STOC-ML, STOC-IC

Chapter 1 **Theoretical Explanation**

1.1. Summary of the STOC System

The STOC system consists of the two following programs:

(1) STOC-ML (multi-layered static dynamics model)

STOC-ML is a quasi-3-dimensional model for calculating fluid dynamics that result from a tsunami by using hydrostatic approximation. We apply STOC-ML to perform calculations for tsunamis that propagate along the Pacific Ocean and other oceans, because it is a known fact that hydrostatic approximation is valid on tsunamis that occur offshore. Here, we have to mention that, in STOC-ML, the fluid can be divided into multi layers along the depth of water; though usually, the single layer model is used in tsunami calculations.

(2) STOC-IC (3-dimensional non-static model)

STOC-IC is a 3-dimensional model for calculating the fluid dynamics resulting from a tsunami, based on the most fundamental equations in fluid dynamics. The model is employed for calculating changes in tsunamis resulting from structures found in coastal areas. In general, STOC-IC is used in connection with STOC-ML. By coupling the two models, it is possible to calculate, with high accuracy, the behavior of tsunami that occur offshore and then propagate oceans and hit coastal areas, such as ports or harbors.

The functions of STOC are listed in Table 0-1-1. With STOC, you couple several STOC-MLs and one STOC-IC to allow for calculations that couple the domains shown in Figure 0-1-1 and Figure 0-1-2. The communication protocol in use is the Message Passing Interface (MPI).

The following externally available programs can also be coupled with STOC. A separate manual for STOC-DM (one of the programs listed below) will be made available.

(3) STOC-DM (debris model)
(4) CADMAS-SURF/3D (3-dimensional non-static dynamical model, VOF Method)
(5) STOC-OIL (oil diffusion model)
(6) AGENT (multi-agent model)

Table 0-1-1 STOC functions

Item	Description
Analyzed	Storm surge and tsunami STOC-ML covers the propagation of waves over a wide range of sea bodies, from the open ocean to the coast. On the other hand, STOC-IC primarily covers the detailed calculation of the behavior of water streams in harbors.
Fundamental equation	An equation extended from the Navier-Stokes equation by using the Porous approximation for 3-dimensional incompressible viscous fluids. - Equation of continuity - Equation of momentum conservation - Equation for free surfaces - Hydrostatic conditions (STOC-ML)
Physical model	- Run-up tip model - Permeable structure model - Turbulent flow model (LES, k-ε) -Transparent boundary model - Dispersive wave model - Breaking wave model - Overflow model - Model for a change in water level due to an earthquake
Discretization	- Difference equations using staggered mesh - Shape approximation using the porous model
Advection term	- Second-order accurate central-differencing scheme - First-order accurate upwind scheme - Hybrid difference, weighted average of the above parameters
Time integration	- Leap-frog method - SMAC (Simplified Marker and Cell) method
Method to solve simultaneous linear equations	- MILU-BiCGStab method

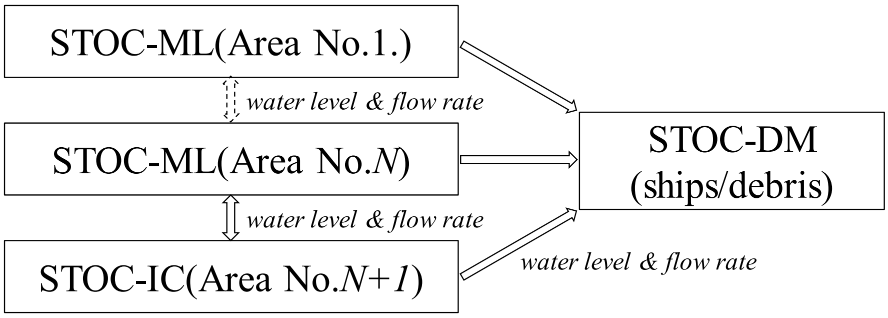


Figure 0-1-1 Relationship among the Calculations Coupled in STOC

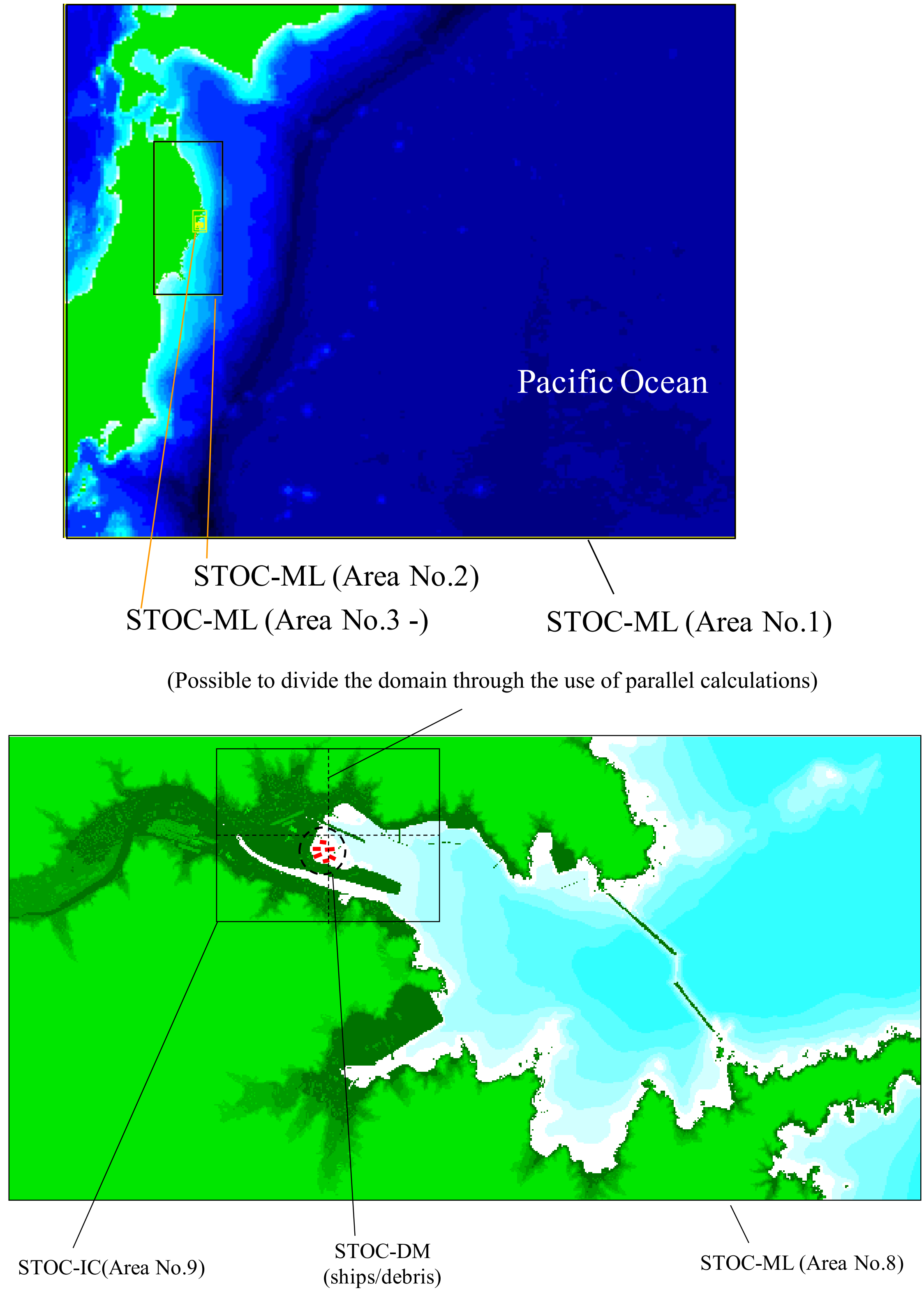


Figure 0-1-2 Example of the Division of Domains in the Calculations Coupled by STOC

1.2. Fundamental Equations

Although STOC-ML is a hydrostatic model and STOC-IC a non-hydrostatic model, both models share a number of things in common. The following sections show all the fundamental equations and describe the differences between the two programs where there are any.

1.2.1. Definitions of Physical Quantities

Below are the descriptions of the symbols used in the equations. All of them are listed as SI units unless otherwise stated.

D : Total water depth [m] (□η□h, defined in Figure 0-1-3)

f_0 : Coriolis parameter [1/s] (= $2\Omega \sin\theta$)

g : Gravitational acceleration [m/s^2] (≈ -9.8)

h : Thickness ratio of water in a computational cell (hereinafter referred to as the "layer thickness ratio") [-] (defined in Figure 0-1-5)

H : Water depth [m]

p : Pressure [Pa]

p_{atm}: Atmospheric pressure [Pa]

R : Earth's radius [m] (assuming that the Earth is a sphere, not an ellipsoid)

u : x-component of flow velocity [m/s]

v : y-component of flow velocity [m/s]

w : z-component of flow velocity [m/s]

ϕ : Longitude [rad] (positive to the east)

γ_v : Porous value (porosity) [-] ($0 \le \gamma_v \le 1$, defined in Figure 0-1-4)

$\gamma_x, \gamma_y, \gamma_z$: Porous values (permeability in each direction) [-] ($0 \le \gamma_x, \gamma_y \le 1$, γ_z □1)

η : Water level [m]

ν_H : Horizontal kinematic viscosity coefficient [m^2/s]

ν_V : Vertical kinematic viscosity coefficient [m^2/s]

Ω : Earth's rotational speed [1/s]

θ : Latitude [rad] (positive to the north)

ρ : Seawater density [kg/m^3]

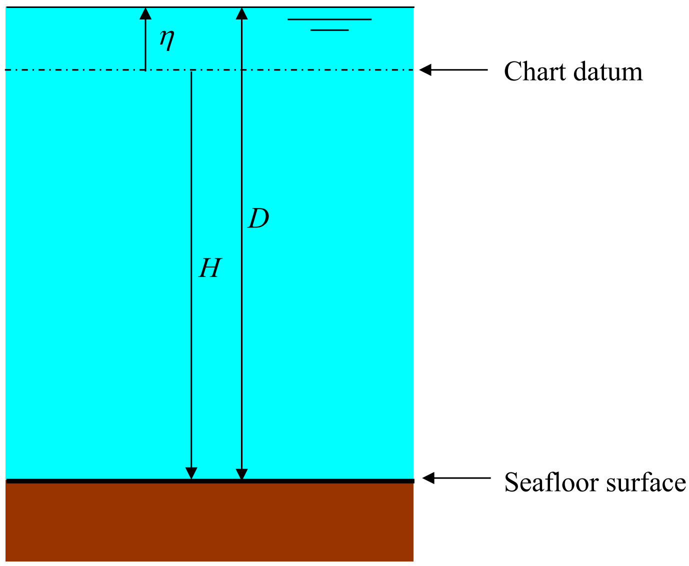


Figure 0-1-3 Definitions of Water Level, Water Depth, and Total Water Depth

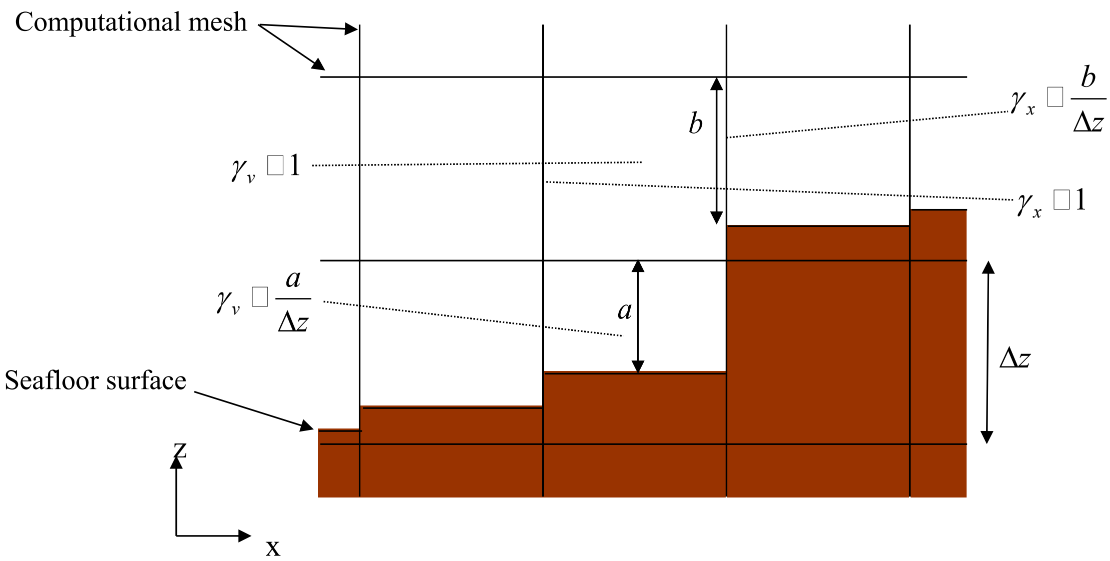


Figure 0-1-4 Definition of Porous Values

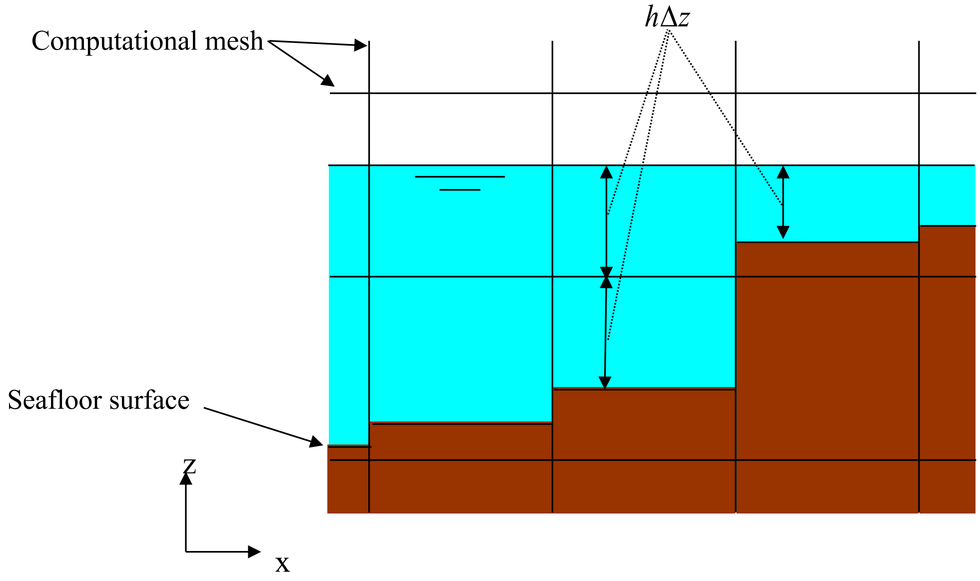


Figure 0-1-5 Definition of Layer Thickness

1.2.2. Fundamental Equations in a Plane coordinate system

The fundamental equation used in STOC-ML and STOC-IC are a three-dimensional continuity equation for porousness and the momentum conservation equation. Porous values are used to represent topography at resolutions less than or equal to those of computational meshes.

1.2.2.1. Continuity Equation

$$\frac{\partial}{\partial x}\left(\gamma_x u\right)+\frac{\partial}{\partial y}\left(\gamma_y v\right)+\frac{\partial}{\partial z}\left(\gamma_z w\right)=0 \qquad (1.2.1)$$

1.2.2.2. Momentum Conservation Equation

(1) x direction

$$\begin{aligned}\gamma_v\frac{\partial u}{\partial t}&+\frac{\partial}{\partial x}\left(\gamma_x uu\right)+\frac{\partial}{\partial y}\left(\gamma_y uv\right)+\frac{\partial}{\partial z}\left(\gamma_z uw\right)=\gamma_v f_0 v\\&-\gamma_v\frac{1}{\rho}\frac{\partial p}{\partial x}+\frac{\partial}{\partial x}\left(\gamma_x\nu_H 2\frac{\partial u}{\partial x}\right)+\frac{\partial}{\partial y}\left\{\gamma_y\nu_H\left(\frac{\partial u}{\partial y}+\frac{\partial v}{\partial x}\right)\right\}\\&+\frac{\partial}{\partial z}\left\{\gamma_z\nu_V\left(\frac{\partial u}{\partial z}+\frac{\partial w}{\partial x}\right)\right\}\end{aligned} \qquad (1.2.2)$$

(2) y direction

$$\begin{aligned}\gamma_v\frac{\partial v}{\partial t}&+\frac{\partial}{\partial x}\left(\gamma_x vu\right)+\frac{\partial}{\partial y}\left(\gamma_y vv\right)+\frac{\partial}{\partial z}\left(\gamma_z vw\right)=-\gamma_v f_0 u\\&-\gamma_v\frac{1}{\rho}\frac{\partial p}{\partial y}+\frac{\partial}{\partial x}\left\{\gamma_x\nu_H\left(\frac{\partial v}{\partial x}+\frac{\partial u}{\partial y}\right)\right\}+\frac{\partial}{\partial y}\left(\gamma_y\nu_H 2\frac{\partial v}{\partial y}\right)\\&+\frac{\partial}{\partial z}\left\{\gamma_z\nu_V\left(\frac{\partial v}{\partial z}+\frac{\partial w}{\partial y}\right)\right\}\end{aligned} \qquad (1.2.3)$$

(3) z direction (STOC-IC only)

$$\begin{aligned}\gamma_v\frac{\partial w}{\partial t}&+\frac{\partial}{\partial x}\left(\gamma_x wu\right)+\frac{\partial}{\partial y}\left(\gamma_y wv\right)+\frac{\partial}{\partial z}\left(\gamma_z ww\right)\\&=-\gamma_v\frac{1}{\rho}\frac{\partial p}{\partial z}-g+\frac{\partial}{\partial x}\left\{\gamma_x\nu_H\left(\frac{\partial w}{\partial x}+\frac{\partial u}{\partial z}\right)\right\}\\&+\frac{\partial}{\partial y}\left(\gamma_y\nu_H\left(\frac{\partial w}{\partial y}+\frac{\partial v}{\partial z}\right)\right)+\frac{\partial}{\partial z}\left\{\gamma_z\nu_V 2\frac{\partial w}{\partial z}\right\}\end{aligned} \qquad (1.2.4)$$

STOC-ML calculates w with the continuity equation without solving the z-direction momentum conservation equation (performs integrations upward from the seafloor surface by assigning u, v to the continuity equation).

1.2.2.3. Free Surface Equation

In STOC-ML and IC, only one water surface is defined at a certain point (x, y).

$$\gamma_z \frac{\partial \eta}{\partial t} + \frac{\partial}{\partial x}\int_{-H}^{\eta} \gamma_x u\, dz + \frac{\partial}{\partial y}\int_{-H}^{\eta} \gamma_y v\, dz = 0 \quad (1.2.5)$$

1.2.2.4. Hydrostatic Pressure Equation (STOC-ML only)

Since the hydrostatic pressure is implicit in STOC-ML, the equation represents pressure as a function of the vertical distance from a water surface as shown below.

$$p(z) = p_{atm} - \rho g(\eta - z) \quad (1.2.6)$$

Meanwhile, STOC-IC couples the continuity equation and momentum conservation equation to calculate pressure.

1.2.3. Fundamental Equations in a Spherical Coordinate System

In a spherical coordinate system, it is assumed that the Earth's radius R is constant irrespective of water depth, with longitude, latitude, longitudinal velocity components (positive to the east), and latitudinal velocity components (positive to the north) denoted by ϕ, θ, u, and v, respectively. Figure 0-1-6 and Figure 0-1-7 provide a comparison in a Plane coordinate system in terms of the length of each side of a control volume (cell). Table 0-1-2 lists quantities related to spatial differentiation in control volumes as a means of comparison.

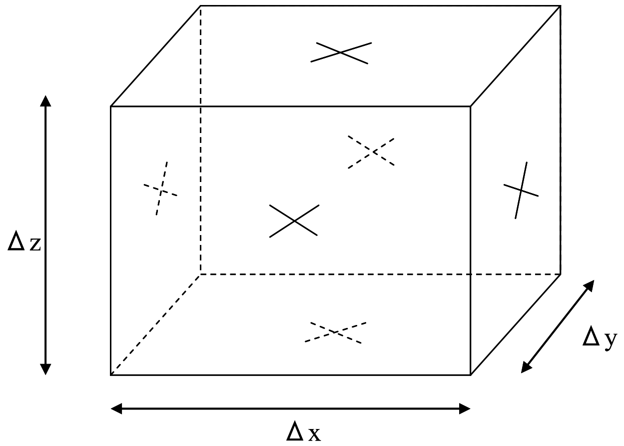


Figure 0-1-6 Length of Each Side of a Cell in a Plane coordinate system

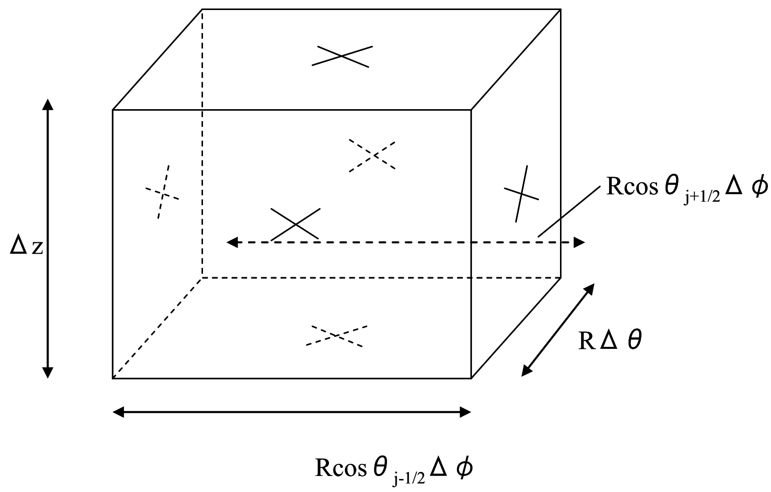


Figure 0-1-7 Length of Each Side of a Cell in a Spherical coordinate system

Table 0-1-2 Comparison of the Cartesian and Spherical coordinate systems

Item	Plane coordinate system	Spherical coordinate system
Cell volume	$\Delta x\,\Delta y\,\Delta z$	$R^2\cos\theta_j\Delta\phi\Delta\theta\Delta z$
-X surface area	$\Delta y\,\Delta z$	$R\,\Delta\theta\,\Delta z$
+X surface area	(Same as above)	(Same as above)
-Y surface area	$\Delta x\ \Delta z$	$R\cos\theta_{j-1/2}\ \Delta\phi\,\Delta z$
+Y surface area	(Same as above)	$R\cos\theta_{j\square 1/2}\ \Delta\phi\,\Delta z$
-Z surface area	$\Delta x\,\Delta y$	$R^2\cos\theta_j\ \Delta\phi\Delta\theta$
+Z surface area	(Same as above)	(Same as above)
(1, 1) component of viscous stress	$2\nu_H\dfrac{\partial u}{\partial x}$	$2\nu_H h\dfrac{1}{R\cos\theta}\dfrac{\partial u}{\partial\phi}$
(1, 2) component of viscous stress	$\nu_H\left(\dfrac{\partial v}{\partial x}\square\dfrac{\partial u}{\partial y}\right)$	$\nu_H\left(\dfrac{1}{R\cos\theta}\dfrac{\partial v}{\partial\phi}\square\dfrac{1}{R}\dfrac{\partial u}{\partial\theta}\right)$
(1, 3) component of viscous stress	$\nu_V\left(\dfrac{\partial w}{\partial x}\square\dfrac{\partial u}{\partial z}\right)$	$\nu_V\left(\dfrac{1}{R\cos\theta}\dfrac{\partial w}{\partial\phi}\square\dfrac{\partial u}{\partial z}\right)$
(2, 1) component of viscous stress	$\nu_H\left(\dfrac{\partial v}{\partial x}\square\dfrac{\partial u}{\partial y}\right)$	$\nu_H\left(\dfrac{1}{R\cos\theta}\dfrac{\partial v}{\partial\phi}\square\dfrac{1}{R}\dfrac{\partial u}{\partial\theta}\right)$
(2, 2) component of viscous stress	$2\nu_H\dfrac{\partial v}{\partial y}$	$2\nu_H\dfrac{1}{R}\dfrac{\partial v}{\partial\theta}$
(2, 3) component of viscous stress	$\nu_V\left(\dfrac{\partial w}{\partial y}\square\dfrac{\partial v}{\partial z}\right)$	$\nu_V\left(\dfrac{1}{R}\dfrac{\partial w}{\partial\theta}\square\dfrac{\partial v}{\partial z}\right)$
(3, 1) component of viscous stress	$\nu_H\left(\dfrac{\partial w}{\partial x}\square\dfrac{\partial u}{\partial z}\right)$	$\nu_H\left(\dfrac{1}{R\cos\theta}\dfrac{\partial w}{\partial\phi}\square\dfrac{\partial u}{\partial z}\right)$
(3, 2) component of viscous stress	$\nu_H\left(\dfrac{\partial w}{\partial y}\square\dfrac{\partial v}{\partial z}\right)$	$\nu_H\left(\dfrac{1}{R}\dfrac{\partial w}{\partial\theta}\square\dfrac{\partial v}{\partial z}\right)$

(3, 3) component of viscous stress	$2\nu_V \dfrac{\partial w}{\partial z}$	$2\nu_V \dfrac{\partial w}{\partial z}$

We can take the above and simplify the fundamental equations for use in a spherical coordinate system as follows. Note that it is impossible to apply a system that includes the north and south poles.

1.2.3.1. Continuity Equation

$$\frac{1}{R\cos\theta}\frac{\partial}{\partial\phi}\left(\gamma_x u\right)+\frac{1}{R\cos\theta}\frac{\partial}{\partial\theta}\left(\gamma_y v\cos\theta\right)+\frac{\partial}{\partial z}\left(\gamma_z w\right)=0 \quad \text{.......................................} (1.2.7)$$

1.2.3.2. Momentum Conservation Equations

(1) Longitudinal direction (positive to the east)

$$\begin{aligned}
&\gamma_v\frac{\partial u}{\partial t}+\frac{1}{R\cos\theta}\frac{\partial}{\partial\phi}\left(\gamma_x uu\right)+\frac{1}{R\cos\theta}\frac{\partial}{\partial\theta}\left(\gamma_y uv\cos\theta\right)+\frac{\partial}{\partial z}\left(\gamma_z uw\right)\\
&-\gamma_v 2\Omega\sin\theta\, v\\
&=-\gamma_v\frac{1}{\rho}\frac{1}{R\cos\theta}\frac{\partial p}{\partial\phi}+\frac{1}{R\cos\theta}\frac{\partial}{\partial\phi}\left(\gamma_x\nu_H\frac{2}{R\cos\theta}\frac{\partial u}{\partial\phi}\right) \qquad \text{.....................} (1.2.8)\\
&\quad+\frac{1}{R\cos\theta}\frac{\partial}{\partial\theta}\left\{\gamma_y\nu_H\cos\theta\left(\frac{1}{R}\frac{\partial u}{\partial\theta}+\frac{1}{R\cos\theta}\frac{\partial v}{\partial\phi}\right)\right\}\\
&\quad+\frac{\partial}{\partial z}\left\{\gamma_z\nu_V\left(\frac{\partial u}{\partial z}+\frac{1}{R\cos\theta}\frac{\partial w}{\partial\phi}\right)\right\}
\end{aligned}$$

(2) Latitudinal direction (positive to the north)

$$\begin{aligned}
&\gamma_v\frac{\partial v}{\partial t}+\frac{1}{R\cos\theta}\frac{\partial}{\partial\phi}\left(\gamma_x vu\right)+\frac{1}{R\cos\theta}\frac{\partial}{\partial\theta}\left(\gamma_y vv\cos\theta\right)+\frac{\partial}{\partial z}\left(\gamma_z vw\right)\\
&+\gamma_v 2\Omega\sin\theta\, u\\
&=-\gamma_v\frac{1}{\rho}\frac{1}{R}\frac{\partial p}{\partial\theta}+\frac{1}{R\cos\theta}\frac{\partial}{\partial\phi}\left\{\gamma_x\nu_H\left(\frac{1}{R\cos\theta}\frac{\partial v}{\partial\phi}+\frac{1}{R}\frac{\partial u}{\partial\theta}\right)\right\} \qquad \text{......................} (1.2.9)\\
&\quad+\frac{1}{R\cos\theta}\frac{\partial}{\partial\theta}\left(\gamma_y\nu_H\cos\theta\frac{2}{R}\frac{\partial v}{\partial\theta}\right)+\frac{\partial}{\partial z}\left\{\gamma_z\nu_V\left(\frac{\partial v}{\partial z}+\frac{1}{R}\frac{\partial w}{\partial\theta}\right)\right\}
\end{aligned}$$

(3) Z direction (STOC-IC only)

$$\begin{aligned}
&\gamma_v\frac{\partial w}{\partial t}+\frac{1}{R\cos\theta}\frac{\partial}{\partial\phi}\left(\gamma_x wu\right)+\frac{1}{R\cos\theta}\frac{\partial}{\partial\theta}\left(\gamma_y wv\cos\theta\right)+\frac{\partial}{\partial z}\left(\gamma_z ww\right)\\
&=-\gamma_v\frac{1}{\rho}\frac{\partial p}{\partial z}-g+\frac{1}{R\cos\theta}\frac{\partial}{\partial\phi}\left\{\gamma_x\nu_H\left(\frac{1}{R\cos\theta}\frac{\partial w}{\partial\phi}+\frac{\partial u}{\partial z}\right)\right\} \qquad \text{................} (1.2.10)\\
&\quad+\frac{1}{R\cos\theta}\frac{\partial}{\partial\theta}\left(\gamma_y\nu_H\cos\theta\left(\frac{1}{R}\frac{\partial w}{\partial\theta}+\frac{\partial v}{\partial z}\right)\right)+\frac{\partial}{\partial z}\left\{\gamma_z\nu_V 2\frac{\partial w}{\partial z}\right\}
\end{aligned}$$

1.2.3.3. Free Surface Equation

In STOC-ML and IC, only one water surface is defined at a certain point (x, y).

$$\gamma_z \frac{\partial \eta}{\partial t} + \frac{1}{R\cos\theta}\frac{\partial}{\partial \phi}\int_{-H}^{\eta}\gamma_x u\,dz + \frac{1}{R\cos\theta}\frac{\partial}{\partial \theta}\int_{-H}^{\eta}\gamma_y v\cos\theta\,dz = 0 \quad \text{(1.2.11)}$$

1.2.4. Boundary Conditions

The general boundary conditions are shown below. Special boundaries such as transparent boundaries, overflow boundaries for storm surge barriers, and nesting boundaries will be described later.

1.2.4.1. Shorelines

The conditions for shorelines (those assumed to be steep) are as follows.

- Water level: Zero gradient
- Flow velocity: Slip, no slip, and fixed flow velocity

Use fixed flow velocity for rivers running into the ocean.

1.2.4.2. Outer Edge of a Computation Region

The conditions for the outer edge of a computation region are as follows. The transparent boundary model will be described later.

- Water level: Fixed water level, transparent boundary, or small-amplitude waves–select one of the three
- Flow velocity: Zero gradient (fixed velocity for small-amplitude waves)

1.2.4.3. Water Surface

The conditions for water surfaces are as follows.

- Flow velocity: Slip or wind stress

Calculate the wind stress as follows.

$$\tau_x = \rho_a \gamma_a^2 W_x \sqrt{W_x^2 + W_y^2}, \ \tau_y = \rho_a \gamma_a^2 W_y \sqrt{W_x^2 + W_y^2} \quad \text{(1.2.12)}$$

where

W_x, W_y refer to the x and y components of wind speed [m/s],

γ_a^2 refers to the surface friction coefficient [-], and

ρ_a refers to air density [kg/m^3]

Specify the surface friction coefficient as an input parameter. Or else, calculate this coefficient by using these functions:

$$\gamma_a^2 = 0.001 \times (1.29 - 0.024\sqrt{W_x^2 + W_y^2} \quad (\text{when } \sqrt{W_x^2 + W_y^2} < 8.0 m/s) \quad \text{(1.2.13)}$$

$$\gamma_a^2 = 0.001\times(0.581 + 0.063\sqrt{W_x^2 + W_y^2} \quad (\text{when} \quad \sqrt{W_x^2 + W_y^2} \geq 8.0 m/s) \qquad (1.2.14)$$

1.2.4.4. Seafloor Surface

The conditions for seafloor surfaces are as follows.

- Flow velocity: Slip, no slip, equation using the Manning's roughness, or friction stress equation– select one of the four

The equation that uses Manning's roughness is as follows:

$$\tau_x = \frac{\rho g n^2 u_b \sqrt{u_b^2 + v_b^2}}{h^{1/3}}, \quad \tau_x = \frac{\rho g n^2 v_b \sqrt{u_b^2 + v_b^2}}{h^{1/3}} \qquad (1.2.15)$$

The friction stress equation is as follows:

$$\tau_x = \rho\gamma_b^2 u_b \sqrt{u_b^2 + v_b^2}, \quad \tau_y = \rho\gamma_b^2 v_b \sqrt{u_b^2 + v_b^2} \qquad (1.2.16)$$

where

n refers to Manning roughness,

u_b, v_b refer to the x and y components of flow velocity of an adjacent mesh on the seafloor surface [m/s], and

γ_b^2 refers to the seafloor friction coefficient [-]

1.3. Physical Models

1.3.1. Run-up Tip Model

The run-up tip model restricts the movement of the leading edge of a tsunami by using altitude differences during tsunami runup to land.

If this model is not used, the following problem occurs:

Suppose, for example, that a vertical mesh is a single-layer mesh for calculation as shown in Figure 0-1-8 and that the meshes of a flatland at an altitude of 0 m and a mountain which is 1,000 m high are right next to each other. In this case, if a wave hits the mountain when water flows leftward along the boundary, water runs up to the cell on the left that is at an elevation of 1,000 m because the total water depth along the boundary is not 0.

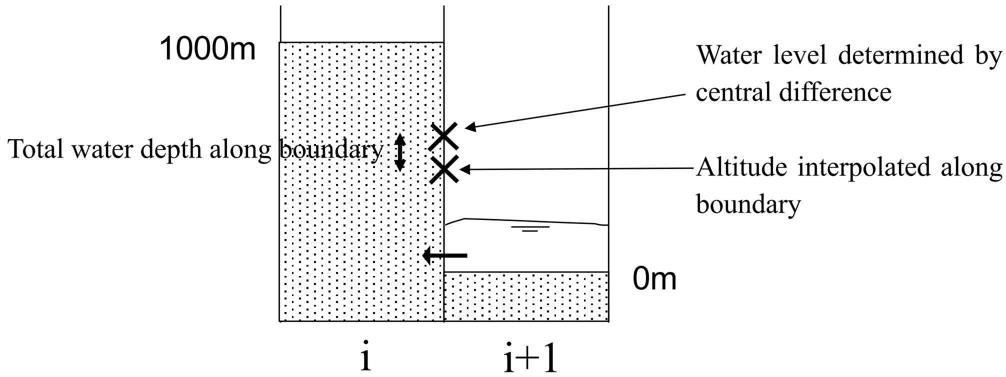


Figure 0-1-8 Run-up Tip Model Not Used

To avoid this problem, the flow velocity should be set to 0 without calculating the flow velocity in the position where the leading edge was detected. The conditions for detecting the leading edge are shown below (the velocity of a flow in x direction for the boundary between cells i and i + 1 is calculated).

$$K \square KF(I,J), K \square KG(I,J),$$
$$\eta_i - z_{G,i} \leq \mathrm{GXB}\,,\ z_{G,i} \geq \eta_{i\square 1} \square z_{G,i\square 1} \square \mathrm{GXB} \quad \cdots\cdots (1.3.1)$$

or

$$K \square KF(I \square 1,J), K \square KG(I \square 1,J),$$
$$\eta_{i\square 1} - z_{G,i\square 1} \leq \mathrm{GXB}\,,\ z_{G,i\square 1} \geq \eta_i \square z_{G,i} \square \mathrm{GXB} \quad \cdots\cdots (1.3.2)$$

where

KF refers to the z-direction index for the cell that includes the water surface ($KF \geq KG$),

KG refers to the z-direction index for the cell that includes the ground surface,

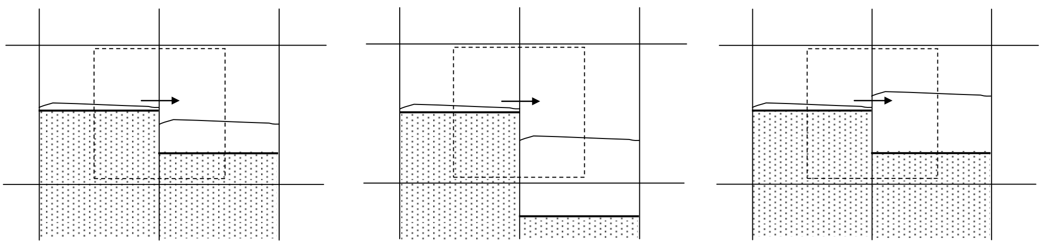
η refers to the water level [m] ($\eta \square z_G$),

z_G refers to the ground surface altitude [m], and

GXB refers to the runup detection parameter (about 10^{-4}[m])

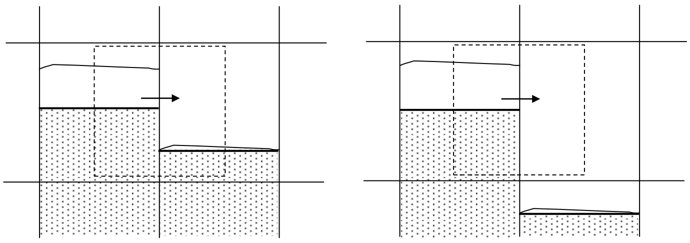
Note that there is a small amount of water in the dry land, and thus $\eta \square z_G \square \varepsilon$ (ε : Small quantity) and $KF \square KG$.

When the detection conditions above are applied to the system below, the flow velocity for the cells on the left and in the middle is not calculated, but that for the cell on the right is calculated.



(The cell on the left contains a small quantity of water.)

The flow velocity for both the cells on the left and right is calculated in the system below.



(The cell on the right contains a small quantity of water.)

1.3.2. Turbulent Flow Models

The turbulent flow models below are incorporated in STOC. The SGS and k-ε models are only incorporated in STOC-IC.

1.3.2.1. LES Model (Zero-Equation Model)

The LES model calculates the turbulent kinematic viscosity ν_t as follows.

$$\nu_t = (C_S \Delta)^2 \sqrt{2 S_{ij} S_{ij}} \quad \text{.......... (1.3.3)}$$

$$\Delta = (\Delta x \Delta y \Delta z)^{1/3} \quad \text{.......... (1.3.4)}$$

$$S_{ij} = \frac{1}{2}\left(\frac{\partial v_j}{\partial x_i} + \frac{\partial v_i}{\partial x_j} \right) \quad \text{.......... (1.3.5)}$$

C_S is a model parameter which can take values approximately equal to 0.1 through 0.2.

1.3.2.2. SGS Model (One-Equation Model)

The SGS model is a one-equation LES model developed by Deardorff. It calculates the turbulent kinematic viscosity ν_t as follows.

$$\nu_t = (C_m \Delta) k^{1/2} \quad \text{.......... (1.3.6)}$$

where

k refers to turbulent kinetic energy [m^2/s^2]

k is obtained by solving the equation below.

$$\gamma_v \frac{\partial k}{\partial t} + \frac{\partial \gamma_x k u}{\partial x} + \frac{\partial \gamma_y k v}{\partial y} + \frac{\partial \gamma_z k w}{\partial z} = \frac{\partial}{\partial x}\left(\gamma_x \frac{\nu_t}{\sigma_k} \frac{\partial k}{\partial x} \right) + \frac{\partial}{\partial y}\left(\gamma_y \frac{\nu_t}{\sigma_k} \frac{\partial k}{\partial y} \right) + \frac{\partial}{\partial z}\left(\gamma_z \frac{\nu_t}{\sigma_k} \frac{\partial k}{\partial z} \right) + P_k - \frac{C_\varepsilon}{l} k^{3/2} \quad \text{.......... (1.3.7)}$$

where P_k is a production term of turbulent kinetic energy produced by shearing and is expressed as follows:

$$P_k = \frac{1}{2}\nu_t\left(\frac{\partial v_i}{\partial x_j} + \frac{\partial v_j}{\partial x_i}\right)^2 \quad \text{(1.3.8)}$$

Note that C_m, C_ε, and σ_k, which are model parameters, equal 0.12, 0.31, and 0.7, respectively.

1.3.2.3. k-ε Model (Two-Equation Model)

The k-ε model calculates the turbulent kinematic viscosity ν_t as follows.

$$\nu_t = C_\mu \frac{k^2}{\varepsilon} \quad \text{(1.3.9)}$$

where

k refers to turbulent kinetic energy [m^2/s^2],

ε refers to turbulent kinetic energy dissipation rate [m^2/s^3], and

C_μ refers to constant (0.09)

k and ε are obtained by solving the equation below.

• Transport equation for k

$$\gamma_v \frac{\partial k}{\partial t} + \frac{\partial \gamma_x k u}{\partial x} + \frac{\partial \gamma_y k v}{\partial y} + \frac{\partial \gamma_z k w}{\partial z} = \frac{\partial}{\partial x}\left(\gamma_x \frac{\nu_t}{\sigma_k}\frac{\partial k}{\partial x}\right) + \frac{\partial}{\partial y}\left(\gamma_y \frac{\nu_t}{\sigma_k}\frac{\partial k}{\partial y}\right) + \frac{\partial}{\partial z}\left(\gamma_z \frac{\nu_t}{\sigma_k}\frac{\partial k}{\partial z}\right) + P_k - \varepsilon \quad \text{(1.3.10)}$$

• Transport equation for ε

$$\gamma_v \frac{\partial \varepsilon}{\partial t} + \frac{\partial \gamma_x \varepsilon u}{\partial x} + \frac{\partial \gamma_y \varepsilon v}{\partial y} + \frac{\partial \gamma_z \varepsilon w}{\partial z} = \frac{\partial}{\partial x}\left(\gamma_x \frac{\nu_t}{\sigma_\varepsilon}\frac{\partial \varepsilon}{\partial x}\right) + \frac{\partial}{\partial y}\left(\gamma_y \frac{\nu_t}{\sigma_\varepsilon}\frac{\partial \varepsilon}{\partial y}\right) + \frac{\partial}{\partial z}\left(\gamma_z \frac{\nu_t}{\sigma_\varepsilon}\frac{\partial \varepsilon}{\partial z}\right) + \frac{\varepsilon}{k}\left(C_{\varepsilon 1} P_k - C_{\varepsilon 2}\varepsilon\right) \quad \text{(1.3.11)}$$

σ_k, σ_ε, $C_{\varepsilon 1}$, $C_{\varepsilon 2}$, which are model parameters, equal 1.0, 1.3, 1.44, and 1.92, respectively.

1.3.2.4. Specification of the Turbulent Model

Specify the turbulent model in the %MODEL block of the analysis condition file by using TURBULENT through K-EPS.COEF.

1.3.3. Transparent Boundary Model

The transparent boundary model is applied to the outer periphery of the outermost region. This model prevents waves that go from the inside of a region to the outside from reflecting off the boundary and returning. STOC uses two methods: the characteristic curve and local virtual boundary.

1.3.3.1. Characteristic Curve Method

The characteristic curve method sets water level η_{bc} for a calculation cell, marked ● in Figure 1-9, as the boundary condition. Define η_{bc} as follows.

(1) Determine the position within the computation region, which is $l \square \sqrt{gH}\Delta t$ away from the position ●. This distance is travelled by a wave in one step (along the flow direction). The destination is indicated by ● in Figure 0-1-9.

(2) Determine water level η_{interp} for the position calculation cell marked ● from the surrounding water level by linear interpolation.

(3) Replace η_{bc} for original position ● with η_{interp}.

