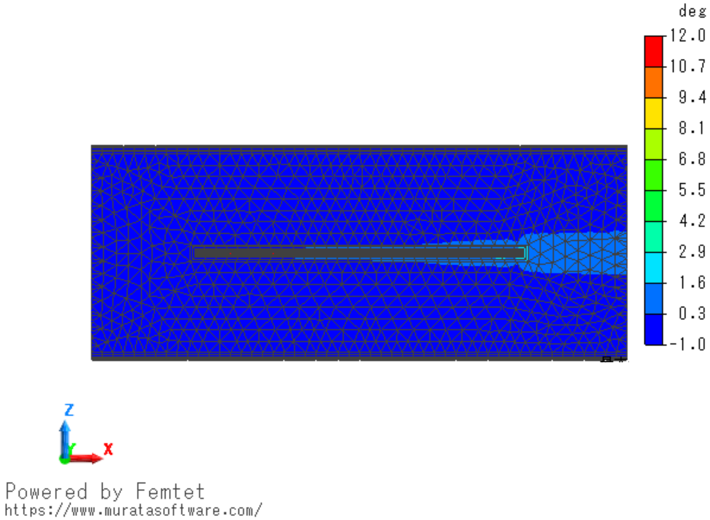
To set fine meshes near the wall face, use set the height of the 1st layer mesh by using the recommended values for $y^+ < 1$、 $y^+ < 5$.
In doing so, pay attention to the number of layers and growth rate as well so that they are optimum as explained on page 78.

To place the 1st layer of the wall face in the logarithmic domain, set the height of the 1st layer mesh by using the recommended values for $y^+ < 200$.

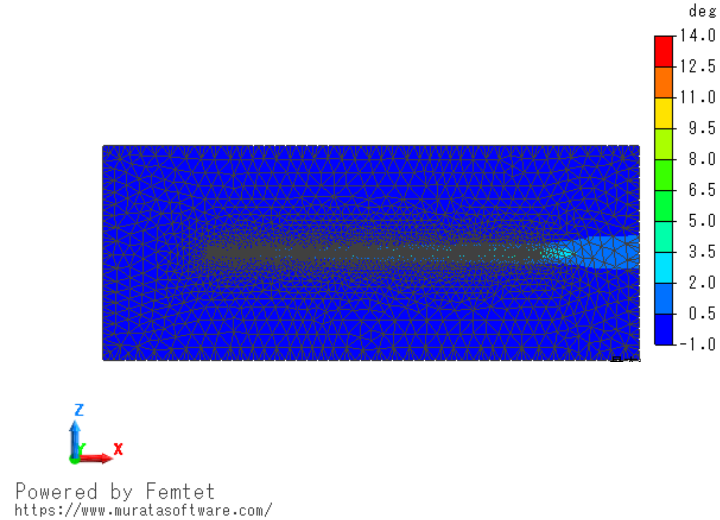
6-5. Effect of Layer Mesh

With/Without Layer Meshes

Fluid-Thermal Analysis Example 2
Analysis Comparison with and without Layer Meshes



With Layer Meshes



Without Layer Meshes

With Layer Meshes

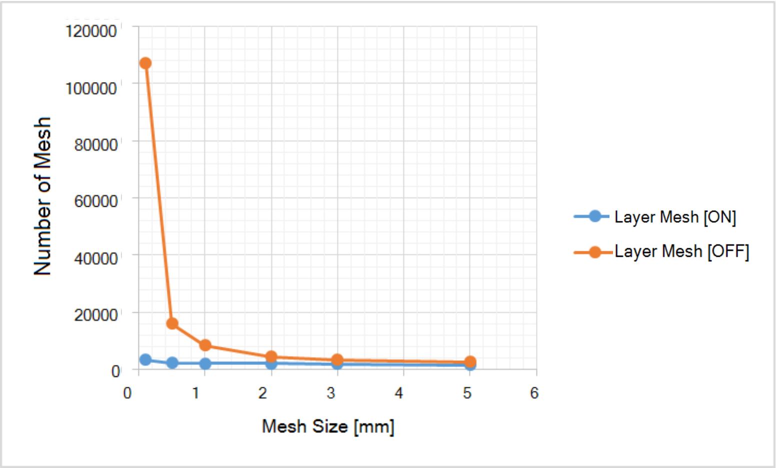
The mesh size is controlled by setting the height of the 1st layer mesh.
Height of the 1st layer mesh: 0.1～5[mm]

Without Layer Meshes

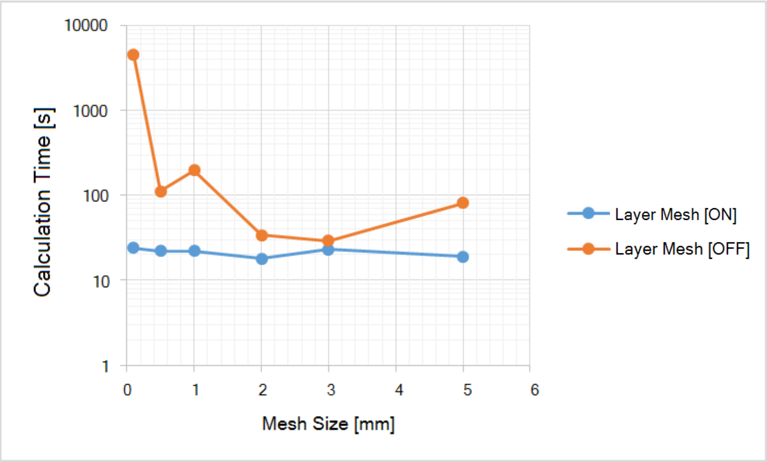
The mesh size is controlled by setting the mesh size of the flat area.
Mesh size of the flat area: 0.1～5[mm]

Number of Meshes / Analysis Time

Murata Software



The Number of Meshes

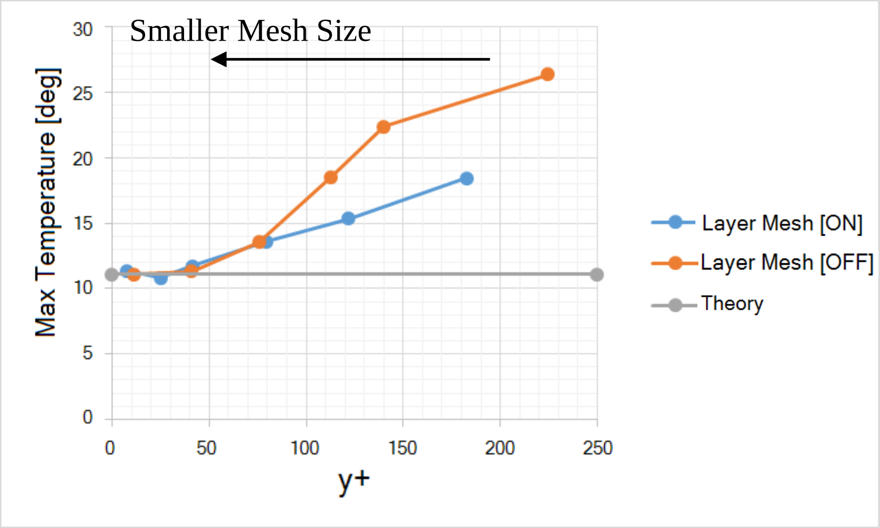


Analysis Time

By using layer meshes, the number of meshes and analysis time can be suppressed even if the mesh size is reduced.

Without layer meshes, the number of meshes and analysis time will increase if the mesh size is reduced.

Accuracy



Maximum Temperature with y+ on the Horizontal Axis

If y+<30, calculation converges near the theoretical value.

If 30<y+<200, divergence is great especially when the layer mesh is not used.

6-6. Problems of Layer Mesh

Problems of Layer Mesh

Murata Software

There are problems in creating layer mesh as it is technically difficult compared with triangular and tetrahedron meshes.

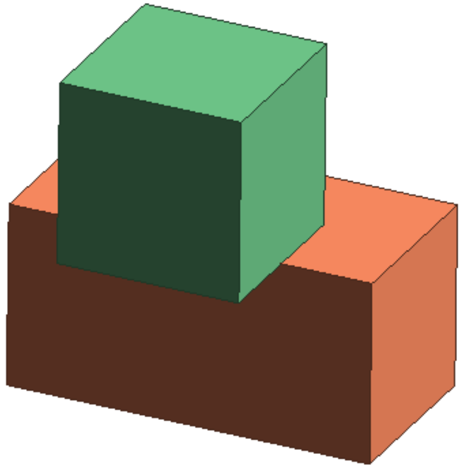
1. Mesh creation may fail in some cases
   ➡ If the failure persists even by changing the mesh size,
   set [layer mesh creation] OFF, which is an only countermeasure.

2. Layer meshes may not be created as specified
   ➡ In a narrow area, mesh cannot be created physically.

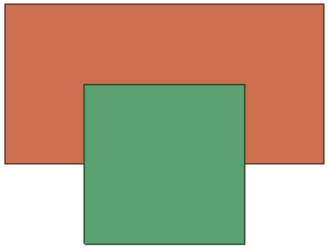
3. In some cases, layer meshes are created on the inflow/outflow faces where they are not supposed to be created.

Case Where Layer Mesh Is Not Created

If the fluid surrounds the form like below, the layer mesh is not created.