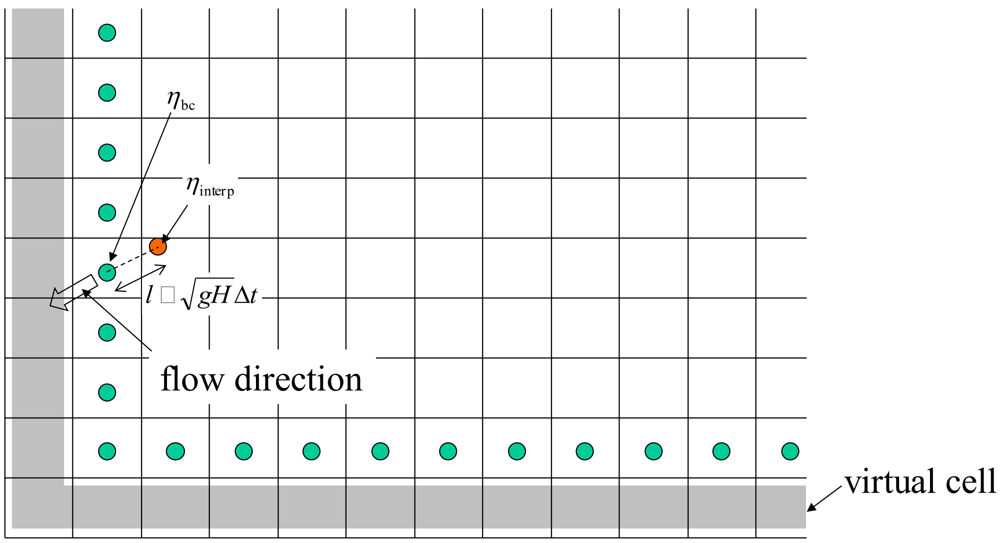


Figure 0-1-9 Transparent Boundary Processing by the Characteristic Curve Method

1.3.3.2. Method Developed by Imamura Et Al.

The "local virtual boundary method" has been adopted. The method is one of Imamura et al.'s models (2001)[1], which are more precise versions of the models (1988)[2] proposed by Hino and Nakaza.

This method obtains a reflected wave that is assumed to exist when there is a virtual wall at the boundary of the outer periphery. Then, it specifies 1/2 of the height of that wave as the wave height, determined at the next time point, of the cell adjacent to the boundary. When there is a virtual wall at the boundary of the outer periphery as shown in Figure 0-1-10, the continuity equation is discretized in one inner cell as follows.

$$\frac{\eta_{i,j}^{n+1}-\eta_{i,j}^{n}}{\Delta t}+\frac{M_{i+1/2,j}^{n}-M_{i-1/2,j}^{n}}{\Delta x}+\frac{N_{i,j+1/2}^{n}-N_{i,j-1/2}^{n}}{\Delta y}=0 \quad \text{.......... (1.3.12)}$$

where

M refers to the flow rate per unit width in the x direction [m²/s] ($=\int_{-H}^{\eta} u dz$) and

N refers to the flow rate per unit width in the y direction [m²/s] ($=\int_{-H}^{\eta} v dz$).

By substituting the virtual-wall condition $M_{i+1/2,j}^{n}=0$ into the equation above, $\eta_{i,j}^{n+1}$ is calculated as follows:

$$\eta_{i,j}^{n+1}=\eta_{i,j}^{n}+\frac{\Delta t}{\Delta x}M_{i-1/2,j}^{n}-\frac{\Delta t}{\Delta y}\left(N_{i,j+1/2}^{n}-N_{i,j-1/2}^{n}\right) \quad \text{.......... (1.3.13)}$$

When this water level is $\eta_{i,j}^{\prime n+1}$ and contains the level of the reflected wave, the water level boundary condition for adjacent cells is expressed as follows, using 1/2 of that water level.

$$\eta_{i,j}^{n+1}=\frac{1}{2}\eta_{i,j}^{\prime n+1}=\frac{1}{2}\eta_{i,j}^{n}+\frac{1}{2}\frac{\Delta t}{\Delta x}M_{i-1/2,j}^{n}-\frac{1}{2}\frac{\Delta t}{\Delta y}\left(N_{i,j+1/2}^{n}-N_{i,j-1/2}^{n}\right) \quad \text{.......... (1.3.14)}$$

Note that as the restriction condition for setting $\eta_{i,j}^{n+1}=\frac{1}{2}\eta_{i,j}^{\prime n+1}$, $\frac{\Delta x}{\Delta t}=\sqrt{g(\eta+H)}$ is applied when the normal direction of the boundary is the x direction, and $\frac{\Delta y}{\Delta t}=\sqrt{g(\eta+H)}$ is applied when that normal direction is the y direction. This is because the wave that hits the virtual wall should return to the same place during one step.

---

1 Fumihiko Imamura, Isao Yoshida, and Andrew Moore; "Numerical Study on the 1771 Meiwa Tsunami at Ishigaki Island, Okinawa and the Movement of the Tsunami Stones "; Proceedings of Coastal Engineering Lecture, JSCE, Vol.48, pp.346-350 (2001)

2 Mikio Hino and Eizo Nakaza; "Applying a New Numerical Wave Simulation Scheme on a Nonreflective Two-dimensional Open Boundary", Proceedings of Coastal Engineering, JSCE, Vol.35, pp.262-266 (1988)

When waves go out toward the boundary, the method developed by Imamura et al. can suppress the reflected waves better than the characteristic curve method. In contrast, when waves propagate along the boundary, the method developed by Imamura et al. might cause an abnormality in water distribution for the boundary.

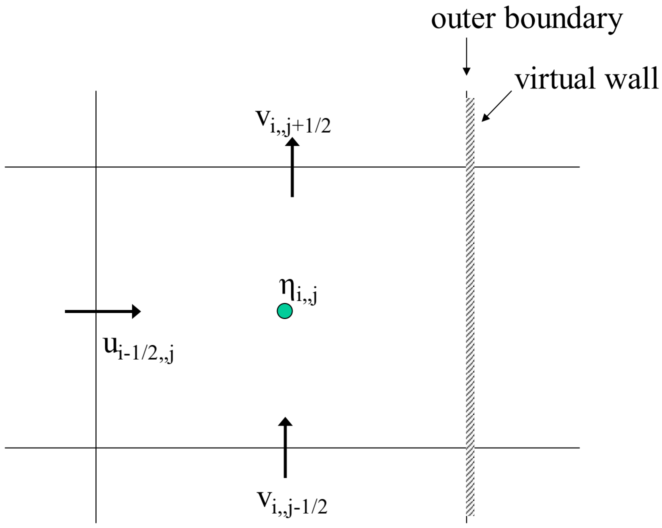


Figure 0-1-10 Method Developed by Imamura Et Al.

1.3.3.3. Transparent Boundary Model Specifications

Specify the transparent boundary model in the %BOUNDARY block of the analysis condition file by using OPEN-SOMMER.

Note that it is necessary to also specify the chart datum for the boundary when using the method developed by Imamura et al. Specify HO when the chart datum is fixed or TYPE-H=TABLE or TIDE when the chart datum is a variable. Then, specify either the time table for water levels or astronomical tidal conditions.

1.3.4. Dispersive Wave Model

The non-linear dispersive wave model can deal with the deformation behaviors of waves with good accuracy by adding dispersion terms based on the non-linear dispersive wave theory to STOC-ML.

Note that STOC-IC, which does not assume pressure is hydrostatic, offers the above capability automatically, thus eliminating the need to implement the dispersive wave model in STOC-IC. But, if you use the dispersive wave model with STOC-ML, you can perform calculations extraordinarily faster than with STOC-IC.

1.3.4.1. Fundamental Equations for the Dispersive Wave Model

When the dispersive wave theory is applied, the momentum conservation equations that are integrated vertically are as follows.

$$\frac{\partial M}{\partial t} \square \alpha H^2\left(\frac{\partial^3 M}{\partial t \partial x^2} \square \frac{\partial^3 N}{\partial t \partial x \partial y}\right) \square \beta {c_0}^2 H^2\left(\frac{\partial^3 \eta}{\partial x^3} \square \frac{\partial^3 \eta}{\partial x \partial y^2}\right) \square f_x \qquad (1.3.15)$$

$$\frac{\partial N}{\partial t} \square \alpha H^2\left(\frac{\partial^3 N}{\partial t \partial y^2} \square \frac{\partial^3 M}{\partial t \partial x \partial y}\right) \square \beta {c_0}^2 H^2\left(\frac{\partial^3 \eta}{\partial y^3} \square \frac{\partial^3 \eta}{\partial y \partial x^2}\right) \square f_y \qquad (1.3.16)$$

where

M refers to the flow rate per unit width in the x direction [m^2/s] ($\square \int_{-H}^{\eta} udz$),

N refers to flow rate per unit width in the y direction [m^2/s] ($\square \int_{-H}^{\eta} vdz$),

${c_0}^2$ refers to wave speed [m/s] (${c_0}^2 \square \sqrt{gh}$), and

f_x, f_y refer to groups of other terms.

Note: α and β are arbitrary constants which satisfy the following relational expression:

$$\alpha - \beta \square \frac{1}{3} \qquad (1.3.17)$$

Madsen-Sørensen (1991) shows that the dispersion relation is relatively well satisfied at high frequencies when $\beta \square 1/15$. Thus, $\beta \square 1/15$ is the default value for STOC.

1.3.4.2. Implementation of a Potential Function

A potential function will be implemented according to Shigihara (2007). Calculations that use a potential function significantly increase the stability of matrix inversion and reduce the operation volume. The following potential function is defined for the x-direction and y-direction dispersion terms:

$$\psi \square \alpha H^2\left(\frac{\partial^2 M}{\partial t\partial x} \square \frac{\partial^2 N}{\partial t\partial y}\right) \square \beta {c_0}^2 H^2\left(\frac{\partial^2\eta}{\partial x^2} \square \frac{\partial^2\eta}{\partial y^2}\right) \text{ .... } (1.3.18)$$

At this time, the motion equation can be changed to the Poisson equation below:

$$\psi - \alpha h^2\left(\frac{\partial^2\psi}{\partial x^2} \square \frac{\partial^2\psi}{\partial y^2}\right) \square \alpha h^2\left(\frac{\partial f_x}{\partial x} \square \frac{\partial f_y}{\partial y}\right) \square \beta g h^3\left(\frac{\partial^2\eta}{\partial x^2} \square \frac{\partial^2\eta}{\partial y^2}\right) \text{ .... } (1.3.19)$$

1.3.4.3. Calculation Procedure

The following shows an algorithm related to the non-linear dispersion term. After solving the motion equation (Poisson equation), allocate the flow rate for each layer. The calculation steps are as follows.

(Step 1)

$$\tilde{u} \square u^n \square \chi_x \delta t \text{ .... } (1.3.20)$$

$$\tilde{v} \square v^n \square \chi_y \delta t \text{ .... } (1.3.21)$$

Calculate $\tilde{u}$ and $\tilde{v}$ explicitly.

(Step 2)

$$\tilde{M} \square \sum_z \tilde{u}\delta z \text{ .... } (1.3.22)$$

$$\tilde{N} \square \sum_z \tilde{v}\delta z \text{ .... } (1.3.23)$$

Calculate flow rates $\tilde{M}$ and $\tilde{N}$.

(Step 3)

$$f_x \square \tilde{M} - M^n \text{ .... } (1.3.24)$$

$$f_y \square \tilde{N} - N^n \text{ .... } (1.3.25)$$

Calculate f_x and f_y.

(Step 4)

Set the land-side boundary conditions by using $\widetilde{M}$ and $\widetilde{N}$. The waters near land are shallow enough for us to ignore the dispersion effects due to the land-side boundary condition for the Poisson equation. As such, use the value of ψ for the land-side boundary condition. This value is determined from M and N, which are explicitly solved.

When the land-side boundary condition is applied, $-H < z_{disp,\lim}$ is established for the current calculation cell and $-H \geq z_{disp,\lim}$ for one of the adjacent cells ($z_{disp,\lim}$ is an input parameter).

By using $\frac{\partial M}{\partial t} = f_x$ and $\frac{\partial N}{\partial t} = f_y$ for explicit solutions, the land-side boundary value of ψ is calculated as follows:

$$\psi = \frac{H^2}{3}\left(\frac{\partial^2 M}{\partial t \partial x} + \frac{\partial^2 N}{\partial t \partial y}\right) = \frac{H^2}{3}\left(\frac{\partial f_x}{\partial x} + \frac{\partial f_y}{\partial y}\right) \quad \text{......} (1.3.26)$$

Note that if the water depth of a boundary cell that makes contact with land suddenly increases, the fixed value for ψ becomes too large. Thus, do not calculate such a calculation cell (the judgment criteria include cells to which the land-side boundary conditions are applicable, and $-H \leq z_{disp,BC,\lim}$. $z_{disp,BC,\lim}$ is an input parameter).

(Step 5)

$$\psi - \alpha H^2\left(\frac{\partial^2 \psi}{\partial x^2} + \frac{\partial^2 \psi}{\partial y^2}\right) = \alpha H^2\left(\frac{\partial f_x}{\partial x} + \frac{\partial f_y}{\partial y}\right) + \beta g H^3\left(\frac{\partial^2 \eta}{\partial x^2} + \frac{\partial^2 \eta}{\partial y^2}\right) \quad \text{......} (1.3.27)$$

Discretize the equation above and solve the resultant equation by using a matrix solver.

(Step 6)

Update u and v by using the potential function ψ you obtained.

$$u^{n+1}(x_i, y_i, z_i) = \widetilde{u}(x_i, y_i, z_i) - 3\frac{1}{z_{i+1} - z_i}\int_{z_i}^{z_{i+1}}\left(\frac{z}{H}\left(1 + \frac{z}{2H}\right)\right)dz \frac{\partial \psi(x_i, y_i)}{\partial x}\delta t \quad \text{....} (1.3.28)$$

$$v^{n+1}(x_i, y_i, z_i) = \widetilde{v}(x_i, y_i, z_i) - 3\frac{1}{z_{i+1} - z_i}\int_{z_i}^{z_{i+1}}\left(\frac{z}{H}\left(1 + \frac{z}{2H}\right)\right)dz \frac{\partial \psi(x_i, y_i)}{\partial y}\delta t \quad \text{....} (1.3.29)$$

1.3.4.4. Dispersive Wave Model Specifications

Specify the dispersive wave model in the %MODEL block of the analysis condition file by using DISPERSIVE-WAVE through DISP-BC-LIMIT.

1.3.5. Breaking Wave Model

The breaking wave model considers energy loss due to breaking waves. The two types of breaking wave models below are incorporated.

1.3.5.1. Kennedy's Model

The Kennedy model[3] includes an energy dissipation term, determined by breaking waves, on the right side of the motion equation. This term is in the same format as the viscosity term. The kinematic viscosity coefficient ν_{BW} for the breaking wave model is expressed as follows.

$$\nu_{BW} = B\delta^2 (\zeta + h) \frac{\partial \zeta}{\partial t} \quad \cdots\cdots (1.3.30)$$

ζ : Water level

h : Water depth (positive for the vertical downward direction)

$$B = \begin{cases} 1 & \zeta_t \geq 2\zeta_t^* \\ \zeta_t / \zeta_t^* - 1 & \zeta_t^* < \zeta_t < 2\zeta_t^* \\ 0 & \zeta_t \leq \zeta_t^* \end{cases} \quad \cdots\cdots (1.3.31)$$

$$\zeta_t^* = \begin{cases} \zeta_t^F & t - t_0 \geq T^* \\ \dfrac{t - t_0}{T^*} \zeta_t^F + \left(1 - \dfrac{t - t_0}{T^*}\right) \zeta_t^I & 0 \leq t - t_0 < T^* \end{cases} \quad \cdots\cdots (1.3.32)$$

$$\zeta_t^F = C_3 \sqrt{g(\zeta + h)} \quad \cdots\cdots (1.3.33)$$

$$\zeta_t^I = C_2 \sqrt{g(\zeta + h)} \quad \cdots\cdots (1.3.34)$$

$$T^* = C_1 \sqrt{\frac{\zeta + h}{g}} \text{ : Time scale for breaking waves} \quad \cdots\cdots (1.3.35)$$

Specify δ, C_1, C_2, C_3 as input parameters.

First, t_0 is initialized at -1. Then, it is set to the time when either of the following expressions for generation of breaking waves is satisfied:

[3] P.J.Lynett, "Nearshore Wave Modeling with High-Order Boussinesq-Type Equations", J. Waterway, Port, Coastal, and Ocean Engineering. (2006)

$$\sqrt{\left(\frac{\partial \zeta}{\partial x}\right)^2 + \left(\frac{\partial \zeta}{\partial y}\right)^2} \geq 0.60 \quad \text{(1.3.36)}$$

or

$$\frac{\partial \zeta}{\partial t} \geq \zeta_t^I \quad \text{(1.3.37)}$$

When $\zeta_t \leq \zeta_t^*$ and $\nu \square 0$, it is set to -1 again.

Note that because this model is also a 2-dimensional model, it does not consider the difference in ν_{BW} due to the height position when multi-layer calculations are performed with STOC.

1.3.5.2. Iwase Et Al.'s Model

The Iwase el al.'s model[4] also includes an energy dissipation term, determined by breaking waves, on the right side of the motion equation. This term is in the same format as the viscosity term.

$$\nu_{BW} = \beta\sqrt{g(\zeta + h)}\zeta' \quad \text{(1.3.38)}$$

β is an input parameter. ζ' is determined through calculations. The method of calculating ζ' will be described later.

(a) Identifying the peak of a wave

Identify the peak of a wave as follows. The flow velocity/wave velocity rate is u_s/C.

(1) Identify the point when u_s/C > 0.59 as the peak of the wave.

(2) Treat this point as the peak of the wave until u_s/C becomes equal to or less than 0.55.

Note that the following equations are established:

$$u_s = \sqrt{\left(\bar{u} - \frac{(\zeta + h)^2}{3}\frac{\partial^2 \bar{u}}{\partial x^2}\right)^2 + \left(\bar{v} - \frac{(\zeta + h)^2}{3}\frac{\partial^2 \bar{v}}{\partial y^2}\right)^2} \quad \text{(1.3.39)}$$

[4]Hiroyuki Iwase et al., "A Numerical Simulation of Run-Up of Nihonkai-Chubu Earthquake Tsunami in Consideration of Dispersive Effects", Proceedings of Coastal Engineering, JSCE, Vol.49, pp.266-270 (2002)

$$C = \sqrt{g(\zeta + h)}\left(1 + \frac{1}{2}\frac{\zeta}{h}\right) \text{ .... } (1.3.40)$$

where $\bar{u}, \bar{v}$ are average cross-sectional flow velocities. $\frac{\zeta}{h}$ in the equation above cannot be calculated for runup areas. Thus, the lower limit on h is 0.0001 m, and the upper limit on $\frac{\zeta}{h}$ is 0.83.

(b) Identifying the bottom of a wave and calculating the kinematic viscosity coefficient

To determine ζ' in the equation (1.3.38), determine the position of the bottom of the wave relative to the peak of the wave as follows.

(1) 4-directional search for the bottom and water level setup

Starting at the peak, search for the point where the level of water flowing in four cardinal directions toward the north, east, south, and west stops changing or starts rising. The detected point is the bottom. There are four bottoms for XY 2-dimensional calculations. The water levels for the four bottoms on the north, east, south, and west sides are $\zeta_N, \zeta_E, \zeta_S, \zeta_W$, respectively.

(2) Determining representative water level ζ_{xy} for the bottom of the wave on the wave travel direction side

Determine the water level for the bottom of the wave on the wave travel direction side for each of the x and y directions as follows.

$$\zeta_x = \begin{cases} \zeta_W & (u \le 0) \\ \zeta_E & (u > 0) \end{cases} \text{ .... } (1.3.41)$$

$$\zeta_y = \begin{cases} \zeta_S & (v \le 0) \\ \zeta_N & (v > 0) \end{cases} \text{ .... } (1.3.42)$$

Determine the lower of these water levels (i.e., the water level which is farthest from that of the peak) as follows:

$$\zeta_{xy} = \min(\zeta_x, \zeta_y) \text{ .... } (1.3.43)$$

(3) Calculating distribution function ζ' for water level differences

Calculate ζ' in the equation (1.3.38) below when the water level of the peak of a wave is ζ_p

($k = N$ for the north point, $k = E$ for the east point, $k = S$ for the south point, and $k = W$ for the west point).

$$\zeta' = \left(\zeta - \zeta_k\right)\frac{\zeta_p - \zeta_{xy}}{\zeta_p - \zeta_k} \quad \text{(1.3.44)}$$

By examining the equation (1.3.44), we can say that because $\zeta = \zeta_k$ at the bottom point for the each direction, the equation (1.3.44) results in 0. For the peak point ($\zeta = \zeta_p$), this equation is identical to $\zeta' = \zeta_p - \zeta_{xy}$ irrespective of direction k. For one of the four cardinal directions, ζ_k and ζ_{xy} match, and the fractional part of this equation results in 1 for that direction. As a result, this equation can be simplified as $\zeta' = \zeta - \zeta_{xy}$.

(4) Calculating the kinematic viscosity coefficient

Using equation (1.3.38), calculate the kinematic viscosity coefficient for an extent from the peak of the wave to the bottoms for the four cardinal directions. $\zeta = h$ as the total water depth in the set position, and ζ' as the value calculated from equation (1.3.44).

Note that the kinematic viscosity might be calculated multiple times in the same position due to differences in the starting point (peak). If so, use the largest of the calculated values.

The Iwase et al. model is also a 2-dimensional model. Thus, it does not consider the difference in ν_{BW} due to the height position when multi-layer calculations are performed with STOC.

1.3.5.3. Upper Limit on the Kinematic Viscosity Coefficient

The kinematic viscosity coefficient ν_{BW} can become quite large, especially in the Kennedy's breaking wave model. In such cases, the stabilization conditions for calculation are not satisfied with the normal time steps, and so the solutions are dispersed. Therefore, the stabilization condition below is applied for calculation of viscosity.

$$\Delta t \le \frac{1}{2\nu\left(\frac{1}{\Delta x^2} + \frac{1}{\Delta y^2} + \frac{1}{\Delta z^2}\right)} \quad \text{(1.3.45)}$$

The upper limit on the kinematic viscosity coefficient is defined as:

$$\nu_{\text{limit}} = \frac{C}{2\Delta t\left(\dfrac{1}{\Delta x^2} + \dfrac{1}{\Delta y^2} + \dfrac{1}{\Delta z^2}\right)} \quad \text{(1.3.46)}$$

As a result, ν_{BW} is restricted (C has a safety factor of about 0.5 that is specified as an input parameter).

1.3.5.4. Breaking Wave Model Specifications

Specify the breaking wave model in the %MODEL block of the analysis condition file by using BREAK-WAVE through DTSAFE-BREAK.

1.3.6. Overflow Model

The overflow model calculates the flow rate of an overflow over a breakwater or seawall from an experimentally-obtained correlation model without solving the fundamental equations.

1.3.6.1. Honma's Formula

When a wave passes over breakwater, its flow rate is expressed as follows by using Honma's formula (see Figure 0-1-11).

$$q = 0.35 h_1 \sqrt{2 g h_1} \qquad \left(h_2 \leq \frac{2}{3} h_1 \right) \quad \text{......} \quad (1.3.47)$$

$$q = 0.91 h_2 \sqrt{2 g (h_1 - h_2)} \qquad \left(h_2 > \frac{2}{3} h_1 \right) \quad \text{......} \quad (1.3.48)$$

Figure 0-1-11 shows the case where $h_1 > 0,\ h_2 > 0$, but when $h_1 \leq 0,\ h_2 \leq 0,\ q = 0$. When $h_1 > 0,\ h_2 \leq 0$, equation (1.3.47) can be applied as is.

Note that Honma's formula is an equation originally applied to a steady state, and so if it is applied to an unsteady state without changing it, the water level might fluctuate before or after the breakwater, because the flow rate is instantly determined without considering inertia. To prevent this problem, the two options below are provided.

(1) Using the flow rate, based on Honma's formula, as is

(2) Calculating the flow rate by using the normal method, and then using the flow rate based on Honma's formula as the upper limit on the calculated flow rate

Note that Honma's formula can only be applied to cases where the breakwater thickness is expressed with a boundary having zero thickness.

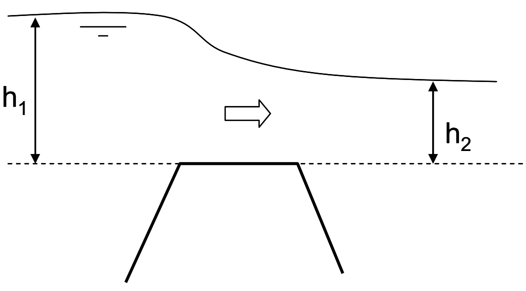


Figure 0-1-11 Symbols for Honma's Formula

1.3.6.2. Aida's Formula and the Drop Formula

When a wave spills over a coastline seawall, its flow rate is expressed by using Aida's formula as follows (see Figure 0-1-12).

$$q = 0.6 h_2 \sqrt{g (h_1 - h_2)} \quad \text{......} \quad (1.3.49)$$

When water flows down the seawall during the backwash phase, its flow rate is expressed with the drop formula below (see Figure 0-1-13). Note that h_2 is 0 when the water level is lower than the height of the seawall.

$$q = 0.544(h_1 - h_2)\sqrt{g(h_1 - h_2)} \quad \text{.......} (1.3.50)$$

A problem with Aida's formula is that if, for example, equation (1.3.49) is applied when h_2 = 0 at the arrival of the first wave, flow rate q becomes 0, thereby preventing overflow. Thus, Aida's formula should not be applied until h_2 exceeds the minimum value ($h_{\text{limit,aida}}$) specified as an input parameter.

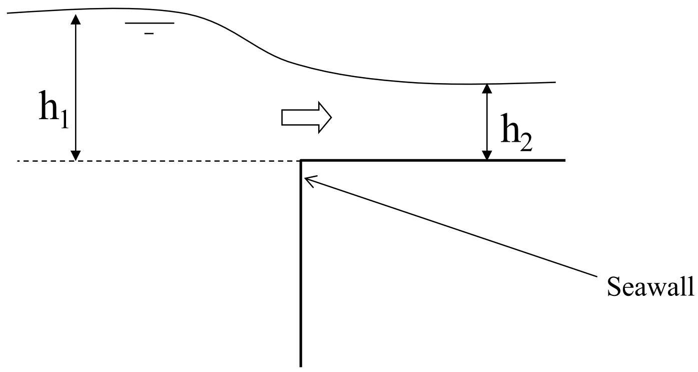


Figure 0-1-12 Symbols for Aida's Formula

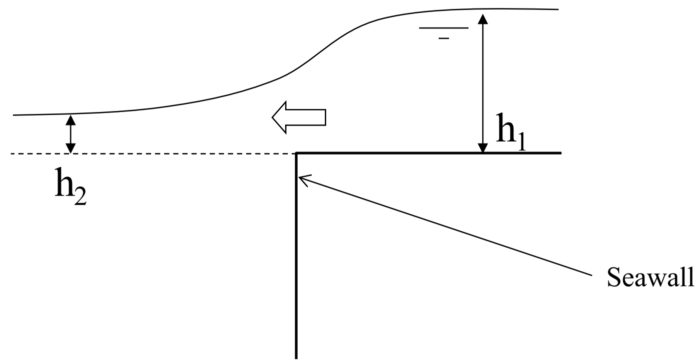


Figure 0-1-13 Symbols for the Drop Formula

1.3.6.3. Specifications for of the Overflow formulas

Specify the overflow formulas in the %MODEL block of the analysis condition file by using OVERFLOW-HONMA through AIDA-LIMIT. Prepare an applicable-position specification data file (*.ofl) for the overflow models and specify the applicable position when using either of these formulas. For information about the format of the ofl file, refer to the description of the applicable-position specification data file (Area.ofl) for the overflow models in “Instruction for Use”.

1.3.7. Permeable Structure Model

The permeable structure model covers energy loss due to wave dissipating blocks, etc. In the model, inertial and drag forces are considered resistance forces applied to the permeable structure. Two permeable structure models (CDM and DF) are adopted. The CDM model uses a turbulent flow resistance that is proportional to the squared flow velocity. The DF model uses a resistance based on the Dupuit-Forchheimer method that is expressed as the sum of laminar and turbulent flow resistances.

Shown below are the fundamental equations to which the Sakakiyama et al.'s porous model in wave field (1990)[5] is applied. Note that the porous value for expressing topography is 1 and the porosity and permeability for expressing the permeable structure are $\gamma_{v,2}$, $\gamma_{x,2}$, $\gamma_{y,2}$, $\gamma_{z,2}$.

1.3.7.1. Fundamental Equations

(1) Continuity equation

$$\frac{\partial \gamma_{x,2} u}{\partial x} + \frac{\partial \gamma_{y,2} v}{\partial y} + \frac{\partial \gamma_{z,2} w}{\partial z} = 0 \qquad (1.3.51)$$

(2) Momentum conservation equation: x direction

$$\begin{aligned}
&\lambda_v \frac{\partial u}{\partial t} + \frac{\partial \lambda_x u^2}{\partial x} + \frac{\partial \lambda_y uv}{\partial y} + \frac{\partial \lambda_z uw}{\partial z} \\
&= -\frac{\gamma_{v,2}}{\rho}\frac{\partial p}{\partial x} + \frac{\partial}{\partial x}\left(2\gamma_{x,2}\nu_H \frac{\partial u}{\partial x}\right) + \frac{\partial}{\partial y}\left\{\gamma_{y,2}\nu_H\left(\frac{\partial v}{\partial x} + \frac{\partial u}{\partial y}\right)\right\} \qquad (1.3.52) \\
&\quad + \frac{\partial}{\partial z}\left\{\gamma_{z,2}\nu_V\left(\frac{\partial w}{\partial x} + \frac{\partial u}{\partial z}\right)\right\} - R_x
\end{aligned}$$

The equation for the y direction is omitted because it is conversely identical to that for the x direction.

(3) Momentum conservation equation: z direction (STOC-IC only)

[5]Tsutomu Sakakiyama, Nobuyuki Abe, Ryouichi Kashima (1990): "Non-linear Wave Analysis, Based on a Porous Model, of Areas Near Permeable Structures", Proceedings of Coastal Engineering, JSCE, Vol. 37, pp.554-558.

$$
\begin{aligned}
&\lambda_v \frac{\partial w}{\partial t} + \frac{\partial \lambda_x u w}{\partial x} + \frac{\partial \lambda_y v w}{\partial y} + \frac{\partial \lambda_z w^2}{\partial z} \\
&= -\frac{\gamma_{v,2}}{\rho}\frac{\partial p}{\partial z} - \gamma_{v,2} g_z + \frac{\partial}{\partial x}\left\{\gamma_{x,2}\nu_H\left(\frac{\partial w}{\partial x} + \frac{\partial u}{\partial z}\right)\right\} \\
&\quad + \frac{\partial}{\partial y}\left\{\gamma_{y,2}\nu_H\left(\frac{\partial w}{\partial y} + \frac{\partial v}{\partial z}\right)\right\} + \frac{\partial}{\partial z}\left(2\gamma_{z,2}\nu_V\frac{\partial w}{\partial z}\right) - R_z
\end{aligned}
\qquad (1.3.53)
$$

λ and R are expressed as follows.

$$
\begin{aligned}
\lambda_v &\equiv \gamma_{v,2} + \left(1-\gamma_{v,2}\right)C_M \\
\lambda_x &\equiv \gamma_{x,2} + \left(1-\gamma_{x,2}\right)C_M \\
\lambda_y &\equiv \gamma_{y,2} + \left(1-\gamma_{y,2}\right)C_M \\
\lambda_z &\equiv \gamma_{z,2} + \left(1-\gamma_{z,2}\right)C_M
\end{aligned}
\qquad (1.3.54)
$$

(a) CDM model

$$
R_i = \frac{1}{2}\frac{C_D}{\Delta x}\left(1-\gamma_{i,2}\right)v_i\left|\mathbf{v}\right| \qquad \left(i = x, y, z\right) \qquad (1.3.55)
$$

(b) DF (Dupuit-Forchheimer) model

$$
R_i = \left(a + b\left|\mathbf{V}\right|\right)V_i \qquad \left(i = x, y, z\right) \qquad (1.3.56)
$$

$$
a = \alpha_0 \frac{\left(1-\gamma_{v,2}\right)^3}{\gamma_{v,2}^{\;2}}\frac{\nu}{D^2} \qquad (1.3.57)
$$

$$
b = \beta_0 \frac{1-\gamma_{v,2}}{\gamma_{v,2}^{\;3}}\frac{1}{D} \qquad (1.3.58)
$$

where

C_M: Inertia force coefficient (input parameter)

C_D: Drag force coefficient (input parameter)

α_0: Coefficient (input parameter)

β_0: Coefficient (input parameter)

D: Representative diameter of the permeable structure (input parameter)

ν : Kinematic viscosity coefficient of the water (constant, 1.00×10^{-6} m^2/s)

Note that the flow velocity used in the DF model is called the Darcy flow velocity and is different from the actual flow velocity. The Darcy flow velocity is expressed as follows.

$$V_i = \gamma_{x_i} v_i \qquad (1.3.59)$$

1.3.7.2. Porous Value for Expressing Topography

STOC normally uses porous values to express topography. The porous values for permeable structures should be treated separately from those used by STOC. Thus, the porous variables for expressing topography are $\gamma_{v,1}, \gamma_{x,1}, \gamma_{y,1}, \gamma_{z,1}$, and those for permeable structures are $\gamma_{v,2}, \gamma_{x,2}, \gamma_{y,2}, \gamma_{z,2}$. When all these porous values are considered, the fundamental equations are as follows.

• Continuity equation

$$\frac{\partial \gamma_{x,1}\gamma_{x,2}u}{\partial x} + \frac{\partial \gamma_{y,1}\gamma_{y,2}v}{\partial y} + \frac{\partial \gamma_{z,1}\gamma_{z,2}w}{\partial z} = 0 \qquad (1.3.60)$$

• Momentum conservation equation: x direction

$$\begin{aligned}
&\gamma_{v,1}\lambda_v \frac{\partial u}{\partial t} + \frac{\partial \lambda_x \gamma_{x,1} u^2}{\partial x} + \frac{\partial \lambda_y \gamma_{y,1} uv}{\partial y} + \frac{\partial \lambda_z \gamma_{z,1} uw}{\partial z} \\
&= -\frac{\gamma_{v,1}\gamma_{v,2}}{\rho}\frac{\partial p}{\partial x} + \frac{\partial}{\partial x}\left(2\gamma_{x,1}\gamma_{x,2}\nu_H \frac{\partial u}{\partial x}\right) + \frac{\partial}{\partial y}\left\{\gamma_{y,1}\gamma_{y,2}\nu_H\left(\frac{\partial v}{\partial x} + \frac{\partial u}{\partial y}\right)\right\} \qquad (1.3.61) \\
&+ \frac{\partial}{\partial z}\left\{\gamma_{z,1}\gamma_{z,2}\nu_V\left(\frac{\partial w}{\partial x} + \frac{\partial u}{\partial z}\right)\right\} - \gamma_{v,1}R_x
\end{aligned}$$

The composite porous variables are defined as follows.

$$\begin{aligned}
\Gamma_v &\equiv \gamma_{v,1}\gamma_{v,2} \\
\Gamma_x &\equiv \gamma_{x,1}\gamma_{x,2} \\
\Gamma_y &\equiv \gamma_{y,1}\gamma_{y,2} \\
\Gamma_z &\equiv \gamma_{z,1}\gamma_{z,2}
\end{aligned} \qquad (1.3.62)$$

$$\begin{aligned}
\Lambda_v &\equiv \gamma_{v,1}\lambda_v \\
\Lambda_x &\equiv \gamma_{x,1}\lambda_x \\
\Lambda_y &\equiv \gamma_{y,1}\lambda_y \\
\Lambda_z &\equiv \gamma_{z,1}\lambda_z
\end{aligned} \qquad (1.3.63)$$

The fundamental equations are changed to the following:

• Continuity equation

$$\frac{\partial \Gamma_x u}{\partial x} + \frac{\partial \Gamma_y v}{\partial y} + \frac{\partial \Gamma_z w}{\partial z} = 0 \qquad (1.3.64)$$

• Momentum conservation equation: x direction

$$\begin{aligned}
&\Lambda_v \frac{\partial u}{\partial t} + \frac{\partial \Lambda_x u^2}{\partial x} + \frac{\partial \Lambda_y uv}{\partial y} + \frac{\partial \Lambda_z uw}{\partial z} \\
&= -\frac{\Gamma_v}{\rho}\frac{\partial p}{\partial x} + \frac{\partial}{\partial x}\left(2\Gamma_x \nu_H \frac{\partial u}{\partial x}\right) + \frac{\partial}{\partial y}\left\{\Gamma_y \nu_H \left(\frac{\partial v}{\partial x} + \frac{\partial u}{\partial y}\right)\right\} \qquad (1.3.65)\\
&+ \frac{\partial}{\partial z}\left\{\Gamma_z \nu_V \left(\frac{\partial w}{\partial x} + \frac{\partial u}{\partial z}\right)\right\} - \gamma_{v,1} R_x
\end{aligned}$$

1.3.7.3. Changes in the Pressure Correction equations

STOC-IC calculates the coefficients for the pressure correction equations. These coefficients are changed as shown below depending on whether or not permeable structures are considered.

• When the permeable structures are not considered:

$$\begin{aligned}
&\gamma_{x,1_{i+1/2}} \frac{\Delta y_j \Delta z_k}{\Delta x_{i+1/2}}\left(\delta p - \delta p_{i+1}\right) + \gamma_{x,,1_{i-1/2}} \frac{\Delta y_j \Delta z_k}{\Delta x_{i-1/2}}\left(\delta p - \delta p_{i-1}\right) \\
&+ \gamma_{y,1_{j+1/2}} \frac{\Delta x_i \Delta z_k}{\Delta y_{j+1/2}}\left(\delta p - \delta p_{j+1}\right) + \gamma_{y,1_{j-1/2}} \frac{\Delta x_i \Delta z_k}{\Delta y_{j-1/2}}\left(\delta p - \delta p_{j-1}\right) \qquad (1.3.66)\\
&+ \gamma_{z,1_{k+1/2}} \frac{\Delta x_i \Delta y_j}{\Delta z_{k+1/2}}\left(\delta p - \delta p_{k+1}\right) + \gamma_{z,1_{k-1/2}} \frac{\Delta x_i \Delta y_j}{\Delta z_{k-1/2}}\left(\delta p - \delta p_{k-1}\right) \\
&= -\frac{\rho\, \Delta x_i \Delta y_j \Delta z_k}{\Delta t}\,\mathrm{div}\left(\gamma_{\mathbf{x},1}\tilde{\mathbf{v}}\right)
\end{aligned}$$

• When the permeable structures are considered:

$$
\begin{aligned}
&\frac{\Gamma_{v_{i+1/2}}\gamma_{x_{i+1/2}}}{\Lambda_{v_{i+1/2}}}\frac{\Delta y_j \Delta z_k}{\Delta x_{i+1/2}}\left(\delta p-\delta p_{i+1}\right)+\frac{\Gamma_{v_{i-1/2}}\gamma_{x_{i-1/2}}}{\Lambda_{v_{i-1/2}}}\frac{\Delta y_j \Delta z_k}{\Delta x_{i-1/2}}\left(\delta p-\delta p_{i-1}\right)\\
&+\frac{\Gamma_{v_{j+1/2}}\gamma_{y_{j+1/2}}}{\Lambda_{v_{j+1/2}}}\frac{\Delta x_i \Delta z_k}{\Delta y_{j+1/2}}\left(\delta p-\delta p_{j+1}\right)+\frac{\Gamma_{v_{j-1/2}}\gamma_{y_{j-1/2}}}{\Lambda_{v_{j-1/2}}}\frac{\Delta x_i \Delta z_k}{\Delta y_{j-1/2}}\left(\delta p-\delta p_{j-1}\right)\\
&+\frac{\Gamma_{v_{k+1/2}}\gamma_{z_{k+1/2}}}{\Lambda_{v_{k+1/2}}}\frac{\Delta x_i \Delta y_j}{\Delta z_{k+1/2}}\left(\delta p-\delta p_{k+1}\right)+\frac{\Gamma_{v_{k-1/2}}\gamma_{z_{k-1/2}}}{\Lambda_{v_{k-1/2}}}\frac{\Delta x_i \Delta y_j}{\Delta z_{k-1/2}}\left(\delta p-\delta p_{k-1}\right)\\
&=-\frac{\rho\,\Delta x_i \Delta y_j \Delta z_k}{\Delta t}\,\mathrm{div}\left(\boldsymbol{\Gamma}_{\mathbf{x}}\tilde{\mathbf{v}}\right)
\end{aligned}
\qquad \text{..............} (1.3.67)
$$

1.3.7.4. Permeable Structures Specifications

Specify the permeable structures in the %OBSTACLE block of the analysis condition file by using D-POROUS. It is possible to specify them by using an external file (D-FILE=YES). Use D-POROUS-MODEL in the %MODEL block of the analysis condition file to switch from the CDM model to the DF model, or vice versa. For information about the format of an external file, refer to the description of the permeable structure file (Area.dpr) in “Instruction for Use”.

1.3.7.5. Notes on the Drag Coefficient for the CDM Model

Drag coefficient C_D for the CDM model depends on the mesh size used for calculation.

For example, a typical drag coefficient of 0.5 is used for spheres in a turbulence range. For spheres, the relationship between typical drag coefficient $C_D{}'$ and drag coefficient C_D for this model is as follows.

$$C_D = \frac{3}{2}\frac{\Delta x}{d} C_D{}' \qquad (1.3.68)$$

d : Diameter of the sphere

1.3.8. Overfall Model

In the overfall model, changes in momentum of water that spills over and falls down breakwater are considered.

1.3.8.1. Momentum of Overfall

Figure 0-1-14 shows an overfall model for overflow.

When the flow rate per unit width is M_{of} and the flow velocity is u_{of} for the overflow position, and the difference between the water levels before and after overflow is $\Delta\zeta$, the mass of water spilling over the breakwater is $\rho(M_{of}\Delta y\Delta t)$ at a certain time step. At this time, the momentum of the overfall in the x direction is $\rho(M_{of}\Delta y\Delta t)u_{of}$. Suppose that water freely falls at an initial speed of 0 and travels distance $\Delta\zeta$. In this case, the flow velocity of water when it reaches the ground is $w \,\square\, \sqrt{2g\Delta\zeta}$, and so the momentum of overfall in the z direction when the water reaches the ground is $\rho(M_{of}\Delta y\Delta t)\sqrt{2g\Delta\zeta}$. Note that when the momentum loss caused by a mixture during overfall is considered, the value multiplied by coefficient C_{fw} ($\geq 0, \leq 1$) is the final momentum of the overfall.

Therefore, the following momentum should be considered:

Momentum in the x direction: $C_{fw}\rho(M_{of}\Delta y\Delta t)u_{of}$ [kg・m/s]

Momentum in the z direction: $C_{fw}\rho(M_{of}\Delta y\Delta t)\sqrt{2g\Delta\zeta}$ [kg・m/s]

Note that, in the overfall model, one does not consider the water level difference expressed as $\Delta\zeta'$ in Figure 0-1-15. The reason is that water level difference $\Delta\zeta'$ is already taken into consideration for calculating the momentum conservation equation for the u_{of} position.

The overfall model is applied not only to the set positions of plate boundaries shown in Figure 0-1-15 but also to the system shown in Figure 0-1-15. Generally speaking, it is applied when the lower water level is not used for calculating the pressure gradient term of a momentum conservation equation. At this time, $\Delta\zeta$ is the water level difference for the part that is ignored in calculation of the pressure gradient term.

If water falls from multiple cells as shown on the right side of Figure 0-1-15, the overfall quantity for each cell height is calculated, and an overfall model is applied to each calculation result. For the example shown on the right side of Figure 0-1-15, the overfall model is calculated with drop distance $\Delta\zeta$ when the quantity of overfall from the top cell is M_{of} and with drop distance $\Delta\zeta_2$ when the quantity of overfall from the second cell is M_{of2}. The total of the calculation results amounts to the momentum of the overfall.

1.3.8.2. Momentum in the X Direction

The result of dividing the momentum above by mass $\rho\Delta x\Delta y\Delta z$ and time step width Δt is added to the right side of the momentum conservation equation. Note that if meshes are vertically formed in multiple layers, momentum in the x direction is proportionally distributed to positions $u_k, u_{k\square 1}$, shown in Figure 0-1-15, according to the layer thickness.

Term added to the momentum conservation equation for the x direction: $C_{fw}\dfrac{hM_{of}u_{of}}{\Delta x\,D}$ [m/s^2]

Note that D is the total water depth in the position, where the momentum is added.

1.3.8.3. Momentum in the Z Direction

Process the momentum in the z direction differently for STOC-IC and STOC-ML. In STOC-IC, use the momentum in the z direction at the position of $w_{k\square 3/2}$ shown in Figure 0-1-15.

On the other hand, in STOC-ML, do not calculate the momentum conservation equation for the z direction, as you cannot add momentum to the right side of this equation. The same can be said about STOC-IC if there is a wall in the position of $w_{k\square 3/2}$ shown in Figure 0-1-15.