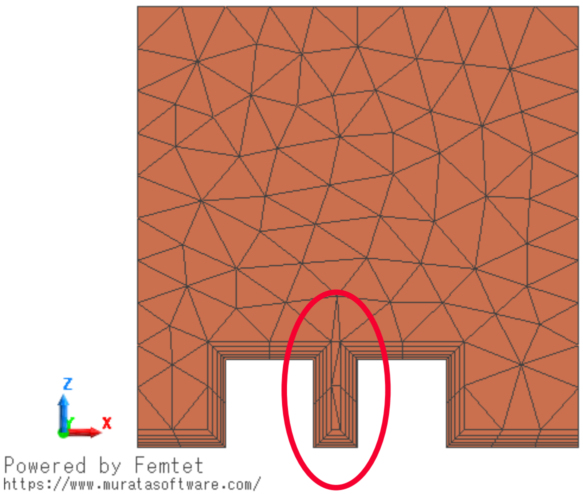
Side View



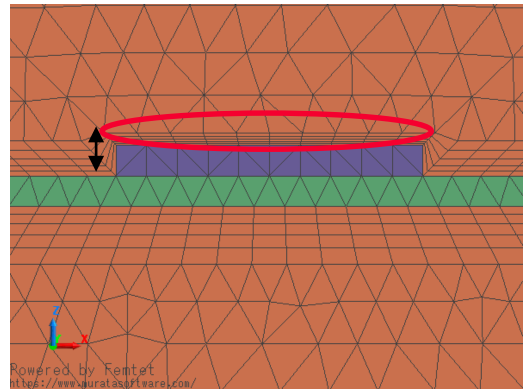
Top View

Front View

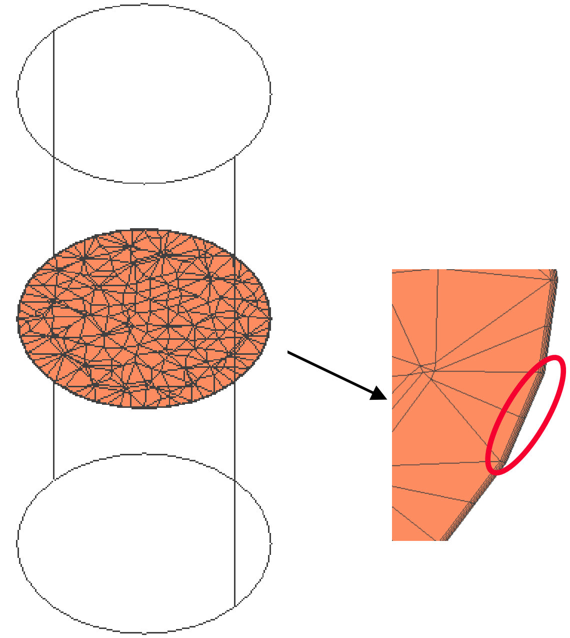
Case Where Thickness Is Not as Specified



Narrow Area



Small Step

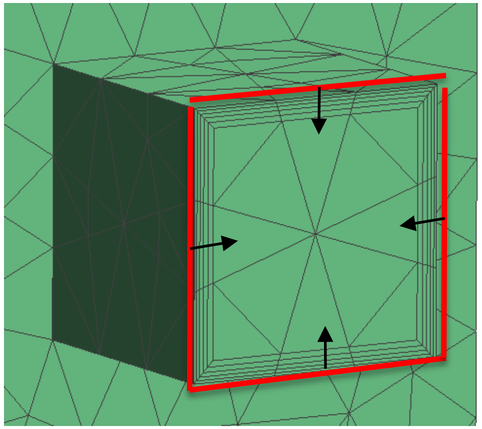


Inside of Cylinder

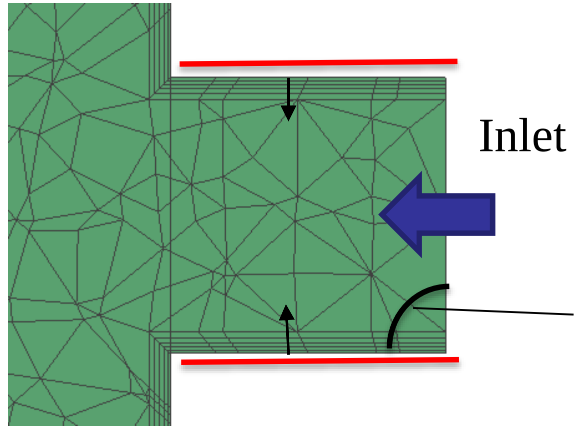
The thickness is automatically adjusted thinner than specified for the cases above.

For the [Narrow Area], the automatic adjustment is necessary since it is physically impossible to set layer meshes.
We continue to improve the issues with [Small Step] and [Inside of Cylinder].

Meshing Inlet/Outlet



Inlet / Outlet



Sectional View

<Proper state of inlet/outlet positioning>

1. Layer meshes are created inside the inlet/outlet edges
2. Layer meshes are created along the inflowing/outflowing direction

Requirements:

・The angle of the inlet/outlet and the adjacent wall faces must be less than 180°.

・Multiple inlet/outlet should not be contacting each other.

・Inflow/outflow face must be flat.

Meshing Inlet/Outlet

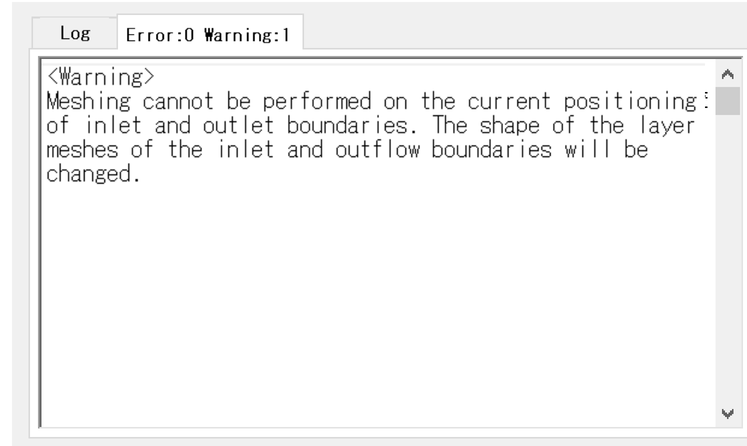
<If the inlet/outlet is not positioned properly>

Such inlet/outlet is not supported.
Meshes are created on the inlet/outlet faces.

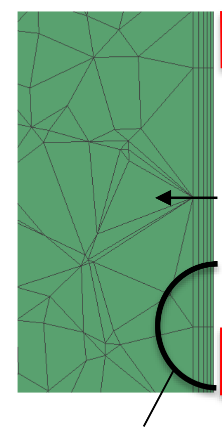
Analysis is possible but there may occur some problems.