In this case, processing is performed as follows.

If the momentum conservation equation for the z direction is not solved, change the momentum in the z direction to momentum in the x and y directions, which you then distribute to computational points u, v. After horizontally distributing the momentums, specify $C_{fw,dist,norm}$ as an input parameter, which is the rate for distribution along the normal direction of the breakwater. Once you have performed the distribution along a tangential direction, specify $C_{fw,dist,\mathrm{tang}}$ as an input parameter, which is the rate for distribution in the tangential direction. Because there are two possible tangential directions, you can distribute momentum in each tangential direction by using rate $C_{fw,dist,\mathrm{tang}}$. If there are multiple layers, proportionally distribute momentum to positions $u_k, u_{k\square 1}$, shown in Figure 0-1-15, according to the layer thickness. In this case, the terms to add to the momentum conservation equations are as follows.

Term to add to the momentum conservation equation for the x direction (normal line):

$$C_{fw} C_{fw,dist,norm} \frac{h M_{of} \sqrt{2g\Delta\zeta}}{\Delta x D} \quad [\mathrm{m/s^2}]$$

Term to add to the momentum conservation equation for the y direction (tangent line):

$$C_{fw} C_{fw,dist,\mathrm{tang}} \frac{h M_{of} \sqrt{2g\Delta\zeta}}{\Delta x D} \quad [\mathrm{m/s^2}]$$

The description of overflow in the y direction is the same as that of overflow in the x direction, except that “y” is used instead of “x”.

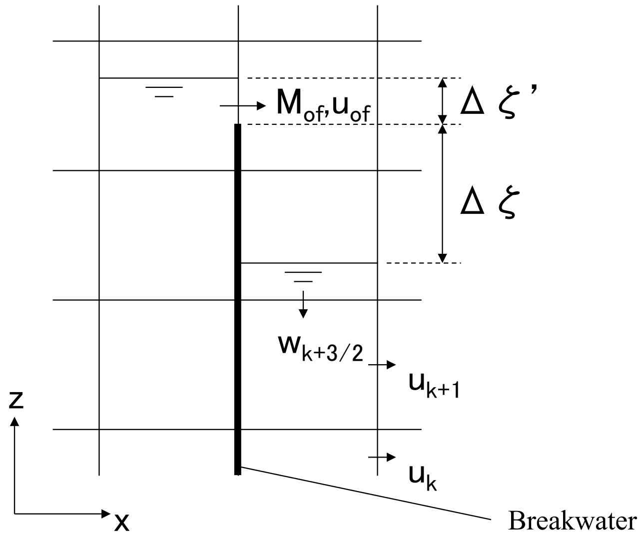


Figure 0-1-14 Changes in Momentum of Water Flowing down Breakwater

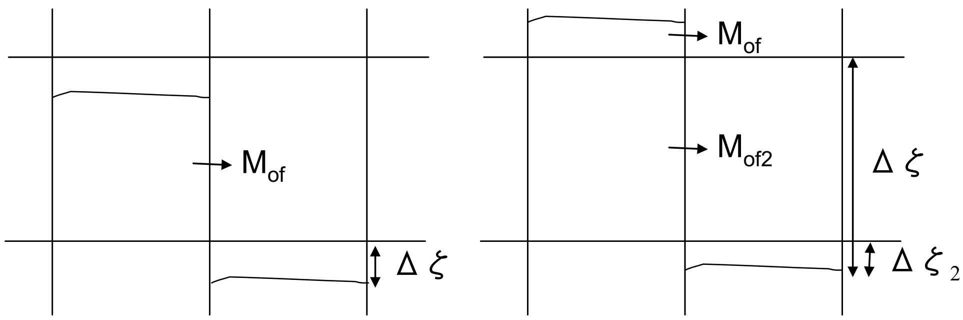


Figure 0-1-15 Positions to Which the Overfall Models Are Applied

1.3.8.4. Overfall Model Specifications

Specify the overfall model in the %MODEL block of the analysis condition file by using TYPE-WALL-WATER through DIST-TANG-FALL-WATER.

1.3.9. Model for a Change in Water Level as a result of an Earthquake

1.3.9.1. Specification of the Water Level Fluctuations

In STOC, the three methods below are available to specify the water level fluctuations that result from an earthquake. Use (b) or (c) when setting the time-dependent water level fluctuations. If you use (b) or (c) to change the water level, the water depth cannot be variable.

(a) Set the initial water level distribution, which exists when an earthquake occurs, in the topography/geometry data file (*.str).
(b) Set the time-dependent water level fluctuations by seismic action in the topography/geometry data file (*.sbt).
(c) Set the time-dependent fault parameters in the fault parameter file (fault.txt) and internally calculate the amount of water level fluctuation.

When using the fault parameter file, there is a need to determine the positional relationship with the STOC grid coordinates, because the fault parameters include longitudes and latitudes. Thus, STOC implements functions for coordinate conversion and geodetic system conversion.

1.3.9.2. Specification of the Model for a Change in Water Level as a results of an Earthquake

Specify the model for a change in water level as a result of an earthquake in the %BOUNDARY block of the analysis condition file by using SEA-BOTTOM.

When SEA-BOTTOM=CALC, the input method varies depending on the COORDINATE settings as follows.

(1) For the plane rectangular coordinate system

```
COORDINATE = JAPAN-PLANE-RECTANGULAR
RECTANGULAR-ZONE = 10          # Specify a number from 1 to 19.
```

(2) For the UTM coordinate system

```
COORDINATE = UTM
UTM-CENTER = 135.0             # Specify the longitude (in degrees) of the central longitude line on the UTM coordinate system.
```

(3) For the latitude/longitude coordinate system

```
COORDINATE = LONGITUDE-LATITUDE
```

1.4. Numerical Solutions

The following describes the discretization, time integration, and coupled calculation methods.

1.4.1. Discretization Methods

1.4.1.1. Overview

Space is discretized by observing the policies below.

- Use a staggered grid (Figure 0-1-16). u (i-1, j, k) is defined for the cell interface on the -x side of cell (i, j, k), and u (i, j, k) defined for the cell interface on the +x side.
- Use a rectangular grid. The mesh width is variable.
- Use a finite volume method for discretization.
- Apply a hybrid scheme that combines the first-order upwind scheme and second-order central-differencing scheme to discretize the advection term.
- Apply the second-order central-differencing scheme to discretize other terms.
- Apply the Leapfrog method to perform time integration.
- Deploy one layer of virtual cells outside of the computation area for processing boundary conditions (Figure 0-1-17). The 3-dimensional array size (MX, MY, MZ) is two meshes greater than the actual computation area size (MX-2, MY-2, MZ-2) for each direction, because this layer is necessary.

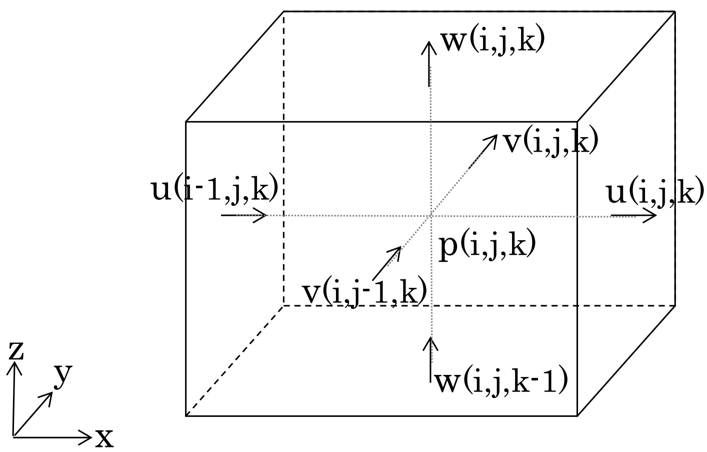


Figure 0-1-16Arrangement of Variables for Staggered Grid

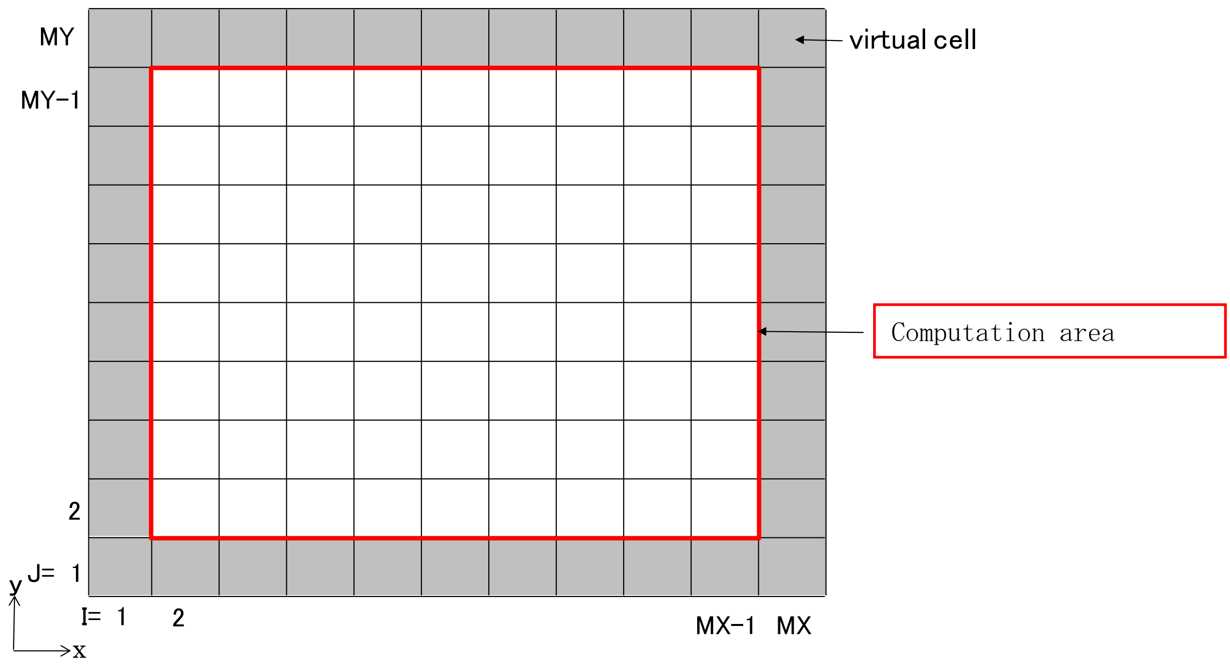


Figure 0-1-17 Computation Area and Virtual Cells

1.4.1.2. Discretization of a Continuity Equation

Equation (1.2.1) is a continuity equation for cells that do not contain a water surface. The generalized equation for cells that contain a water surface is as follows.

$$\frac{\partial V}{\partial t} + \frac{\partial (hu)}{\partial x} + \frac{\partial (hv)}{\partial y} + \frac{\partial \gamma_z w}{\partial z} = \frac{\partial}{\partial t}\Delta V \qquad (1.4.1)$$

where

V refers to the volume ratio of water within the cells

$$(hu) = \max(h_F - 1 + \gamma_x, 0)\, u \qquad (1.4.2)$$

$$(hv) = \max(h_F - 1 + \gamma_y, 0)\, v \qquad (1.4.3)$$

$$h_F \equiv \begin{cases} 1 & \eta \geq z_T \\ (\eta - z_B)/\Delta z & z_B \leq \eta < z_T \\ 0 & \eta < z_B \end{cases} \qquad (1.4.4)$$

(z_T: Height of the +z plane for the computational cell, and z_B: Height of the –z plane for the computational cell)

ΔV refers to the amount of discharge from the water surface cell when water that spills over the storm surge barrier and flows into the cells above the water surface is forcibly dropped to the cell that contains the water surface.

The equation below can be established when the water level does not vary from cell to cell.

$$\frac{\partial V}{\partial t} = \gamma_v \frac{\partial h_F}{\partial t} \qquad (1.4.5)$$

Equation (1.4.1) is vertically integrated to obtain the following:

$$\frac{\partial}{\partial t}\left(h_{F,KG}\Delta z_{KG} + \sum_{k=KG+1}^{MZM} \gamma_{v,k} h_{F,k} \Delta z_k \right) + \frac{\partial}{\partial x}\sum_{k=KG}^{MZM}(hu)_k \Delta z_k + \frac{\partial}{\partial y}\sum_{k=KG}^{MZM}(hv)_k \Delta z_k = 0 \qquad (1.4.6)$$

where KG is the index for the cell that contains the seafloor surface.

Calculate water level index KF, h_F in the relevant position, and water level η with this equation.

1.4.1.3. Discretization of a Horizontal Momentum Conservation Equation

Equations (1.2.2) and (1.2.3) are horizontal momentum conservation equations for cells that do not contain a water surface. Considering the control volume shown in Figure 0-1-18, the generalized equation for cells that contain a water surface is as follows.

$$\frac{\partial hu}{\partial t} + \frac{\partial hu^2}{\partial x} + \frac{\partial huv}{\partial y} + \frac{\partial uw}{\partial z} - f_0 hv = -\frac{h}{\rho}\frac{\partial p}{\partial x}$$
$$+ \frac{\partial}{\partial x}\left(2\nu_H h\frac{\partial u}{\partial x}\right) + \frac{\partial}{\partial y}\left\{\nu_H h\left(\frac{\partial v}{\partial x} + \frac{\partial u}{\partial y}\right)\right\} + \frac{\partial}{\partial z}\left\{\nu_V\left(\frac{\partial w}{\partial x} + \frac{\partial u}{\partial z}\right)\right\} + \frac{\tau_{wind}}{\rho}\frac{1}{\Delta z} \quad \text{..... (1.4.7)}$$

Note that because h does not appear in the advection, viscosity, and wind stress terms for the z direction; the numerical solution when $h \to 0$ should not become $u \to \infty$ at the time of explicit evaluation.

The wind stress term is as follows.

$$\frac{\tau_{wind,x}}{\rho} = \frac{\rho_{air}}{\rho}C_{fric}(W_x - u)\left|\vec{W} - \vec{u}\right| = \frac{\rho_{air}C_{fric}W_x\left|\vec{W} - \vec{u}\right|}{\rho} - \frac{\rho_{air}C_{fric}\left|\vec{W} - \vec{u}\right|}{\rho}u \quad \text{.... (1.4.8)}$$

To prevent the dispersion of surface velocity, the time differentiation term is modified to the following:

$$\frac{\partial hu}{\partial t} = h\frac{\partial u}{\partial t} + u\frac{\partial h}{\partial t} \quad \text{........ (1.4.9)}$$

As a result, the equation is as follows.

$$h\frac{\partial u}{\partial t} + \frac{\partial hu^2}{\partial x} + \frac{\partial huv}{\partial y} + \frac{\partial uw}{\partial z} - f_0 hv = -\frac{h}{\rho}\frac{\partial p}{\partial x} - u\frac{\partial h}{\partial t}$$
$$+ \frac{\partial}{\partial x}\left(2\nu_H h\frac{\partial u}{\partial x}\right) + \frac{\partial}{\partial y}\left\{\nu_H h\left(\frac{\partial v}{\partial x} + \frac{\partial u}{\partial y}\right)\right\} + \frac{\partial}{\partial z}\left\{\nu_V\left(\frac{\partial w}{\partial x} + \frac{\partial u}{\partial z}\right)\right\} + \frac{\tau_{wind}}{\rho}\frac{1}{\Delta z} \quad \text{...... (1.4.10)}$$

This equation is discretized into the following form:

$$
\begin{aligned}
& h_{i+1/2}^{n+1/2}\,\frac{u_{i+1/2}^{*}-u_{i+1/2}^{n}}{\Delta t} \\
& \square\,\frac{\langle u\rangle_{i+1}^{n}\,(hu)_{i+1}^{n}-\langle u\rangle_{i}^{n}\,(hu)_{i}^{n}}{\Delta x_{i+1/2}} \\
& \square\,\frac{\langle u\rangle_{j+1/2}^{n}\,(hv)_{j+1/2}^{n}-\langle u\rangle_{j-1/2}^{n}\,(hv)_{j-1/2}^{n}}{\Delta y_{j}} \\
& \square\,\frac{\langle u\rangle_{k+1/2}^{n}\,w_{k+1/2}^{n}-\langle u\rangle_{k-1/2}^{n}\,w_{k-1/2}^{n}}{\Delta z_{k}} - f_0 h_{i+1/2}^{n+1/2} v_{i+1/2}^{n} \\
& \square -\frac{h_{i+1/2}^{n+1/2}}{\rho}\,\frac{p_{i+1}^{n}-p_{i}^{n}}{\Delta x_{i+1/2}} \\
& \square\,\frac{1}{\Delta x_{i+1/2}}\left(2\nu_H h_{i+1}^{n+1/2}\,\frac{u_{i+3/2}^{n}-u_{i+1/2}^{n}}{\Delta x_{i+1}} - 2\nu_H h_{i}^{n+1/2}\,\frac{u_{i+1/2}^{n}-u_{i-1/2}^{n}}{\Delta x_{i}}\right) \\
& \square\,\frac{1}{\Delta y_{j}}\left\{\nu_H h_{j+1/2}^{n+1/2}\left(\frac{v_{i+1,j+1/2}^{n}-v_{i,j+1/2}^{n}}{\Delta x_{i+1/2}} + \frac{u_{i+1/2,j+1}^{n}-u_{i+1/2,j}^{n}}{\Delta y_{j+1/2}}\right)\right. \\
& - \qquad \left.\nu_H h_{j-1/2}^{n+1/2}\left(\frac{v_{i+1,j-1/2}^{n}-v_{i,j-1/2}^{n}}{\Delta x_{i+1/2}} + \frac{u_{i+1/2,j}^{n}-u_{i+1/2,j-1}^{n}}{\Delta y_{j-1/2}}\right)\right\} \\
& \square\,\frac{1}{\Delta z_{k}}\left\{\nu_V\left(\frac{w_{i+1,k+1/2}^{n}-w_{i,k+1/2}^{n}}{\Delta x_{i+1/2}} + \frac{u_{i+1/2,k+1}^{n}-u_{i+1/2,k}^{n}}{\Delta z_{k+1/2}}\right)\right. \\
& - \qquad \left.\nu_V\left(\frac{w_{i+1,k-1/2}^{n}-w_{i,k-1/2}^{n}}{\Delta x_{i+1/2}} + \frac{u_{i+1/2,k}^{n}-u_{i+1/2,k-1}^{n}}{\Delta z_{k-1/2}}\right)\right\} \qquad \text{.......................} \ (1.4.11) \\
& \square\,\frac{\rho_{air}}{\rho} C_{fric}\left|\vec{W}-\vec{v}\right|_{i+1/2}^{n}\left(W_{x,i+1/2}^{n}-u_{i+1/2}^{n\ \text{or}\ n+1}\right) \\
& \square\,u_{i+1/2}^{n}\left(\frac{(hu)_{i+1}^{n}-(hu)_{i}^{n}}{\Delta x_{i+1/2}} + \frac{(hv)_{j+1/2}^{n}-(hv)_{j-1/2}^{n}}{\Delta y_{j}} + \frac{w_{k+1/2}^{n}-w_{k-1/2}^{n}}{\Delta z_{k}}\right)
\end{aligned}
$$

where

$$
\langle u\rangle_{i}^{n}\,(hu)_{i}^{n} = \begin{cases}
\dfrac{u_{i-1/2}^{n}+u_{i+1/2}^{n}}{2}(hu)_{i}^{n} & (\text{2nd}-\text{order central differencing scheme}) \\
u_{i-1/2}^{n}(hu)_{i}^{n} & (\text{1st}-\text{order upwind scheme}\ \&\ (\text{hu}) \ge 0) \\
u_{i+1/2}^{n}(hu)_{i}^{n} & (\text{1st}-\text{order upwind scheme}\ \&\ (\text{hu}) < 0)
\end{cases}
$$

.... (1.4.12)

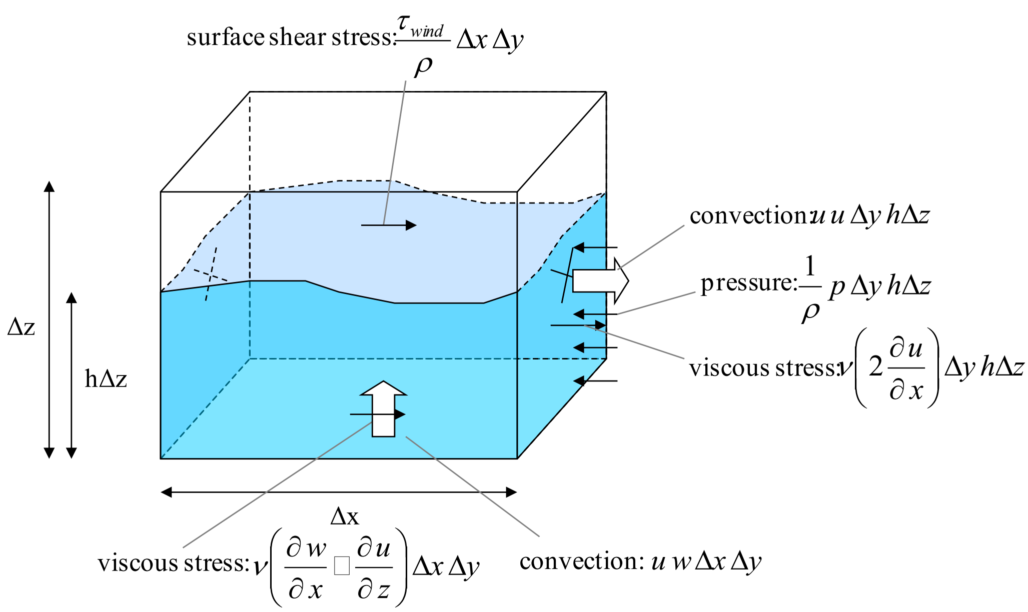


Figure 0-1-18 Control Volume: Horizontal Momentum Conservation Equation

1.4.1.4. Discretization of a Vertical Momentum Conservation Equation

Equation (1.2.4) is a vertical momentum conservation equation for cells that do not contain a water surface. Considering the control volume shown in Figure 0-1-19, the generalized equation for cells that contain a water surface is as follows.

$$\frac{\partial h'w}{\partial t}+\frac{\partial h'wu}{\partial x}+\frac{\partial h'wv}{\partial y}+\frac{\partial w^2}{\partial z}=-\frac{h'}{\rho}\frac{\partial p}{\partial z}+g_z h' +\frac{\partial}{\partial x}\left\{\nu_H h'\left(\frac{\partial w}{\partial x}+\frac{\partial u}{\partial z}\right)\right\}+\frac{\partial}{\partial y}\left\{\nu_H h'\left(\frac{\partial w}{\partial y}+\frac{\partial v}{\partial z}\right)\right\}+\frac{\partial}{\partial z}\left(2\nu_V\frac{\partial w}{\partial z}\right) \quad \text{(1.4.13)}$$

where

$$h'=\frac{h_k\Delta z_k+h_{k+1}\Delta z_{k+1}}{\Delta z_k+\Delta z_{k+1}} \quad \text{(1.4.14)}$$

The time differentiation term is modified to the following like the one described in 1.4.1.3:

$$h'\frac{\partial w}{\partial t}+\frac{\partial (hu)w}{\partial x}+\frac{\partial (hv)w}{\partial y}+\frac{\partial w^2}{\partial z}=-\frac{h'}{\rho}\frac{\partial p}{\partial z}+g_z h' -w\frac{\partial h'}{\partial t} +\frac{\partial}{\partial x}\left\{\nu_H h'\left(\frac{\partial w}{\partial x}+\frac{\partial u}{\partial z}\right)\right\}+\frac{\partial}{\partial y}\left\{\nu_H h'\left(\frac{\partial w}{\partial y}+\frac{\partial v}{\partial z}\right)\right\}+\frac{\partial}{\partial z}\left(2\nu_V\frac{\partial w}{\partial z}\right) \quad \text{(1.4.15)}$$

This equation is discretized into the following form:

$$
\begin{aligned}
& h_{k\square 1/2}^{n\square 1/2}\, \frac{w_{k\square 1/2}^{*} - w_{k\square 1/2}^{n}}{\Delta t} \\
& \square\, \frac{\square\, w \,\square_{i\square 1/2}^{n}\, (hu)_{i\square 1/2}^{n} - \square\, w \,\square_{i-1/2}^{n}\, (hu)_{i-1/2}^{n}}{\Delta x_i} \\
& \square\, \frac{\square\, w \,\square_{j\square 1/2}^{n}\, (hv)_{j\square 1/2}^{n} - \square\, w \,\square_{j-1/2}^{n}\, (hv)_{j-1/2}^{n}}{\Delta y_j} \\
& \square\, \frac{\square\, w \,\square_{k\square 1}^{n}\, w_{k\square 1}^{n} - \square\, w \,\square_{k}^{n}\, w_{k}^{n}}{\Delta z_{k\square 1/2}} \\
& \square - \frac{h_{k\square 1/2}^{n\square 1/2}}{\rho}\, \frac{p_{k\square 1}^{n} - p_{k}^{n}}{\Delta z_{k\square 1/2}} \square\, g_z\, h_{k\square 1/2}^{n\square 1/2} \\
& \square\, \frac{1}{\Delta x_i} \left\{ \nu_H h_{i\square 1/2}^{n\square 1/2} \left(\frac{u_{i\square 1/2,k\square 1}^{n} - u_{i\square 1/2,k}^{n}}{\Delta z_{k\square 1/2}} \square \frac{w_{i\square 1,k\square 1/2}^{n} - w_{i,k\square 1/2}^{n}}{\Delta x_{i\square 1/2}} \right) \right. \\
& - \qquad \left. \nu_H h_{i-1/2}^{n\square 1/2} \left(\frac{u_{i-1/2,k\square 1}^{n} - u_{i-1/2,k}^{n}}{\Delta z_{k\square 1/2}} \square \frac{w_{i,k\square 1/2}^{n} - w_{i-1,k\square 1/2}^{n}}{\Delta x_{i-1/2}} \right) \right\} \\
& \square\, \frac{1}{\Delta y_j} \left\{ \nu_H h_{j\square 1/2}^{n\square 1/2} \left(\frac{v_{j\square 1/2,k\square 1}^{n} - v_{j\square 1/2,k}^{n}}{\Delta z_{k\square 1/2}} \square \frac{w_{j\square 1,k\square 1/2}^{n} - w_{j,k\square 1/2}^{n}}{\Delta y_{j\square 1/2}} \right) \right. \\
& - \qquad \left. \nu_H h_{j-1/2}^{n\square 1/2} \left(\frac{v_{j-1/2,k\square 1}^{n} - v_{j-1/2,k}^{n}}{\Delta z_{k\square 1/2}} \square \frac{w_{j,k\square 1/2}^{n} - w_{j-1,k\square 1/2}^{n}}{\Delta y_{j-1/2}} \right) \right\} \qquad \text{......................} (1.4.16) \\
& \square\, \frac{1}{\Delta z_{k\square 1/2}} \left(2\nu_V\, \frac{w_{k\square 3/2}^{n} - w_{k\square 1/2}^{n}}{\Delta z_{k\square 1}} - 2\nu_V\, \frac{w_{k\square 1/2}^{n} - w_{k-1/2}^{n}}{\Delta z_{k}} \right) \\
& \square\, w_{k\square 1/2}^{n} \left(\frac{(hu)_{i\square 1/2}^{n} - (hu)_{i-1/2}^{n}}{\Delta x_i} \square \frac{(hv)_{j\square /1/2}^{n} - (hv)_{j-/1/2}^{n}}{\Delta y_j} \square \frac{w_{k\square 1}^{n} - w_{k}^{n}}{\Delta z_{k\square 1/2}} \right)
\end{aligned}
$$

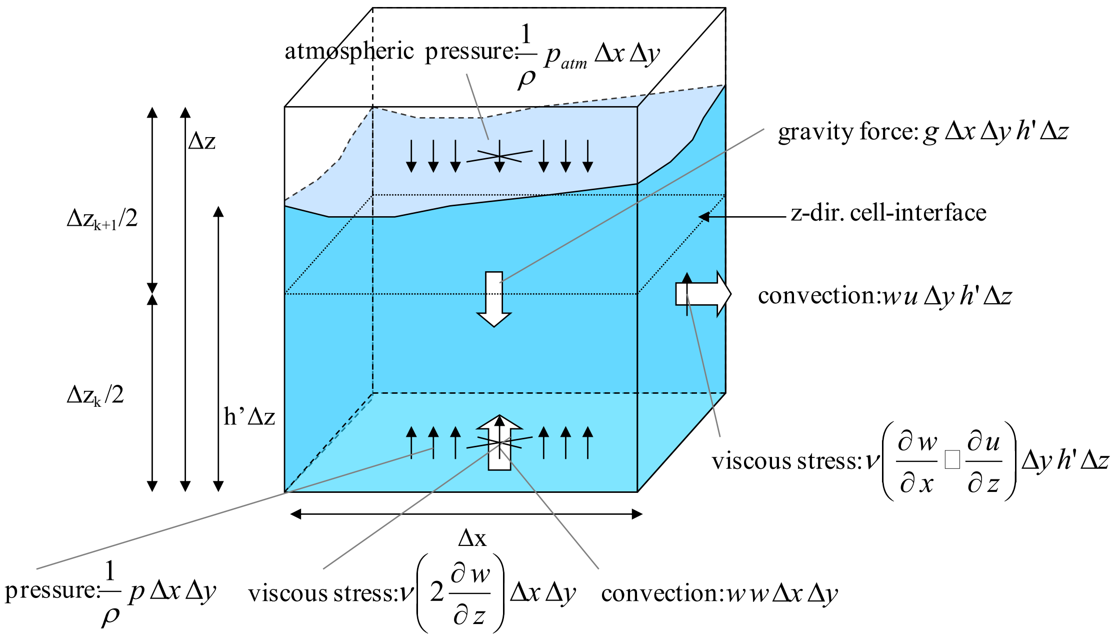


Figure 0-1-19 Control Volume: Vertical Momentum Conservation Equation

1.4.1.5. Transportation Equation for Scalar Amounts

The transportation equation for scalar amounts for cells that do not contain a water surface is already described in the topic of the turbulent model. Considering the control volume shown in Figure 0-1-20, the generalized equation for cells that contain a water surface is as follows (C represents various scalar amounts).

$$\frac{\partial hC}{\partial t}+\frac{\partial (hu)C}{\partial x}+\frac{\partial (hv)C}{\partial y}+\frac{\partial wC}{\partial z} = \frac{\partial}{\partial x}\left(hK_H\frac{\partial C}{\partial x}\right)+\frac{\partial}{\partial y}\left(hK_H\frac{\partial C}{\partial y}\right)+\frac{\partial}{\partial z}\left(K_Z\frac{\partial C}{\partial z}\right)+\frac{q}{\Delta z}+hS_C \quad \text{.................................... (1.4.17)}$$

When h→0 for the cell that contains the water surface, C becomes undefined. Thus, the values for this cell and the cell immediately below it are actually averaged. Note that if the cell that contains the water surface touches the ground when h→0, then the scalar amount should be determined from the boundary conditions for the ground.

The equation above is discretized into the following form:

$$\begin{aligned}
&\frac{h^{n+1/2}C^{n+1/2}-h^{n-1/2}C^{n-1/2}}{\Delta t}\\
&+\frac{\{C\}^{n-1/2}_{i+1/2}(hu)^n_{i+1/2}-\{C\}^{n-1/2}_{i-1/2}(hu)^n_{i-1/2}}{\Delta x_i}\\
&+\frac{\{C\}^{n-1/2}_{j+1/2}(hv)^n_{j+1/2}-\{C\}^{n-1/2}_{j-1/2}(hv)^n_{j-1/2}}{\Delta y_j}\\
&+\frac{\{C\}^{n-1/2}_{k+1/2}w^n_{k+1/2}-\{C\}^{n-1/2}_{k-1/2}w^n_{k-1/2}}{\Delta z_k}\\
&=\frac{1}{\Delta x_i}\left(h^{n-1/2}_{i+1/2}K_{H,i+1/2}\frac{C^{n-1/2}_{i+1}-C^{n-1/2}_{i}}{\Delta x_{i+1/2}}-h^{n-1/2}_{i-1/2}K_{H,i-1/2}\frac{C^{n-1/2}_{i}-C^{n-1/2}_{i-1}}{\Delta x_{i-1/2}}\right)\\
&+\frac{1}{\Delta y_j}\left(h^{n-1/2}_{j+1/2}K_{H,j+1/2}\frac{C^{n-1/2}_{j+1}-C^{n-1/2}_{j}}{\Delta y_{j+1/2}}-h^{n-1/2}_{j-1/2}K_{H,j-1/2}\frac{C^{n-1/2}_{j}-C^{n-1/2}_{j-1}}{\Delta y_{j-1/2}}\right)\\
&+\frac{1}{\Delta z_k}\left(K_{Z,k+1/2}\frac{C^{n+1/2}_{k+1}-C^{n+1/2}_{k}}{\Delta z_{k+1/2}}-K_{Z,k-1/2}\frac{C^{n+1/2}_{k}-C^{n+1/2}_{k-1}}{\Delta z_{k-1/2}}\right)\\
&-S_AC^{n+1/2}+S_B \quad \text{................. (1.4.18)}
\end{aligned}$$

(*This shows an implicit calculation of the vertical diffusion term; it can also be explicitly calculated.)

Note that the generation term is defined as $S\equiv -S_AC^{n+1/2}+S_B$. The equation below is used to define $\widetilde{C}$.

$$
\begin{aligned}
&\frac{h^{n+1/2}\widetilde{C} - h^{n-1/2}C^{n-1/2}}{\Delta t} \\
&+ \frac{\{C\}_{i+1/2}^{n-1/2}\,(hu)_{i+1/2}^{n} - \{C\}_{i-1/2}^{n-1/2}\,(hu)_{i-1/2}^{n}}{\Delta x_i} \\
&+ \frac{\{C\}_{j+1/2}^{n-1/2}\,(hv)_{j+1/2}^{n} - \{C\}_{j-1/2}^{n-1/2}\,(hv)_{j-1/2}^{n}}{\Delta y_j} \\
&+ \frac{\{C\}_{k+1/2}^{n-1/2}\,w_{k+1/2}^{n} - \{C\}_{k-1/2}^{n-1/2}\,w_{k-1/2}^{n}}{\Delta z_k} \qquad \text{.................} (1.4.19) \\
&= \frac{1}{\Delta x_i}\left(h_{i+1/2}^{n-1/2} K_{H,i+1/2} \frac{C_{i+1}^{n-1/2} - C_{i}^{n-1/2}}{\Delta x_{i+1/2}} - h_{i-1/2}^{n-1/2} K_{H,i-1/2} \frac{C_{i}^{n-1/2} - C_{i-1}^{n-1/2}}{\Delta x_{i-1/2}} \right) \\
&+ \frac{1}{\Delta y_j}\left(h_{j+1/2}^{n-1/2} K_{H,j+1/2} \frac{C_{j+1}^{n-1/2} - C_{j}^{n-1/2}}{\Delta y_{j+1/2}} - h_{j-1/2}^{n-1/2} K_{H,j-1/2} \frac{C_{j}^{n-1/2} - C_{j-1}^{n-1/2}}{\Delta y_{j-1/2}} \right) \\
&- S_A \widetilde{C} + S_B
\end{aligned}
$$

After $\widetilde{C}$ is obtained from equations (1.4.18) and (1.4.19), $C^{n+1/2}$ can easily be obtained by calculating the following triple diagonal matrix:

$$
\begin{aligned}
&\left(1 + A\frac{K_{Z,k+1/2}}{\Delta z_{k+1/2}} + A\frac{K_{Z,k-1/2}}{\Delta z_{k-1/2}} \right) C^{n+1/2} - A\frac{K_{Z,k+1/2}}{\Delta z_{k+1/2}} C_{k+1}^{n+1/2} - A\frac{K_{Z,k-1/2}}{\Delta z_{k-1/2}} C_{k-1}^{n+1/2} \qquad \text{............} (1.4.20) \\
&= \widetilde{C}
\end{aligned}
$$

$$
A \equiv \frac{\Delta t}{\left(h^{n+1/2} + S_A \Delta t \right) \Delta z} \qquad \text{..} (1.4.21)
$$

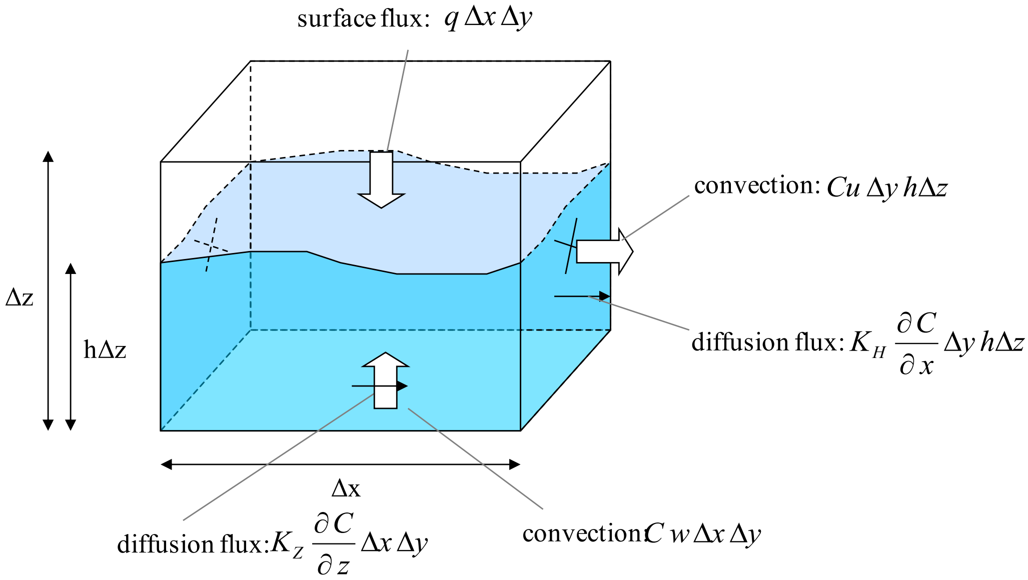


Figure 0-1-20 Control Volume: Transportation Equation for Scalar Amounts

1.4.2. Algorithm for Time Integration

Time integration is performed using the Leapfrog method in which there is a difference of $\Delta t/2$ between the time to define the flow velocity and the time to define the pressure or water surface position. A specific procedure for time integration is shown below. Note that the discretized equation below is in simplified form.

(1) Calculate water level η^n at time n from the equation below.

$$\frac{\eta^n - \eta^{n-1}}{\Delta t} + \frac{\partial}{\partial x}\int_{-h}^{\eta}(hu)^{n-\frac{1}{2}}dz + \frac{\partial}{\partial y}\int_{-h}^{\eta}(hv)^{n-\frac{1}{2}}dz = 0 \quad \text{......} (1.4.22)$$

(2) Discretize the momentum conservation equation as follows to calculate temporary flow velocity ${}^{l}\tilde{u}_i^{n+\frac{1}{2}}$ at time (n).

$$\frac{u_i^{n+\frac{1}{2}} - u_i^{n-\frac{1}{2}}}{\Delta t} = -\frac{1}{\rho}\frac{\partial p^n}{\partial x_i} + f_{ui}\left(u,v,w\right)^n, \quad \frac{\partial u_i^{n+\frac{1}{2}}}{\partial x_i} = 0, \quad u_i^n = \frac{u_i^{n+\frac{1}{2}} + u_i^{n-\frac{1}{2}}}{2} \quad \text{......} (1.4.23)$$

Repeatedly solve the non-linear equations above by performing the following calculation.

$${}^{(l)}u_i^n = \frac{{}^{(l)}u_i^{n+\frac{1}{2}} + u_i^{n-\frac{1}{2}}}{2} \quad \text{......} (1.4.24)$$

$$\frac{{}^{(l+1)}u_i^{n+\frac{1}{2}} - u_i^{n-\frac{1}{2}}}{\Delta t} = -\frac{1}{\rho}\frac{\partial\,{}^{(l+1)}p^n}{\partial x_i} + f_{ui}\,{}^{(l)}\left(u,v,w\right)^n \quad \text{......} (1.4.25)$$

$$\frac{\partial\,{}^{(l+1)}u_i^{n+\frac{1}{2}}}{\partial x_i} = 0 \quad \text{......} (1.4.26)$$

where *l* is the number of iterations. When *l*=1, ${}^{(l)}p^n = p^{n-1}$, ${}^{(l)}u_i^{n+\frac{1}{2}} = u_i^{n-\frac{1}{2}}$.

(2-1) Calculate temporary flow velocity ${}^{(l+1)}\tilde{u}_i^{n+\frac{1}{2}}$ with the equation below.

$$\frac{{}^{(l+1)}\tilde{u}_i^{n+\frac{1}{2}} - u_i^{n-\frac{1}{2}}}{\Delta t} = -\frac{1}{\rho}\frac{\partial\,{}^{(l)}p^n}{\partial x_i} + f_{ui}\,{}^{(l)}\left(u,v,w\right)^n \quad \text{......} (1.4.27)$$

(2-2) Subtract equation (1.4.27) from the first equation in (1.4.23). The equations below are obtained. Substitute the obtained equations into the continuity equation represented by the second equation in (1.4.23) and rearrange this continuity equation. Poisson equation (1.4.29) is obtained. Solve this Poisson equation by using an iterative method.

$$ {}^{(l+1)}u_i^{n+1/2} = {}^{(l+1)}\tilde{u}_i^{n+1/2} - \Delta t \frac{1}{\rho}\frac{\partial \Delta p}{\partial x_i}, \qquad {}^{(l+1)}p^n = {}^{(l)}p^n + \Delta p \quad \text{......} \quad (1.4.28) $$

$$ \frac{\partial^2 \Delta p}{\partial x_i^2} = \frac{\Delta t}{\rho} \cdot \frac{\partial {}^{(l+1)}\tilde{u}_i^{n+1/2}}{\partial x_i} \quad \text{......} \quad (1.4.29) $$

(2-3) Using the correction pressure calculated in equation (1.4.29), update the flow velocity and pressure.

$$ {}^{(l+1)}u_i^{n+1/2} = {}^{(l)}\tilde{u}_i^{n+1/2} - \Delta t \frac{1}{\rho}\frac{\partial \Delta p}{\partial x_i}, \qquad {}^{(l+1)}p^n = {}^{(l)}p^n + \Delta p \quad \text{......} \quad (1.4.30) $$

(2-4) If Δp is close enough to zero or if the number of iterations exceeds the maximum, move on to the next step. In all the other cases, increment *l* by 1 and return to step (2-1). Note that when the maximum number of iterations is 1, no iteration occurs, and thus the following scheme is implemented:

$$ \frac{u_i^{n+1/2} - u_i^{n-1/2}}{\Delta t} = -\frac{1}{\rho}\frac{\partial p^n}{\partial x_i} + f_{ui}(u, v, w)^{n-1/2}, \quad \frac{\partial u_i^{n+1/2}}{\partial x_i} = 0 \quad \text{......} \quad (1.4.31) $$

(3) Update the time and, if the updated time is not yet the time to terminate the processing, return to step (1).

1.4.3. Matrix Solutions

Simultaneous linear equations with symmetrical coefficient matrixes are formulated as a result of multiplying both sides of equation (1.4.29) by the volume of a cell. STOC uses an asymmetrical-matrix solution to allow for greater flexibility in the process of handling boundary conditions. The BiCGSTAB method is known as a stable and fast solution to simultaneous linear equations with asymmetrical coefficient matrixes. Adding an incomplete LU decomposition process as the preprocess to this method makes the ILU/BiCGSTAB method. STOC uses the ILU/BiCGSTAB method. Figure 0-1-21 shows the algorithm for the ILU/BiCGSTAB method.

(Iteration (Process before Iteration)

Set initial value x_0 and calculate $r_o = (LU)^{-1}(b - Ax_o)$.

Select arbitrary vector r_s to meet $(r_s, r_o) \neq 0$. (For example, select $r_s = r_o$.)

Set $p_o = r_0$.

(Iteration) $i = 1, 2, 3, \ldots$.

$\alpha = (r_s, r_{i-1}) / (r_s, (LU)^{-1} A p_{i-1})$

$s_i = r_{i-1} - \alpha (LU)^{-1} A p_{i-1}$

$\omega = ((LU)^{-1} A s_i, s_i) / ((LU)^{-1} A s_i, (LU)^{-1} A s_i)$

$x_i = x_{i-1} + \alpha p_{i-1} + \omega s_i$

$r_i = s_i - \omega (LU)^{-1} A s_i$

if ($\|r_i\| \leq \varepsilon$) then

Perform the termination process, because convergence was achieved.

else

$\beta = ((r_s, r_i)\alpha) / ((r_s, r_{i-1})\omega)$

$p_i = r_i + \beta (p_{i-1} - \omega (LU)^{-1} A p_{i-1})$

$i = i + 1$

Return to the first step for iteration.