1. Error occurs if the meshes are coarse.
Meshes must be finer. Analysis time will become longer.

2. Abnormal values may appear in the distribution near the inlet/outlet faces.
Calculation may not converge.



Warning Message



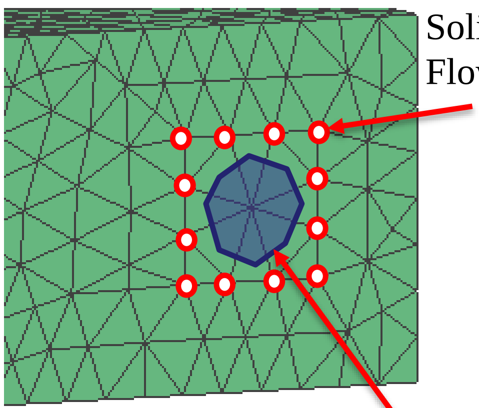
Inlet/Outlet

Angle of the wall face and inlet/outlet is 180°

Improper Inlet/Outlet (1)

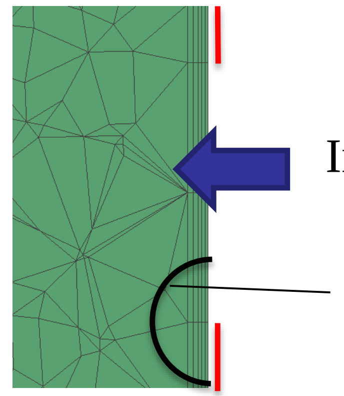
Murata Software

If the layer meshes are created partially on the face, the requirement of 180° is not met. The layer meshes along the surrounding edges are not created.



Solid Wall
Flow Velocity: 0

Domain where inflow/outflow occurs (effective area)

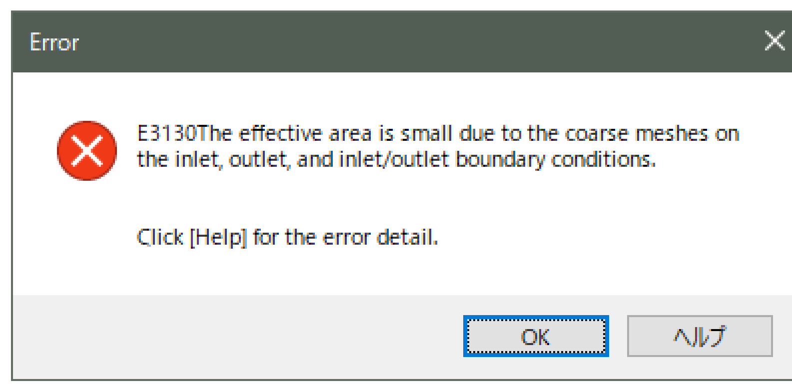


Inlet/Outlet

Angle of the wall face and inlet/outlet is 180°

Nodes on the surrounding edges are treated as solid wall (flow velocity: 0). The effective area of the inlet/outlet becomes smaller.

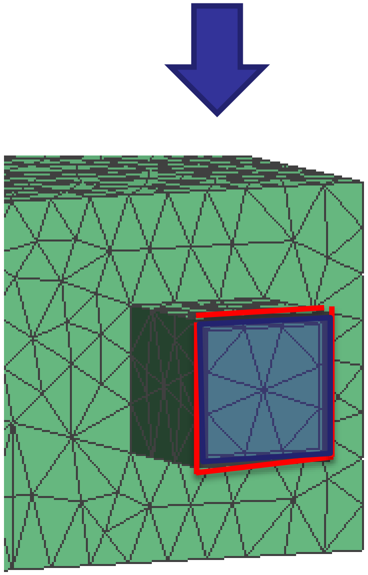
If the mesh is coarse, the effective area becomes smaller. If its occupancy ratio is less than 80%, an error will occur.



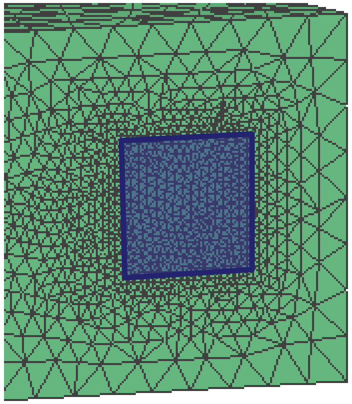
Counter Measure

Output Window When an Error Occurred

```
Bernoulli
The effective cross-sectional area of the boundary condition [in] is small due to the coarse meshes. (Occupancy ratio < 80 [%]
Refine the meshes on the boundary condition.
Effective cross-sectional are= 4.524e-05 [m2]
Cross-sectional area=1.996e-04 [m2]
Occupancy ratio= 22[%]
Recommended mesh size=7.005e-01 [mm]
```



(1) Inlet/Outlet Pushed Out



(2) Finer Meshes

<u>(1) If inlet/outlet face is pushed out</u>

Number of meshes: 23154

Analysis time: 1min23s

<u>(2) If the meshes are made finer</u>

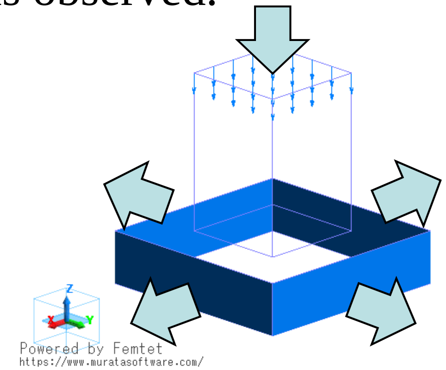
Number of meshes: 69497

Analysis time: 5min43s

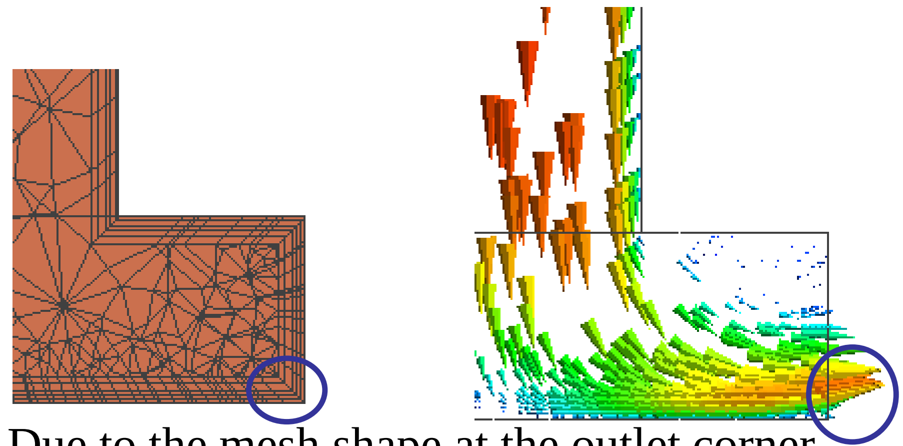
Analysis time tends to be longer with finer meshes.
Method (1) is more efficient.

Improper Inlet/Outlet (2)

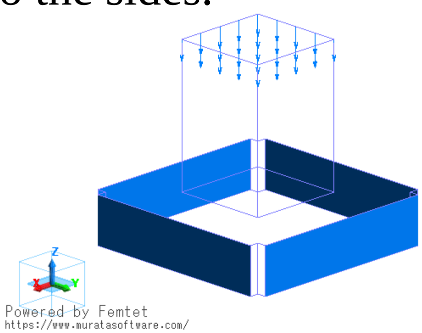
If multiple inlet/outlet are connected each other, the layer meshes are not created along the surrounding edges. Due to the mesh shape, disturbance in the results near the inlet/outlet is observed.



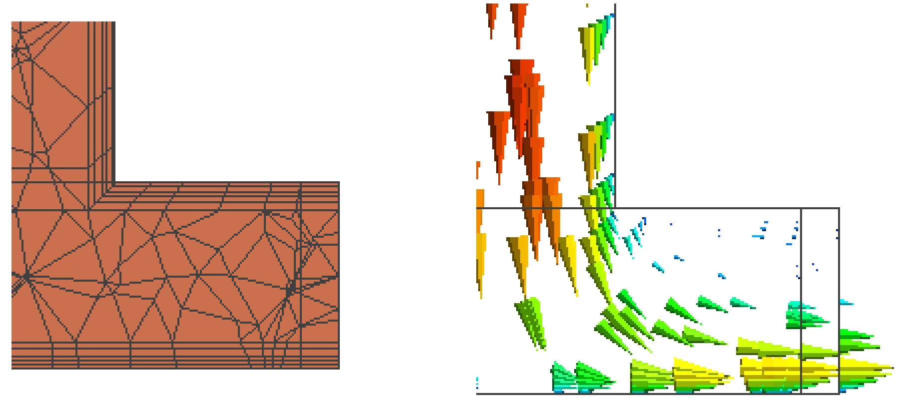
The flows come in from the top and go out at the bottom to the sides.



Due to the mesh shape at the outlet corner, the flow direction is affected.



The four sides are pushed outward.



Flows out straight

7. Appendix

7-1. List of Functionalities

Functionalities

Murata Software

Item	Functionality
Analysis Function	Steady-state Analysis
Analysis Flow	Incompressible, Laminar, Non-temperature dependent (no buoyancy)
Material	Density, Viscosity
Boundary Condition	Solid wall, Slip wall, Forced inflow (velocity-specified, pressure-specified, fan), Forced outflow (velocity-specified, pressure-specified, Fan), Natural inflow, Natural outflow
Result Output	Flow velocity, Pressure, Turbulent energy K, Energy dissipation rate ε, y+, Force on wall, Volumetric flow rate, Pressure loss, etc.
Laminar / Turbulent Flow	Laminar flow, Turbulent flow (Realizable K-ε model)
Analysis Space	2D, 3D (axisymmetry is not supported)
Analysis Method	Finite volume method Steady-state analysis: SIMPLE method
Advection Scheme	1st-order upwind differencing scheme 2nd-order upwind differencing scheme
Mesh	1st-order element Wall surface: layer mesh (rectangular, triangular prism)

Functionalities

Item	Functionality
Analysis Function	Fluid steady-state analysis⇒Thermal steady-state analysis (Forced convection) Fluid steady-state analysis⇒Thermal transient analysis (Forced convection)
Analysis Flow	Incompressible, Laminar, Non-temperature dependent (no buoyancy)
Material	Density, Viscosity, Thermal conductivity, Specific heat
Boundary Condition	Solid: Temperature, Heat flux, Heat dissipation/Ambient radiation, Radiation, Thermal resistance Fluid: Heat flux on the wall, Wall temperature, Temperature of inflowing fluid
Result Output	Temperature, Heat flux, Heat flux of wall, Heat balance, Thermal flow rate, etc.
Laminar / Turbulent Flow	Laminar flow, Turbulent flow (Realizable K-ε model)
Analysis Space	2D, 3D (axisymmetry is not supported)
Analysis Method	Solid: Finite element method Fluid: Finite volume method
Advection Scheme	1st-order upwind differencing scheme 2nd-order upwind differencing scheme
Mesh	1st-order element Wall surface: layer mesh (rectangular, triangular prism)

7-2. Laminar Flow Analysis

For the details of variables settings, refer to
Home > Technical Note > Fluid Analysis/Fluid-Thermal Analysis > Differential Equations in Fluid Analysis/Fluid-Thermal Analysis

Governing Equations (Laminar Flow/Incompressible)

Murata Software

Navier-Stokes Equation

$$\frac{\partial \boldsymbol{u}}{\partial t} + (\boldsymbol{u} \cdot \nabla)\boldsymbol{u} = -\frac{1}{\rho}\nabla p + \nabla \cdot (2\nu \boldsymbol{s})$$

Acceleration | Advection | Pressure Gradient | Viscosity Diffusion

Continuity Equation

$$\nabla \cdot \boldsymbol{u} = 0$$

Transport Equation of Thermal Energy

$$\rho C_p \frac{\partial \theta}{\partial t} + \rho C_p (\boldsymbol{u} \cdot \nabla)\theta = \nabla \cdot (\lambda \nabla \theta) + Q$$

Heat Storage | Advection | Thermal Conduction | Heat Source

- **Acceleration**

Flow velocity remains unchanged over time.
It is zero in the steady-state analysis.