Figure 0-1-21 Algorithm for the ILU/BiCGSTAB Method

1.4.4. Special Processing of Free Surfaces

1.4.4.1. Storm Surge Barrier

1.4.4.1.1. Basic Concept

The philosophy behind the basic concept of the storm surge barrier model is that if there is a storm surge barrier at the interface of a computation cell, the flow rate is maintained at 0 until the water level exceeds the storm surge barrier.

1.4.4.1.2. Determining Presence/Absence of a Storm Surge Barrier

The conditions for determining presence/absence of a storm surge barrier are as follows.

Storm surge barrier absent: $\gamma_{x,i\pm1/2} \geq \gamma_{v,i\pm1/2}$ .... (1.4.32)

Storm surge barrier present: $\gamma_{x,i\pm1/2} < \gamma_{v,i\pm1/2}$ .... (1.4.33)

where $\gamma_{v,i\pm1/2} \equiv \dfrac{\gamma_{v,i}\Delta x_i + \gamma_{v,i\pm1}\Delta x_{i\pm1}}{\Delta x_i + \Delta x_{i\pm1}}$.

1.4.4.1.3. Processing When a Storm Surge Barrier Is Present

For the water levels shown in Figure 0-1-22, the flow velocity on the cell interface is set to 0 without solving the momentum conservation equation.

For the water levels shown in Figure 0-1-23 (A), the momentum conservation equation is calculated assuming that the layer thickness for the cell on the left is “a” and that for the cell on the right is “b”. If the water level exceeds the height of the cell on the right as shown inFigure 0-1-23 (B), the same processing as that for Figure 0-1-23 (A) is performed assuming that this water level is at cell height “b”. If the water level for the cell on the right is lower than the height of the storm surge barrier as shown in Figure 0-1-23 (C), the same processing as that for Figure 0-1-23 (A) is performed assuming that this water level is at storm surge barrier height b = 0. For the water levels shown in Figure 0-1-23 (D), the same processing as that for Figure 0-1-23 (A) is performed by correcting “a” and “b” as described above.

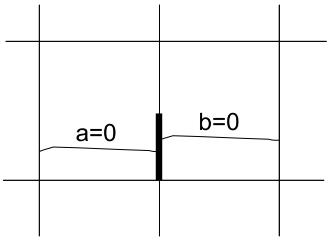


Figure 0-1-22 When the Water Levels on Both Sides of the Storm Surge Barrier Are Lower than the Storm surge barrier Height

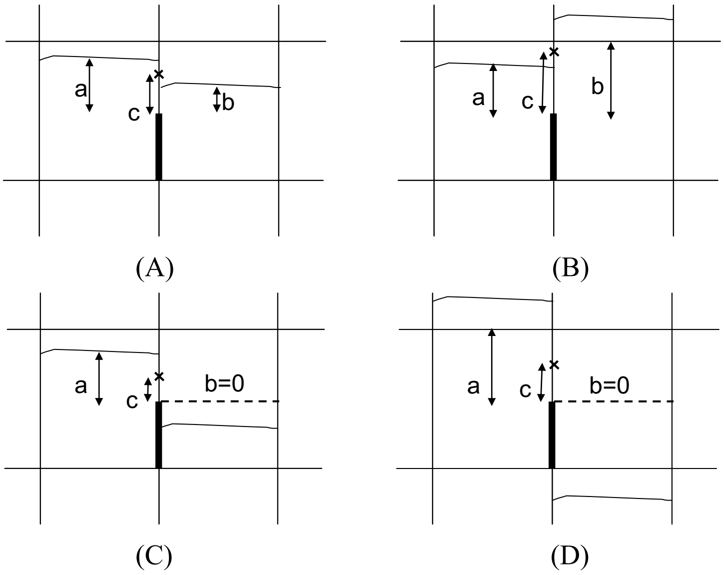


Figure 0-1-23 When the Water Levels on Both Sides of the Storm Surge Barrier or Water Level on Either Side Is Higher than the Storm Surge Barrier Height

(“a”, “b”, and “c” are overflow water levels, and “c” is the water level interpolated with the central difference.)

1.4.4.1.4. Calculating a Flow Rate per Unit Width

For normal cells that do not contain a storm surge barrier, the flow rate per unit width is calculated as follows.

$$(hu)_i \square h_{i\square 1/2}\, u_{i\square 1/2} \square \max(h_{F,i\square 1/2} - 1 \square \gamma_{x,i\square 1/2}, 0)\, u_{i\square 1/2} \quad \text{(1.4.34)}$$

$$
h_{F,i+1/2} = \begin{cases} h_{F,i} & \text{(1st order upwind, } u_{i+1/2} \geq 0) \\ h_{F,i+1} & \text{(1st order upwind,} u_{i+1/2} < 0) \\ \dfrac{h_{F,i}\Delta x_{i+1} + h_{F,i+1}\Delta x_i}{\Delta x_i + \Delta x_{i+1}} & \text{(central differntial)} \end{cases} \quad \text{.............................. (1.4.35)}
$$

Similar processing is possible with cells that contain a storm surge barrier; however, it involves correcting "b" shown in Figure 0-1-23 (C) and (D). Thus, instead of $h_{F,i}$ and $h_{F,i+1}$, $h'_{F,i}$ and $h'_{F,i}$ are used in equation (1.4.35). $h'_{F,i}$ and $h'_{F,i}$ are defined as follows.

$$
\begin{aligned} h'_{F,i} &= \max(h_{F,i},\ 1-\gamma_{x,i+1/2}) \\ h'_{F,i+1} &= \max(h_{F,i+1},\ 1-\gamma_{x,i+1/2}) \end{aligned} \quad \text{.............................. (1.4.36)}
$$

1.4.4.1.5. Calculating the Pressure Gradient

For locations that do not contain a storm surge barrier (normal cells), the pressure gradient is calculated as follows.

$$
-\frac{h}{\rho}\frac{\partial p}{\partial x} = -\frac{h_{i+1/2}}{\rho}\frac{p_{i+1} - p_i}{\Delta x_{i+1/2}} \quad \text{.............................. (1.4.37)}
$$

p_i, p_{i+1} for a water surface cell are determined by linear interpolation with atmospheric pressure and pressure on the lower cells.

Similar processing is possible with locations that contain a storm surge barrier. If the water levels for the cells on both sides of the storm surge barrier are higher than the storm surge barrier height as shown in Figure 0-1-23 (A) and (B), these cells are treated the same as normal cells. On the other hand, if the water level on either side of the storm surge barrier is lower than the storm surge barrier height as shown in Figure 0-1-23 (C) and (D), the pressure gradient is calculated assuming that this water level is at the height of the storm surge barrier.

In summary, a pressure gradient term for storm surge barriers is calculated as follows.

$$
-\frac{h}{\rho}\frac{\partial p}{\partial x} = -\frac{h_{i+1/2}}{\rho}\frac{p'_{i+1} - p'_i}{\Delta x_{i+1/2}} \quad \text{.............................. (1.4.38)}
$$

$$p'_i \square \begin{cases} p_i & (h_{F,i} \geq 1-\gamma_{x,i\square 1/2}) \\ p_{atm,i} - \rho g_z (1-\gamma_{x,i\square 1/2})\Delta z_k \square \rho g_z \dfrac{\Delta z_k}{2} & (h_{F,i} \square 1-\gamma_{x,i\square 1/2}) \end{cases} \quad \text{........................} (1.4.39)$$

$$p'_{i\square 1} \square \begin{cases} p_{i\square 1} & (h_{F,i\square 1} \geq 1-\gamma_{x,i\square 1/2}) \\ p_{atm,i\square 1} - \rho g_z (1-\gamma_{x,i\square 1/2})\Delta z_k \square \rho g_z \dfrac{\Delta z_k}{2} & (h_{F,i\square 1} \square 1-\gamma_{x,i\square 1/2}) \end{cases} \quad \text{.................} (1.4.40)$$

1.4.4.2. Correction of a Pressure Gradient Term with Topography and Bore

The pressure gradient term for topographies with different heights and for bore water level distributions is corrected in the same manner as described in 1.4.4.1 for storm surge barriers.

(1) Restrictions per the topography

If the water level on one side is lower than the height of the other side as shown in Figure 0-1-24, the pressure gradient is calculated assuming that this water level is at the height of the other side.

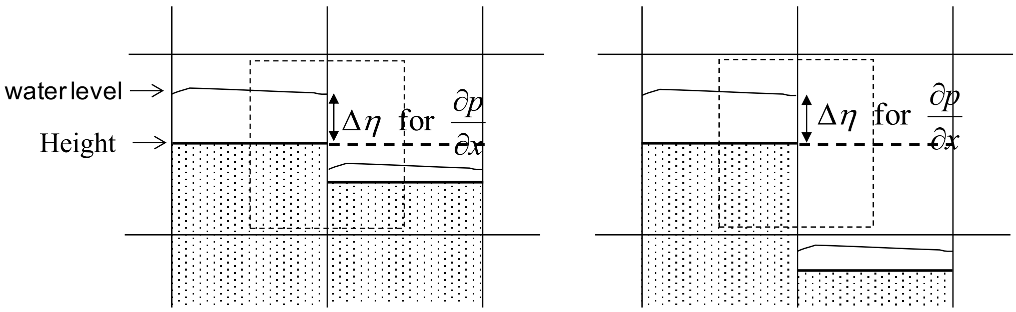


Figure 0-1-24 Pressure Gradient Calculated for Topographies That Are Different in Heigh

(2) Restrictions per bore

If the water level on one side is lower than the height of a cell as shown in Figure 0-1-25, the pressure gradient is calculated assuming that this water level is at the bottom of the cell.

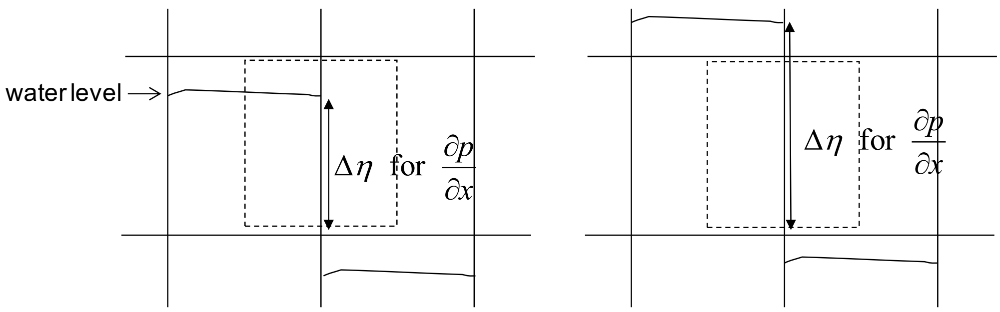


Figure 0-1-25 Calculation of Pressure Gradient for Bore Water Level Distribution

In summary, p'_i, $p'_{i\square 1}$ below are used for calculating equation (1.4.38).

$$p'_i = \begin{cases} p_{atm,i} + \rho g_z \dfrac{\Delta z}{2} & (K = KF(I,J)) \\ p_{atm,i} - \rho g_z \left(1-\gamma_{v,i+1}\right)\Delta z + \rho g_z \dfrac{\Delta z}{2} & (h_{F,i} < 1-\gamma_{v,i+1}) \\ p_i & (other) \end{cases} \quad (1.4.41)$$

$$p'_{i+1} = \begin{cases} p_{atm,i+1} + \rho g_z \dfrac{\Delta z}{2} & (K = KF(I+1,J)) \\ p_{atm,i+1} - \rho g_z \left(1-\gamma_{v,i}\right)\Delta z + \rho g_z \dfrac{\Delta z}{2} & (h_{F,i+1} < 1-\gamma_{v,i}) \\ p_{i+1} & (other) \end{cases} \quad (1.4.42)$$

Note that currently, STOC is not subject to any restrictions that arise from topography, and it calculates the equations below.

$$p'_i = \begin{cases} p_{atm,i} + \rho g_z \dfrac{\Delta z}{2} & (K = KF(I,J)) \\ p_i & (other) \end{cases} \quad (1.4.43)$$

$$p'_{i+1} = \begin{cases} p_{atm,i+1} + \rho g_z \dfrac{\Delta z}{2} & (K = KF(I+1,J)) \\ p_{i+1} & (other) \end{cases} \quad (1.4.44)$$

1.4.5. Model Coupling

1.4.5.1. Nesting Function

The following combinations of products can perform coupled calculations by nesting calculation:

- STOC-ML and STOC-ML
- STOC-ML and STOC-IC

For example, they compute the outer wide regions with coarse meshes and compute the inner target points with fine meshes as shown in Figure 0-1-6. Note that the time step should be the same for all the regions.

A different process is used for each region for calculation, and data is exchanged between regions through Message Passing Interface (MPI).

1.4.5.2. Flow of Nesting Processing

The basic flow of processing for the nesting function is as follows.

(1) Calculate flow velocities “u” and “v” at the updated time.

(2) Transfer the “u” and “v” values, required for processing of the boundary conditions, between the parent and child.

(3) Calculate flow velocity “w” and the water level at the updated time.

(4) Transfer the “w” and water level values, required for processing of the boundary conditions, between the parent and child.

(5) If the current time is not yet the time to terminate the processing, return to step (1).

(6) Perform the termination processing.

1.4.5.3. Processing of the Boundary Conditions for the Nesting Section (Overlapped Area)

The area filled with gray in Figure 0-1-26 is called an overlapped area for nesting. In this area, the flow velocity and water level values are transferred between the parent and child, and their boundary conditions are set. Specify the width of this area as an input parameter. Typically, it should be equal to a single parent mesh. In this document, the boundary (- - -) for transferring the boundary conditions from the parent to the child is called the connection boundary, and the boundary (═══) for transferring the boundary conditions from the child to the parent is called the child connection boundary.

The method of updating the values in an overlapped area is as follows.

(1) Parent mesh calculation

In the parent mesh, the child connection boundary (═══) is treated the same as a free inflow/outflow boundary, and the region within that boundary is not calculated. The areas filled with slashes in Figure 0-1-26 are processed as obstacles.

(1-1) Flow velocity calculation

In the parent mesh, calculate the flow velocity (in the position indicated by ⟹ in Figure 0-1-26) on both the connection boundary surface and overlapped area. Set the inner flow velocity (in the position indicated by ⟶) required for this calculation to the average value of flow velocities computed for the associated child meshes.

(1-2) Water level calculation

In the parent mesh, set the water level (○) in the overlapped area and the inner water level (△) to the average value of water levels computed for the associated child meshes.

For nesting calculations, it is strongly recommended to match the water depth in both the overlapped area and the area immediately inside this overlapped area to the water depth for the child mesh.

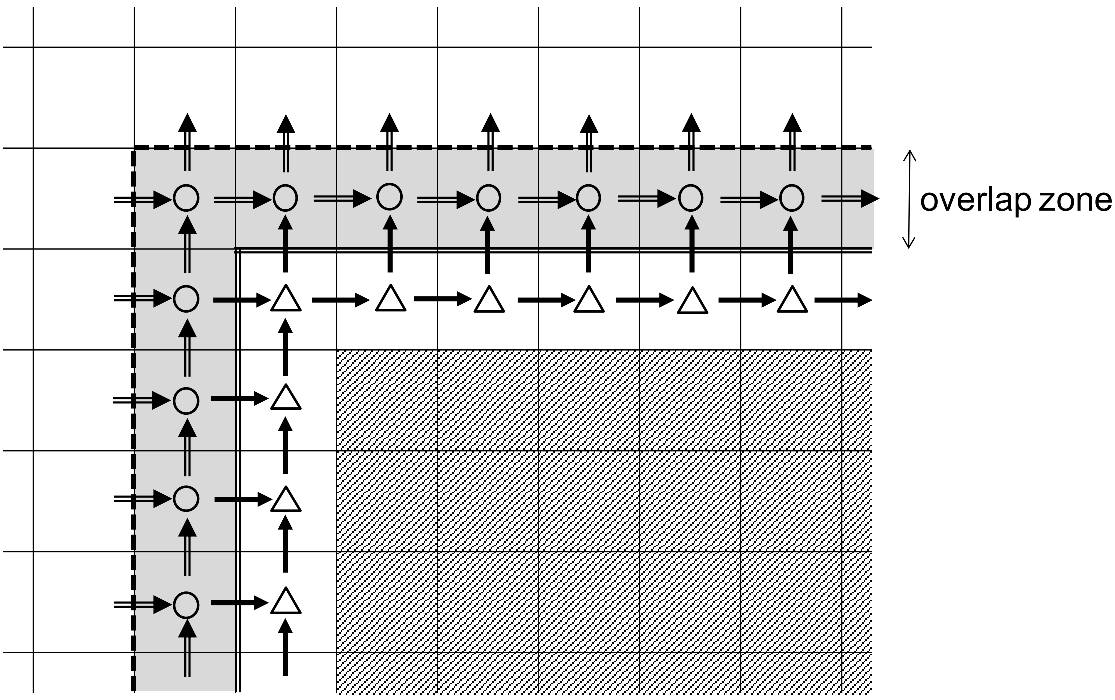


Figure 0-1-26 Parent Mesh Calculation

(2) Child mesh calculation

In the child mesh, the connection boundary (– – –) is treated the same as a fixed flow velocity boundary.

(2-1) Flow velocity calculation

In the child mesh, calculate the flow velocity inside the connection boundary shown in Figure 0-1-27. Set the flow velocity (in the position indicated by → in Figure 0-1-27) at the connection boundary to the value obtained by linear interpolation of the flow velocity for the parent mesh. Note that the data is corrected after linear interpolation so the flow rate at the connection boundary for the parent mesh and the outer boundary for the child mesh can match each other.

(2-2) Water level calculation

The water levels for the child mesh are calculated in the regular way. For the overlapped area, the weighted average between the water levels calculated in the parent and child meshes is calculated. The weighting coefficients are configured to set the weight for the parent to 1 and that for the child to 0 at the connection boundary (– – –), as well as to set the weight for the parent to 0 and that for the child to 1 at the child connection boundary (══).

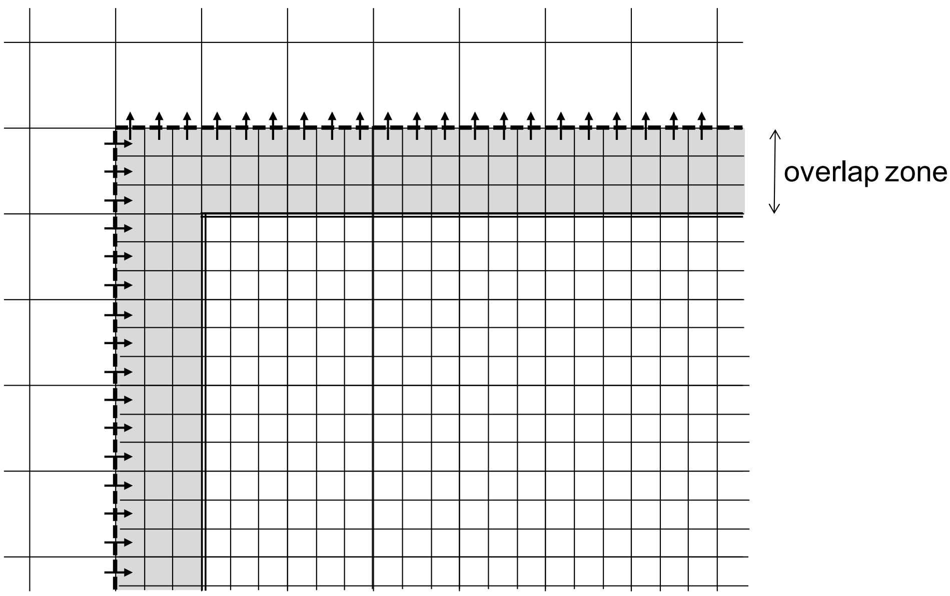


Figure 0-1-27 Child Mesh Calculation

1.4.5.4. Setting of Vertical Flow Velocity Distributions at a Connection Boundary

If there is one parent mesh and multiple child meshes along the vertical direction, set one of these vertical distributions of boundary flow velocities for the position indicated by $\rightarrow$ in Figure 0-1-27:

(1) Uniform vertical distribution (see Figure 0-1-28)
(2) Vertical distribution that is similar to the one in the reference position (see Figure 0-1-29)

If (2), an inner position that is one to several meshes apart from the connection boundary is selected as the reference position, and the distribution characteristics in this position are applied to the boundary position.

If the flow velocity complemented from the parent mesh is $\overline{u}_{IC}$, the flow velocity for each cell height is $u_{IC,k}$, and the layer thickness for each cell height is $\delta_{IC,k}$ (k=2, 3, etc.); then, the restriction condition for the connection boundary is established as follows.

$$\Sigma u_{IC,k}\delta_{IC,k} \square \overline{u}_{IC}\Sigma\delta_{IC,k} \quad \text{(1.4.45)}$$

If the flow velocity and layer thickness in the reference position are $u'_{IC,k}$ and $\delta'_{IC,k}$ (k=2, 3, etc.), respectively, then average flow velocity $\overline{u'_{IC}}$ is calculated as follows.

$$\overline{u'_{IC}} \square \frac{\Sigma_k u'_{IC,k}\delta'_{IC,k}}{\Sigma_k \delta'_{IC,k}} \quad \text{(1.4.46)}$$

In this case, flow velocity $u_{IC,k}$ at the connection boundary is calculated from the equation below by assuming that the water level and water depth at the connection boundary are the same as those in the reference position.

$$u_{IC,k} - \overline{u}_{IC,k} \square u'_{IC,k} - \overline{u'_{IC,k}} \quad \text{(1.4.47)}$$

In actuality, the water level and water depth at the connection boundary are different from those in the reference position. Thus, the equation above is applied after the flow velocity distribution in the reference position is scaled according to the range between the surface and bottom of the sea at the connection boundary as shown in Figure 0-1-30

.

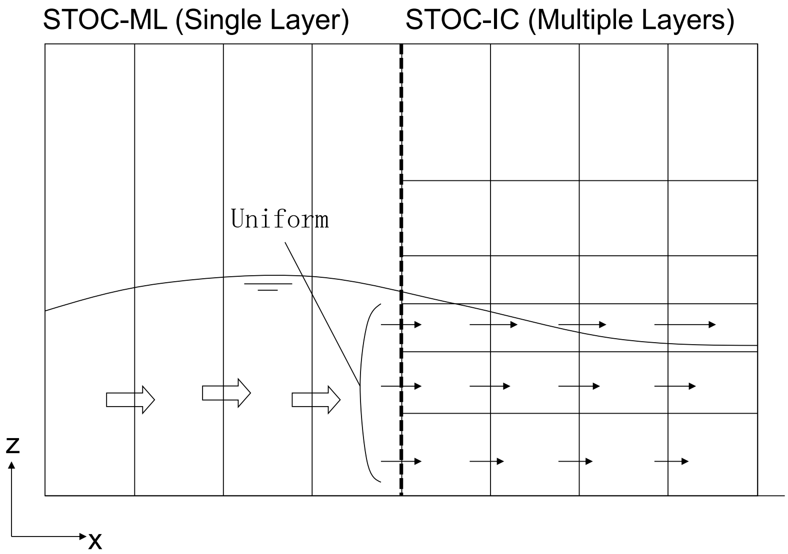


Figure 0-1-28 Vertical Flow Velocity Distribution (Uniform) at Connection Boundary

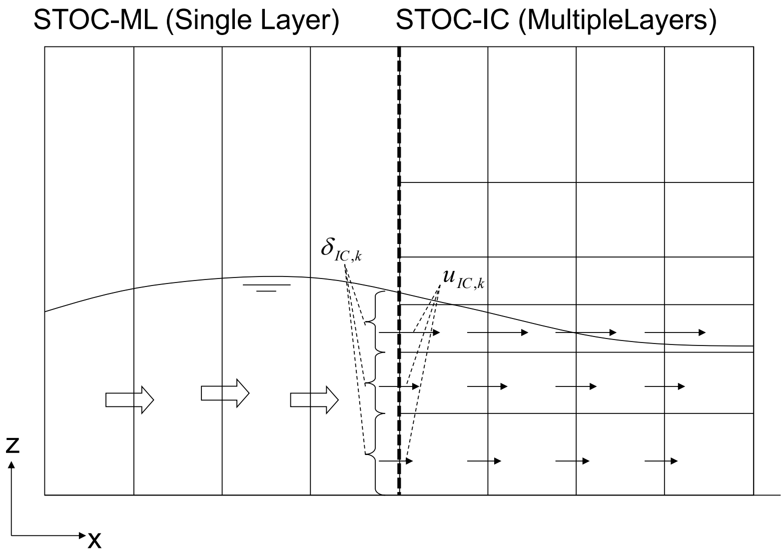


Figure 0-1-29 Vertical Flow Velocity Distribution (Similar to the One in Reference Position) at Connection Boundary

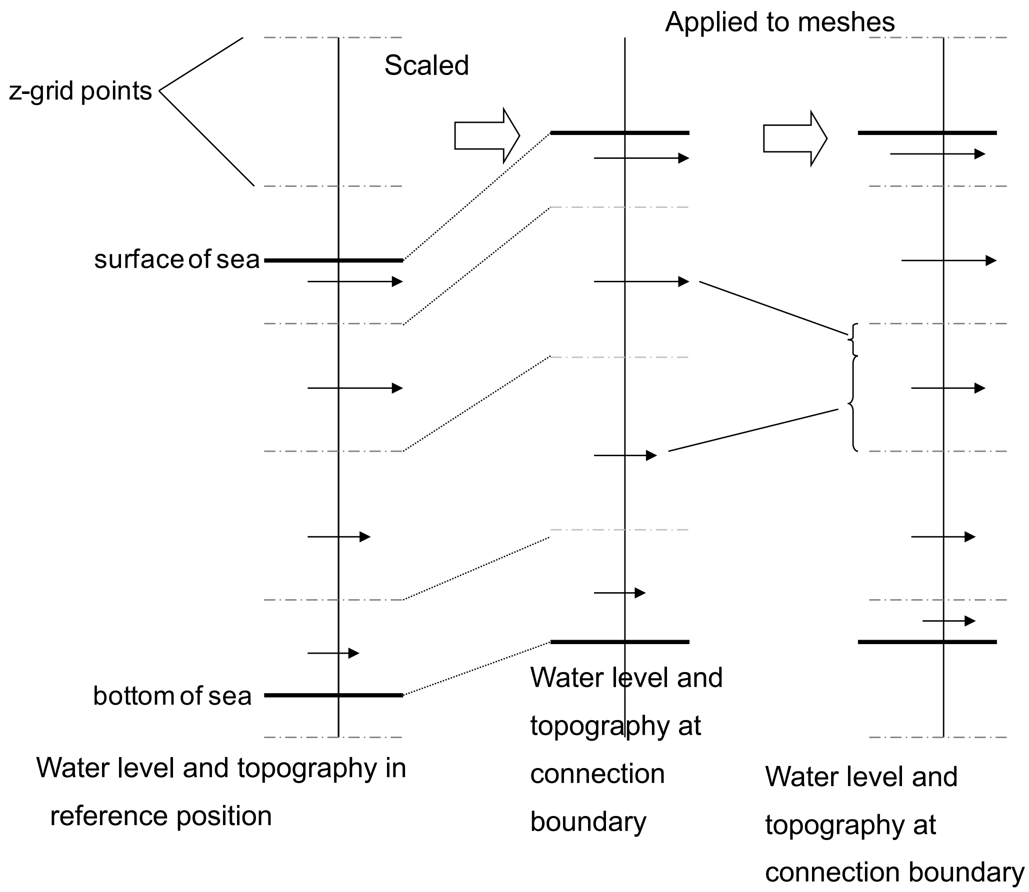


Figure 0-1-30 Corrected Flow Velocity Distribution in Reference Position

Chapter 2 Instructions for Use

2.1. STOC System Configuration

Figure 0-2-1 shows a typical I/O data flow in STOC (where the italic *Area* indicates the name of user-defined data; the other italic *NN* shows a processor number in parallel computations). "Data.in" defines the relationship between connections across areas and exists solely on its own. On the other hand, there is as many of the other files as the number of areas to be computed.

Table 0-2-1 shows a list of I/O data files used in STOC. Section 2.2 describes these files.

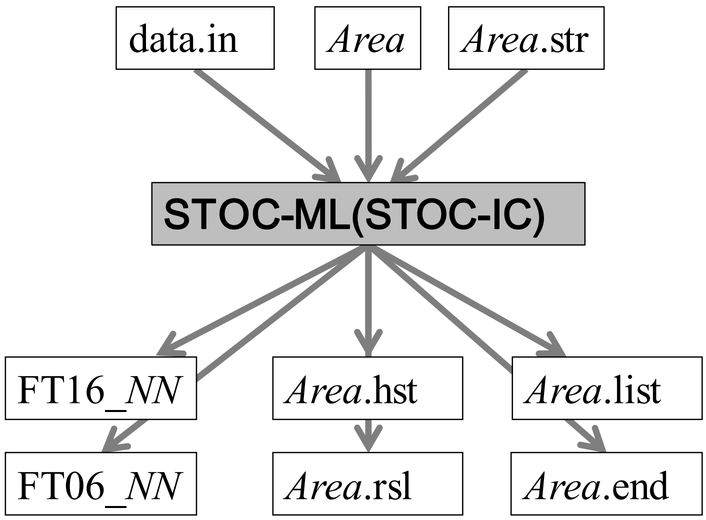


Figure 0-2-1 I/O Correlation Diagram in STOC

Table 0-2-1 List of I/O Files in STOC

File	I/O	Contents	Remarks
data.in	I	Defines the relationship between connections across areas	Mandatory
Area	I	Analysis condition	Mandatory
Area.str	I	Topography/geometry data	As required
Area.sbt	I	The amount of data for the time-dependent water level fluctuations by seismic action	As required
fault.txt	I	Seismic fault parameters	As required
Area.rsi	I	Data for restart (input)	As required
Area.bci	I	Water level/flow velocity data at connection boundary (input)	As required
Area.tim	I	Flow velocity/water level time-series input data	As required
Area.dpr	I	Permeable structure data	As required
Area.ofl	I	Data for specifying the location on which to apply the overflow model	As required
Area.fwc	I	Coefficient data for overfall model	As required
Area.ini	I	Initial distribution data of water temperature and chlorine level	As required
FT16_*NN*	O	Computation intermediate information	Output required
FT06_*NN*	O	Debugging information 1 (maximum velocity, etc.)	Output required
Area.hst	O	Time-series output data	As required
Area.lst	O	Spatial distribution data of water level/flow velocity	As required
Area.end	O	Aggregate data, including the maximum water level, etc.	Output required
Area.rso	O	Data for restart (output)	As required
Area.bco	O	Water level/flow velocity data at connection boundary (output)	As required
data.in_debug	O	Debugging information 2 (relation between connections across areas, etc.)	Output required
Area.dbg	O	Debugging information 3 (value of common variables, etc.)	Output required
Area.ars	I/O	Data for automatic restart	As required

*The italic *Area* indicates the name of user-defined data. The other italic *NN* shows a processor number in parallel computations.

2.2. I/O File Format

2.2.1. Input Data Format 1: Defines Relationship between Connections across Areas (data.in File)

The data.in file defines the relationship between connections across areas and the name of the analysis condition files for each area. A single data.in file is provided for all areas.

Specifically, the parent and child relationship of an area and the name of data files to type are as shown in Figure 0-2-2(in the case of the example in Figure 0-1-2). This is a space-separated free format.

```
1 -99    2   -99   -99   -99   -99   0   Area01
2    1    3   -99   -99   -99   -99   0   Area02
3    2    4   -99   -99   -99   -99   0   Area03
4    3    5   -99   -99   -99   -99   0   Area04
5    4    6   -99   -99   -99   -99   0   Area05
6    5    7   -99   -99   -99   -99   0   Area06
7    6    8   -99   -99   -99   -99   0   Area07
8    7    9   -99   -99   -99   -99   0   Area08
9    8 -99   -99   -99   -99   -99   1   Area09
```

Figure 0-2-2 Example 1 of data.in File

Type one line for each area to be used in computations. The meaning of each row is shown below. Type -99 when no relevant area is found.

1st row: Area number (areas are numbered starting at “1”)
2nd row: Area number of the parent area (one outer area that includes your own area)
3rd row: Area number of the child area (one inner area that is inside your own area)
* When specifying -1, couple it with CADMAS-SURF/3D.
4th to 7th row: Specification for domain decomposition (described later)
8th row: Computation model (0 when computing with STOC-ML and 1 when with STOC-IC)
9th row: Analysis condition file name

After data.in is loaded, the file(s) specified in the 9th row will load.

(1) Specification method at domain decomposition 1

To compute an area in parallel with domain decomposition in the current STOC, all you have to do is specify, I-DIV and J-DIV as the X-directional and Y-directional partitioning methods in the %GRID block of the analysis condition file,. There is no need to update data.in.

(2) Specification method at domain decomposition 2

Method (1) has not yet been introduced in earlier versions of STOC, and some existing input data have an area number specified on the 4th through 7th row. The following shows the meaning of the respective row:

4th row: Area number on the south of your area

5th row: Area number on the west of your area

6th row: Area number on the east of your area

7th row: Area number on the north of your area

For example, to partition the area as shown in Figure 0-2-3, the 4th to 7th rows are specified as shown in Figure 0-2-4. Compared to Method 1, Area08 and Area09 files and Area08.str and Area09.str files must be named differently for each partial area; therefore, Method 2 has practically no advantages for use now.

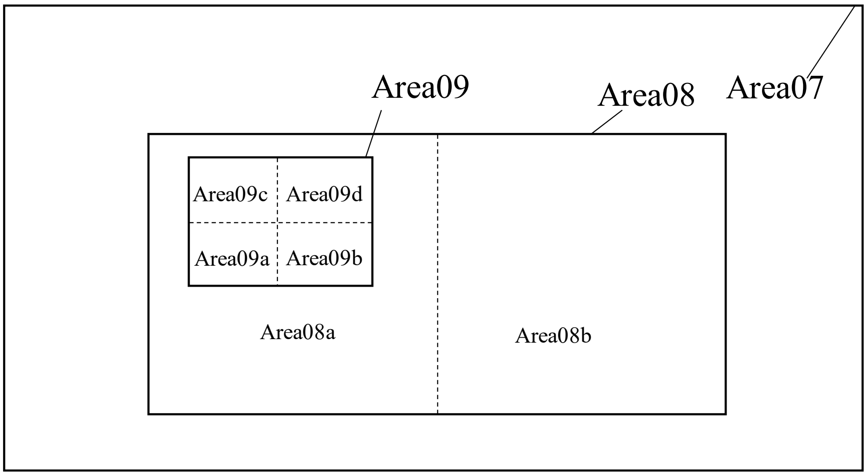


Figure 0-2-3 Example of Domain Decomposition

```
(First half omitted)
 7   6    8  -99  -99  -99  -99  0   Area07
 8   7   10  -99  -99    9  -99  0   Area08a
 9   7  -99  -99    8  -99  -99  0   Area08b
10   8  -99  -99  -99   11   12  1   Area09a
11   8  -99  -99   10  -99   13  1   Area09b
12   8  -99   10  -99   13  -99  1   Area09c
13   8  -99   11   12  -99  -99  1   Area09d
```

Figure 0-2-4 Example 2 of data.in File

2.2.2. Input Data Format 2: Analysis Condition File (Area File)

Create an analysis condition file for each area described in data.in. The actual file name is the one specified in the data.in file.

2.2.2.1. Coding Rules

The data coding rules are as follows.

- Describe input data in the form of “variable=value” in a block sandwiched between %Block name and %END as shown in the sample below. Enclose more than one value, if any, (array type data) in parentheses “( )”.
- Separate data by a space character or line feed. The number of space characters or line feeds is not limited.
- You cannot use a tab character or double-byte space in place of a space.
- One line can be up to 132 characters.
- Lines starting with a pound sign (#) are regarded as comments and are not processed.
- When the same variable is entered more than once, the latest value overrides the previous value.

2.2.2.2. Coding Examples

Figure 0-2-5 through Figure 0-2-7 show sample data for the outmost area. The %BOUNDARY block statement in the inner area changes as shown inFigure 0-2-8.

Moreover, if the turbulence model is set to OFF, input a value that includes turbulence viscosity in VISCOSITY-H as shown in the example. This value depends on horizontal mesh size, and its setting example is shown in Figure 0-2-10.

```
############################
## Indicates computation case names ##
############################
%CASE
   CASE = Area01                    # Case name (up to 32 characters); Used as an I/O file name
%END

############################
## Indicates computation grid data ##
############################
%GRID
  X = (                                # X-directional grid point coordinate value
         -183912.0 -178512.0 -173112.0 -167712.0 -162312.0
Omitted
         1193088.0 1198488.0
       )
  Y = (                                # Y-directional grid point coordinate value
         -728684.0 -723284.0 -717884.0 -712484.0 -707084.0
Omitted
          378316.0
       )
  Z = (                                # Z-directional grid point coordinate value
         -10000.00         50.00
       )
%END

########################
## Indicates geometric data ##
########################
%OBSTACLE
   FILE = YES      # read from "*.str" file
%END

#################################
## Indicates time integration control information ##
#################################
%TIME
   START     =          0.0D0
   END         =    3600.1D0
   MAXSTEP =    1000000
   TYPE       = CONSTANT
   DT           = 0.05
# If TYPE=AUTO, specify safety factor DTSAFE and time-step upper/lower limit: DTMAX and
DTMIN.
   MAX-ITERATION = 1
%END
```

Figure 0-2-5 Example of Analysis Condition File (1/3)

```
######################################
## Specifies the model to use and parameters ##
######################################
%MODEL
   MAX-VELOCITY    = 20.0D0
   GRAVITY          = -9.8D0
   GAMMAS            =    0.0D0
   GAMMAB            = -1.0D0
   CORIOLI          =    9.1D-5
   SURFACE          = ON
   TURBULENT        = OFF
   PARAM-SCHEME-F= 0.0D0
   PARAM-SCHEME-V=  1.0D0        # Weighting of difference scheme in a convection term of
momentum conservation equation
                                   # Central difference for 0.0 and the first-order upwind scheme for 1.0
   ISW = ( 0 0 0 0 0 )
   RUNUP-GXB        = 1.D-4
   RUNUP-EPSH       = 1.D-4
   RUNUP-GLH        = 1.D+4
%END

########################
## Indicates the physicality value of the fluids ##
########################
%PROPERTY
   DENSITY             = 1025.0D0
   VISCOSITY-H         = 1025.D+2    # horizontal
   VISCOSITY-V         = 1025.D-6    # vertical
%END

####################
## Indicates boundary conditions ##
####################
%BOUNDARY
   SURFACE-TYPE           = CONSTANT
   SURFACE-PRESSURE     = 0.0
   SURFACE-WIND-U        = 0.0
   SURFACE-WIND-V        = 0.0
   OPEN-SOMMER = ( 1 1 1 1 ) # Specifies transparent boundary
   SEA-BOTTOM = ON
   FREE-I   = (       1       2    206 2 2 )
   FREE-I   = (    257       2    206 2 2 )
   FREE-J   = (       1       2    257 2 2 )
   FREE-J   = (    206       2    257 2 2 )
   DEFAULT-TYPE-V = SLIP
%END
```

Figure 0-2-6 Example of Analysis Condition File (2/3)

```
####################
## Indicates initial conditions ##
####################
%INITIAL
   U = 0.0D0
   V = 0.0D0
   W = 0.0D0
   H = 0.0D0
%END

##################################
## Specifies matrix solver's parameters ##
##################################
%MATRIX
   EPS                    = 1.0D-20
   EPS-R                  = 1.0D-10
   MAX-ITERATION = 500
   PRINT                  = NO
%END

########################
## Specifies file output ##
########################
%OUTPUT
     RESTART-TIME = ( 1800.0 2100.0 2400.0 3000.0 3600.0 )
     LIST-TIME       = (     0.00    900.0 1800.0 2700.0 3600.0 )
     LIST-PHYS       = ( H U V )
     LIST-SECT-K    = 2
     HISTORY-TIME = 10.0
     HISTORY-PHYS = ( H U V )
     HISTORY-CELL = ( 376 519 2 )
%END
```

Figure 0-2-7 Example of Analysis Condition File (3/3)

```
%BOUNDARY
   OVERLAP = ( 3 3 3 3 )    # Indicates four mesh sizes compared to those in the parent area
   SURFACE-TYPE            = CONSTANT
   SURFACE-PRESSURE     = 0.0
   SURFACE-WIND-U         = 0.0
   SURFACE-WIND-V         = 0.0

   SEA-BOTTOM = ON

   DEFAULT-TYPE-V = SLIP
%END
```

Figure 0-2-8 Example of Analysis Condition File (Inner Area)

DENSITY [kg/m³]
1025

VISCOSITY-H [Pa・s]	ν_H [m/s²]	Δx [m]		
1.025E+05	100.0	1001	～	
5.125E+04	50.0	501	～	1000
2.050E+04	20.0	201	～	500
1.025E+04	10.0	101	～	200
5.125E+03	5.0	51	～	100
2.050E+03	2.0	21	～	50
1.025E+03	1.0	11	～	20
5.125E+02	0.5		～	10

Figure 0-2-9 Setting Example of VISCOSITY-H

2.2.2.3. Detailed Description of Coding

Table 0-2-2 shows the details of code that can be specified in the analysis condition file. The following shows the meaning of each type. The array sizes for array values are specified in the Descriptions column of the table.

C: Character string

I: Integer

R: Real value

T: Real value or time-series table (see 2.2.2.6 for entry method)

X: Combinatorial entry of C, I, and R (see the Descriptions in the table)

Table 0-2-2 Entering Code in Analysis Condition File

Block	Variable	Type	Descriptions
CASE	CASE	C	Computation case name (up to 32 characters) *Recommended to use a name identical to that of the analysis condition file
GRID	X	R	X-directional grid point coordinate value [m] Array size: MXM *Enter in ascending order.
	Y	R	Y-directional grid point coordinate value [m] Array size: MYM
	Z	R	Z-directional grid point coordinate value [m] Array size: MZM *Enter both the lower and upper limit coordinate for perpendicular single layer computation.
	LONGITUDE	R	X-directional grid point coordinate value [degree] Array size: MXM *Enter the east longitude in degrees. Use either the (X,Y) or (LONGITUDE, LATITUDE) set.
	LATITUDE	R	Y-directional grid point coordinate value [degree] Array size: MYM *Enter the north latitude in degrees.

	ORIGIN	R	Origin coordinate value (XG,YG) Array size: 2 Default value: (0.0　0.0) *The coordinate value specified in ORIGIN is added to (X,Y) or (LONGITUDE, LATITUDE). The result is the actual coordinate value used.
	REGION	R	Range of the child area (LONG1, LAT1, LONG2, LAT2) LONG1: Longitude of the west end point LATI1: Latitude of the south end point LONG2: Longitude of the east end point LATI2: Latitude of the north end point Array size: 4 *Specify REGION only if you want to determine a relative positional relationship when your own area is a spherical coordinate and the child area is a Plane coordinate. The value to specify must match one of the grid point coordinates in your own area.
	HLIMIT	R	Lower limit of water depth [m] Default value:　0.0 *Change to HLIMIT when the water depth is a positive value and less than HLIMIT.
	I-DIV	I	Number of cells in each of the X-directional partial areas Array size: Number of X-directional domain decomposition *Specify I-DIV so that the total equals the number of cell partitions, or MXM-1. In addition, when the parent area is present, align a partitioning position with the mesh line of the parent area. To do so, specify a value integral that is a multiple of the proportion of the mesh size to the parent area.

	J-DIV	I	Number of cells in each of the Y-directional partial areas Array size: Number of Y-directional domain decomposition *Specify J-DIV so that the total equals the number of cell partitions, or MYM-1. The same rules apply as with I-DIV when the parent area is present.
OBSTACLE	FILE	C	Whether to load topography/geometry data (.str) or not =YES Load. =NO Do not load. Default value: NO
	D-FILE	C	Whether to load permeable structure data (*.dpr) or not =YES Load. =NO Do not load. Default value: NO
	SOLID	I	Position of a solid obstacle (IS, IE, JS, JE, KS, KE) IS, IE: Start and end positions of cell index I JS, JE: Start and end positions of cell index J KS, KE: Start and end positions of cell index K Array size: 6 *Specify a cell index that includes virtual cells (see 2.2.2.4 for details).
	PLATE-I	I	Position of a plane obstacle when the X direction is normal (I, JS, JE, KS, KE) I: Grid point index I JS, JE: Start and end positions of cell index J KS, KE: Start and end positions of cell index K Array size: 5 (See 2.2.2.4 for details.)

	PLATE-J	I	Position of a plane obstacle when the Y direction is normal (J, IS, IE, KS, KE) J: Grid point index J IS, IE: Start and end positions of cell index I KS, KE: Start and end positions of cell index K Array size: 5 (See 2.2.2.4 for details.)
	PLATE-K	I	Position of a plane obstacle when the Z direction is normal (K, IS, IE, JS, JE) K: Grid point index K IS, IE: Start and end positions of cell index I JS, JE: Start and end positions of cell index J Array size: 5 (See 2.2.2.4 for details.)
	POROUS	X	Position of porous structure and porous value (IS, IE, JS, JE, KS, KE, GV, GX-, GX, GX+, GY-, GY, GY+, GZ-, GZ, GZ+) IS, IE: Start and end positions of cell index I JS, JE: Start and end positions of cell index J KS, KE: Start and end positions of cell index K GV: Porosity GX-: X-directional permeability (on the outer periphery at -X) GX: X-directional permeability (inside the area) GX+: X-directional permeability (on the outer periphery at +X) GY and GZ are similar to GX. Array size: 16 *The same specifications in SOLID apply to IS through KE.

	D-POROUS	X	Position of permeable structure, porous value, and coefficient (IS, IE, JS, JE, KS, KE, GV, GX-, GX, GX+, GY-, GY, GY+, GZ-, GZ, GZ+, CM, CD, α, β, D) Array size: 21 IS through GZ+ are the same as POROUS. CM: Inertia force coefficient CD: Drag coefficient α: Coefficient used in Dupuit-Forchheimer model β: Coefficient used in Dupuit-Forchheimer model D: Characteristic particle diameter used in Dupuit-Forchheimer model
	FRIC	X	Specified range of friction drag and drag coefficient (IS, IE, JS, JE, KS, KE, RFRIC) IS, IE: Start and end positions of cell index I JS, JE: Start and end positions of cell index J KS, KE: Start and end positions of cell index K RFRIC: Drag coefficient Array size: 7
	SEA-FLAG	I	Positional index for changing non-computation cell to computation cell (I, J) I: Cell index I J: Cell index J Array size: 2 (See 2.2.2.5 for details.)
TIME	START	R	Computation start time [s]
	END	R	Computation end time [s]
	MAXSTEP	I	Maximum number of steps
	RESTART-TIME	R	Restart start time [s] *Specify either RESTART-TIME or RESTART-STEP when computating a restart.
	RESTART-STEP	I	Restart start step [s]

	TYPE	C	Time step control method =CONSTANT: Constant-based =AUTO: Variable hour-based Default value: CONSTANT
	DT	R	Time step value [s] *DT is used as an initial value if TYPE=AUTO.
	DTSAFE	R	Safety factor when the variable time step setting is used Default value: 0.4 *Specify DTSAFE if TYPE=AUTO.
	DTMIN	R	Variable time step lower limit *Specify DTMIN if TYPE=AUTO.
	DTMAX	R	Variable time step upper limit *Specify DTMAX if TYPE=AUTO. Stability conditions that results from flow velocity and viscosity are considered when a variable time step value is specified; however, the wave propagation speed is not considered. Therefore, the upper limit should be less than the horizontal mesh size divided by the wave propagation speed.
	MAX-ITERATION	I	Iteration count by the Leap-frog method Default value: 1 *Apply MAX-ITERATION on when using STOC-IC.
MODEL	SURFACE	C	Presence of free surface computation =OFF Do not compute. =ON Compute

	TURBULENT	C	Type of turbulence model =OFF Do not use a turbulence model. =ON or LES Use the LES model. =K-E Use the k-ε model (for STOC-IC only). =SGS Use the SGS model (for STOC-IC only). Default value: OFF *See 1.3.2 for details.
	TURB-VISC	R	Upper and lower limits of turbulence kinematic viscosity [m^2/s] (TVSMAX, TVSMIN, TVSVMX) TVSMAX: Upper limit of turbulence kinematic viscosity TVSMIN: Lower limit of turbulence kinematic viscosity TVSVMX: Additional upper limit applied to vertical turbulence kinematic viscosity only Default value: (100.0 1e-6 0.1) Array size: 3
	TURB-SMAGO	R	Parameter C_S of LES model Default value: 0.2 *See 1.3.2.1 for details.
	SGS-COEF	R	Parameters (C_S C_ε σ_k Sc_t) for SGS One-Equation model Default value: (0.12, 0.31, 0.7, 0.7) Array size: 4 *See 1.3.2.2 for details.

	K-EPS.COEF	R	Parameters ($C_{\square}$ σ_k σ_ε σ_T $C_{\varepsilon,1}$ $C_{\varepsilon,2}$ $C_{\varepsilon,3}$ C_ε $k_{\min}$ $\varepsilon_{\min}$) of k-ε model Default value: (0.09, 1.0 1.3 1.0 1.44 1.92 0.0 0.0 1e-20 1e-20) Array size: 10 *See 1.3.2.3 for details.
	GRAVITY	R	Gravity acceleration [m^2/s] Default value: -9.8
	GAMMAS	R	Surface friction coefficient γ_a^2 <0 Use the friction coefficient as a function of wind speed. =0 Do not consider surface friction. >0 Use the friction coefficient value as input. Default value: -1 *See 1.2.4.3 for details.
	GAMMAB	R	Seabed friction coefficient γ_b^2 <0 Compute using roughness of manning. =0 Do not consider seabed friction. >0 Use the friction coefficient value as input. Default value: -1 *See 1.2.4.4 for details.
	D-POROUS-MODEL	C	Selection of permeable structure model =CDM: Use the CDM model. =DF: Use the DF (Dupuit-Forchheimer) model. *See 1.3.7 for details.
	CORIOLI	R	Coriolis' parameters Default value: 0.0 *Specify either CORIOLI or LATITUDE.

	LATITUDE	R	Reference latitude (north latitude) θ_0 [°] *If you enter the reference latitude in a Plane coordinate system, define Coriolis' parameters as $2\Omega\sin\theta_0$. *Specify either LATITUDE or CORIOLI.
	PARAM-SCHEME-F	R	Weighting parameters for central difference and first-order upwind scheme in the computation of the advective term in Function F Default value: 1.0 *Weighted so that 0.0 indicates the central difference and 1.0 indicates the first-order upwind scheme.
	PARAM-SCHEME-V	R	Weighting parameters for central difference and first-order upwind scheme in the computation of the advective term in momentum conservation equation Default value: 1.0 *Weighted so that 0.0 indicates the central difference and 1.0 indicates the first-order upwind scheme.
	PARAM-SCHEME-K	R	Weighting parameters for central difference and first-order upwind scheme in the computation of the advective term in the advection-diffusion equation used in the turbulence model Default value: 1.0 *Weighted so that 0.0 indicates the central difference and 1.0 indicates the first-order upwind scheme.
	RUNUP-GXB	R	The GXB [m] value used to determine the tip used in the run-up tip model Default value: 1e-4 *See 1.3.1 for details.

	MAX-VELOCITY	R	Limited value of velocity [m/s] Default value: 20.0 *Use RUNUP-GLH is to determine whether limitations should be applied or not.
	RUNUP-GLH	R	The GLH [m] value used to determine whether velocity limitations should be applied or not Default value: 1e+4 *The velocity is limited to VVMAX m/s when layer thickness $h\Delta z$ is GLH or less and the velocity exceeds VVMAX m/s. *Enter the VVMAX value in MAX-VELOCITY.
	RUNUP-GZH	R	The GZH [m] value used to determine whether the surface layer velocity should be computed or not Default value: 1e-3 *Do not computer the surface layer velocity when the thickness of surface layer $h\Delta z$ is GZH or less.
	RUNUP-EPSH	R	Additional thickness [m] of water level when the tide ebbs away Default value: 1e-5
	RUNUP-EPST	R	Tsunami arrival parameter [m] Default value: 1e-2 *Time at which the water level rose above this value from the initial level is assumed to be the arrival time (for post-processing).
	RIMP	R	Implicit parameters Default value: 0.5 *Parameters used to evaluate seabed friction and sea friction drag. Set it to 0.0 for explicit computation, and to 1.0 for implicit computation, or a value between 0 and 1 for weighted averaging.

	SEA-WALL	C	A flag used to determine whether the height of a storm surge barrier should be considered at the flow rate computation of the storm surge barrier =ON Consider. =OFF Do not consider. Default value: OFF *Use OFF to reduce the significant velocity at the storm surge barrier.
	OVERFLOW-HONMA	C	A flag used to determine whether the Honma formula should be applied when there's overflow at the breakwater =OFF Do not apply. =FIX Fixed at the flow rate computed by the Honma formula. =LIMIT Limit the flow rate using the flow rate computed by the Honma formula as the upper limit. Default value: OFF *If anything from OVERFLOW-HONMA to OVERFLOW-BACKSTEP is set to a value other than OFF, the *.ofl file is required for positioning. See 1.3.6.1 for details.
	OVERFLOW-AIDA	C	A flag used to determine whether the Aida formula should be applied when there's overflow at seawall =OFF Do not apply. =ON Fixed at the flow rate computed by the Aida formula. Default value: OFF *See 1.3.6.2 for details.

	OVERFLOW-BACKSTEP	C	A flag used to determine whether a drop formula should be applied when there's a drop at seawall =OFF Do not apply. =ON Fixed at the flow rate computed by a drop formula. Default value: OFF * See 1.3.6.2 for details.
	AIDA-LIMIT	R	Minimum value $h_{\mathrm{limit,aida}}$ [m], the downstream water level in the Aida formula Default value: 0.1 *See 1.3.6.2 for details.
	BREAK-WAVE	C	A flag used to determine whether a breaking wave model should be applied =OFF Do not apply =KENNEDY Apply Kennedy's breaking wave model. =IWASE Apply Iwase's breaking wave model. Default value: OFF *See 1.3.5 for details.
	KENNEDY-DELTA	R	Mixing length coefficient δ in Kennedy's breaking wave model (amplification coefficient) Default value: 6.5 *See 1.3.5.1 for details.
	KENNEDY-COEF1	R	Coefficient C_1 associated with transition time in Kennedy's model Default value: 8.0
	KENNEDY-COEF2	R	Coefficient C_2 of the breaking wave start condition expression in Kennedy's model Default value: 0.65
	KENNEDY-COEF3	R	Coefficient C_3 of the breaking wave end condition expression in Kennedy's model Default value: 0.08

	IWASE-BETA	R	Coefficient β associated with the kinematic eddy viscosity coefficient in Iwase's breaking wave model Default value: 0.37 *See 1.3.5.2 for details.
	DTSAFE-BREAK	R	Safety factor used to determine the upper limit of viscosity from stability conditions in the computation when viscosity increases in the breaking wave model Default value: 0.5 *See 1.3.5.3 for details.
	FALL-WATER-MODEL	C	A flag used to determine whether the Overfall model should be applied =OFF Do not apply. =ON Apply. Default value: OFF *See 1.3.8 for details.
	TYPE-FALL-WATER	C	Method for specifying that coefficients in the Overfall model =CONSTANT Constant value =FILE Load from a file (*.fwc) Default value: CONSTANT
	COEF-FALL-WATER	R	Loss coefficient C_{fw} of momentum in the Overfall model Default value: 1.0 *See 1.3.8.1 for details.
	DIST-NORM-FALL-WATER	R	Distribution coefficient $C_{fw,dist,norm}$ towards the normal direction when vertical momentum is distributed horizontally in the Overfall model Default value: 1.0 *See 1.3.8.3 for details.

	DIST-TANG-FALL-WATER	R	Distribution coefficient $C_{fw,dist,\tan g}$ towards the tangential direction when vertical momentum is distributed horizontally in the Overfall model Default value: 0.0
	DISPERSIVE-WAVE	C	A flag used to determine whether the dispersive wave model should be applied =OFF Do not apply. =ON Apply. *Allowed only for STOC-ML. See 1.3.4 for details.
	DISPBETA	R	Parameters for a dispersive wave model Default value: 0.06666666667 (=1/15) *Set to 0 for Boussinesq expressions, 1/15 for Madsen-Sørensen expressions, or -1/3 for explicit Boussinesq expressions.
	DISP-LIMIT	R	Lower limit $z_{disp,\lim}$ [m] of the z coordinate in which fixed boundary conditions are applied for landside boundary in the dispersive wave model Default value: -0.5 *See 1.3.4.3 for details.
	DISP-BC-LIMIT	R	Upper limit $z_{disp,BC,\lim}$ [m] of the z coordinate in which landside boundaries are not computed in the dispersive wave model Default value: -2.0 *See 1.3.4.3 for details.

	MANNING-FM	X	Roughness of manning and settings range (n, IS, IE, JS, JE) n: Roughness of manning IS, IE: Start and end positions of cell index I within which roughness of manning is set JS, JE: Start and end positions of cell index J within which roughness of manning is set Array size: 1 (only roughness specified) or 5 * The whole range applies only when roughness is specified. The value of the str file supersedes when loading a topography/geometry * file (.str).
	STOC-DS-MODE	C	A flag used to determine whether the destruction/blocking function should be applied =OFF Do not apply. =ON Apply. *Allowed only for STOC-ML.

	ISW	I	Processing flags ISW(1) : Not yet used ISW(2) : Not yet used ISW(3) : Surface velocity computation flag =1 Do not compute the velocity of a position including the water level. =0 Compute the velocity of a position including the water level. ISW(4) : Flag for an advective term in the momentum conservation formula =1 Do not compute the advective term in momentum conservation equation. =0 Compute the advective term in the momentum conservation equation. ISW(5) : Not yet used Default value: (0 0 1 0 0) Array size: 5 *The velocity computed in the lower cell is copied if ISW(3)=1. However, the velocity is computed regardless of ISW(3) when no velocity computation point is present in the lower cell.
PROPERTY	DENSITY	R	Density of sea water [kg/m^3] Default value: 1026.0
	VISCOSITY-H	R	Horizontal viscosity coefficient [Pa.s] *Set a coefficient of molecular viscosity (approx. 0.001) when using the turbulence model (LES, k-ε).
	VISCOSITY-V	R	Vertical viscosity coefficient [Pa.s] *Set a coefficient of molecular viscosity (approx. 0.001) when using the turbulence model (LES, k-ε).

BOUNDARY	OPEN-2D	C	Whether to load outer boundary conditions from a file or not =OFF Do not load. =ON Load Default value: OFF *Water level/velocity data at connection boundary (*.bci) can be loaded to set the water level and velocity at the boundary for STOC-ML only. Use BC-FILE in %OUTPUT block to output the data to a file.
	OPEN-SOMMER	I	Specify a transparent boundary condition at the outmost computation area periphery (JM, IM, IP, JP) JM: Southern boundary condition 0 to 3 IM: Western boundary condition 0 to 3 IP: Eastern boundary condition 0 to 3 JP: Northern boundary condition 0 to 3 Default value: (0 0 0 0) Array size: 4 *See 1.3.3 for details. Enter a value that is an integer ranging from 0 to 3 as follows: 0: No transparent boundary 1: Characteristic curve method 2: Imamura et al.'s (2001) method 3: Apply Imamura et al.'s (2001) method to the water level table and astronomical tide. If you specify 2, enter normal water level H0; if you specify 3, specify the tidal condition with a table or an astronomical tide.

	OVERLAP	I	Number of meshes in the range where the water level overlaps with the connection boundary between the parent area and your own area (JM IM IP JP) JM: Number of meshes where the level overlaps with the southern boundary IM: Number of meshes where the level overlaps with the western boundary IP: Number of meshes where the level overlaps with the eastern boundary JP: Number of meshes where the level overlaps with the northern boundary Default value: (0 0 0 0) Array size: 4 *When the proportion of the horizontal mesh size of the parent area to that of your own area is N, we recommend specifying N N N N.
	MODIFY-DEPTH	C	Whether to automatically modify topography that overlaps with a nesting boundary or not =OFF Do not automatically modify. =ON Automatically modify. Default value: ON *Automatic modification helps replace the depth value of overlapped topography in your own area with the depth value of that in the parent area. The replacement can result in an error. See 2.2.2.5 in that case.

	VERTICAL-PROFILE	C	Method for specifying the profile of normal direction components of velocity in the nesting boundary =NABOR　Distribute vertically =FLAT　Vertically-even Default value:　FLAT *For NABOR, align the distribution profile with the velocity distribution profile separated for the amount of NABOR found below.
	NABOR	I	Distance which references the vertical distribution profile of velocity (number of cells) Default value: 1 *Specify if VERTICAL-PROFILE=NABOR.
	NEST-TANGENTIAL-VELOCITY	C	Method for specifying the tangential direction components of velocity in the nesting boundary =PARENT　Fixed at the parent's velocity interpolated value =FREE　Use the velocity value of the adjacent cell. =HYBRID Specify PARENT condition for inflow and FREE condition for outflow. Default value: PARENT
	SURFACE-PRESSURE	T	Sea-level pressure [Pa] Default value: 0
	SURFACE-WIND-U	T	x direction component of surface wind speed [m/s] Default value: 0
	SURFACE-WIND-V	T	y direction component of surface wind speed [m/s] Default value: 0

	DEFAULT-TYPE-V	C	Default boundary condition of flow velocity =SLIP Slip condition =NO-SLIP Non-slip condition =SLIP-XY: Apply the slip condition on sides and non-slip condition on the bottom. *Apply DEFAULT-* setting to the surface that was not specified in INLET-I through WALL-K.
	U	T	x direction component of flow velocity [m/s]
	V	T	y direction component of flow velocity [m/s]
	W	T	z direction component of flow velocity [m/s]
	H	T	Water level [m]
	TYPE-V	C	Boundary condition of flow velocity =SLIP: Slip condition =NO-SLIP: Non-slip condition =WALL-FUNCTION: Wall function condition (STOC-IC only) =CONSTANT: Flow velocity fixing condition
	TYPE-H	C	Boundary condition of water level =FREE: Zero gradient condition =FIX: Constant value =TABLE: Supply in a time variation table
	INLET-I	I	Position of x-directional flow velocity fixed boundary surface (I, JS, JE, KS, KE) I: Grid point index I JS, JE: Start and end positions of cell index J KS, KE: Start and end positions of cell index K Array size: 5 *When the velocity fixed boundary surface is specified, the U, V, W, or H that was specified immediately before will be the fixed boundary condition at that position. (Specify the position in the same way as with PLATE-I in 2.2.2.4.)

	INLET-J	I	Position of y-directional flow velocity fixed boundary surface (J, IS, IE, KS, KE) J: Grid point index J IS, IE: Start and end positions of cell index I KS, KE: Start and end positions of cell index K Array size: 5 *Same as INLET-I
	INLET-K	I	Position of z-directional flow velocity fixed boundary surface (K, IS, IE, JS, JE) K: Grid point index K IS, IE: Start and end positions of cell index I JS, JE: Start and end positions of cell index J Array size: 5 *Same as INLET-I
	FREE-I	I	Position of x-directional free inflow/outflow boundary surface (I, JS, JE, KS, KE) I: Grid point index I JS, JE: Start and end positions of cell index J KS, KE: Start and end positions of cell index K Array size: 5 * If you specify the free inflow/outflow boundary surface, the TYPE-H (H) that was specified immediately before will be the water-level boundary condition at that position. Zero gradient condition is applied to the velocity.
	FREE-J	I	Position of y-directional free inflow/outflow boundary surface (J, IS, IE, KS, KE) J: Grid point index J IS, IE: Start and end positions of cell index I KS, KE: Start and end positions of cell index K Array size: 5 *Same as FREE-I

	FREE-K	I	Position of z-directional free inflow/outflow boundary surface (K, IS, IE, JS, JE) K: Grid point index K IS, IE: Start and end positions of cell index I JS, JE: Start and end positions of cell index J Array size: 5 *Same as FREE-I
	WALL-I	I	Position of x-directional wall surface boundary (I, JS, JE, KS, KE) I: Grid point index I JS, JE: Start and end positions of cell index J KS, KE: Start and end positions of cell index K Array size: 5 *If you specify the wall surface boundary, TYPE-V (V) that was specified immediately before will become the velocity boundary condition at that position. Zero gradient condition is applied to the water level. In addition, normal direction components of velocity will be fixed as 0.
	WALL-J	I	Position of y-directional wall surface boundary (J, IS, IE, KS, KE) J: Grid point index J IS, IE: Start and end positions of cell index I KS, KE: Start and end positions of cell index K Array size: 5 *Same as WALL-I
	WALL-K	I	Position of z-directional wall surface boundary (K, IS, IE, JS, JE) K: Grid point index K IS, IE: Start and end positions of cell index I JS, JE: Start and end positions of cell index J Array size: 5 *Same as WALL-I

	SEA-BOTTOM	C	A flag used to determine whether the water level fluctuation computation model should be used =OFF Do not use =ON Load the fluctuation amount from a file =CALC Compute the fluctuation amount with fault parameters Default value: OFF * See 1.3.9 for details. If you flag =ON, prepare an *.sbt file. If you flag =CALC, prepare a fault.txt and also specify COORDINATE through GRID-SYSTEM.
	COORDINATE	C	Specify a computation grid coordinate system =JAPAN-PLANE-RECTANGULAR: Japan Plane Rectangular coordinate system =UTM: UTM coordinate system =LONGITUDE-LATITUDE: Latitude/longitude coordinate system *Specify COORDINATE if SEA-BOTTOM=CALC.
	RECTANGULAR-ZONE	I	Area number of Japan Plane Rectangular coordinate system (1 to 19) *Specify RECTANGULAR-ZONE if COORDINATE= JAPAN-PLANE-RECTANGULAR.
	UTM-CENTER		Longitude of the central meridian in the UTM coordinate system [°] * Specify UTM-CENTER if COORDINATE= UTM.

	FAULT-SYSTEM	C	Geodetic system of fault parameters =TOKYO Old Tokyo Datum =JGD2000 World Datum 1 =WGS84 World Datum 2 Default value: JGD2000 *The difference between JGD2000 and WGS84 can be as small as 10 cm and can be ignored in the fault computation. Specify FAULT-SYSTEM if SEA-BOTTOM=CALC.
	GRID-SYSTEM	C	Geodetic system of computation grid =TOKYO Old Tokyo Datum =JGD2000 World Datum 1 =WGS84 World Datum 2 Default value: JGD2000 *The difference between JGD2000 and WGS84 can be as small as 10 cm and can be ignored in the fault computation. Specify GRID-SYSTEM if SEA-BOTTOM=CALC.
	WAVE	R	Wave-making condition with small amplitude wave (AMP, TTT, ALL, HHH, AXX) AMP: Half-amplitude [m] TTT: Frequency [s] ALL: Wavelength [m] HHH: Water depth [m] AXX: Mesh width [m] Array size: 5 *Available only for XY 2D computation

INITIAL	H	X	Initial water level fluctuation amount [m] and setting range (H, IS, IE, JS, JE) H: Water level fluctuation amount [m] IS, IE: Start and end positions of cell index I within which water level is set JS, JE: Start and end positions of cell index J within which water level is set Array size: 1 (only water level specified) or 5 Default value: 0 *The whole range applies when you have specified a water level. If you set the water level with the str file, the level will be updated for the amount specified with H.
	U	R	x direction component of initial flow velocity [m/s]
	V	R	y direction component of initial flow velocity [m/s]
	W	R	z direction component of initial flow velocity [m/s]
	K	R	Initial turbulence kinetic energy k [m^2/s^2]
	EP	R	Initial turbulence kinetic energy dissipation rate ε [m^2/s^3]
MATRIX	EPS	R	Convergence test condition with the absolute value of a matrix solver Default value: 1e-10 *The setting of the MATRIX block is for use with STOC-IC. EPS is not referenced when specified with STOC-ML.
	EPS-R	R	Convergence test condition with the relative value of a matrix solver Default value: 1e-10
	MAX- ITERATION	I	Maximum iteration count of a matrix solver Default value: 100

	PRINT	R	A flag used to determine whether convergence status of a matrix solver should be output =YES Output =NO Do not output Default value: NO
OUTPUT	RESTART-TIME	R	Time at which restart data is output [s] Array size: Number of output times *Specify either RESTART-TIME or RESTART-STEP.
	RESTART-STEP	I	Output step of restart data Array size: Specify for the amount of steps to output. *Specify either RESTART-STEP or RESTART-TIME.
	LIST-TYPE	C	Specify the output format of the computation result list. =ASCII: The list is output in ASCII format to an FT16_NN file. =BINARY: The list is output in binary format to an Area.lst file. Default value: ASCII
	LIST-TIME	R	Specify the time at which the computation result list is output [s] Array size: Number of output times When the array size is 3, set this variable to one of the following values: TS: Output start time [s] TE: Output end time [s] TI: Output interval [s] *Specify either LIST-TIME or LIST-STEP.
	LIST-STEP	I	Specify the output steps of the computation result list [s] Array size: Number of output times *Specify either LIST-STEP or LIST-TIME.

	LIST-PHYS	C	Name of the variable that is output to the computation result list Array size: Number of variables to output *Allowed variable names are as follows: H: Water level DEP: z coordinate value on the seafloor surface U,V,W, P: Velocity and pressure TMU,F: Turbulence kinematic viscosity, volume occupancy of fluid KF,KP: Free surface index KG: Seafloor surface index K,E: Turbulence kinetic energy k, dissipation rate ε
	LIST-SECT-I	I	X direction index on YZ surface to be listed Array size: Number of cross-section surfaces to output
	LIST-SECT-J	I	Y direction index on XZ surface to be listed Array size: Number of cross-section surfaces to output
	LIST-SECT-K	I	Z direction index on XY surface to be listed Array size: Number of cross-section surfaces to output
	HISTORY-TIME	R	Time interval at which time-series data is output [s]
	HISTORY-STEP	I	Step interval at which time-series data is output
	HISTORY-PHYS	C	Physical quantity output in the time-series data Array size: Number of variables to output *Allowed variable names are as follows: H: Water level DEP: z coordinate value on the seafloor surface U,V,W, P: Flow velocity and pressure TMU, F: Turbulence kinematic viscosity, volume occupancy of fluid K, E: Turbulence kinetic energy k, dissipation rate ε

	HISTORY-CELL	I	Index of the cell to be output to a time-series file (I, J, K) Array size: 3 *Keep entering HISTORY-CELL as many times as the number of points if more than one point should be output.
	END-TIME	R	Output time of end file (s) Array size: Number of output times *Specify either END-TIME or END-STEP.
	END-STEP	I	Output step of end file (s) Array size: Number of output steps *Specify either END-STEP or END-TIME.
	BC-FILE	R	Specify the time at which a boundary value file is output (TS, TE, TI) TS: Output start time [s] TE: Output end time [s] TI: Output interval [s] Array size: 3

2.2.2.4. Supplementary Explanation 1: Text-Entry Geometry

Consider the case where geometry shown in Figure 0-2-10 is entered using the SOLID variable of the %OBSTACLE block. In general, the index of a call located at the lower left of the computation area (I J K) is set to (1 1 1); however, STOC assumes an additional virtual cell at the periphery of the computation area, resulting in (2 2 2). Therefore, enter the four obstacles that touch the floor shown in Figure 0-2-10 as shown below. The same principle applies to the positioning of a porous structure.

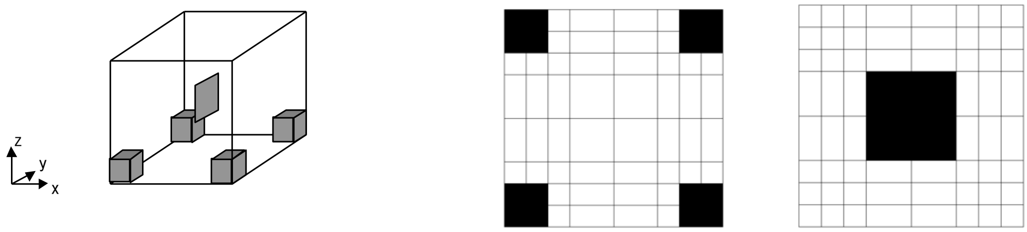
SOLID = (2 3 2 3 2 3) # (IS, IE, JS, JE, KS, KE)
SOLID = (8 9 2 3 2 3)
SOLID = (2 3 8 9 2 3)
SOLID = (8 9 8 9 2 3)

In addition, enter the flat structure around the center of the computation area as:

PLATE-I = (5 5 6 5 6) # (I, JS, JE, KS, KE)

where I specifies the index of a grid point. The grid point index is calculated as follows:

- Grid point index of a surface on a negative (-) cell–Value of the cell index minus 1
- Grid point index of a surface on a positive (+) cell–Equivalent to the cell index



Computation area k=2, 3 cross-sectional grid i=5 cross-sectional grid

Figure 0-2-10 Sample Geometry Entry

2.2.2.5. Supplementary Explanation 2: Error Due to Inconsistency of the Geometry in the Nesting Computation

If topographic/geometric data is not consistent between the parent and your own areas during the computation of a vertical 1 layer, (a) “your own area cells might not fall in the range of computation when the parent area cells are within the range” or (b) “vice versa” in the overlapped area. In the case of (a) or (b), the processing terminates with the following error message in FT16 file:

(a)

```
## CHILD-PARENT (LAND-SEA)POINT = ( i1 j1 )          # PARENT(I,J) = i2 j2
```

(b)

```
## CHILD-PARENT (SEA-LAND)POINT = ( i1 j1 )          # PARENT(I,J) = i2 j2
```

In the case of (a) or (b), we generally recommended correcting the topographic/geometric data. However, for simplified correction, use (i1 j1) from the error message to add the following in your own area:

(a): SEA-FLAG = (i1 j1)

(b): SOLID = (i1 i1 j1 j1 2 2)

This is a possible workaround for the error.

2.2.2.6. Supplementary Explanation 3: Real Value or Time-series Table Entries

Some entry items related to boundary conditions allow a value to be specified with one real value or time-series table. For example, when the water level is entered as variable H, as far as there is no time variation detected, set one real value as follows.

H = 0.5

On the other hand, for example, when you vary the water level with time as shown in Figure 0-2-11, enter array pairs for the time [s] and water level [m] values alternately as shown below. The array sizes should amount to even numbers.

```
H=(
     0.0   0.0
   100.0   0.5
   200.0   0.5
   300.0   0.0
  9999.0   0.0
  )
```

In this example, a linefeed is inserted after every pair for clarity, but it works without the linefeed as follows:

H=(0.0 0.0 100.0 0.5 200.0 0.5 300.0 0.0 9999.0 0.0)

The 9999.0 at the end means that water level 0 should be retained and kept after 300 seconds or more. In the time-series table entry, to show a cyclic change, revert time to 0 seconds after the final data exceeds the time defined. That is, if there is no pair 9999.0 in the above example, the function looks like the one shown in Figure 0-2-12.

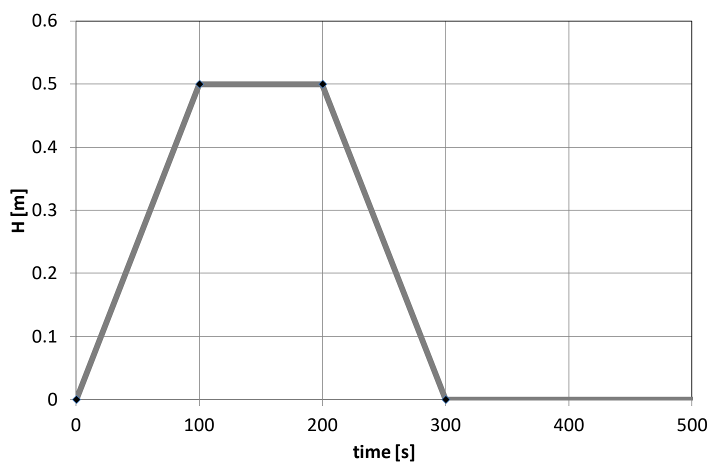


Figure 0-2-11 Water Level Fluctuations over Time

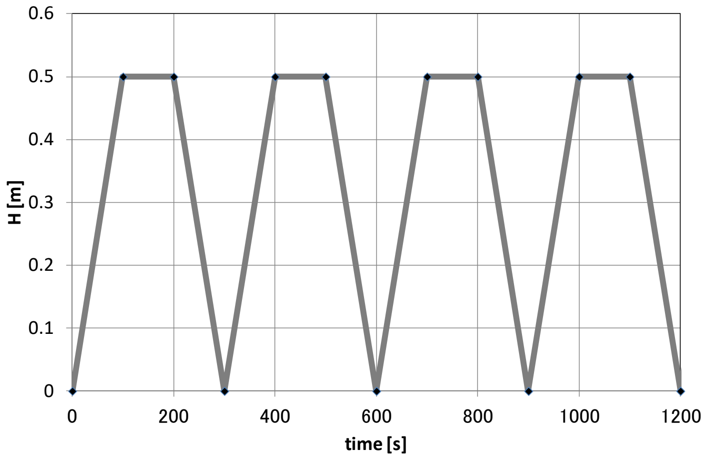


Figure 0-2-12 Water Level Fluctuations over Time (Cyclic Change)

2.2.3. Input Data Format 3: Topography/Geometry Data (Area.str File)

The Area.str file can be read by specifying the following in the %OBSTACLE block of the analysis condition file (Area):

FILE= YES

The Area.str file defines the porosity, permeability of cell interface, water depth, water level, roughness, etc., as FORTRAN unformatted data. Figure 0-2-13 shows specific read processing. エラー！参照元が見つかりません。 shows the meanings of the variables.

For the definitions of the porous values (GV, GX, GY, and GZ), refer to Figure 0-1-4. Set GV for the cells (solid-filled in Figure 0-1-4 エラー！参照元が見つかりません。) to 1. Also, set GX, GY, and GZ for the boundary of these cells to 1. Note that if HDEP and GV are not consistent, the computation results will be abnormal.

Table 0-2-3 Topography/Geometry Data (Content of Area.str File)

Variable	Type	Meaning
MX	I	Number of cells arranged in the X direction in the analysis area (actual cell count NX + 2). *The virtual cells for each layer on both sides are considered.
MY	I	Number of cells arranged in the Y direction in the analysis area (actual cell count NY + 2).
MZ	I	Number of cells arranged in the Z direction in the analysis area (actual cell count NZ + 2).
INDC	I	Flag indicating whether the cell is a computational cell =1 Computational cell =0 Non-computational cell
GV	R*8	Porosity for each cell
GX	R*8	X-directional permeability
GY	R*8	Y-directional permeability
GZ	R*8	Z-directional permeability
HDEP	R*8	z coordinate value on the seafloor surface [m]
ZZ	R*8	Water level [m]
AMNG	R*8	Roughness of manning [$m^{-1/3}s$]
The three records below are only used when the DS model is ON.		
IDST	I	Flag for destruction processing >0 Wooden building present

		=0 No building present =-1 Non-wooden building present
HTDST1	R*8	Height of the ground after destruction of the building (HTDST2<=HTDST1<HDEP)
HTDST2	R*8	Height of the ground after complete removal of the building (HTDST2<HDEP)

```
OPEN(istr,FILE='aXX.str',FORM='UNFORMATTED')
READ(istr) NX,NY,NZ
MX=NX+2
MY=NY+2
MZ=NZ+2
READ (istr) (((INDC(I,J,K),I=2,MX-1),J=2,MY-1),K=2,MZ-1)
READ (istr) (((GV(I,J,K),   I=2,MX-1),J=2,MY-1),K=2,MZ-1)
READ (istr) (((GX(I,J,K),   I=1,MX-1),J=2,MY-1),K=2,MZ-1)
READ (istr) (((GY(I,J,K),   I=2,MX-1),J=1,MY-1),K=2,MZ-1)
READ (istr) (((GZ(I,J,K),   I=2,MX-1),J=2,MY-1),K=1,MZ-1)
READ (istr)   ((HDEP (I,J),     I=2,MX-1),J=2,MY-1)
READ (istr)   ((HH(I,J),        I=2,MX-1),J=2,MY-1)
READ (istr)   ((AMNG(I,J),      I=2,MX-1),J=2,MY-1)
CLOSE(istr)
```

Figure 0-2-13 Read Processing of Topography/Geometry Data (Area.str File)

2.2.4. Input Data Format 4: Water Level Fluctuation Amount Data per Hour by Seismic Action (Area.sbt File)

The Area.sbt file is available in two formats: Format 1 and Format 2. When reading the first line, STOC automatically determines whether this file is in Format 1 or Format 2.

(1) Format 1

The file in Format 1 is read in FORTRAN free format as an ASCII file. Figure 0-2-14 shows the format for file read. Figure 0-2-15 shows a sample of water level fluctuation amount data (Area.sbt). Note that for the Y direction, indexes are read in ascending order (starting on the south side).

(2) Format 2

Like the file in Format 1, the file in Format 2 is read in FORTRAN free format as an ASCII file. As Figure 0-2-16 shows, portion (2) of Format 1 is extracted to a separate file, and the Area.sbt file includes only portion (1) and the name of the separate file. The contents of the separate file are identical to portion (2) shown in Figure 0-2-16.

```
        OPEN(isb,FILE='aXX.sbt,FORM='FORMATTED')
        N=0
        DO
          READ(IFLSB,*,END=900) TIMESB(N),IS,IE,JS,JE                    ! (1)
          N=N+1
          READ(IFLSB,*,END=900) ((DELH(I,J,N),I=IS,IE),J=JS,JE)          ! (2)
        ENDDO

    900 CONTINUE
```

Figure 0-2-14 Read Processing of an SBT File

*TIMESB: Time [s]

IS: Index (= 2) of the start position for the X direction

IE: Index (= computation cell count for the X direction + 1) of the end position for the X direction

JS: Index (= 2) of the start position for the Y direction

JE: Index (= computation cell count for the Y direction + 1) of the end position for the Y direction

DELH: Water fluctuation amount [m]

(1)

(2)

```
      6        2      351        2      351
-0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.0
-0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.0
-0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.0
-0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.0
-0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.0
-0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.0
-0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.0
-0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.0
-0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.0
-0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.0
-0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.0
-0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.0
-0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.0
-0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.0
-0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.0
-0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.0
-0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.0
-0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.0
-0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.0
-0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.0
-0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.0
-0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.0
-0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.0
-0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.0
-0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.0
-0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.0
-0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.0
-0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.0
-0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.0
-0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.0
-0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.0
-0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.006   -0.0
```

Figure 0-2-15 Sample SBT File (Format 1)

(1) Added portion (file name)

6	2	257	2	206	‘sbtsub/a01.sbt_0006’
10	2	257	2	206	‘sbtsub/a01.sbt_0010’
12	2	257	2	206	‘sbtsub/a01.sbt_0012’
14	2	257	2	206	‘sbtsub/a01.sbt_0014’
16	2	257	2	206	‘sbtsub/a01.sbt_0016’
20	2	257	2	206	‘sbtsub/a01.sbt_0020’
24	2	257	2	206	‘sbtsub/a01.sbt_0024’
30	2	257	2	206	‘sbtsub/a01.sbt_0030’
32	2	257	2	206	‘sbtsub/a01.sbt_0032’
36	2	257	2	206	‘sbtsub/a01.sbt_0036’
38	2	257	2	206	‘sbtsub/a01.sbt_0038’
42	2	257	2	206	‘sbtsub/a01.sbt_0042’
44	2	257	2	206	‘sbtsub/a01.sbt_0044’
50	2	257	2	206	‘sbtsub/a01.sbt_0050’
54	2	257	2	206	‘sbtsub/a01.sbt_0054’
58	2	257	2	206	‘sbtsub/a01.sbt_0058’
60	2	257	2	206	‘sbtsub/a01.sbt_0060’
62	2	257	2	206	‘sbtsub/a01.sbt_0062’
66	2	257	2	206	‘sbtsub/a01.sbt_0066’
68	2	257	2	206	‘sbtsub/a01.sbt_0068’

Figure 0-2-16 Sample SBT File (Format 2)

2.2.5. Input Data Format 5: Seismic Fault Parameters (fault.txt File)

The fault.txt file contains parameters concerning seismic faults. Unlike the Area.sbt file, you do not need to prepare a fault.txt file for each area.

This file is in FORTRAN free format. One fault parameter is defined on each line. Figure 0-2-17 shows a sample fault.txt file. There are 10 parameters on each line. You can find parameters (a) through (j) shown below from left to right. Note that, sometimes, publicly available fault parameters indicate the length, width, and depth in kilometers, but the fault parameters in the fault.txt file indicate values in meters.

(a) Fault length [m]
(b) Fault width [m]
(c) Fault depth [m]
(d) Strike [°]
(e) Dip [°]
(f) Rake [°]
(g) Slip [m]
(h) Latitude [°]
(i) Longitude [°]
(j) Fault destruction time [s] *Time elapsed since the start of computation

Note that the fault parameters do not need to be arranged in order of fault destruction time (j), and there may be multiple fault parameters determined at the same time.

```
1500.0  1500.0  16000.0  190.0  20.0  80.0 14.8  33.64823  136.15535  30.0
1200.0  1200.0  20000.0  191.0  20.0  80.0 12.0  33.82355  136.06754  50.0
1200.0  1200.0  24000.0  192.0  20.0  80.0 11.1  33.90650  136.02585  70.0
1200.0  1200.0  27000.0  193.0  20.0  80.0  8.8  33.98531  135.98616  90.0
1200.0  1200.0  29000.0  194.0  20.0  80.0  7.7  34.02163  135.96784  90.0
1200.0  1200.0  30000.0  195.0  20.0  80.0  7.0  34.04402  135.95654 110.0
1200.0  1200.0  30000.0  196.0  20.0  80.0  6.7  34.05160  135.95272 110.0
1200.0  1200.0  30000.0  197.0  20.0  80.0  6.0  34.07471  135.94104 270.0
1200.0  1200.0  31000.0  198.0  20.0  80.0  5.7  34.08255  135.93708 270.0
1200.0  1200.0  31000.0  199.0  20.0  80.0  5.5  34.09045  135.93309 270.0
```

Figure 0-2-17 Sample fault.txt File

2.2.6. Input Data Format 6: Flow Velocity/Water Level Time-series Input Data (Area.tim File)

When time-series data is input into the analysis condition file (Area) as shown in 2.2.2.6, the time-series table can be extracted to a separate file.

As an example, suppose that the analysis condition file (Area) contains the following data:

```
H=(
     0.0   0.0
   100.0   0.5
   200.0   0.5
   300.0   0.0
  9999.0   0.0
  )
 FREE-J=(1 2 51 2 2 )
H=(
     0.0   0.0
   100.0   0.6
   200.0   0.6
   300.0   0.0
  9999.0   0.0
  )
 FREE-J=(1 52 101 2 2 )
```

By using the Area.tim file, you can change the above contents to the following:

Contents of the analysis condition file (Area):

```
 H=FILE
 FREE-J=(1 2 51 2 2 )
 H=FILE
 FREE-J=(1 52 101 2 2 )
```

Contents of the Area.tim file:

```
H=(
     0.0   0.0
   100.0   0.5
   200.0   0.5
   300.0   0.0
  9999.0   0.0
  )
H=(
     0.0   0.0
```

```
  100.0    0.6
  200.0    0.6
  300.0    0.0
 9999.0    0.0
 )
```

The data in the Area.tim file is arranged in the same order as the FILE in the analysis condition file (Area) is referenced.

You can input only U, V, W, and H for the %BOUNDARY block when using the Area.tim file.

2.2.7. Input Data Format 7: Permeable Structure Data (Area.dpr File)

The Area.dpr file is read by specifying D-FILE=YES in the %OBSTACLE block of the analysis condition file (Area). Figure 0-2-18 shows specific read processing. Table 0-2-4 shows the meanings of the variables (which are the same as those specified with D-POROUS in the %OBSTACLE block of the analysis condition file (Area)).

Note that variables GVD through GZD for the position where a permeable structure is not specified should be 1.