- **Advection**

Along with the fluid flow, its nature (momentum and thermal energy) is moved.

- **Pressure Gradient**

Fluid is accelerated or decelerated due to the unbalanced force.

- **Viscosity Diffusion / Thermal Conduction**

Velocity and temperature are made uniform.

- **Continuity Equation**

Balance of flow rate is zero.
(Incoming flow always flows out)

- **Heat Storage**

Change of temperature (thermal energy) over time.
It is zero in the steady-state analysis.

- **Heat Source**

Heat generation (solid only).

7-3. Turbulent Flow Analysis

For the details of variables settings, refer to
Home > Technical Note > Fluid Analysis/Fluid-Thermal Analysis >
Differential Equations in Fluid Analysis/Fluid-Thermal Analysis

Classification of Turbulent Flow Analysis

Murata Software

Turbulent Flow

DNS Direct Numerical Simulation

Navier–Stokes equations are numerically solved with numerous meshes representing the smallest vortex. Used in the transient analysis.

LES Large Eddy Simulation

The effect of the smallest vortices that cannot be represented by meshing is statistically taken into account. Used in the transient analysis.

RANS Reynolds-averaged Navier-Stokes

Averaged ensemble is applied the Navier-Stokes equations. As the temporal average is considered, it is useful for the steady-state analysis. Depending on how to model the Reynolds stress, analysis method varies.

Stress Equation Model

Transport equation of the Reynolds stress is solved.

Eddy Viscosity Model

The Reynolds stress is replaced by the increased viscosity.
The material mixing becomes active because of the turbulent flow and vortex.

Femtet uses RANS/Eddy viscosity model (Calculation load is the smallest)

Reynolds Average and Eddy Viscosity Model

Murata Software

Laminar Flow

Navier-Stokes Equation

$$\frac{\partial \boldsymbol{u}}{\partial t} + (\boldsymbol{u} \cdot \nabla)\boldsymbol{u} = -\frac{1}{\rho}\nabla p + \nabla \cdot (2\nu \boldsymbol{s})$$

Continuity Equation

$$\nabla \cdot \boldsymbol{u} = 0$$

Transport Equation of Thermal Energy

$$\rho C_p \frac{\partial \theta}{\partial t} + \rho C_p (\boldsymbol{u} \cdot \nabla)\theta = \nabla \cdot (\lambda \nabla \theta) + Q$$

Reynolds Average

Turbulent Flow

Reynolds-averaged Navier-Stokes Equation

$$\frac{\partial \boldsymbol{U}}{\partial t} + (\boldsymbol{U} \cdot \nabla)\boldsymbol{U} = -\frac{1}{\rho}\nabla P + \nabla \cdot (2\boxed{(\nu + \nu_t)}\boldsymbol{S})$$

Eddy viscosity model
Its kinematic viscosity increases according to the state of turbulent flow.

Reynolds-averaged Continuity Equation

$$\nabla \cdot \boldsymbol{U} = 0$$

Transport Equation of Reynolds-averaged Thermal Energy

$$\rho C_p \frac{\partial T}{\partial t} + \rho C_p (\boldsymbol{U} \cdot \nabla)T = \nabla \cdot \left(\boxed{(\lambda + \lambda_t)}\nabla T\right) + Q$$

Eddy viscosity model
Its thermal conductivity increases according to the state of turbulent flow.

<Reynolds Average>
Variables like flow velocity, pressure, and temperature are separated to the averaged components and the variable components. The converted governing equation is for the averaged quantities.

<Eddy Viscosity Model>
The variable components of turbulent flow are converted to the increase of viscosity and thermal conductivity. (The material mixing becomes active because of the turbulent flow and vortex)

- The governing equation's form is similar to the laminar flow.
- Turbulent viscosity coefficient(Eddy viscosity) ν_t and turbulent thermal conductivity λ_t must be solved.

Classification of Eddy Viscosity Model

Murata Software

There are models for each method of the turbulent kinematic viscosity (Eddy viscosity) ν_t.
Classified by the number of equations added.

Eddy Viscosity Model

- **Zero-Equation Model (Algebraic Model)**

  Determined by the averaged-velocity gradient and distance from the wall, etc.

- **One-Equation Model**

  - Transport equation of turbulent energy K
  - Transport equation of turbulent kinematic viscosity ν_t

- **Two-equation Model**

  - K-ε model: transport equation of turbulent energy K and energy dissipation rate ε
  - K-ω model: transport equation of turbulent energy K and relative dissipation rate ω

Femtet uses hybrid model of one- and two-equation models.

Analysis domain is divided to two layers:
- Near wall face ⇒ Wolfshtein's one-equation model
- Fully turbulent ⇒ Two-equation model (Realizable K-ε model)

- Reducing the mesh height near the wall face, detailed analysis of the flow near the wall is possible.
- Switching turbulent model is not required.
- The impact on the results by the mesh height near the wall face is minimized.

There are various K-ε models. In Femtet, Realizable K-ε model is used.

Realizable K-ε model
Considering the realizability, setup is made so that K and ε do not become negative.
Accuracy is high for the complex flows as well.