Table 0-2-4 Permeable Structure Data (Content of the Area.dpr File)

Variable	Type	Meaning
GVD	R*8	Porosity for the permeable structure
GXD	R*8	X-directional permeability of the permeable structure
GYD	R*8	Y-directional permeability of the permeable structure
GZD	R*8	Z-directional permeability of the permeable structure
CM	R*8	Inertia force coefficient
CD	R*8	Drag coefficient
ALPHA	R*8	Coefficient α used for Dupuit-Forchheimer model
BETA	R*8	Coefficient β used for Dupuit-Forchheimer model
DIAM	R*8	Characteristic particle diameter used for Dupuit-Forchheimer model

```
OPEN(idpr,FILE='aXX.dpr',FORM='UNFORMATTED')
READ (idpr) (((GVD(I,J,K),   I=2,MX-1),J=2,MY-1),K=2,MZ-1)
READ (idpr) (((GXD(I,J,K),   I=1,MX-1),J=2,MY-1),K=2,MZ-1)
READ (idpr) (((GYD(I,J,K),   I=2,MX-1),J=1,MY-1),K=2,MZ-1)
READ (idpr) (((GZD(I,J,K),   I=2,MX-1),J=2,MY-1),K=1,MZ-1)
READ (idpr) (((CM (I,J,K),   I=2,MX-1),J=2,MY-1),K=2,MZ-1)
READ (idpr) (((CD (I,J,K),   I=2,MX-1),J=2,MY-1),K=2,MZ-1)
READ (idpr) (((ALPHA(I,J,K),   I=2,MX-1),J=2,MY-1),K=2,MZ-1)
READ (idpr) (((BETA(I,J,K),   I=2,MX-1),J=2,MY-1),K=2,MZ-1)
READ (idpr) (((DIAM(I,J,K),   I=2,MX-1),J=2,MY-1),K=2,MZ-1)
CLOSE(idpr)
```

Figure 0-2-18 Read Processing of Permeable Structure Data (Area.dpr File)

2.2.8. Input Data Format 8: Data for Specifying a Location on which to Apply the Overflow Model (Area.ofl File)

The Area.ofl file should be a text file in FORTRAN free format. The syntax for this format is as follows.

Syntax: Cell index I   Cell index J   Flag

Note that the first line in the file is ignored. Cell indexes I and J should consider a virtual cell's one layer like those for specifying the time-series output location. The flag has one of these values:

1 …..   Honma formula is applied to the east side of a cell.
2 …..   Honma formula is applied to the north side of a cell.
3 …..   Aida formula and a drop formula are applied to the east side of a cell.
4 …..   Aida formula and a drop formula are applied to the north side of a cell.
Other values ….. Nothing is done.

Figure 0-2-19 shows sample data. Note that the flag is ignored if input OVERFLOW-HONMA, OVERFLOW-AIDA, or OVERFLOW-BACKSTEP in the %MODEL block of the analysis condition file (Area) is set to OFF.

Ensure consistency with the topography/geometry data (Area.str) to satisfy the following conditions:

① $\gamma_{x,i\square 1/2} \square \gamma_{v,i}$ and $\gamma_{x,i\square 1/2} \square \gamma_{v,i\square 1}$ should be established for the location to which Honma formula is applied.

② $\gamma_{x,i\square 1/2} \square \gamma_{v,i} \square \gamma_{v,i\square 1}$ or $\gamma_{x,i\square 1\backslash 2} \square \gamma_{v,i\square 1} \square \gamma_{v,i}$ should be established for the location to which Aida formula and a drop formula are applied.

# I	J	FLAG
3	3	2
4	3	2
2	4	1
5	5	4
6	4	4
5	5	3
4	6	3

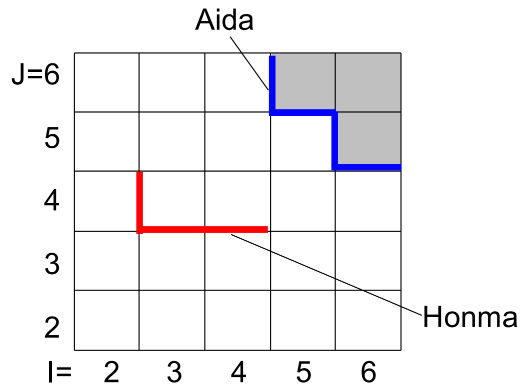


Figure 0-2-19 Sample Data for Specifying a Location on which to Apply Overflow Model (Area.ofl File)

2.2.9. Input Data Format 9: Coefficient Data for Overfall Model (Area.fwc File)

The Area.fwc file is read by specifying TYPE-FALL-WATER=FILE in the %MODEL block of the analysis condition file (Area). A different coefficient can be set for each location when coefficients are read from the file.

The Area.fwc file should be a text file in FORTRAN free format. Figure 0-2-20 shows specific read processing. Table 0-2-5 shows the meanings of the variables. The variables with X attached are defined (in the same position as U) for the cell interface in the X direction. Those with Y attached are defined (in the same position as V) for the cell interface in the Y direction.

Table 0-2-5 Coefficient Data for Overfall Model (Content of Area.fwc File)

Variable	Type	Meaning
CX,CY	R*8	Loss coefficient C_{fw} of momentum in the overfall model
DNX,DNY	R*8	Distribution coefficient $C_{fw,dist,norm}$ towards the normal direction when vertical momentum is distributed horizontally in the overfall model
DTX,DTY	R*8	Distribution coefficient $C_{fw,dist,\mathrm{tang}}$ towards the tangential direction when vertical momentum is distributed horizontally in the overfall model

```
OPEN(ifwc,FILE='aXX.fwc',FORM='FORMATTED')
READ (ifwc,*) ! Comment line
READ (ifwc,*) ((CX(I,J,K),   I=1,MX-1),J=2,MY-1)
READ (ifwc,*) ! Comment line
READ (ifwc,*) ((DNX(I,J,K),   I=1,MX-1),J=2,MY-1)
READ (ifwc,*) ! Comment line
READ (ifwc,*) ((DTX(I,J,K),   I=1,MX-1),J=2,MY-1)
READ (ifwc,*) ! Comment line
READ (ifwc,*) ((CY(I,J,K),   I=2,MX-1),J=1,MY-1)
READ (ifwc,*) ! Comment line
READ (ifwc,*) ((DNX(I,J,K),   I=2,MX-1),J=1,MY-1)
READ (ifwc,*) ! Comment line
READ (ifwc,*) ((DNY(I,J,K),   I=2,MX-1),J=1,MY-1)
CLOSE(ifwc)
```

Figure 0-2-20 Read Processing of Coefficient Data for Overfall Model (Area.fwc File)

2.2.10. Output Data Format 1: Computation Intermediate Information (FT16_NN File)

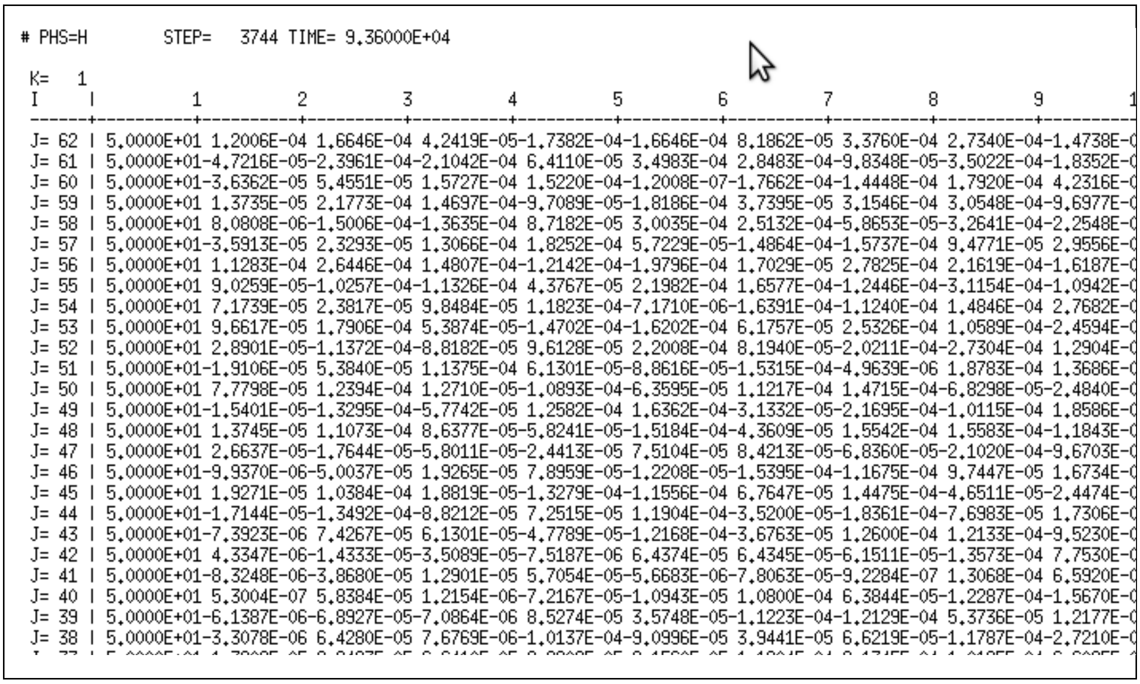
The FT16_NN file constitutes a standard output produced in ASCII text format by STOC. It contains the data shown below (NN is a processor number, and FT16_00 is output by computation that involves only one processor).

- Input data block name
- Error message
- Computation intermediate information (time step and maximum flow velocity)
- CPU time
- Data specified in LIST-* in the %OUTPUT block of the analysis condition file (Area)
- Others

Figure 0-2-21 shows sample output that indicates computation intermediate information and CPU time. Figure 0-2-22 shows sample output that indicates data specified in LIST-*.

```
(skip)
 +--------------------------------------+
 |   START TIME INTEGRATION (Leap frog)   |
 +--------------------------------------+
STEP=         1   TIME= 1.00000D+00   DT= 1.00D+00   MAX-VELOCITY= 3.4220191153512D-03
STEP=         2   TIME= 2.00000D+00   DT= 1.00D+00   MAX-VELOCITY= 6.8437008910172D-03
STEP=         3   TIME= 3.00000D+00   DT= 1.00D+00   MAX-VELOCITY= 1.0264927279857D-02
(skip)
STEP=   21599   TIME= 2.15990D+04   DT= 1.00D+00   MAX-VELOCITY= 6.8400687491982D-01
STEP=   21600   TIME= 2.16000D+04   DT= 1.00D+00   MAX-VELOCITY= 6.8231788477961D-01
 CLOSE HISTORY FILE
 OPEN TSUNAMI-RESULT FILE
 FILE NAME    =a01.end
 CLOSE TSUNAMI-RESULT FILE
+----------------+
 |    NORMAL END    |
 +----------------+
#### CPU INFORMATION (SEC) ####
      TOTAL                    = 3.61335D+04
         PRE                   = 2.41602D+01
(skip)
         SOLVER                = 3.61086D+04
         <TIME ZERO>           = 2.51211D+01
         <TIME LOOP>           = 3.60942D+04
            SETTBL             = 4.29688D-02
(skip)
```

Figure 0-2-21 Sample 1 of Computation Intermediate Information (FT16_NN File)



```
# PHS=H        STEP=   3744 TIME= 9.36000E+04

 K=   1
 I     |          1          2          3          4          5          6          7          8          9          1
 ------+----------+----------+----------+----------+----------+----------+----------+----------+----------+----------
 J= 62 | 5.0000E+01 1.2006E-04 1.6646E-04 4.2419E-05-1.7382E-04-1.6646E-04 8.1862E-05 3.3760E-04 2.7340E-04-1.4738E-
 J= 61 | 5.0000E+01-4.7216E-05-2.3961E-04-2.1042E-04 6.4110E-05 3.4983E-04 2.8483E-04-9.8348E-05-3.5022E-04-1.8352E-
 J= 60 | 5.0000E+01-3.6362E-05 5.4551E-05 1.5727E-04 1.5220E-04-1.2008E-07-1.7662E-04-1.4448E-04 1.7920E-04 4.2316E-
 J= 59 | 5.0000E+01 1.3735E-05 2.1773E-04 1.4697E-04-9.7089E-05-1.8186E-04 3.7395E-05 3.1546E-04 3.0548E-04-9.6977E-
 J= 58 | 5.0000E+01 8.0808E-06-1.5006E-04-1.3635E-04 8.7182E-05 3.0035E-04 2.5132E-04-5.8653E-05-3.2641E-04-2.2548E-
 J= 57 | 5.0000E+01-3.5913E-05 2.3293E-05 1.3066E-04 1.8252E-04 5.7229E-05-1.4864E-04-1.5737E-04 9.4771E-05 2.9556E-
 J= 56 | 5.0000E+01 1.1283E-04 2.6446E-04 1.4807E-04-1.2142E-04-1.9796E-04 1.7029E-05 2.7825E-04 2.1619E-04-1.6187E-
 J= 55 | 5.0000E+01 9.0259E-05-1.0257E-04-1.1326E-04 4.3767E-05 2.1982E-04 1.6577E-04-1.2446E-04-3.1154E-04-1.0942E-
 J= 54 | 5.0000E+01 7.1739E-05 2.3817E-05 9.8484E-05 1.1823E-04-7.1710E-06-1.6391E-04-1.1240E-04 1.4846E-04 2.7682E-
 J= 53 | 5.0000E+01 9.6617E-05 1.7906E-04 5.3874E-05-1.4702E-04-1.6202E-04 6.1757E-05 2.5326E-04 1.0589E-04-2.4594E-
 J= 52 | 5.0000E+01 2.8901E-05-1.1372E-04-8.8182E-05 9.6128E-05 2.2008E-04 8.1940E-05-2.0211E-04-2.7304E-04 1.2904E-
 J= 51 | 5.0000E+01-1.9106E-05 5.3840E-05 1.1375E-04 6.1301E-05-8.8616E-05-1.5315E-04-4.9639E-06 1.8783E-04 1.3686E-
 J= 50 | 5.0000E+01 7.7798E-05 1.2394E-04 1.2710E-05-1.0893E-04-6.3595E-05 1.1217E-04 1.4715E-04-6.8298E-05-2.4840E-
 J= 49 | 5.0000E+01-1.5401E-05-1.3295E-04-5.7742E-05 1.2582E-04 1.6362E-04-3.1332E-05-2.1695E-04-1.0115E-04 1.8586E-
 J= 48 | 5.0000E+01 1.3745E-05 1.1073E-04 8.6377E-05-5.8241E-05-1.5184E-04-4.3609E-05 1.5542E-04 1.5583E-04-1.1843E-
 J= 47 | 5.0000E+01 2.6637E-05-1.7644E-05-5.8011E-05-2.4413E-05 7.5104E-05 8.4213E-05-6.8360E-05-2.1020E-04-9.6703E-
 J= 46 | 5.0000E+01-9.9370E-06-5.0037E-05 1.9265E-05 7.8959E-05-1.2208E-05-1.5395E-04-1.1675E-04 9.7447E-05 1.6734E-
 J= 45 | 5.0000E+01 1.9271E-05 1.0384E-04 1.8819E-05-1.3279E-04-1.1556E-04 6.7647E-05 1.4475E-04-4.6511E-05-2.4474E-
 J= 44 | 5.0000E+01-1.7144E-05-1.3492E-04-8.8212E-05 7.2515E-05 1.1904E-04-3.5200E-05-1.8361E-04-7.6983E-05 1.7306E-
 J= 43 | 5.0000E+01-7.3923E-06 7.4267E-05 6.1301E-05-4.7789E-05-1.2168E-04-3.6763E-05 1.2600E-04 1.2133E-04-9.5230E-
 J= 42 | 5.0000E+01 4.3347E-06-1.4333E-05-3.5089E-05-7.5187E-06 6.4374E-05 6.4345E-05-6.1511E-05-1.3573E-04 7.7530E-
 J= 41 | 5.0000E+01-8.3248E-06-3.8680E-05 1.2901E-05 5.7054E-05-5.6683E-06-7.8063E-05-9.2284E-07 1.3068E-04 6.5920E-
 J= 40 | 5.0000E+01 5.3004E-07 5.8384E-05 1.2154E-06-7.2167E-05-1.0943E-05 1.0800E-04 6.3844E-05-1.2287E-04-1.5670E-
 J= 39 | 5.0000E+01-6.1387E-06-6.8927E-05-7.0864E-06 8.5274E-05 3.5748E-05-1.1223E-04-1.2129E-04 5.3736E-05 1.2177E-
 J= 38 | 5.0000E+01-3.3078E-06 6.4280E-05 7.6769E-06-1.0137E-04-9.0996E-05 3.9441E-05 6.6219E-05-1.1787E-04-2.7210E-
```

Figure 0-2-22 Sample 2 of Computation Intermediate Information (FT16_NN File)

2.2.11. Output Data Format 2: Spatial Distribution Data of Water Level/Flow Velocity (Area.lst File)

Data specified in LIST-* in the %OUTPUT block of the analysis condition file (Area) is usually output as ASCII text to the FT16_NN file. If LIST-TYPE=BINARY is specified, only the water level and flow velocity are output in FORTRAN UNFORMATTED format to the Area.list file. At this time, the cross-sectional position (specified in LIST-SECT-*) is ignored, and all the 3-dimensional variables are output. Note that if the area is split into partitions by using the I-DIV and J-DIV specifications, then Area.lst data is output to a different file for each partition, and each file is named Area_MM.lst (MM is the number for the partition).

Figure 0-2-23 shows specific output processing. Table 0-2-6 shows the meanings of the variables. Note that when variables of real number type are output to an Area_MM.lst file, they are converted into single-precision real numbers to reduce the file size. (To output double-precision real numbers to an Area_MM.lst file, change REAL (PHYS) in output.f,dbwr2d.f to PHYS.)

Output data, such as pressure, can be in the ranges of 2<=I<=MX-1, 2<=J<=MY-1, and 2<=K<=MZ-1. However, because there are variables defined as staggered variables, such as U, V, and W, output data should be in the ranges of 1<=I<=MX-1, 1<=J<=MY-1, and 1<=K<=MZ-1.

Table 0-2-6 Spatial Distribution Data of Water Level/Flow Velocity (Content of Area.lst File)

Variable	Type	Meaning
MX	I*4	Number of cells arranged in the X direction in the analysis area (actual cell count NX + 2). *The virtual cells for each layer on both sides are considered.
MY	I	Number of cells arranged in the Y direction in the analysis area (actual cell count NY + 2).
MZ	I	Number of cells arranged in the Z direction in the analysis area (actual cell count NZ + 2).
XX	R*4	X-directional grid point coordinate value
YY	R*4	Y-directional grid point coordinate value
ZZ	R*4	Z-directional grid point coordinate value
HDEP	R*4	z coordinate value on the seafloor surface [m]
CLIST	C*8	Name of variable
TIME	R*4	Time
ISTEP	I	Number of steps
PHYS	R*4	Output physical quantities such as: Water level for CLIST=H X-directional component of flow velocity [m/s] for CLIST=U

		Y-directional component of flow velocity [m/s] for CLIST=V

```
        OPEN(ilst,FILE='aXX.lst',FORM='UNFORMATTED')
! Header : 8 records (Axis of grid points & -water depth)
        WRITE(ilst)   MX-1
        WRITE (ilst) (XX (I), I=1,MX-1)
        WRITE(ilst)   MY-1
        WRITE (ilst) (YY (J), J=1,MY-1)
        WRITE(ilst)   MZ-1
        WRITE (ilst) (ZZ (K), K=1,MZ-1)
        WRITE(ilst)   MX-1, MY-1, 1
        WRITE (ilst) ((HDEP (I,J), I=1,MX-1),J=1,MY-1)

! Results
        DO N=1,NTIME    ! each time
        DO M=1,NPHYS    ! each physical value

        WRITE(ilst) CLIST(M),ISTEP,TIME
        WRITE(ilst) MX-1,MY-1,MZ-1              ! size
        WRITE (ilst) (((PHYS(I,J,K), I=1,MX-1) ,J=1,MY-1) ,K=1,MZ-1)

        ENDDO
        ENDDO

        CLOSE(ilst)
```

Figure 0-2-23 Output Processing of Topography/Geometry Data (Area.lst File)

2.2.12. Output Data Format 3: Time-series Output Data (Area.hst File)

Time-series output data (Area.hst), specified with HISTORY-* in the %OUTPUT block of the analysis condition file, is output as ASCII text data at fixed intervals. Figure 0-2-24 shows sample output data. The first comment lines (starting with #) specify the physical-quantity names and positions of columns. Graphs are created using external programs such as MS-EXCEL and gnuplot.

```
# START IMPORT AT ROW   17
# COLUMN: VARIABLE     I     J     K
#      1: H           26    26     2
#      2: H           26    26     3
#      3: H           26    77     2
#      4: H           26    77     3
#      5: H           77    26     2
#      6: H           77    77     2
#      7: U           26    26     2
#      8: U           26    26     3
#      9: U           26    77     2
#     10: U           26    77     3
#     11: U           77    26     2
#     12: U           77    77     2
#
#        TIME            1            2            3            4            5            6            7
  0.00000E+00  0.00000E+00  0.00000E+00  0.00000E+00  0.00000E+00  0.00000E+00  0.00000E+00  0.00000E+00  0.
  1.50000E+00  0.00000E+00  0.00000E+00  0.00000E+00  0.00000E+00  0.00000E+00  0.00000E+00  0.00000E+00  0.
  3.00000E+00  3.00000E-03  3.00000E-03  3.00000E-03  3.00000E-03  2.00000E-03  2.00000E-03  0.00000E+00  0.

(skip)

  5.39850E+03  1.70517E-02  1.70517E-02  3.51181E-02  3.51181E-02 -1.25326E-01 -8.08674E-02  1.05421E-02  1.
  5.40000E+03  1.88987E-02  1.88987E-02  3.63539E-02  3.63539E-02 -1.26607E-01 -8.26286E-02  1.04145E-02  1.
  5.40150E+03  2.07083E-02  2.07083E-02  3.74072E-02  3.74072E-02 -1.27944E-01 -8.43674E-02  1.02803E-02  1.
```

Figure 0-2-24 Sample Time-series Output Data (Area.hst File)

2.2.13. Output Data Format 4: Aggregate Data that Includes the Maximum Water Level, Etc. (Area.end File)

Aggregate data that includes the maximum water level, etc. (Area.end) is always output as ASCII text data upon completion of computation. Figure 0-2-25 shows specific output processing. Table 0-2-7 shows the meanings of the variables. Figure 0-2-26 shows sample output data.

Note that the first-wave arrival time shown in Table 0-2-7 indicates the time when the difference between the initial water level and a wave-driven water level exceeds threshold EPST (which is 0.01m by default).

Table 0-2-7 Aggregate Data that Includes the Maximum Water Level, Etc. (Content of Area.end File)

Variable	Type	Meaning
MX	I*4	Number of cells arranged in the X direction in the analysis area (actual cell count NX + 2). *The virtual cells for each layer on both sides are considered.
MY	I	Number of cells arranged in the Y direction in the analysis area (actual cell count NY + 2).
MZ	I	Number of cells arranged in the Z direction in the analysis area (actual cell count NZ + 2).
COMMENT	C	Output data is shown below. MAX FREE-SURFACE VALUE: Max. water level for each cell MAX FREE-SURFACE TIME: Time when the water level reaches its max. for each cell MIN FREE-SURFACE VALUE: Min. water level for each cell MIN FREE-SURFACE TIME: Time when the water level reaches its min. for each cell TSUNAMI TIME: Time when the first wave arrives for each cell MAX VELOCITY VALUE: Max. flow velocity for each cell MAX VELOCITY TIME: Time when the flow velocity reaches its max. for each cell
PHYS	R*8	Output physical quantity

```
        OPEN(iend,FILE='aXX.end',FORM='FORMATTED')

        DO N=1,NPHYS
        WRITE(iend,*)    '#',COMMENT
        WRITE(iend,610) ((PHYS(I,J),I=1,MX),J=1,MY)
610     FORMAT(1P,10E10.3)
        ENDDO

        CLOSE(iend)
```

Figure 0-2-25 Output Processing of Aggregate Data that Includes the Maximum Water Level, Etc. (Area.end File)

```
 # MAX FREE-SURFACE VALUE
 0.000E+00 0.000E+00 0.000E+00 0.000E+00 0.000E+00 0.000E+00 0.000E+00 0.000E+00 0.000E+00 0.000E+00
 0.000E+00 0.000E+00 0.000E+00 0.000E+00 0.000E+00 0.000E+00 0.000E+00 0.000E+00 0.000E+00 0.000E+00
 0.000E+00 0.000E+00 0.000E+00 0.000E+00 0.000E+00 0.000E+00 0.000E+00 0.000E+00 0.000E+00 0.000E+00
 0.000E+00 0.000E+00 0.000E+00 0.000E+00 0.000E+00 0.000E+00 0.000E+00 0.000E+00 0.000E+00 0.000E+00
 0.000E+00 0.000E+00 0.000E+00 0.000E+00 0.000E+00 0.000E+00 0.000E+00 0.000E+00 3.052E-03 3.192E-03
 3.391E-03 3.539E-03 3.771E-03 3.948E-03 4.023E-03 4.123E-03 4.277E-03 4.413E-03 4.529E-03 4.623E-03
 4.684E-03 4.710E-03 4.776E-03 4.875E-03 4.953E-03 5.013E-03 5.051E-03 5.064E-03 5.055E-03 5.022E-03
(skip)
 0.000E+00 0.000E+00 0.000E+00 0.000E+00 0.000E+00 0.000E+00 0.000E+00 0.000E+00 0.000E+00 0.000E+00
 0.000E+00 0.000E+00 0.000E+00 0.000E+00 0.000E+00 0.000E+00 0.000E+00 0.000E+00 0.000E+00 0.000E+00
 0.000E+00 0.000E+00 0.000E+00 0.000E+00
 # MAX FREE-SURFACE TIME
(skip)
 # MIN FREE-SURFACE VALUE
(skip)
 # MIN FREE-SURFACE TIME
(skip)
 # TSUNAMI TIME
(skip)
 # MAX VELOCITY VALUE
(skip)
 # MAX VELOCITY TIME
(skip)
```

Figure 0-2-26 Sample Aggregate Data that Includes the Maximum Water Level, Etc. (Area.end File)

2.3. Installation and Execution

2.3.1. Installation and Execution on Linux

2.3.1.1. Compilation

STOC-ML and STOC-IC should be compiled separately. So, the compilation process described below must be conducted twice for running STOC-ML and STOC-IC. The STOC directory structure is shown below.

ML .... Subroutine used for STOC-ML only
NS .... Subroutine used for STOC-IC only
COM .... Subroutine used for both STOC-ML and STOC-IC
Include .... Header file that contains common variables

At the time of compilation, move to the ML and NS directories and modify the makefile. Then, execute a make command. In the makefile, set the compile option to COPT.

2.3.1.2. Execution

Because the stack size often becomes insufficient, add the following settings to the Shell environment configuration file in advance:

(a) Settings when using bash:

ulimit -s unlimited

(b) Settings when using csh:

unlimit stacksize

For parallel computation, access should be enabled through ssh or rsh without using a password. Below is an example of execution with mpich1.

mpirun -p4pg appfile /home/stoc/bin/ml.out-opt

appfile constitutes a combination of a host names and execution commands like those shown below. The second column includes flags for the hosts that should be started. Only the first row includes 0. The second and subsequent rows include 1.

host01	0	/home/stoc/bin/ml.out
host01	1	/home/stoc/bin/ml.out
host02	1	/home/stoc/bin/ml.out
host02	1	/home/stoc/bin/ns.out

If the options are unavailable when mpi2 is used, you can execute the program by delimiting the strings with colons as follows.

mpiexec -n 2 –host host01 /home/stoc/bin/ml.out : -n 1 –host host02 /home/stoc/bin/ml.out : -n 1 –host host02 /home/stoc/bin/ns.out

2.3.2. Installation and Execution on Windows

In Microsoft Visual Studio, you can build and execute the programs by using Intel Visual FORTRAN as described below. Suppose that MPICH2 is already installed in the MPI environment.

To use something other than MPICH2, modify the names of the libraries to be linked to build time, paths, and execution methods to suit the environment.

2.3.2.1. Setting up the MPI Environment

After starting Microsoft Visual Studio, open the Options window from the [Tools] menu. Enter the following in the Includes and Libraries fields for Intel Visual FORTRAN setup (Figure 0-2-27). Note that you must select the x64 tab, and then enter the necessary data in the selected tab on a 64-bit environment.

- Includes C:¥Program Files¥MPICH2¥include
- Libraries C:¥Program Files¥MPICH2¥lib

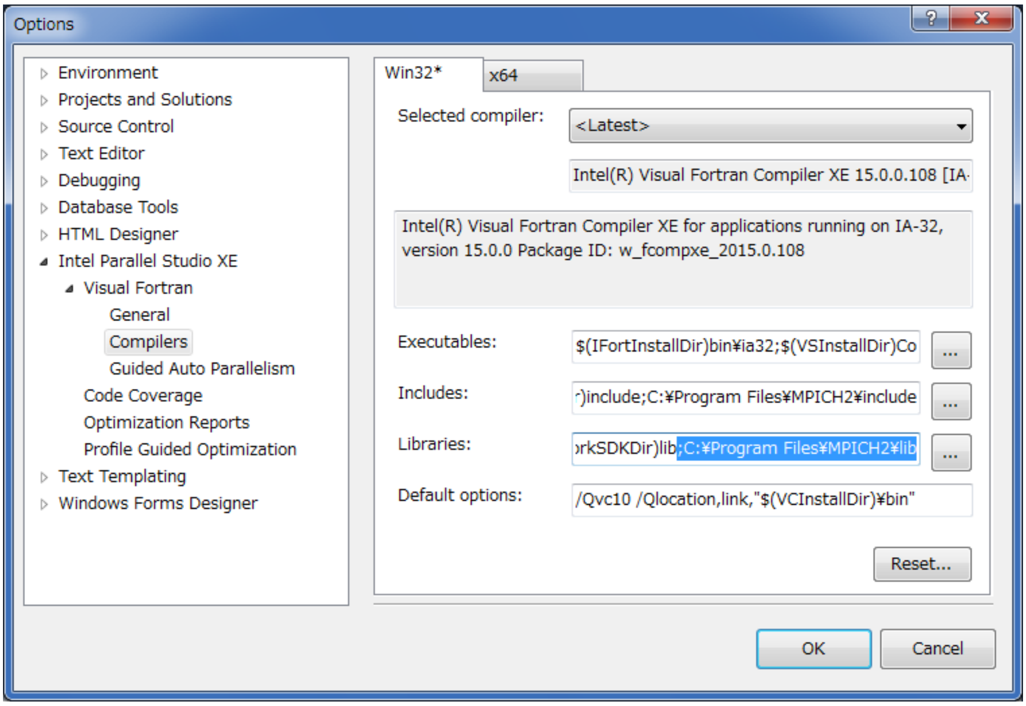


Figure 0-2-27 Options Window

2.3.2.2. Creating STOC Projects

STOC-ML and STOC-IC should be compiled separately. So, the project creation process described below must be conducted twice for running STOC-ML and STOC-IC.

(1) Open a new project, and then configure it (Figure 0-2-28).

(File Menu — New — Project)

Project type: Intel Visual FORTRAN —— Console application

Template: Empty Project

Project name: (Enter name of choice)

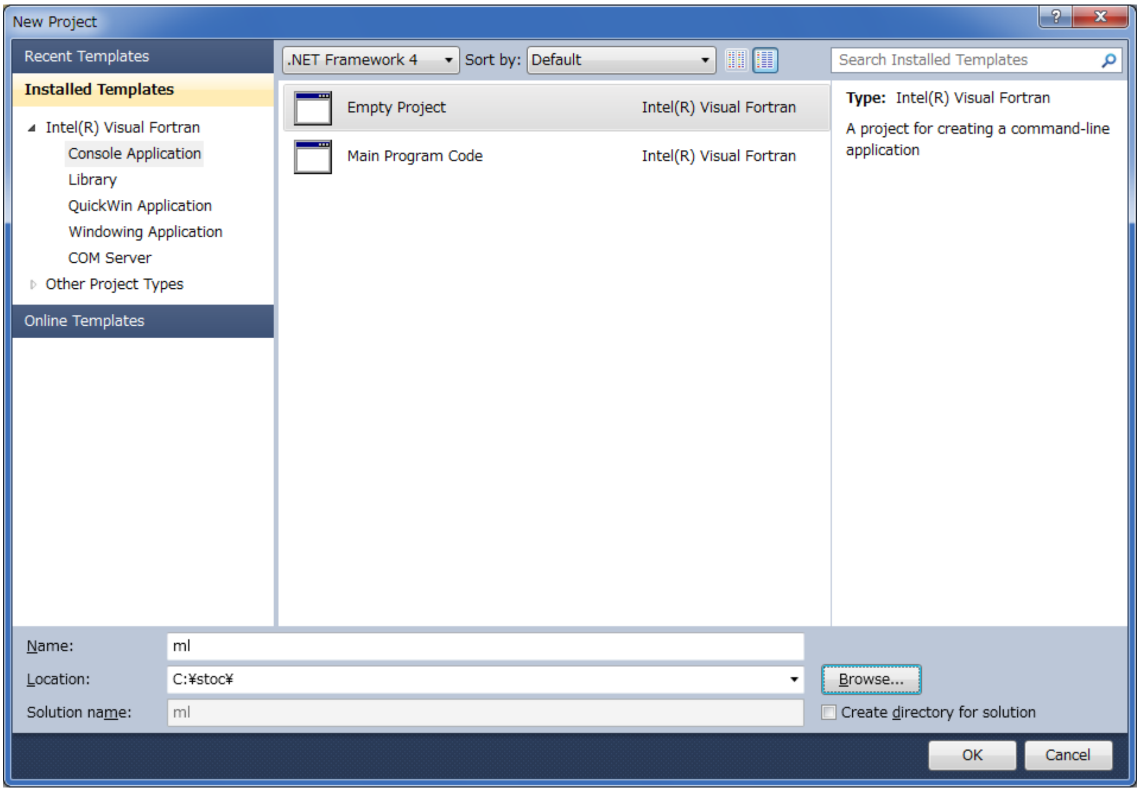


Figure 0-2-28 New Project Window

(2) Saving source files (Figure 0-2-29)

Right-click "Source Files" in Solution Explorer and add all the source files (*.f and *.f90) located under the ML/NS and COM folders for STOC. Similarly, right-click "Header Files" and add the header file (*.h) located under the Include folder for STOC.

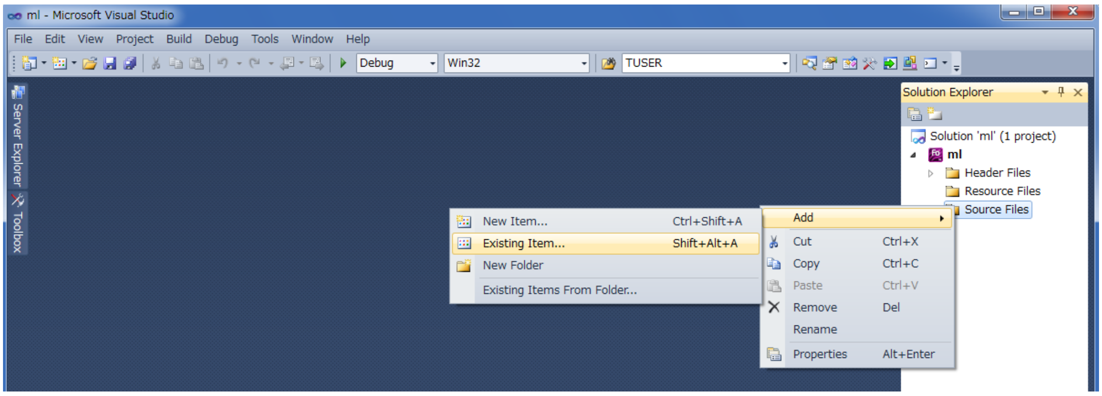


Figure 0-2-29 Saving Source Files

(3) Changing the configuration (Figure 0-2-30)

(Build Menu — Configuration Manager)

Change “Active solution configuration” from Debug to Release. For a 64-bit environment, change “Platform” from Win32 to x64.

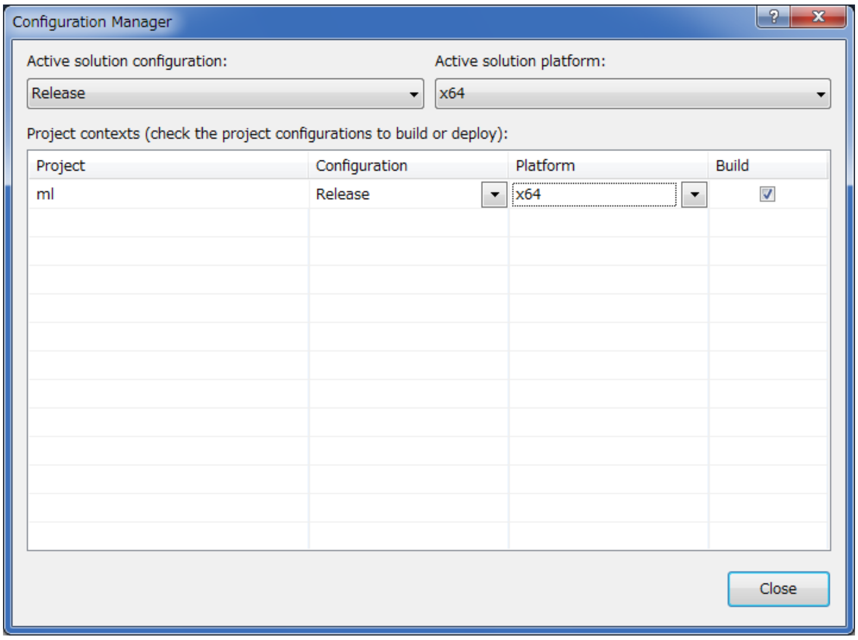


Figure 0-2-30 Configuration Manager Window

(4) Specifying the MPICH2 library (Figure 0-2-31, Figure 0-2-32, Figure 0-2-33, Figure 0-2-34)

(Project Menu — Properties)

Open Property Pages, select the General option under “Fortran”, and specify the STOC’s Include folder path for “Additional Include Directories”.

Select the Input option under “Linker” and specify “fmpich2.lib” as the additional library. Select the System option under “Linker” and set the stack size in bytes to 10 MB (for both reserve and commit). If a stack overflow error occurs at the time of execution, set this size to a larger value.

When you want to compile with “Debug” configuration, it is necessary to avoid the checking of routine interfaces in the diagnostics option under “Fortran”.

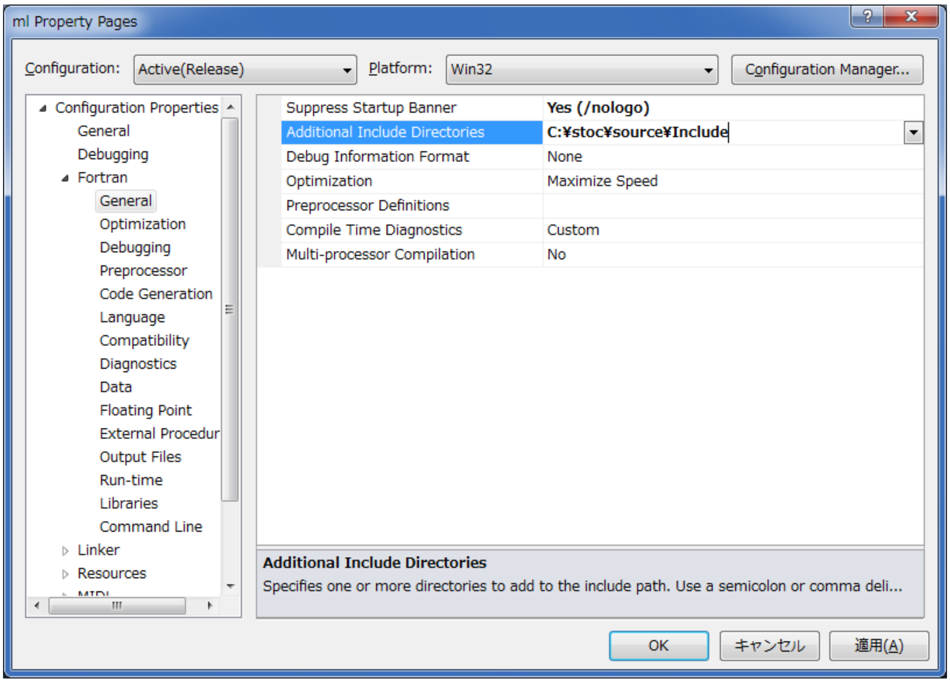


Figure 0-2-31 Property Pages Window 1

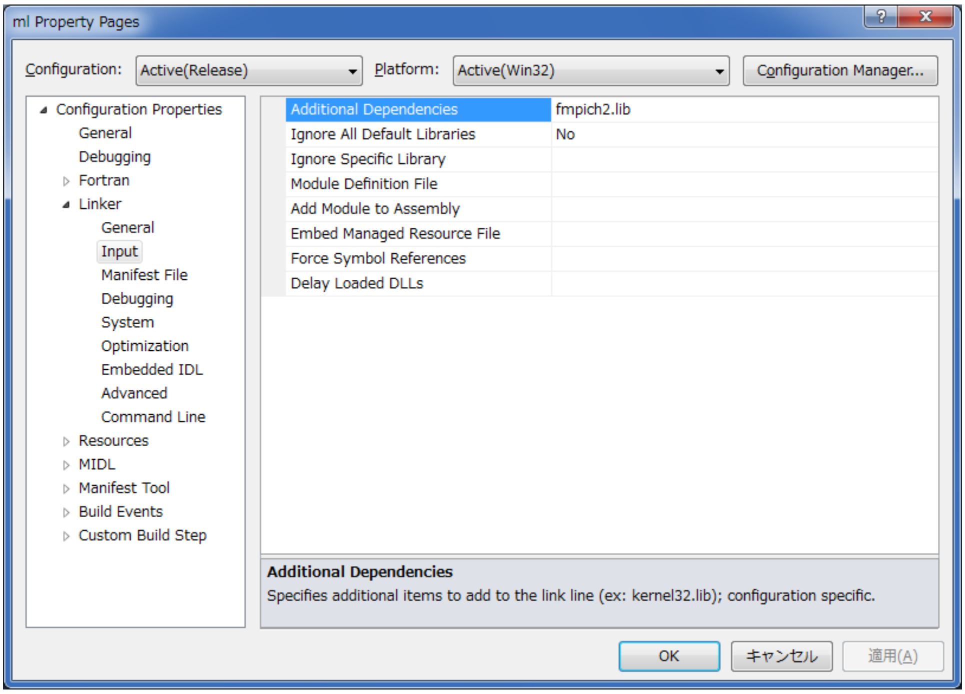


Figure 0-2-32 Property Pages Window 2

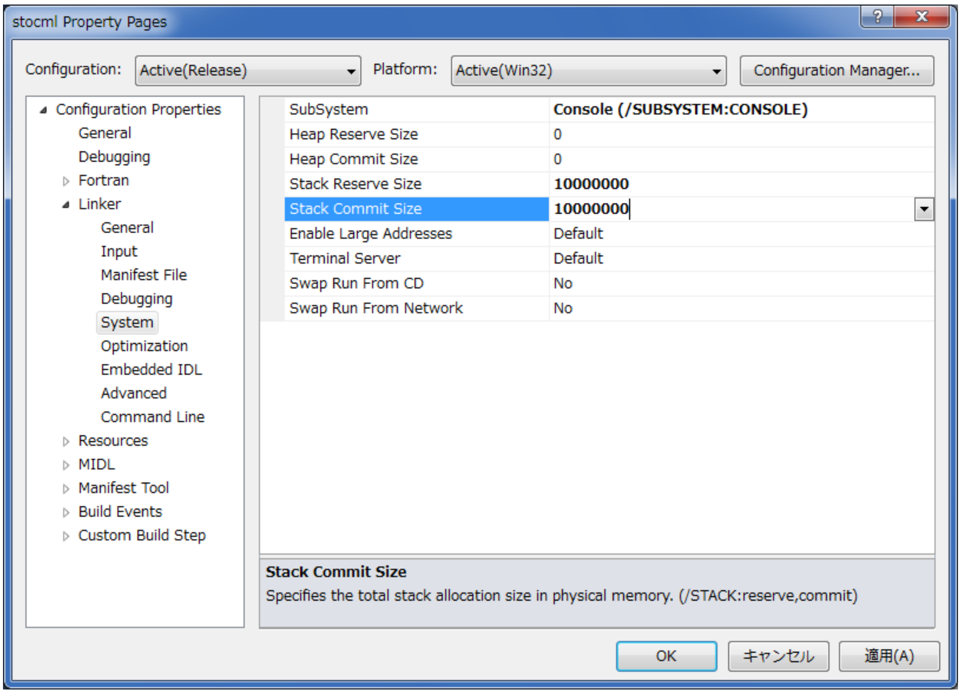


Figure 0-2-33 Property Pages Window 3

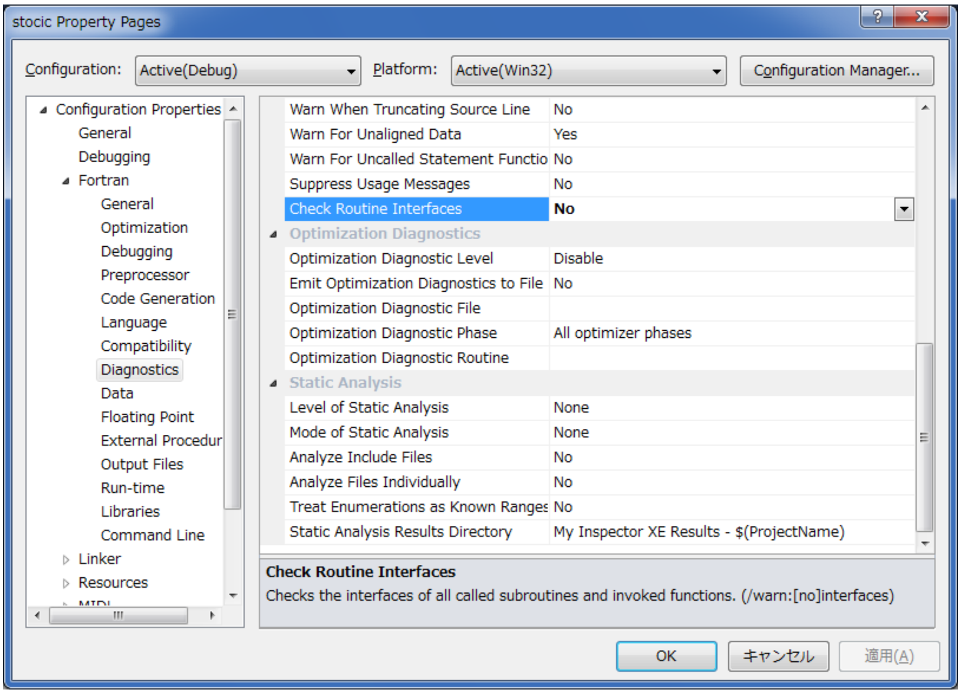


Figure 0-2-34 Property Pages Window 4

(5) Conduct the build process.