Turbulent Kinematic Viscosity Coefficient

$$\nu_t = C_\mu \frac{K^2}{\varepsilon} [m^2/s]$$

Turbulent Thermal Conductivity (proportional to the turbulent kinematic viscosity coefficient)

$$\lambda_t = \frac{\rho C_p \nu_t}{Pr_t} [W/m/deg]$$

Transport Equation of Turbulent Energy

$$\frac{\partial K}{\partial t} + (\boldsymbol{U} \cdot \nabla) K = G - \varepsilon + \nabla \cdot \left(\left(\nu + \frac{\nu_t}{\sigma_K} \right) \nabla K \right)$$

Transport Equation of Energy Dissipation Rate

$$\frac{\partial \varepsilon}{\partial t} + (\boldsymbol{U} \cdot \nabla) \varepsilon = C_{\varepsilon 1} S \varepsilon - C_{\varepsilon 2} \frac{\varepsilon^2}{K + \sqrt{\nu \varepsilon}} + \nabla \cdot \left(\left(\nu + \frac{\nu_t}{\sigma_\varepsilon} \right) \nabla \varepsilon \right)$$

Variables of Model

$$C_\mu = \frac{1}{A_0 + A_s U^* \frac{K}{\varepsilon}}$$

Displayable items are:
Turbulent energy K, Energy dissipation rate ε , Energy generation rate G ,
Turbulent viscosity rate $(\rho\nu_t)$, Turbulent viscosity ratio ν_t/ν,
and Turbulent thermal conductivity λ_t

Calculation of Turbulent Viscosity Coefficient near Wall Face

Murata Software

Wolfshtein's One-equation Model

The transport equation of turbulent energy K is solved to obtain the turbulent viscosity coefficient near the wall face.
The equations for turbulent kinematic viscosity coefficient $\nu_{t_{near\_wall}}$ and energy dissipation rate ε_{near_wall} are applied according to K and height y from the wall.
The Reynolds number of the wall Re_y is used to judge the nearness to the wall. ($Re_y < 200$)

Reynolds Number of the Wall

$$Re_y = \frac{\sqrt{K}y}{\nu}$$

Height from the Wall

$y[m]$

Transport Equation of Turbulent Energy

$$\frac{\partial K}{\partial t} + (\boldsymbol{U} \cdot \nabla)K = G - \varepsilon + \nabla \cdot \left(\left(\nu + \frac{\nu_t}{\sigma_K}\right)\nabla K\right)$$

Turbulent Kinematic Viscosity Coefficient

$$\nu_{t_{near\_wall}} = C_{\mu_{const}}^{1/4}\sqrt{K}l_\nu \; [m^2/s]$$

Turbulent Thermal Conductivity

(Proportional to turbulent kinematic viscosity coefficient)

$$\lambda_{t_{near\_wall}} = \frac{\rho C_p \nu_{t_{near\_wall}}}{Pr_t} \; [W/m/deg]$$

Energy Dissipation Rate

$$\varepsilon_{near\_wall} = C_{\mu\_const}^{3/4} \frac{K^{3/2}}{l_\varepsilon} [m^2/s^3]$$

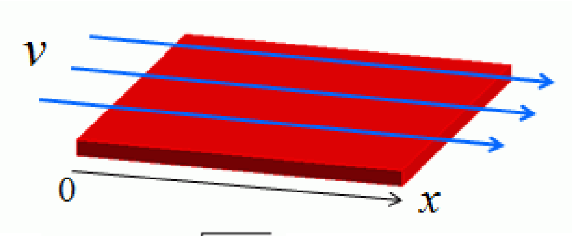
Displayable items are:
height from the wall, Reynolds number of the wall, and distribution of layers near the wall ($Re_y < 200$)

7-4. Simple Fluid-Thermal Analysis

Simple Fluid-Thermal Analysis

Murata Software

Assuming the plate, heat transfer coefficient is given to the solid surface.



$$h = 1.93\sqrt{v/x}$$

v: Flow velocity over the solid surface obtained by the analysis
x: Path length given by the analysis results

Heat transfer coefficient becomes smaller as the path length becomes larger (farther away from the upwind side)

<u>Features</u>

- -Effective if the model form is close to flat plate and temperature is constant.
- -Assumption is that;
  - the surrounding space is wide.
  - the fluid is not viscous.
  - the flow is laminar.
  - the temperature of the flow is constant (given as a main flow temperature)
- -Calculation load is smaller than the fluid-thermal analysis by more than 10 times.

<u>The case where the accuracy deteriorates</u>

- -The model form is not smooth but rough
- -Narrow path like a heatsink exists
- - Flow velocity is large

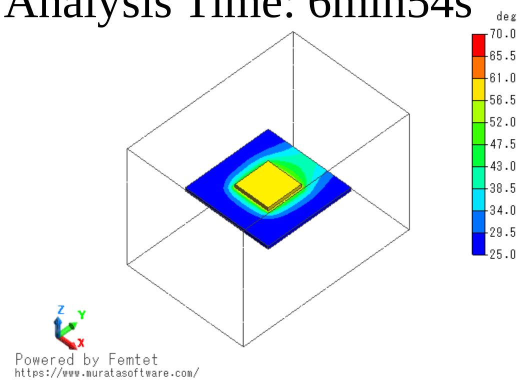
Comparison

Murata Software

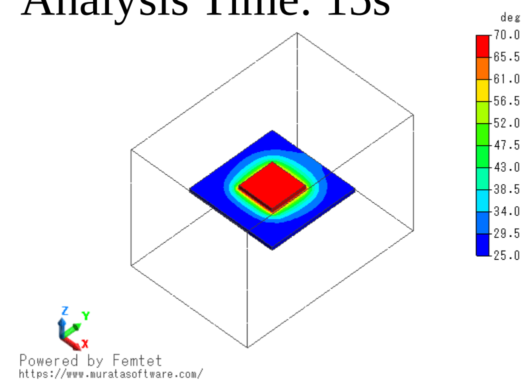
Fluid-Thermal Analysis
Analysis Time: 6min54s