(Build Menu — Build Solution)

2.3.2.3. Execution

(a) Executing STOC-ML

Launch the Command Prompt and specify the following:

mpiexec -n 4 C:¥stoc¥ml.exe > log.txt

The number that follows -n indicates the number of Processor Elements to use. Enter the name of the executable file with a full path. To execute STOC-ML on multiple PCs, create machinefile.txt, and then enter the following:

mpiexec -n 8 –machinefile machinefile.txt C:¥stoc¥ml.exe > log.txt

Below is an example of machinefile.txt.

```
host01:2
host02:2
```

(b) Performing coupled calculations with STOC-ML and STOC-IC

Launch the Command Prompt and enter the following:

mpiexec –file appfile.txt > log.txt

appfile.txt is a text file that contains host and machine names to be used like those shown below.

```
hosts
host01 0 C:¥stoc¥ml.exe
host01 1 C:¥stoc¥ml.exe
host02 1 C:¥stoc¥ml.exe
host02 1 C:¥stoc¥ml.exe
host03 1 C:¥stoc¥ic.exe
host03 1 C:¥stoc¥ic.exe
host04 1 C:¥stoc¥ic.exe
host04 1 C:¥stoc¥ic.exe
```

The -file option might be unavailable in MPIs other than MPICH2. In such cases, enter a separate command for each host by delimiting the strings with colons as follows.

mpiexec -n 2 –host host01 C:¥stoc¥ml.exe : -n 2 –host host02 C:¥stoc¥ml.exe : -n 2 –host host03 C:¥stoc¥ic.exe : -n 2 –host host04 C:¥stoc¥ic.exe > log.txt

*Precautions

Multiple vendors offer MPI programs. Different MPI programs might be installed on one PC. For example, Intel FORTRAN might be installed together with Intel MPI on one PC. In such cases, if the vendor of the mpiexec command differs from that of the link MPI library at the time of construction, calculations cannot be performed. Note that for mpiexec, the install folder name can be displayed by entering where mpiexec in the Command Prompt window.

Chapter 3 **Program Explanation**

3.1. Organization of Programs

3.1.1. Program Structures

Figure 0-3-1 shows the tree structure of the main part of STOC-ML. Figure 0-3-2 shows the tree structure of the main part of STOC-IC.

```
                  1                   2                   3                   4
 1 ML_MAIN        -+-INCNCT         -+-INIT_MPIENV
 2                 |                 +-INPUT
 3                 +-DEFALT
 4                 +-INPUT (2,3)
 5                 +-ARCHLD
 6                 +-ARCONE
 7                 +-MKGRID
 8                 +-CP_MKIND3
 9                 +-AUTONEST
10                 +-MKIND1
11                 +-MKIND2
12                 +-MKPORS
13                 +-MKINIT
14                 +-CP_NEIBIND
15                 +-CP_SNDML2NS
16                 +-CP_RCVML2NS
17                 +-CP_MODDEP
18                 +-CP_NEIBCOM
19                 +-CP_BCML2NS
20                 +-CP_BCNS2ML
21                 +-CP_SNDNS2ML
22                 +-CP_RCVNS2ML
23                 +-MKDPRS
24                 +-SOLVER         -+-CP_RCVML2NS (16,2)
25                                   +-CP_INTMLVAL
26                                   +-SEABOT         -+-FAULTI
27                                   |                 +-FAULTT
28                                   |                 +-KFSURF
29                                   +-SETTBL
30                                   +-BCINLF
31                                   +-BCINLV
32                                   +-BCSURF
33                                   +-CLTMU
34                                   +-CP_NEIBCOM (18,2)
35                                   +-OUTPUT         -+-OUTRST
36                                   |                 +-OUTLST
37                                   |                 +-OUTHST
38                                   +-SETDT
39                                   +-BCTIDE
40                                   +-KFSURF (28,4)
41                                   +-BCSUR2
42                                   +-CLHUVWD
43                                   +-CLBRKW
44                                   +-CLFALLW
45                                   +-CLUEQ          -+-CLDP
46                                   |                 +-RUNUPX
47                                   +-CLVEQ          -+-CLDP (45,4)
48                                   |                 +-RUNUPY
49                                   +-CLHUVW
50                                   +-DISP_CLUV
51                                   +-DISP_DIVUV
52                                   +-CP_SNDML2NS (15,2)
53                                   +-CP_BCML2NS (19,2)
54                                   +-CP_BCNS2ML (20,2)
55                                   +-CP_SNDNS2ML (21,2)
56                                   +-CP_RCVNS2ML (22,2)
57                                   +-CP_UVML2NS
58                                   +-CLWEQ
59                                   +-CLSURF
60                                   +-BOUND
61                                   +-QBICGS
62                                   +-DSTRY
63                                   +-BLOCK
64                                   +-CLPRES
```

Figure 0-3-1 STOC-ML Program Structure

```
        1                   2                   3                   4
 1 NS_MAIN         -+-INCNCT            -+-INIT_MPIENV
 2                  |                    +-INPUT
 3                  +-DEFALT
 4                  +-INPUT (2,3)
 5                  +-ARCHLD
 6                  +-ARCONE
 7                  +-MKGRID
 8                  +-MKIND1
 9                  +-MKIND2
10                  +-CP_MKIND3
11                  +-MKPORS
12                  +-MKINIT
13                  +-CP_NEIBIND
14                  +-CP_SNDML2NS
15                  +-CP_RCVML2NS
16                  +-CP_MODDEP
17                  +-CP_NEIBCOM
18                  +-CP_BCML2NS
19                  +-CP_BCNS2ML
20                  +-CP_SNDNS2ML
21                  +-CP_RCVNS2ML
22                  +-MKDPRS
23                  +-SOLVER            -+-SEABOT           -+-FAULTI
24                                       |                   +-FAULTT
25                                       |                   +-KFSURF
26                                       +-SETTBL
27                                       +-BCINLF
28                                       +-BCINLV
29                                       +-BCSURF
30                                       +-CLTMU
31                                       +-CP_NEIBCOM (17,2)
32                                       +-OUTPUT           -+-OUTRST
33                                       |                   +-OUTLST
34                                       |                   +-OUTHST
35                                       +-MKCOE1
36                                       +-SETDT
37                                       +-BCTIDE
38                                       +-KFSURF (25,4)
39                                       +-BCSUR2
40                                       +-CLHUVWD
41                                       +-CLBRKW
42                                       +-CLFALLW
43                                       +-CLUEQ            -+-CLDP
44                                       |                   +-RUNUPX
45                                       +-CLVEQ            -+-CLDP (43,4)
46                                       |                   +-RUNUPY
47                                       +-CP_RCVML2NS (15,2)
48                                       +-CP_BCML2NS (18,2)
49                                       +-CP_BCNS2ML (19,2)
50                                       +-CP_SNDNS2ML (20,2)
51                                       +-CLWEQ
52                                       +-CP_UVML2NS
53                                       +-MKCOE2
54                                       +-QBICGS
55                                       +-UPUVWP
56                                       +-CLHUVW
57                                       +-CLSURF
58                                       +-BOUND
59                                       +-CLKEPS           -+-CLKEGN
60                                       |                   +-CLKEQ
61                                       |                   +-CLEEQ
62                                       +-CLSGSM           -+-CLKEGN (59,4)
63                                                           +-CLSGE
```

Figure 0-3-2 STOC-IC Program Structure

3.1.2. Subroutine Lists

Table 0-3-1 through Table 0-3-12 are lists of subroutines by category.

Table 0-3-1 Subroutines (Solvers: Common)

Routine name	Processing
CELLSC	Multiplies a scalar quantity by the volume of water in the computational cell
CHKCNS	Adds up the values calculated by CELLSC for the entire calculation area
CHKVAL	Replaces the scalar quantity, determined in the computational cell characterized by a small layer thickness, with the specified value
CLBRKW	Calculates data on a breaking wave model
CLDP	Calculates the difference in pressure between adjacent cells
CLFALLW	Calculates data on an overfall model
CLHUVW	Sets the flow rate per unit width, which is determined from the flow velocity and water level
CLHUVWD	Sets the flow rate per unit width, which is determined from the flow velocity and water level (applicable to permeable structures)
CLTMU	Calculates the turbulent kinematic viscosity when an LES model or a k-ε model is applied
CLSIMP	Calculates the vertical diffusion term of an advection-diffusion equation for scalar quantities by an implicit method
CLSNEW	Updates the value by solving an advection-diffusion equation for scalar quantities
CLSURF	Updates the water level
DISP_CLUV	Calculates the potential function for dispersive-wave model calculation
DISP_DIVUV	Corrects the flow velocity by using the potential function for dispersive-wave model calculation
FAULTI	Performs initial processing to calculate the water level fluctuations from the fault parameters
FAULTT	Calculates the water level fluctuations from the fault parameters
FLUXPL	Corrects the flux at the plate boundary
FLUXSX	Calculates the X-directional flux of an advection-diffusion equation for scalar quantities
FLUXSY	Calculates the Y-directional flux of an advection-diffusion equation for scalar quantities

FLUXSZ	Calculates the Z-directional flux of an advection-diffusion equation for scalar quantities
FLUXUP	Calculates the shear stress at the plate boundary when solving an X-directional motion equation
FLUXUX	Calculates the X-directional in-plane momentum flux when solving an X-directional motion equation
FLUXUY	Calculates the Y-directional in-plane momentum flux when solving an X-directional motion equation
FLUXUZ	Calculates the Z-directional in-plane momentum flux when solving an X-directional motion equation
FLUXVP	Calculates the shear stress at the plate boundary when solving a Y-directional motion equation
FLUXVX	Calculates the X-directional in-plane momentum flux when solving a Y-directional motion equation
FLUXVY	Calculates the Y-directional in-plane momentum flux when solving a Y-directional motion equation
FLUXVZ	Calculates the Z-directional in-plane momentum flux when solving a Y-directional motion equation
FLUXWP	Calculates the shear stress at the plate boundary when solving a Z-directional motion equation
FLUXWX	Calculates the X-directional in-plane momentum flux when solving a Z-directional motion equation
FLUXWY	Calculates the Y-directional in-plane momentum flux when solving a Z-directional motion equation
FLUXWZ	Calculates the Z-directional in-plane momentum flux when solving a Z-directional motion equation
KFSURF	Sets the KF and KP values
LOGLAW	Calculates the friction velocity by using logarithmic law
RUNUPX	Restricts the X-directional flow velocity at either the leading edge of a running-up wave or the overflow on a storm surge barrier
RUNUPY	Restricts the Y-directional flow velocity at either the leading edge of a running-up wave or the overflow on a storm surge barrier
SEABOT	Processes an earthquake-induced water level fluctuation model
SETDT	Sets the time step size
SETTBL	Sets the current value by linear interpolation of the time series table

Table 0-3-2 Subroutines (Solvers: Available for STOC-ML only)

Routine name	Processing
ML_MAIN	Main routine for STOC-ML
CLPRES	Calculates the hydrostatic pressure distribution
CLUEQ	Updates flow velocity U by explicitly solving an X-directional momentum conservation equation
CLVEQ	Updates flow velocity V by explicitly solving a Y-directional momentum conservation equation
CLWEQ	Updates flow velocity W by solving a continuity equation.
SOLVER	Main routine for calculation with STOC-ML

Table 0-3-3 Subroutines (Solvers: Available for STOC-IC only)

Routine name	Processing
NS_MAIN	Main routine for STOC-IC
CLEEQ	Calculates the ε advection-diffusion equation
CLKEGEN	Calculates the production terms of k and ε equations
CLKEP	Main routine for k-ε model
CLKEQ	Calculates the k advection-diffusion equation
CLSGE	Calculates the k advection-diffusion equation for the SGS one-equation
CLSGSM	Main routine for SGS one-equation model
CLUEQ	Calculates the temporary flow velocity by explicitly solving an X-directional momentum conservation equation
CLVEQ	Calculates the temporary flow velocity by explicitly solving a Y-directional momentum conservation equation
CLWEQ	Calculates the temporary flow velocity by explicitly solving a Z-directional momentum conservation equation
MKCOE1	Creates a coefficient matrix for the pressure correction equation (without considering the water surface)
MKCOE2	Creates a coefficient matrix and right side for the pressure correction equation
SOLVER	Main routine for calculation with STOC-IC
UPUVWP	Updates the flow velocity and pressure
WALBND	Calculates the k and ε wall boundary values
WLCONT	Performs preprocessing for WALBND (i.e., counts the number of planes necessary for wall boundary processing)

Table 0-3-4 Subroutines (Related to Boundary Condition Processing)

Routine name	Processing
BCINLF	Sets the F value for both a fixed-flow-velocity boundary and free-outflow boundary
BCINLV	Sets the normal component of flow velocity on a fixed-flow-velocity boundary
BCSUR2	Sets the boundary conditions for flow velocity on a water surface
BCSURF	Sets the boundary conditions for pressure and flow velocity on a water surface
BCUUYZ	Sets tangential component U of flow velocity on Y- and Z-directional walls and on fixed-flow-velocity and free-outflow boundaries
BCVVXZ	Sets tangential component V of flow velocity on X- and Z-directional walls and on fixed-flow-velocity and free-outflow boundaries
BCWWXY	Sets tangential component W of flow velocity on X- and Y-directional walls and on fixed-flow-velocity and free-outflow boundaries
BOUND	Performs transparent-boundary processing at the outermost periphery of a calculation area

Table 0-3-5 Subroutines (Related to Matrix Solutions)

Routine name	Processing
QAX	Calculates the product of matrix A and vector X
QAX2D	2-dimensional version of QAX (used to calculate the potential function of a dispersive wave model)
QBICGS	Solves an asymmetrical matrix by using the BiCGstab method
QBICGS2D	2-dimensional version of QBICGS (used to calculate the potential function of a dispersive wave model)
QILUDC	Performs incomplete LU decomposition.
QILUDC2D	2-dimensional version of QILUDC (used to calculate the potential function of a dispersive wave model)
QIP	Calculates inner products
QIP2D	2-dimensional version of QIP (used to calculate the potential function of a dispersive wave model)
QMINV	Performs forward elimination and backward substitution
QMINV2D	2-dimensional version of QMINV (used to calculate the potential function of a dispersive wave model)

Table 0-3-6 Subroutines (Others)

Routine name	Processing
COMOUT	Outputs the common variables for debugging
DEFALT	Initializes the common variables
ERRMS2	Outputs the common format of warning messages
ERRMSG	Outputs the common format of error messages
FTIMER	Calculates total computation time
HSORT	Sorts the array variables (hash sorting)
MKDPRS	Reads and sets the porous value of a permeable structure
MKGRID	Creates grid constants XC, YC, and ZC
MKIND1	Creates indexes INDP, INDU, INDV, and INDW
MKIND2	Creates both list LLWALL for wall boundary processing and list LLWALP for plate boundary processing
MKIND4	Creates both list LLWALB for storm surge barrier processing and list LLOFL for overflow boundary processing
MKINIT	Sets the initial values of water levels, flow velocities, etc.
MKPORS	Reads and sets the porous value
ZERCLI	Sets constants in an array of integer type
ZERCLR	Sets constants in an array of double-precision real data type

Table 0-3-7 Subroutines (Related to Analysis Condition File Read)

Routine name	Processing
GET1	Takes out one space-delimited character string from input data and returns the string's start position IS and end position IE; also converts lower case letters to upper case
GET2	Same as GET1 except that GET2 does not convert lower case letters to upper case
GETC	Reads one data item of character type from input data
GETC2	Same as MGETC except that GETC2 reads only one character string and calls GET2 instead of GET1
GETD	Takes out one space-delimited character string from input data and returns the string's start position IS and end position IE; also converts lower case letters to upper case (functions the same way as GET1)
GETI	Reads one data item of integer type from input data
GETR	Reads one data item of real data type from input data

INBOUN	Reads the %BOUNDARY block
INCASE	Reads the %CASE block
INGRID	Reads the %GRID block
ININIT	Reads the %INITIAL block
INMODL	Reads the %MODEL block
INMTRX	Reads the %MATRIX block
INOBST	Reads the %OBSTACLE block
INOUTP	Reads the %OUTPUT block
INPROP	Reads the %PROPERTY block
INPUT	Reads data block names that start with % from the analysis condition file and calls the read processing subroutine for each block
INTIME	Reads the %TIME block.
MGETC	Reads multiple data items of character type from input data
MGETD	Reads multiple data items of real data type from input data (functions the same way as MGETER)
MGETI	Reads multiple data items of integer type from input data
MGETR	Reads multiple data items of real data type from input data

Table 0-3-8 Subroutines (Related to Calculation Results Output)

Routine name	Processing
DBWI2D	Outputs 2-dimensional cross-section data from a 3-dimensional array of integer type
DBWIXY	Outputs a 2-dimensional array of integer type
DBWR2D	Outputs 2-dimensional cross-section data from a 3-dimensional array of double-precision real data type
DBWRXY	Outputs a 2-dimensional array of double-precision real data type
OUTHST	Outputs time series data
OUTLST	Outputs spatial distribution data which includes water levels and flow velocities
OUTPUT	Controls file output
OUTRST	Outputs data for restart

Table 0-3-9 Subroutines (Related to Parallel Computation)

Routine name	Processing
INCNCT	Loads the data.in file and sets the parent-child relationship between calculation areas
INIT_MPIENV	Initializes the MPI environment and splits the communicator
FLOPEN	Sets a different file name for each area (i.e., changes the NN portion of FT16_NN and FT06_NN).
ABORT1	Forcibly terminates parallel computation

Table 0-3-10 Subroutines (Related to Domain Decomposition)

Routine name	Processing
ARCONE	Determines the range of cells where subdomains are located in the overall system when a domain is decomposed
CP_BCINT	Same as CP_DSR_DC2 except that CP_BCINT uses an array sized to fit a dispersive wave model
CP_COMHHML	Communicates HH_ML when a domain is decomposed
CP_DSR_DC2	Sends/receives the values of cells adjacent to the domain boundary when a domain is decomposed
CP_DSR_DC3	Same as CP_DSR_DC2 except that CP_DSR_DC3 uses an array sized to fit a dispersive wave model
CP_DSR_FFF	Sends/receives the value of the corner of a virtual cell when a domain is decomposed
CP_KFEXPND	Creates array KFD that is larger than array KF when a domain is decomposed
CP_LIST_CMMBND	Creates an index and list array for communication between subdomains when a domain is decomposed
CP_LIST_INDCMM	Creates an index for communication between subdomains when a domain is decomposed
CP_NEIBCOM	Sends/receives multiple variables of cells adjacent to the domain boundary when a domain is decomposed
CP_NEIBIND	Sends/receives multiple variables of cells adjacent to the domain boundary when a domain is decomposed (same as CP_NEIBCOM except that CP_NEIBIND sends/receives variables different from those above)
CP_NEIBREV	Sends DU (1, J, K) and DV (I, 1, K) to DU (MXM, J, K) and DV (I, MYM, K)
FLNAM	Sets a different file name for each subdomain when an automatic domain

	decomposition is applied
MODIJ	Converts the grid indexes in the overall system into those in subdomains when an automatic domain decomposition is applied
RDSUB*	Reads only data on your own subdomain from the file containing data on the overall system when an automatic domain decomposition is applied

Table 0-3-11 Subroutines (Related to Nesting Processing)

Routine name	Processing
ARCHLD	Determines the range of cells where child areas are located in a parent area
AUTONEST	Automatically sets the nesting region of child areas in a parent area
CP_AVEUVW	Averages the flow velocity and other data for a child area so the averaged data can be sent to a parent area
CP_BCHHML	Averages the water level of a child area so the averaged data can be sent to a parent area
CP_BCHHNS	Sets the water level of the child area overlap zone by interpolating the water level of a parent area
CP_BCML2NS	Sets the data for the child area overlap zone by interpolating the data for a parent area
CP_BCNS2ML	Averages the data for a child area so the averaged data can be sent to a parent area
CP_BCUVWI	Sets the flow velocity and other data for the X-directional cross section in a child area by interpolating the flow velocity and other data for a parent area (sets the data in the cross section for I = 1 and I = MXM)
CP_BCUVWJ	Sets the flow velocity and other data for the Y-directional cross section in a child area by interpolating the flow velocity and other data for a parent area (sets the data in the cross section for J = 1 and J = MYM)
CP_BCVVML	Sets the average of flow velocity and other data by calling CP_AVEUVW for the north, east, south, and west sides.
CP_HHML2NS	Determines the weighted average of the water levels of parent and child areas in the overlap zone
CP_INTMLVAL	Sets the current value by interpolating values of two times when reading the water level and flow velocity for the connection boundary from the file
CP_MKIND3	Creates an index for cross-reference between parent and child areas
CP_MODDEP	Corrects the water depth of a child area in the overlap zone with the calculation flag and water depth of a parent area

CP_RCVDEP	Allows child area in the overlap zone to receive the water depth of a parent area
CP_RCVIND	Allows child area in the overlap zone to receive the cell index of a parent area
CP_RCVML2NS	Allows child area to receive the water level, flow velocity, and other data for a parent area
CP_RCVNS2ML	Allows parent area to receive the water level, flow velocity, and other data for a child area
CP_SEAFLG	Outputs an error message when the topography does not match between parent and child areas (see Section 2.2.2.5)
CP_SNDDEP	Sends the water depth of a parent area from the parent area in the overlap zone
CP_SNDIND	Sends the cell index of a parent area from the parent area in the overlap zone
CP_SNDML2NS	Sends the water level, flow velocity, and other data for a parent area to a child area
CP_SNDNS2ML	Sends the water level, flow velocity, and other data for a child area to a parent area
ZDNEST	Sets the vertical velocity distribution at the boundary when connecting from a single layer to multiple layers by nesting

Table 0-3-12 Subroutines (Related to Coupled Calculation with STOC-DM)

Routine name	Processing
BLOCK	Performs blocking processing for coupled calculation with STOC-DM
com_drift1	Sends/receives domain information and calculation conditions to/from STOC-DM
com_drift2	Sends the height, water level, and flow velocity to STOC-DM; also sends/receives the destruction and blocking flags to/from STOC-DM
com_drift3	Sends the current time for STOC-ML and STOC-IC to STOC-DM
com_drift4	Sends/receives the calculation termination flag to/from STOC-DM
DSTRY	Performs destruction processing for coupled calculation with STOC-DM

3.1.3. Module Lists

Table 0-3-13 through Table 0-3-18 list the modules used by STOC.

Table 0-3-13 Modules for the MPMD Parallel Environment

Module name	Routine name	Processing
MOD_COMM		Defines the communicator, etc., for the MPMD parallel environment
	INIT_MPMD	Initializes the MPMD parallel environment

Table 0-3-14 Modules for Communication When a Domain Is Decomposed

Module name	Routine name	Processing
cp_module_indcmm		Configures the inter-domain communication settings
	MKINDEX	Sets the index, etc., for inter-subdomain communication when a domain is decomposed

Table 0-3-15 Modules for Output Processing When a Domain Is Decomposed Automatically

Module name	Routine name	Processing
MOD_GATHER		Data is dispersed among subdomains when a domain is automatically decomposed, this module brings the dispersed data together for output of time series data (Area.hst) and totalization data (Area.end) such as the highest water level.
	GATHERV_PRE	Allocates memory to the communication buffer array and storage array for GATHERV processing
	GATHERV	Totalization data is dispersed among subdomains when a domain is automatically decomposed, this routine brings the dispersed data together into one process.
	GATHERVI	Interface for variables of integer type for GATHERV
	SETRV2DG	Stores data into the output array
	GATHERH_PRE	Preprocessing routine that creates a communication list for GATHERH processing
	GATHERH	Time series output point data is dispersed among subdomains when a domain is automatically decomposed, this routine brings the dispersed data together into one process.

Table 0-3-16 Modules for Calculating Water Level Fluctuations with Fault Parameters

Module name	Routine name	Processing
MOD_FAULT		Module that contains the variables and functions necessary for calculating the water level fluctuations with the fault parameters
	SET_PARAM_FAULT	Initializes the calculation parameters
	SET_UTM	Sets the origin of the UTM coordinate system
	EN2LB	Converts the x and y coordinate values in the UTM and Japan plane rectangular coordinate system into longitude and latitude information
	LB2LB	Converts the geodesic system for longitude and latitude information
	SHIGOSEN	Calculates the meridian line length from the equator

	DISPLACE	Calculates the water level fluctuations from the fault parameters
	USCAL	Processing routine 1 called from DISPLACE
	UDCAL	Processing routine 2 called from DISPLACE
	ATN	Corrected version of atan2

Table 0-3-17 Modules for Dispersive Wave Model Calculation

Module name	Routine name	Processing
MOD_PSI		Defines the variables for potential function PSI, etc., when a dispersive wave model is used
	INIT_PSI	Allocates memory to arrays PSI, INDEXPSI, and DHDX2
	FIN_PSI	Deallocates memory from arrays PSI, INDEXPSI, and DHDX2

Table 0-3-18 Modules for Changing the Sizes of List Arrays

Module name	Routine name	Processing
MOD_LIST		Changes the sizes of arrays such as LLWALL when the topography changes
	ALLOC_LIST	Allocates memory to arrays LLWALL and LLWALP
	ALLOC_LIST2	Allocates memory to arrays LLWALB, LLOFL, and HHOFL
	DEALLOC_LIST	Deallocates memory from arrays LLWALL and LLWALP or arrays LLWALB, LLOFL, and HHOFL
	COUNT_MLWAL	Calculates the sizes of arrays LLWALL and LLWALP

3.2. Organization of Data

3.2.1. Common Variables

Table 0-3-19 shows the common variables. The list below shows the meaning of each type.

C: Character type

I: Integer type

R: Real data type (which is a double precision type unless otherwise noted)

L: Logical type

Table 0-3-19 Common Variables

Label name			AREA_I (variables related to the positions specified for boundary conditions, etc.)
Variable name	Type	Size	Description
NARASZ	I	1	Upper limit on the number of areas that can be specified with an input parameter =2000 (fixed parameter)
NAREA	I	1	Number of areas specified with an input parameter
IAREA	I	7,NARASZ	Ranges of the indexes for the area specified with an input parameter (1,*) Minimum value of the X-directional index (2,*) Maximum value of the X-directional index (3,*) Minimum value of the Y-directional index (4,*) Maximum value of the Y-directional index (5,*) Minimum value of the Z-directional index (6,*) Maximum value of the Z-directional index (7,*) Type of area =0 3-dimensional area =1 Plane area whose X direction is a normal direction =2 Plane area whose Y direction is a normal direction =3 Plane area whose Z direction is a normal direction *See Section 2.2.2.4. For a 3-dimensional area, all the indexes are cell indexes. For a plane area, only

			the normal components are grid indexes, and the remaining components are cell indexes. The minimum and maximum values of the normal components are equal.

Label name			AUTODECOMP (variables related to automatic domain decomposition)
Variable name	Type	Size	Description
NDIVX	I	1	Number of X-directional domain decompositions
NDIVY	I	1	Number of Y-directional domain decompositions
IDIV1	I	MAXPE	X-directional range of the partial area (start point cell index)
IDIV2	I	MAXPE	X-directional range of the partial area (end point cell index)
JDIV1	I	MAXPE	Y-directional range of the partial area (start point cell index)
JDIV2	I	MAXPE	Y-directional range of the partial area (end point cell index)
ICRDC	I	1	Connection flag for different coordinate systems =0 Connection of identical coordinate systems =1 A spherical coordinate system for your own area and a plane coordinate system for child areas
IDIVX	I	MAXPE	Number of X-directional cell partitions for partial areas
JDIVY	I	MAXPE	Number of Y-directional cell partitions for partial areas
XDIV1	R	MAXPE	X-directional range of the partial area. (X coordinate value of the start point)
XDIV2	R	MAXPE	X-directional range of the partial area. (X coordinate value of the end point)
YDIV1	R	MAXPE	Y-directional range of the partial area. (Y coordinate value of the start point)
YDIV2	R	MAXPE	Y-directional range of the partial area. (Y coordinate value of the end point)
XCAD1	R	1	X-directional range of the CADMAS-SURF area for connection with CADMAS-SURF (X

			coordinate value of the start point)
XCAD2	R	1	X-directional range of the CADMAS-SURF area for connection with CADMAS-SURF (X coordinate value of the end point)
YCAD1	R	1	X-directional range of the CADMAS-SURF area for connection with CADMAS-SURF (X coordinate value of the start point)
YCAD2	R	1	X-directional range of the CADMAS-SURF area for connection with CADMAS-SURF (X coordinate value of the end point)
REGN	R	4	Range of the child area when ICRDC = 1 (1) X-directional range of the child area. (Longitude of the west end point) (2) Y-directional range of the child area. (Latitude of the south end point) (3) X-directional range of the child area. (Longitude of the east end point) (4) Y-directional range of the child area. (Latitude of the north end point)

Label name			BLOCK (variables related to a blocking model)
Variable name	Type	Size	Description
NBFRSZ	I	1	Settings for the location at which to apply blocking and destruction models *Blocking and destruction models are not applied at a location within the distance specified as a number of cells equal to NBFRSZ from the connection boundary. =10 (fixed parameter)
GLIMIT	R	1	Correction value for permeability of blocking cells =0.6 (fixed parameter)

Label name			BOUNDI and BOUNDR (variables related to boundary conditions)
Variable name	Type	Size	Description
NINLSZ	I	1	=100 (fixed parameter)

NOTFSZ	I	1	=100 (fixed parameter)
NWLLSZ	I	1	=500 (fixed parameter)
ISURF	I	3	Settings for the sea level boundary conditions (1) Atmospheric pressure (2) X-directional component of ocean surface wind speed (3) Y-directional component of ocean surface wind speed The following are available for setting each of the items above: <0 User function =0 Constant value *Corresponds to SURFACE-TYPE for input of analysis conditions. >0 Time-series table (time-series table number) Default value: 0 *When using the user function, specify both the atmospheric pressure and ocean surface wind speed with that user function.
NINLT	I	1	Number of velocity fixed boundary surfaces
MINLT	I	NINLSZ	Position of a velocity fixed boundary surface (pointer to array IAREA)
IINLT	I	8,NINLSZ	Settings for a velocity fixed boundary surface (1,*) X-directional component of inflow velocity (2,*) Y-directional component of inflow velocity (3,*) Z-directional component of inflow velocity (4,*) Inflow water level (5,*) Inflow temperature (6,*) Inflow chlorine concentration (7,*) Not used (8,*) Not used The following are available for setting each of the items above: =0 Constant value >0 Time-series table (time-series table number)
NOUTLT	I	1	Number of free inflow/outflow boundaries
MOUTLT	I	2,NOTFSZ	Position of a free inflow/outflow boundary

			(pointer to array IAREA)
IOUTLT	I	3,NOTFSZ	Settings for a free inflow/outflow boundary (1,*) Inflow temperature (2,*) Inflow chlorine concentration (3,*) Boundary water level The following are available for setting items (1,*) and (2,*): =0 Constant value >0 Time-series table (time-series table number) The following are available for setting item (3,*): =0 Zero gradient >0 Time-series table (time-series table No.)
NWALL	I	1	Number of wall surface boundaries
MWALL	I	3,NWLLSZ	Positions and types of wall surface boundaries (1,*) Position of a wall surface boundary (pointer to array IAREA) (2,*) Boundary conditions for flow velocity =0 Slip =1 No slip =2 Wall function =3: Fixed-flow velocity in the tangential direction (3,*) Boundary conditions for temperature and chlorine concentration =0 Zero gradient =1 Fixed value
IWALL	I	5,NWLLSZ	Settings for a wall surface boundary (1,*) X-directional component of tangential velocity (2,*) Y-directional component of tangential velocity (3,*) Z-directional component of tangential velocity (4,*) Temperature (5,*) Chlorine concentration The following are available for setting each of the

			items above: =0 Constant value >0 Time-series table (time-series table number)
MDWALV	I	1	Predetermined values for wall surface conditions related to flow velocity (These values are applied to wall surfaces for which wall surface boundary conditions are not specified as input parameters.) =0 Slip =1 No slip =2 Wall function =3 slip condition on sides and non-slip condition on the bottom. Default value: 0 *Corresponds to DEFAULT-TYPE-V for input of analysis conditions
IPFLG	I	1	Specify whether to read the external-boundary conditions from the *.bci file. =0 Does not read the data =1 Reads the data *Corresponds to OPEN-2D for input of analysis conditions
NOVRLP	I	4	Number of overlapped meshes with the connection area (1) Number of overlapped meshes with the southern connection boundary (2) Number of overlapped meshes with the western connection boundary (3) Number of overlapped meshes with the eastern connection boundary (4) Number of overlapped meshes with the northern boundary *Corresponds to OVERLAP for input of analysis conditions
NSOMER	I	4	Settings for the transparent boundary of the external periphery of a computational area

			(1) Southern boundary (2) Western boundary (3) Eastern boundary (4) Northern boundary The following are available for setting each of the items above: =0 No transparent boundary =1 Characteristic curve method =2 Imamura et al's (2001) method =3 Apply Imamura et al's (2001) method, applicable to both the water level table and astronomical tide. *Corresponds to OPEN-SOMMER for input of analysis conditions.
ILGLWL	I	1	Specify whether there is a wall surface (other than a plane obstacle) to which logarithmic law is applicable. =0 There is no wall surface. =1 There is a wall surface. *Corresponds to LLWALL (6, *)
ILGLWP	I	1	Specify whether there is a wall surface (plane obstacle) to which logarithmic law is applicable. =0 There is no wall surface. =1 There is a wall surface. *Corresponds to LLWALL (6, *)
NBOT	I	1	Specify whether to use a water level fluctuation model. =-1 Uses the water level fluctuation model (Calculates the fluctuations from the fault parameter) =0 Does not use the water level fluctuation model =1 Uses the water level fluctuation model (Reads the fluctuations from the *.sbt file) *Corresponds to SEA-BOTTOM for input of analysis conditions
LVPNAB	I	1	Specify the vertical distribution of the normal

			component of flow velocity at the nesting boundary. =0 Uniform vertical distribution =1 Non-uniform vertical distribution *Corresponds to VERTICAL-PROFILE for input of analysis conditions
IVPNAB	I	1	Distance (cell count) for referencing the vertical distribution of flow velocity when LVPNAB=1 *Corresponds to NABOR for input of analysis conditions.
LNTANG	I	1	Specify the tangential flow velocity at the nesting boundary. =0 Interpolated flow velocity in the parent area =1 Flow velocity in the adjacent cell =2 Interpolated flow velocity in the parent area for inflow or flow velocity in the adjacent cell for outflow *Corresponds to NEST-TANGENTIAL-VELOCITY for input of analysis conditions
LMODDEP	I	1	Specify whether to automatically correct the topography in the overlap zone at the nesting boundary. =0 Does not automatically correct the topography =1 Automatically corrects the topography *Corresponds to MODIFY-DEPTH for input of analysis conditions
RSURF	R	3	Sea-level boundary conditions (set with constant values) (1,*) Atmospheric pressure [Pa] (2,*) X-directional component of ocean surface wind speed [m/s] (3,*) Y-directional component of ocean surface wind speed [m/s]
RINLT	R	8,NINLSZ	Settings for a velocity fixed boundary surface (set with constant values) (1,*) X-directional component of inflow velocity

			[m/s] (2,*) Y-directional component of inflow velocity [m/s] (3,*) Z-directional component of inflow velocity [m/s] (4,*) Inflow water level [m] (5,*) Inflow temperature [°C] (6,*) Inflow chlorine concentration [‰] (7,*) Not used. (8,*) Not used.
ROUTLT	R	2,NOTFSZ	Settings for a free inflow/outflow boundary (set with constant values) (1,*) Inflow temperature [°C] (2,*) Inflow chlorine concentration [‰]
RWALL	R	5,NWLLSZ	Settings for a wall surface boundary (set with constant values) (1,*) X-directional component of tangential velocity [m/s] (2,*) Y-directional component of tangential velocity [m/s] (3,*) Z-directional component of tangential velocity [m/s] (4,*) Temperature [°C] (5,*) Constant value of chlorine concentration [‰]
HTIDE0	R	1	Reference water level [m] that is used when OPEN-SOMMER=2 is specified for input of analysis conditions *Corresponds to HO for input of analysis conditions

Label name			CADMAS1, CADMAS2, and CADMAS3 (variables related to coupled calculations with CADMAS-SURF)
Variable name	Type	Size	Description
MAX_STOC	I	1	Upper limit on NB_STOC

			=256 (fixed parameter)
MAX_CADMAS	I	1	Upper limit on NB_CADMAS =256 (fixed parameter)
MAX_NIST	I	1	Upper limit on NIST =500 (fixed parameter)
MAX_NJST	I	1	Upper limit on NJST =500 (fixed parameter)
MAX_NKST	I	1	Upper limit on NKST =100 (fixed parameter)
MAX_CADBUF	I	1	Upper limit on the communication buffer size =200000 (fixed parameter) *MAX_CADBUF≧MAX(NIST,NJST)*NKST*6
NB_STOC	I	1	Number of STOC processes associated with CADMAS
LB_STOC	I	1	Order of your own process within the STOC processes associated with CADMAS *A number from 1 to NB_STOC; used when referencing IB_STOC; 0 is assigned to processes not associated with CADMAS.
IB_STOC	I	MAX_STOC	Rank of an STOC process in MPI_COMM_WORLD; this process is associated with CADMAS.
NB_CADMAS	I	1	Number of CADMAS processes associated with STOC
LB_CADMAS	I	1	Order of your own process within the CADMAS processes associated with STOC *A number from 1 to NB_CADMAS; used when referencing IB_CADMAS or IB_SC; 0 is assigned to processes not associated with STOC. *This number is always 0 in STOC.
IB_CADMAS	I	MAX_CADMAS	Rank of a CADMAS process in MPI_COMM_WORLD; this process is associated with STOC.
NB_SC	I	1	Specify whether your own process is involved in coupled calculation with STOC-CADMAS. =0 The process is not involved.

			>0 The process is involved.
ITAGSC	I	1	Value for setting the communication tag
NIST	I	1	Number of X-directional STOC meshes into which the CADMAS area is divided
NJST	I	1	Number of Y-directional STOC meshes into which the CADMAS area is divided
NKST	I	1	Number of Z-directional STOC meshes into which the CADMAS area is divided
IWCAD	I	1	Specify which STOC mesh corresponds to the cell at the southwest end of the CADMAS area. (X-directional cell index I) *0 for the location outside of your own area
IECAD	I	1	Specify which STOC mesh corresponds to the cell at the northeast end of the CADMAS area. (X-directional cell index I) *0 for the location outside of your own area
JSCAD	I	1	Specify which STOC mesh corresponds to the cell at the southwest end of the CADMAS area. (Y-directional cell index J) *0 for the location outside of your own area CADMAS
JNCAD	I	1	Specify which STOC mesh corresponds to the cell at the northeast end of the CADMAS area. (Y-directional cell index J) *0 for the location outside of your own area
KBCAD	I	1	Specify which STOC mesh corresponds to the cell at the lower end in the vertical direction of the CADMAS area. (Z-directional cell index K)
KTCAD	I	1	Specify which STOC mesh corresponds to the cell at the upper end in the vertical direction of the CADMAS area. (Z-directional cell index K)
JJOFF	I	2	Offset value for setting the communication range when the west or east boundary of the CADMAS area connects to your own area. (1) 1 when the south end of the connection line matches that of the CADMAS area, 0 otherwise

			(2) 1 when the north end of the connection line matches that of the CADMAS area, 0 otherwise
IIOFF	I	2	Offset value for setting the communication range when the south or north boundary of the CADMAS area connects to your own area. (1) 1 when the west end of the connection line matches that of the CADMAS area, 0 otherwise (2) 1 when the east end of the connection line matches that of the CADMAS area, 0 otherwise
JJCAD	I	6,NB_CADMAS	Variables for setting the range of communication with CADMAS (1,*) Number of Y-directional cells for the connection boundary when the west boundary of the CADMAS area connects to your own area (2,*) Number of Y-directional cells for the connection boundary when the east boundary of the CADMAS area connects to your own area Items (3,*) through (6,*) are set when the west or east boundary of the CADMAS area connects to your own area. (3,*) Cell index J at the south end of the connection boundary (4,*) Cell index J at the north end of the connection boundary (5,*) 1 when the south end of the connection boundary is a corner or 0 when it is a point on the side. (6,*) 1 when the south end of the connection boundary is a corner or 0 when it is a point on the side. *Items (1,*) through (6,*) are 0 when no connection is made.
IICAD	I	6,NB_CADMAS	Variables for setting the range of communication with CADMAS

			(1,*) Number of X-directional cells for the connection boundary when the south boundary of the CADMAS area connects to your own area (2,*) Number of X-directional cells for the connection boundary when the north boundary of the CADMAS area connects to your own area Items (3,*) through (6,*) are set when the north or south boundary of the CADMAS area connects to your own area. (3,*) Cell index I at the west end of the connection boundary (4,*) Cell index I at the east end of the connection boundary (5,*) 1 when the west end of the connection boundary is a corner or 0 when it is a point on the side (6,*) 1 when the east end of the connection boundary is a corner or 0 when it is a point on the side *Items (1,*) through (6,*) are 0 when no connection is made.
UWCAD	R	MAX_NJST, MAX_NKST,6	West boundary values received from CADMAS (*,*,1) Flow velocity U at the boundary (*,*,2) Flow velocity V at the boundary (*,*,3) Flow velocity W at the boundary (*,*,4) F value of the boundary (*,*,5) Water level of the inside cell of the boundary (*,*,6) Water level of the outside cell of the boundary *Only (*, 1, 5) is used for (*, *, 5), and neither (*, K, 5) nor K≠1 is used for (*, *, 5). The same can be said for (*, *, 6).
UECAD	R	MAX_NJST,	East boundary value received from CADMAS

		MAX_NKST,6	*Same as UWCAD
VSCAD	R	MAX_NIST, MAX_NKST,6	South boundary value received from CADMAS *Same as UWCAD
VNCAD	R	MAX_NIST, MAX_NKST,6	North boundary value received from CADMAS *Same as UWCAD
CADBUF	R	MAX_CADBUF	Buffer for communication with CADMAS

Label name			CONNEC (variables related to area connection)
Variable name	Type	Size	Description
NMFILE	C	MAXPE	Analysis condition file name *See Section 2.2.1.
MAXPE	I	1	Upper limit on the number of STOC processes =256 (fixed parameter)
IDCON	I	7	Data related to area connection (1) Your own area number (2) Parent area number (3) Child area number (4) South area number (5) West area number (6) East area number (7) North area number
IPECON	I	8,MAXPE	Ranks of all STOC processes (1,*) Rank of a process (2,*) Rank of the process in the parent area for (1, *) (3,*) Rank of the process in the child area for (1, *) (4,*) Rank of the process in the south area for (1, *) (5,*) Rank of the process in the west area for (1, *) (6,*) Rank of the process in the east area for (1, *) (7,*) Rank of the process in the north area for (1, *) (8,*) Calculation model used for (1, *) =0 STOC-ML

			=1 STOC-IC *Rank refers to a communication rank in the MPI communicator comm_model.
IDTABL	I	2,MAXPE	Table that shows the association between area numbers and process ranks (1,*) Area number (2,*) Process rank *Typically, Area number – 1 = Process rank.
NUMPE	I	2,0:MAXPE	Data on the child area (1,*) Number of processes in the child area (2,*) Pointer to the second element of NUMCOM *Ranks of processes in all child areas can be retrieved with NUMCOM (1, NUMPE (2, *)) through NUMCOM (1, NUMPE (2, *) + NUMPE (1, *) -1).
NUMCOM	I	5,MAXPE	Data on the child area (1,*) Rank of a process in the child area (2～5,*) Not used
NSIZEALL	I	1	Same as NSIZE
NSIZE	I	1	Total number of processes for STOC-ML and STOC-IC.
NRANK	I	1	Rank of your own process

Label name			CP_NESTBC_I and CP_NESTBC_R (variables related to nesting)
Variable name	Type	Size	Description
NESTFL	I	1	Specify whether there is a parent area. =0 There is no parent area. =1 There is a parent area.
NXY	I	1	Number of X-directional or Y-directional meshes into which the area is divided, whichever is greater * NXY = MAX (MX, MY)
NPNTML	I	1	Not used
NESNS	I	4	Number of overlapped meshes with the connection boundary for the parent area

			(1) Number of overlapped meshes with the southern connection boundary (2) Number of overlapped meshes with the western connection boundary (3) Number of overlapped meshes with the eastern connection boundary (4) Number of overlapped meshes with the northern connection boundary
NESML	I	4	Number of overlapped meshes with the connection area for the child area (1) Number of overlapped meshes with the southern connection boundary (2) Number of overlapped meshes with the western connection boundary (3) Number of overlapped meshes with the eastern connection boundary (4) Number of overlapped meshes with the northern connection boundary
TIMVB	R	1	Time 1 for the boundary value that was read when IPFLG≠0
TIMVF	R	1	Time 2 for the boundary value that was read when IPFLG≠0
TIMHB	R	1	Time 1 for the boundary value that was read when IPFLG≠0
TIMHF	R	1	Time 2 for the boundary value that was read when IPFLG≠0

Label name			CPUCHK (variables that store the processing time for calculation)
Variable name	Type	Size	Description
NCPUSZ	I	1	CPUSEC array size =100 (fixed parameter)
CPUSEC	R	2,NCPUSZ	Processing time for each part of the program (1,*) Total processing time (2,*) Measurement start time for the current measurement section

			*When measurement terminates, Termination time - Start time is added to the processing time.