Simple Fluid-Thermal Analysis
Analysis Time: 15s

Temperature Distribution

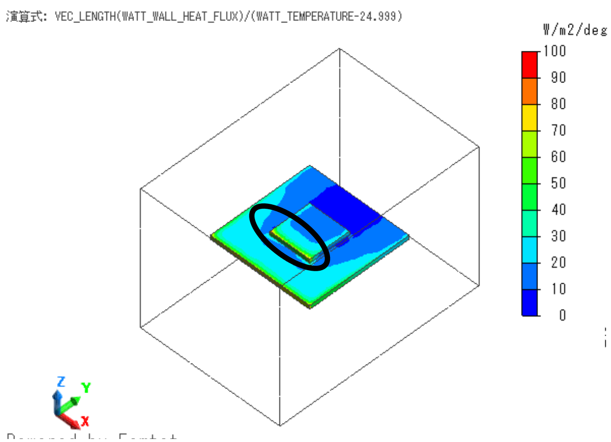


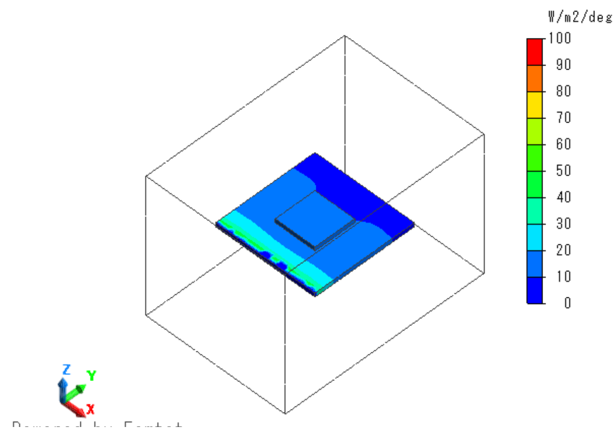
Max Temperature: 60.7°C



Max Temperature: 71.6°C

Heat Transfer Coefficient Distribution





As the increase of the heat transfer coefficient due to the unevenness of the surface is not taken into account, the calculation tends to result in the higher temperature.

Comparison

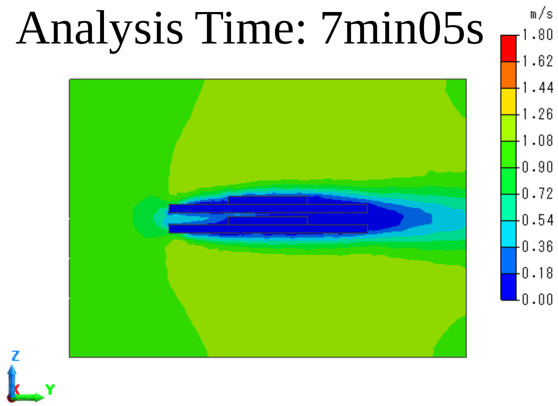
Murata Software

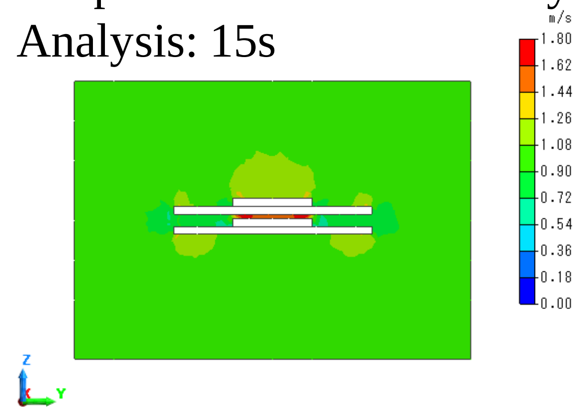
Two substrates are placed in short distance with narrow flow path in between.

Fluid-Thermal Analysis
Analysis Time: 7min05s

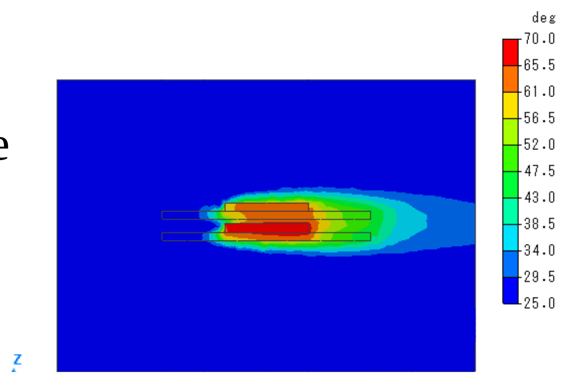
Simple Fluid-Thermal Analysis
Analysis: 15s

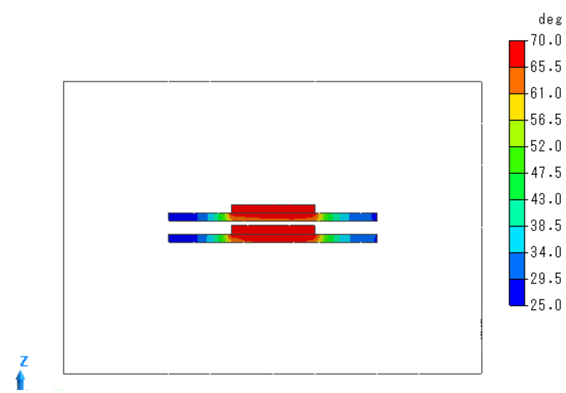
Flow
Velocity
Distributio





Temperature
Distribution





Mutual interference increases the temperature: 60°C⇒67°C

Flow calculation in the path results in the large velocity and the temperature decreases: 71°C⇒69°C