Label name			DST01 (destruction model parameters)
Variable name	Type	Size	Description
NBFRSZ	I	1	Settings for the location at which to apply blocking and destruction models *Blocking and destruction models are not applied at a location within the distance specified as a number of cells equal to NBFRSZ from the connection boundary. =10 (fixed parameter)
NDSTSZ	I	1	Number of types of values used to judge destruction by the depth of inundation =5 (fixed parameter)
AMNGDST	R	1	Manning's roughness set for the destroyed land =0.05 (fixed parameter)
DSTLMT	R	NDSTSZ	Value [m] used to judge destruction by the depth of inundation (1) Judgment value for wooden buildings =2.0 (fixed parameter) (2) Judgment value for reinforced concrete buildings =99.9 (fixed parameter) (3) - (5) Not used

Label name			DOMAIN (number of meshes into which the area is divided and array sizes)
Variable name	Type	Size	Description
MX	I	1	Number of X-directional cells that include virtual cells (See エラー！参照元が見つかりません。.)
MY	I	1	Number of Y-directional cells that include virtual cells
MZ	I	1	Number of Z-directional cells that include virtual cells

MXM	I	1	Number of X-directional gird points *=MX-1
MYM	I	1	Number of Y-directional gird points *=MY-1
MZM	I	1	Number of Z-directional gird points *=MZ-1
MXYZ	I	1	Number of cells which include virtual cells *=MX*MY*MZ
MXY	I	1	Number of cells which include virtual cells in the XY cross section *=MX*MY
NXYZ	I	1	Number of computational cells *=(MX-2)*(MY-2)*(MZ-2)
MLWALL1	I	1	Number of wall surfaces (other than plane obstacles) at the computational cell boundary
MLWALL	I	1	=10*MLWALL1 (for STOC-DS-MODE)
MLWALP	I	1	Number of wall surfaces (plane obstacles) at the computational cell boundary
MLWALB	I	1	Number of locations at the computational cell boundary where storm surge barrier processing is performed
MLWALBX	I	1	Number of locations in MLWALB where the X direction is a normal direction
MLOFL	I	1	Number of locations at the computational cell boundary where the overflow formula is applied
MLOFLX	I	1	Number of locations in MLOFL where the X direction is a normal direction

Label name			DRIFT (variables related to coupled calculations with STOC-DM)
Variable name	Type	Size	Description
NB_SD	I	1	Communication rank of STOC-DM for coupled calculations with STOC-DM (-1 when coupled calculations are not performed)
NB_SD_MAIN	I	1	Specify whether the process for coupled calculations with STOC-DM is a representative

			process. =1 The process is a representative process. =0 The process is not a representative process. *Representative process refers to an STOC process of the lowest rank for coupled calculations with STOC-DM. Only the representative process exchanges STOC's common data, such as time and seawater density, with STOC-DM.
NOCALDM	L	1	Specify whether to perform debris calculation with STOC-DM. =f Performs debris calculation =t Does not perform debris calculations *Set as "t" when writing water level and flow velocity data to the file for pre-calculation to perform offline calculation with STOC-DM.
OFF_INTERVAL	R	1	Interval [s] at which you want to write water level and flow velocity data for pre-calculation
OFF_START	R	1	Time [s] to start writing water level and flow velocity data to the file for pre-calculation
OFF_NEXT	R	1	Time [s] to write the next water level and flow velocity data to the file for pre-calculation

Label name			FILEDV (unit numbers for file input/output)
Variable name	Type	Size	Description
INP	I	1	Unit number of the analysis condition input file =15 (fixed parameter)
LP	I	1	Unit number of the FT16 file =16 (fixed parameter)
IFLRI	I	1	Unit number of the restart input file (rsi) =11 (fixed parameter)
IFLST	I	1	Unit number of the topology/geometry input file (str) =12 (fixed parameter)
IFLTM	I	1	Unit number of the time-series input file (tim) =17 (fixed parameter)

IFLRO	I	1	Unit number of the restart output file (rso) =21 (fixed parameter)
IFLHS	I	1	Unit number of the time-series output file (hst) =23 (fixed parameter)
IFLEN	I	1	Unit number of the aggregate processing output file (end) =24 (fixed parameter)
IFLSB	I	1	Unit number of the water level fluctuation input file (sbt) =25 (fixed parameter)
IFLBO		1	Unit number of the connection boundary water level/flow velocity output file (bco) =26 (fixed parameter)
IFLBI	I	1	Unit number of the connection boundary water level/flow velocity input file (bci) =27 (fixed parameter)
IFLDB	I	1	Unit number of the debug output file (dbg) =29 (fixed parameter)
IFLDP	I	1	Unit number of the permeable structure input file (dpr) =30 (fixed parameter)
IFLOF	I	1	Unit number of the overflow model application location input file (ofl) =31 (fixed parameter)
IFLSB2	I	1	Unit number of the water level fluctuation input file (sbt_NNNN) =35 (fixed parameter) *File specified in the right most column in エラー! 参照元が見つかりません。.
IFLFW	I	1	Unit number of the overfall model coefficient input file (fwc) =37 (fixed parameter)
IFLLP	I	1	Unit number of the lst file =40 (fixed parameter)

Label name	FILEC and FILEI (variables related to file read

			processing)
Variable name	Type	Size	Description
CLINE	C	1	Variable to store one line from the analysis condition file
CNUL	C	1	Blank character string for CLINE data deletion
CFLNM	C	1	Name of a file without its extension *Area01 for the Area01.str file
I1DD	I	1	Position in the time-series input file (tim) from which data is being read
IFLNM	I	1	Length of an input/output file name

Label name			GLOBAL (variables related to domain decomposition)
Variable name	Type	Size	Description
MXG	I	1	Number of X-directional cells in the overall domain *MXG=MX when the domain is not decomposed.
MYG	I	1	Number of Y-directional cells in the overall domain *MYG=MY when the domain is not decomposed.
MZG	I	1	Number of Z-directional cells *MZG=MZ
NPROC	I	1	Number of subdomains after domain decomposition
INDCOM	I	6,MAXPE	Range of a subdomain in the overall domain (1,*) X-directional start cell index (2,*) X-directional end cell index (3,*) Y-directional start cell index (4,*) Y-directional end cell index (5,*) Z-directional start cell index (6,*) Z-directional end cell index
INDCM2	I	6,MAXPE	Range of a subdomain in the overall domain *INDCM2 is an index seen from the parent domain. Using the parent's cell index, INDCM2 indicates the range of the subdomain when your own child domain is decomposed.

CHILDCOMM	I	1	Communicator for communication between subdomains that have been decomposed
IAUTOD	I	1	Specify whether to automatically decompose the domain. =0 Manual decomposition =1 Automatic decomposition (as specified with I-DIV or J-DIV)
MYPROC	I	1	Your own domain number in array INDCOM
MYIS	I	1	X-directional start position (Items MYIS through MYJE are cell indexes for the range of your own domain in the overall domain. MYIS=2 and MYIE=MXM when your own domain is not decomposed.)
MYIE	I	1	X-directional end position
MYJS	I	1	Y-directional start position
MYJE	I	1	Y-directional end position

Label name			GRID (variables related to calculation grids)
Variable name	Type	Size	Description
NGRDSZ=	I	1	Upper limit on the number of grids =10001 (fixed parameter)
ICORDTYPE	I	1	Type of coordinate system =1 Plane coordinate system =2 Spherical coordinate system
REARTH	R	1	Radius of the Earth [m] =6.37e+6 (fixed parameter)
CEARTH	R	1	Rotation velocity of the Earth [rad/s] =7.29212e-5 (fixed parameter)
XGRID	R	NGRDSZ	X-coordinate of the grid point [m] or [°]
YGRID	R	NGRDSZ	Y-coordinate of the grid point [m] or [°]
ZGRID	R	NGRDSZ	Z-coordinate of the grid point [m] or [°]
HLMT	R	1	Lower limit on water depth [m] *Corresponds to HLIMIT for input of analysis conditions
XORG	R	1	X-coordinate of the origin

			*Corresponds to ORIGIN for input of analysis conditions
YORG	R	1	Y-coordinate of the origin *Corresponds to ORIGIN for input of analysis conditions
XCEN	R	1	Reference longitude (for a spherical coordinate system)
YCEN	R	1	Reference latitude (for a spherical coordinate system)
REGION	R	4	Range of the child area REGION(1) Longitude of the west end point REGION(2) Latitude of the south end point REGION(3) Longitude of the east end point REGION(4) Latitude of the north end point *Corresponds to REGION for input of analysis conditions

Label name			INITL (variables related to initial conditions)
Variable name	Type	Size	Description
NINTSZ	I	1	Upper limit on the number of areas specified for the initial values =100 (fixed parameter)
NHINIT	I	1	Number of areas specified for the initial value of a water level
IHINIT	I	4,NINISZ	Cell indexes for the area for setting the initial value of a water level (1,*) X-directional start cell index (2,*) X-directional end cell index (3,*) Y-directional start cell index (4,*) Y-directional end cell index
UUINIT	R	1	Initial value [m/s] of the X-directional component of flow velocity
VVINIT	R	1	Initial value [m/s] of the Y-directional component of flow velocity
WWINIT	R	1	Initial value [m/s] of the Z-directional component of flow velocity

AKINIT	R	1	Initial value [m^2/s^2] of turbulence kinetic energy
EPINIT	R	1	Initial value [m^2/s^3] of turbulence kinetic energy dissipation
HHINIT	R	NINITSZ	Initial value [m] of a water level

Label name			MATRXI and MATRXR (variables related to the matrix solver)
Variable name	Type	Size	Description
ITRMTX	I	1	Iteration count for the matrix solver
MAXMTX	I	1	Maximum iteration count
LPRMTX	I	1	Output flag for matrix solver convergence behavior =0 Does not produce output =1 Produces output
EPSMTX	R	1	Judgment value 1 (absolute value) for matrix solution convergence
EPRMTX	R	1	Judgment value 2 (relative value) for matrix solution convergence
RNRMTX	R	1	Residual norm for matrix calculation

Label name			MODELI and MODELR (variables related to the calculation model)
Variable name	Type	Size	Description
NFNSIZ	I	1	Upper limit on the number of areas specified for Manning's roughness =100 (fixed parameter)
MLNS	I	1	Flag for discrimination between STOC-ML and STOC-IC =0 STOC-ML =1 STOC-IC
LSURF	I	1	Flag for free surface computation =0 Does not perform calculation =1 Performs calculation
LTURB	I	1	Flag for turbulence models =0 Does not perform calculation

			=1 LES model =2 k-ε model =3 Mellor-Yamada model =4 SGS model
ISW	I	5	Processing flags (1), (2), and (5): Not used (3) Flag for surface flow velocity calculation =1 Does not perform calculation =0 Performs calculation (4) Flag for calculating the advective term of a momentum conservation equation =1 Does not perform calculation =0 Perform calculation *Corresponds to ISW for input of analysis conditions
NFN	I	1	Number of areas specified for setting Manning's roughness
IFNTBL	I	4,NFNSIZ	Cell indexes for the area for setting Manning's roughness (1,*) X-directional start cell index (2,*) X-directional end cell index (3,*) Y-directional start cell index (4,*) Y-directional end cell index
IGM2S	I	1	Flag for wind stress computation =-1 Performs calculation (Calculates the friction coefficient from the model equation) =0 Does not perform calculation =1 Performs calculation (The friction coefficient is constant.)
IGM2B	I	1	Seabed friction coefficient =-1 Performs calculation (based on Manning's roughness) =0 Does not perform calculation =1 Performs calculation (The friction coefficient is constant.)
ISEAWL	I	1	Method for calculating the flow rate per unit width

			for overflow water on the storm surge barrier =0 Performs calculation without considering the storm surge barrier =1 Performs calculation in consideration of the storm surge barrier
IHONMA	I	1	Specify whether to apply Honma formula at the overflow at the storm surge barrier. =0 Does not use Honma formula =1 Uses Honma formula =2 Uses the flow rate, computed by Honma formula, as a limit value
IAIDA	I	1	Specify whether to use Aida formula at the overflow at the seawall. =0 Does not use Aida formula =1 Uses Aida formula
IBKSTP	I	1	Specify whether to use a drop formula at the drop at the seawall. =0 Does not use the formula. =1 Uses the formula
IMVERT	I	1	Method for calculating the vertical diffusion term of the transport equation for a scalar =0 Calculates the term explicitly =1 Calculates the term implicitly
LDISS	I	1	Flag for permeable structures =0 Uses the CDM model. =1 Uses the DF model.
IFALLW	I	1	Flag for overfall model calculation =0 Does not perform calculation =1 Performs calculation
JFALLW	I	1	Method for setting the coefficients of an overfall model =0 Uses constant values =1 Reads the coefficients from the file
LBRKW	I	1	Flag for breaking-wave model calculation =0 Does not perform calculation =1 Kennedy’s model

			=2 Iwase et al.'s model
LDISP	I	1	Flag for dispersive-wave model calculation =0 Does not perform calculation =1 Performs calculation
LSTOCDS	I	1	Flag for destruction/blocking model calculation =0 Does not perform calculation =1 Performs calculation
GRAV	R	1	Gravity acceleration [m/s^2]
GM2S	R	1	Surface friction coefficient
GM2B	R	1	Seabed friction coefficient
PARAMF	R	1	Weighting parameters for central difference and first-order upwind scheme in the computation of the advective term in function F. *Weighted so that 0.0 indicates the central difference and 1.0 indicates the first-order upwind scheme
PARAMF2	R	1	=1-PARAMF
PARAMV	R	1	Difference parameter for the advective term of a momentum conservation equation *Same as PARMF
PARAMV2	R	1	=1-PARAMV
PARAMK	R	1	Difference parameter for advective terms k and ε equations *Same as PARAMF
PARAMK2	R	1	=1-PARAMK
PRT	R	1	Turbulent Prandtl number
SCT	R	1	Turbulent Schmidt number
CORI	R	1	Coriolis coefficient
CSMG	R	1	LES model parameter (Smagorinsky constant)
HAIDA	R	1	Minimum value of the downstream water level in Aida formula
DKENNEDY	R	1	Mixing length coefficient (amplification coefficient) δ in Kennedy's breaking-wave model
DKENN1	R	1	Coefficient C_1 associated with transition time in Kennedy's model

DKENN2	R	1	Coefficient C_2 of a breaking-wave start condition expression in Kennedy's model
DKENN3	R	1	Coefficient C_3 of a breaking-wave end condition expression in Kennedy's model
BETAIWA	R	1	Coefficient β associated with a kinematic eddy viscosity coefficient of Iwase et al.'s breaking-wave model
SAFEBRKW	R	1	Safety factor used to determine the upper limit of viscosity from stability conditions in the computation when viscosity increases in the breaking-wave model
GXB	R	1	Value [m] used to determine the tip used in the run-up tip model
GLH	R	1	Value [m] used to determine whether or not to apply velocity limitations
GZH	R	1	Value [m] used to determine whether or not to compute a surface layer velocity
EPSH	R	1	Additional thickness [m] of water level when water has dried up
EPST	R	1	Tsunami arrival parameter [m]
RIMP	R	1	Implicit parameter for seabed friction calculation
FNVAL	R	NFNSIZ	Manning's roughness
CFALLW0	R	1	Loss coefficient C_{fw} of momentum in the overfall model
DFALLWN0	R	1	Distribution coefficient $C_{fw,dist,norm}$ towards the normal direction when vertical momentum is distributed horizontally in the overfall model
DFALLWT0	R	1	Distribution coefficient $C_{fw,dist,\mathrm{tang}}$ towards the tangential direction when vertical momentum is distributed horizontally in the overfall model
DISPBETA	R	1	Parameter for a dispersive wave model =0 Boussinesq expression

			=1/15 Madsen-Sørensen expression =-1/3 Explicit Boussinesq expression
DISPLIM	R	1	Lower limit $z_{disp,\lim}$ [m] of the z coordinate in which fixed boundary conditions are applied for landside boundary in the dispersive wave model
DISPBCLIM	R	1	Upper limit $z_{disp,BC,\lim}$ [m] of the z coordinate in which landside boundaries are not computed in the dispersive wave model

Label name			OBSTI and OBSTR (variables related to geometry/structure)
Variable name	Type	Size	Description
NOBSSZ	I	1	Upper limit on the number of solid obstacles =600 (fixed parameter)
NOBPSZ	I	1	Upper limit on the number of plane obstacles =700 (fixed parameter)
NPRSSZ	I	1	Upper limit on the number of porous structures =700 (fixed parameter)
NFRCSZ	I	1	Upper limit on the number of friction resistance setting ranges =100 (fixed parameter)
NSEASZ	I	1	Upper limit on the number of points for changing from non-computational cells to computational cells =1000 (fixed parameter)
LOBST	I	1	Specify whether to read the topography/geometry data file (*.str). =0 Does not read the file =1 Reads the file
NOBSS	I	1	Number of solid obstacles
NOBSP	I	1	Number of plane obstacles
NPORS	I	1	Number of porous structures
NFRIC	I	1	Number of friction resistance setting ranges
NSEA	I	1	Number of points for changing from non-

			computational cells to computational cells
IOBSS	I	6,NOBSSZ	Range of cell indexes for solid obstacles (1,*) X-directional start cell index (2,*) X-directional end cell index (3,*) Y-directional start cell index (4,*) Y-directional end cell index (5,*) Z-directional start cell index (6,*) Z-directional end cell index
IOBSP	I	7,NOBPSZ	Range of cell indexes for plane obstacles (1,*) X-directional start index (2,*) X-directional end index (3,*) Y-directional start index (4,*) Y-directional end index (5,*) Z-directional start index (6,*) Z-directional end index (7,*) Normal direction of a plane =1 X direction =2 Y direction =3 Z direction *Indexes in the normal direction are grid indexes. The other indexes are cell indexes.
IPORS	I	7,NPRSSZ	Range of cell indexes for porous structures (1,*) X-directional start cell index (2,*) X-directional end cell index (3,*) Y-directional start cell index (4,*) Y-directional end cell index (5,*) Z-directional start cell index (6,*) Z-directional end cell index (7,*) Type =0 Normal porous obstacle =2 Permeable structure
IFRIC	I	6,NFRCSZ	Range of cell indexes for setting friction resistance (1,*) X-directional start cell index (2,*) X-directional end cell index (3,*) Y-directional start cell index

			(4,*) Y-directional end cell index (5,*) Z-directional start cell index (6,*) Z-directional end cell index
ISEA	I	2,NSEASZ	Cell index for the point for changing from non-computational cells to computational cells (1,*) I (2,*) J
LDPRS	I	1	Specify whether to read the permeable structure data file (*.dpr). =0 Does not read the file. =1 Reads the file
RPORS	R	10,NPRSSZ	Porous value (1,*) Porosity (2,*) X-directional permeability (on the peripheral surface in the –X direction) (3,*) X-directional permeability (inside the area) (4,*) X-directional permeability (on the peripheral surface in the +X direction) (5,*) Y-directional permeability (on the peripheral surface in the –Y direction) (6,*) Y-directional permeability (inside the area) (7,*) Y-directional permeability (on the peripheral surface in the +Y direction) (8,*) Z-directional permeability (on the peripheral surface in the –Z direction) (9,*) Z-directional permeability (inside the area) (10,*) Z-directional permeability (on the peripheral surface in the +Z direction) The following are used for permeable structures only: (11,*) Inertia force coefficient (12,*) Drag coefficient (13,*) DF model's α (14,*) DF model's β (15,*) DF model's representative diameter
RFRIC	R	NFRCSZ	Drag coefficient

Label name			OUTPUC, OUTPUI, and OUTPUR (variables related to file output)
Variable name	Type	Size	Description
CLIST	C	NPHYSZ	Name of a physical quantity to be output to results list
CHIST	C	NPHYSZ	Name of a physical quantity to be output as time-series data
NPHYSZ	I	1	Upper limit on the number of physical quantities to be output =60 (fixed parameter)
NRSTSZ	I	1	Upper limit on the number of output times/steps for restart data =50 (fixed parameter)
NOUTSZ	I	1	Upper limit on the number of output times/steps for results list =1300 (fixed parameter)
NPNTSZ	I	1	Upper limit on the number of output cells for time-series data =750 (fixed parameter)
LREST	I	1	Method for specifying restart data to be output =0 Step =1 Time
NREST	I	1	Number of output times/steps for restart data
IREST0	I	1	Pointer to either the next output step or time for restart data
IREST	I	NRSTSZ	Output steps for restart data
LLIST	I	1	Method for specifying results list to be output =0 Step =1 Time
NLIST	I	1	Number of output times/steps for results list
ILIST0	I	1	Pointer to either the next output step or time for lsit data
ILIST	I	NOUTSZ	Output steps for results list
MLIST	I	1	Number of physical quantities to be output to results list

LISTT	I	1	Output format for results list =0 ASCII format =1 Binary format (*.lst file)
NLSECT	I	1	Number of output cross sections for results list
LHIST	I	1	Method for specifying the time-series data to be output =0 Step =1 Time
IHIST0	I	1	Next output step for time-series data
IHIST	I	1	Output step interval for time-series data
MHIST	I	1	Number of physical quantities to be output as time-series data
NHCELL	I	1	Number of output cells for time-series data
IHCELL	I	3, NPNTSZ	Index for the output cell for time-series data (1,*) I (2,*) J (3,*) K
NHCELLSUM	I	1	Number of output cells for time-series data (total number of overall domains for automatic domain decomposition)
LHCELL	I	NPNTSZ	Order in which the output cells for time-series data are specified as input data
LENDF	I	1	Method for specifying the end file to be output =0 Step =1 Time
NENDF	I	1	Number of output times/steps for end files
IENDF0	I	1	Pointer to either the next output step or time for end files
IENDF	I	NOUTSZ	Output steps for end files
RREST	R	NRSTSZ	Output times for restart data
RLIST	R	NOUTSZ	Output times for results list
RHIST0	R	1	Next output time for time-series data
RHIST	R	1	Time interval at which time-series data is output
RENDF	R	NOUTSZ	Output times for end files
RFILE	R	3	Specify the output time for a boundary value file

			(1) Output start time [s] (2) Output end time [s] (3) Output interval [s]

Label name			PROPTY (physical properties)
Variable name	Type	Size	Description
RHO	R	1	Density of seawater [kg/m^3]
RHOA	R	1	Density of air [kg/m^3]
ADRHO	R	1	=RHOA/RHOW
AMUH	R	1	Horizontal viscosity coefficient of seawater [Pa･s]
AMUV	R	1	Vertical viscosity coefficient of seawater [Pa･s]
ANUH	R	1	Horizontal kinematic viscosity coefficient of seawater [m^2/s]
ANUV	R	1	Vertical kinematic viscosity coefficient of seawater [m^2/s]

Label name			RGWAVE (small-amplitude wave parameters)
Variable name	Type	Size	Description
AMP	R	1	Half of the amplitude of the small-amplitude wave [m]
TTT	R	1	Period of the small-amplitude wave [s]
ALL	R	1	Wavelength of the small-amplitude wave [m]
HHH	R	1	Water depth [m]
AXX	R	1	Mesh width [m]

Label name			TABLEI and TABLER (variables related to time-seris input data)
Variable name	Type	Size	Description
NTIMSZ			Upper limit on the number of time-variable pairs in a time-series input table =50000 (fixed parameter)
NTBLSZ			Upper limit on the number of time-series input tables =200 (fixed parameter)
NTABLE	I	1	Number of time-series input tables

ITABLE	I	NTBLSZ	Number of time-variable pairs in each table
TTABLE	R	NTIMSZ, NTBLSZ	Time data in each table
VTABLE	R	NTIMSZ, NTBLSZ	Value of a variable in each table
TABLE	R	NTBLSZ	Current-time value that was interpolated from each table

Label name			TIMEI and TIMER (variables related to time and steps)
Variable name	Type	Size	Description
ISTEP	I	1	Current step
MAXSTP	I	1	Maximum number of steps for calculation
IDT	I	1	Method for setting the time step =0 Uses a constant value =1 Sets each time step based on the conditions for stabilizing numerical values
NITER	I	1	Iteration count for the Leap frog method for motion equations
MXITER	I	1	Upper limit on the iteration count for the Leap frog method for motion equations
LSTART	I	1	Flag for restart calculation =0 Performs restart calculation, starting from the beginning =1 Performs restart calculation, starting from the specified start step =2 Performs restart calculation, starting from the specified start time
TIME	R	1	Current time [s]
DT	R	1	Time step [s]
DTOLD	R	1	Immediately preceding time step [s]
DTV	R	1	=DT
RSTART	R	1	Calculation start time [s]
REND	R	1	Calculation end time [s]
DTCNST	R	1	Constant value of the time step [s]

DTSAFE	R	1	Safety factor when setting the variable time step
DTMIN	R	1	Lower limit on the variable time step [s]
DTMAX	R	1	Upper limit on the variable time step [s]
VELMAX	R	1	Maximum flow velocity at the current time [m/s]

Label name			TURBR (parameters for k-ε and SGS models)
Variable name	Type	Size	Description
AKAR	R	1	Karman constant κ(=0.4)
TAA	R	1	Constant used for logarithmic law (=5.5)
CMU	R	1	C_{μ} (=0.09)
SGK	R	1	σ_k (=1.0)
SGE	R	1	σ_ε (=1.3)
TC1	R	1	$C_{\varepsilon 1}$ (=1.44)
TC2	R	1	$C_{\varepsilon 2}$ (=1.92)
TCE	R	1	SGS model's C_m(=0.31)
AKMIN	R	1	Minimum k value
EPMIN	R	1	Minimum ε value
TVSMAX	R	1	Maximum vertical turbulent kinematic viscosity coefficient
TVSMIN	R	1	Minimum vertical turbulent kinematic viscosity coefficient

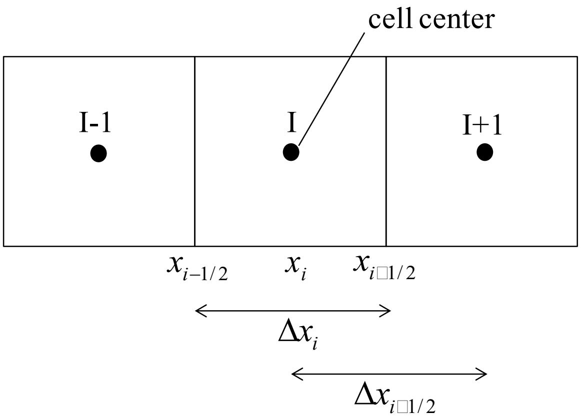
Label name			VVMAX (upper limit on flow velocity)
Variable name	Type	Size	Description
VVMAX	R	1	Upper limit on flow velocity [m/s]

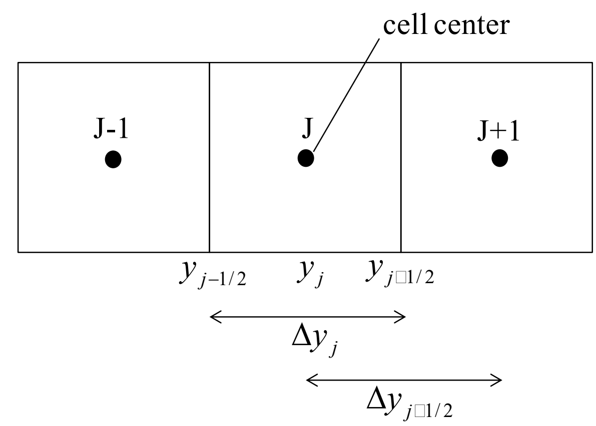
3.2.2. Major Array Variables

エラー！ 参照元が見つかりません。 through エラー！ 参照元が見つかりません。 list the major array variables passed as arguments. MX_PARNT, MY_PARNT, and MZ_PARNT in the array size column indicate the number of mesh partitions in the overlap portion between the parent area and your own area.

Table 0-3-20 Array Variables of Double-precision Real Data Type

Array name	Size	Description
XC	8,MX,MY	X-directional grid constant (1,I,J) Coordinate value $x_{i+1/2}$ of a grid point For a plane coordinate system, $x_{i+1/2}$ is the input value. For a spherical coordinate system, $x_{i+1/2} = \frac{\pi}{180} R \left(\phi_{i+1/2} - \phi_{CEN}\right) \cos\theta_j$ when the longitude of the input value is $\phi_{i+1/2}$ and the latitude of the Y-directional cell center is θ_j, where R is the radius of the Earth and ϕ_{CEN} is the longitude of the center of the innermost area. The values below are calculated from $x_{i+1/2}$. (2,I,J) Coordinate value of a cell center: $x_i \equiv \left(x_{i-1/2} + x_{i+1/2}\right)/2$ (3,I,J) Distance between cell centers: $\Delta x_{i+1/2} \equiv x_{i+1} - x_i$ (4,I,J) Distance between grid points: $\Delta x_i \equiv x_{i+1/2} - x_{i-1/2}$ (5,I,J) $1/\Delta x_{i+1/2}$ (6,I,J) $1/\Delta x_i$

		(7,I,J) $\Delta x_{i+1} / (\Delta x_i + \Delta x_{i+1})$ (8,I,J) $\Delta x_i / (\Delta x_i + \Delta x_{i+1})$  *For a plane coordinate system, the values remain unchanged even you change “J”.
XCP	8, MX,MY	X-directional grid constant 2 Same as XC except that coordinate value $x_{i+1/2}$ of a grid point in a spherical coordinate system is computed as follows. $x_{i+1/2} = \frac{\pi}{180}(\phi_{i+1/2} - \phi_{CEN}) R \cos\theta_{j+1/2}$ when the longitude of the input value is $\phi_{i+1/2}$ and the latitude of the Y-directional grid point is $\theta_{j+1/2}$. *For a plane coordinate system, XCP is exactly the same as XC.
YC	8,MY	Y-directional grid constants (1,J) Coordinate value $y_{j+1/2}$ of a grid point For a plane coordinate system, $y_{j+1/2}$ is the input value. For a spherical coordinate system, $y_{j+1/2} = \frac{\pi}{180} R(\theta_{i+1/2} - \theta_{CEN})$ when the latitude of the input

		value is $\theta_{j+1/2}$ where R is the radius of the Earth and θ_{CEN} is the latitude of the center of the innermost area. The values below are calculated from $y_{j+1/2}$. (2,J) Coordinate value of a cell center: $y_j \equiv \left(y_{j-1/2} + y_{j+1/2}\right)/2$ (3,J) Distance between cell centers: $\Delta y_{j+1/2} \equiv y_{j+1} - y_j$ (4,J) Distance between grid points: $\Delta y_j \equiv y_{j+1/2} - y_{j-1/2}$ (5,J) $1/\Delta y_{j+1/2}$ (6,J) $1/\Delta y_j$ (7,J) $\Delta y_{j+1}/\left(\Delta y_j + \Delta y_{j+1}\right)$ (8,J) $\Delta y_j/\left(\Delta y_j + \Delta y_{j+1}\right)$ 
ZC	8,MZ	Z-directional grid constants (1,K) Coordinate value $z_{k+1/2}$ of a grid point The values below are calculated from $z_{k+1/2}$.

		(2,K) Coordinate value of a cell center: $z_k \equiv (z_{k-1/2} + z_{k+1/2})/2$ (3,K) Distance between cell centers: $\Delta z_{k+1/2} \equiv z_{k+1} - z_k$ (4,K) Distance between grid points: $\Delta z_k \equiv z_{k+1/2} - z_{k-1/2}$ (5,K) $1/\Delta z_{k+1/2}$ (6,K) $1/\Delta z_k$ (7,K) $\Delta z_{k+1}/(\Delta z_k + \Delta z_{k+1})$ (8,K) $\Delta z_k/(\Delta z_k + \Delta z_{k+1})$
YCOS	MY	Correction parameter for latitudes. $\cos\theta_j$ YCOS=1 for a plane coordinate system.
YCOSP	MY	Correction parameter for latitudes. $\cos\theta_{j+1/2}$ YCOSP=1 for a plane coordinate system.
YSIN	MY	Parameter for calculating Coriolis forces. $\sin\theta_j$
YSINP	MY	Parameter for calculating Coriolis forces. $\sin\theta_{j+1/2}$
XC_REF	8,MX	X-directional grid constant (for visualization)
YC_REF	8,MY	Y-directional grid constant (for visualization)
XC_ML	8,MX_ML	XC for the parent area
YC_ML	8,MY_ML	YC for the parent area
ZC_ML	8,MZ_ML	ZC for the parent area
GV	MX,MY,MZ	Fluid porosity γ_v * γ_v for obstacles and virtual cells is 1.0.
GX	MX,MY,MZ	X-directional permeability γ_x

		* γ_x for obstacle insides and wall surfaces is 1.0.
GY	MX,MY,MZ	Y-directional permeability γ_y * γ_y for obstacle insides and wall surfaces is 1.0.
GZ	MX,MY,MZ	Z-directional permeability γ_z * γ_z for obstacle insides and wall surfaces is 1.0.
GV0	MX,MY,MZ	γ_v that is determined without considering a permeable structure *If there is no permeable structure, GV0, GX0, GY0, and GZ0 are equal to GV.
GX0	MX,MY,MZ	γ_x that is determined without considering a permeable structure
GY0	MX,MY,MZ	γ_y that is determined without considering a permeable structure
GZ0	MX,MY,MZ	γ_z that is determined without considering a permeable structure
GVD	MX,MY,MZ	γ_v for permeable structures *If there is no permeable structure, GVD, GXD, GYD, and GZD are equal to 1.
GXD	MX,MY,MZ	γ_x for permeable structures
GYD	MX,MY,MZ	γ_y for permeable structures
GZD	MX,MY,MZ	γ_z for permeable structures
CMD	MX,MY,MZ	Inertia force coefficient C_M for permeable structures
CDD	MX,MY,MZ	Drag coefficient C_D for permeable structures
COE1D	MX,MY,MZ	Coefficient a used for the Dupuit-Forchheimer model $a = \alpha_0 \frac{(1-\gamma_{v,2})^3}{\gamma_{v,2}{}^2} \frac{\nu}{D^2}$
COE2D	MX,MY,MZ	Coefficient b used for the Dupuit-Forchheimer model $b = \beta_0 \frac{1-\gamma_{v,2}}{\gamma_{v,2}{}^3} \frac{1}{D}$

GX_ML	MX_PARNT, MY_PARNT, MZ_PARNT	GX for the parent area
GY_ML	MX_PARNT, MY_PARNT, MZ_PARNT	GY for the parent area
GZ_ML	MX_PARNT, MY_PARNT, MZ_PARNT	GZ for the parent area
UU	MX,MY,MZ	X-directional component of flow velocity [m/s]
VV	MX,MY,MZ	Y-directional component of flow velocity [m/s]
WW	MX,MY,MZ	Z-directional component of flow velocity [m/s]
HU	MX,MY,MZ	□hu□ [m/s] *See Section 1.4.1.2.
HV	MX,MY,MZ	□hv□ [m/s]
HW	MX,MY,MZ	□hw□ [m/s]
PP	MX,MY,MZ	Pressure [Pa]
AK	MX,MY,MZ	Turbulence kinetic energy [m^2/s^2)
EP	MX,MY,MZ	Turbulence kinetic energy dissipation rate [m^2/s^3]
TMU	MX,MY,MZ	Turbulent kinematic viscosity [m^2/s]
FF	MX,MY,MZ	Volume ratio h_F of water within a cell *Refer to Section 1.4.1.2.
RHOW	MX,MY,MZ	Density [kg/m^3]
HH	MX,MY	Water level [m]
TMUBW	MX,MY	Kinematic viscosity coefficient ν_{BW} [m^2/s] of a breaking-wave model
TIMBW	MX,MY	Time t_0 [s] when a wave starts breaking
PATM	MX,MY	Atmospheric pressure [Pa] at sea level
DPS	MX,MY	Work area for pressure calculation
WX	MX,MY	X-directional component of ocean surface wind speed [m/s]
WY	MX,MY	Y-directional component of ocean surface wind speed [m/s]
UN	MX,MY,MZ	UU for the immediately preceding step
VN	MX,MY,MZ	VV for the immediately preceding step

WN	MX,MY,MZ	WW for the immediately preceding step
TN	MX,MY,MZ	TT for the immediately preceding step
CN	MX,MY,MZ	CC for the immediately preceding step
AKN	MX,MY,MZ	AK for the immediately preceding step
EPN	MX,MY,MZ	EP for the immediately preceding step
RL	MX,MY,MZ	Turbulence length scale [m]
AD0	MX,MY,MZ	Diagonal component of Poisson's equation
AL0	3,MX,MY,MZ	Non-diagonal component of Poisson's equation
AD	MX,MY,MZ	Diagonal component of the pressure correction equation
AL	3,MX,MY,MZ	Non-diagonal component (lower left) of the pressure correction equation
AU	3,MX,MY,MZ	Non-diagonal component (upper right) of the pressure correction equation
BB	MX,MY,MZ	Right-hand side of the pressure correction equation
DP	MX,MY,MZ	Pressure correction value [Pa]
WRK1	MX,MY,MZ	Work area
WRK2	MX,MY,MZ	Work area
WRK3	MX,MY,MZ	Work area
WRK4	MX,MY,MZ	Work area
WRK5	MX,MY,MZ	Work area
WRK6	MX,MY,MZ	Work area
WRK7	MX,MY,MZ	Work area
WRK8	MX,MY,MZ	Work area
VLEND	MX,MY,16	Aggregate data for the end file (*,*, 1) Water depth (*,*, 2) Initial water level (*,*, 3) Highest water level (*,*, 4) Lowest water level (*,*, 5) Maximum flow velocity (absolute value) (*,*, 6) Maximum flow velocity (X-directional component) (*,*, 7) Maximum flow velocity (Y-directional component) (*,*, 8) Maximum wind speed (absolute value) (*,*, 9) Maximum wind speed (X-directional component) (*,*,10) Maximum wind speed (Y-directional component)

		(*,*,11) Lowest atmospheric pressure
TMEND	MX,MY,6	Aggregate data 2 for the end file (*,*,1) Time when the water level reaches the highest point (*,*,2) Time when the water level reaches the lowest point (*,*,3) Time when the first tsunami wave arrives (*,*,4) Time when the flow velocity reaches the highest value (*,*,5) Time when the wind speed reaches the highest value (*,*,6) Time when the atmospheric pressure reaches the lowest value
FRIC	MX,MY,MZ	Friction coefficient
HDEP	MX,MY	Water depth (z coordinate value of the seabed surface)
AMNG	MX,MY	Manning's roughness
CD	MX,MY	Wind friction coefficient
HX	MX,MY	Water level (work area)
HY	MX,MY	Water level (work area)
HHW	MX,MY	Weighting coefficient for the overlap zone at the nesting boundary
UU_ML	MX_PARNT, MY_PARNT, MZ_PARNT	UU for the parent area
VV_ML	MX_PARNT, MY_PARNT, MZ_PARNT	VV for the parent area
WW_ML	MX_PARNT, MY_PARNT, MZ_PARNT	WW for the parent area
AK_ML	MX_PARNT, MY_PARNT, MZ_PARNT	AK for the parent area
EP_ML	MX_PARNT, MY_PARNT, MZ_PARN	EP for the parent area

	T	
HH_ML	MX_PARNT, MY_PARNT	HH for the parent area
HDEP_ML	MX_PARNT, MY_PARNT	HDEP for the parent area
BUF	NBUFSIZE	MPI communication buffer
HHBCN	MX,MY	HH_ML interpolation value
UUBCN	NXY,MZ,4	UU_ML interpolation value (*,*,1) Southern boundary (*,*,2) Western boundary (*,*,3) Eastern boundary (*,*,4) Northern boundary
VVBCN	NXY,MZ,4	VV_ML interpolation value
WWBCN	NXY,MZ,4	WW_ML interpolation value
AKBCN	NXY,MZ,4	AK_ML interpolation value
EPBCN	NXY,MZ,4	EP_ML interpolation value
HTDST1	MX,MY	Elevation [m] of the ground where the building has been destroyed (HTDST2<=HTDST1<HDEP)
HTDST2	MX,MY	Elevation [m] of the ground where the building has been completely eliminated (HTDST2<HDEP)
TMDST	MX,MY	Time [s] when the building is destroyed
FALLWX	MX,MY	Quantity [m^3/s^2] added to the X-directional momentum conservation equation in the overfall model
FALLWY	MX,MY	Quantity [m^3/s^2] added to the Y-directional momentum conservation equation in the overfall model
FALLWZ	MX,MY	Quantity [m^3/s^2] added to the Z-directional momentum conservation equation in the overfall model
DHX	MX,MY	Total water depth [m] in the position where FALLWX is added
DHY	MX,MY	Total water depth [m] in the position where FALLWY is added

CFALLWX	MX,MY	Loss coefficient C_{fw} (for X-directional boundary surfaces) of momentum due to overfall
CFALLWY	MX,MY	Loss coefficient C_{fw} (for Y-directional boundary surfaces) of momentum due to overfall
DFALLWNX	MX,MY	Distribution rate $C_{fw,dist,norm}$ (for X-directional boundary surfaces) towards the normal direction when momentum due to overfall is distributed horizontally
DFALLWNY	MX,MY	Distribution rate $C_{fw,dist,norm}$ (for Y-directional boundary surfaces) towards the normal direction when momentum due to overfall is distributed horizontally
DFALLWTX	MX,MY	Distribution rate $C_{fw,dist,\mathrm{tang}}$ (for X-directional boundary surfaces) towards the tangential direction when momentum due to overfall is distributed horizontally
DFALLWTY	MX,MY	Distribution rate $C_{fw,dist,\mathrm{tang}}$ (for Y-directional boundary surfaces) towards the tangential direction when momentum due to overfall is distributed horizontally

Table 0-3-21 Array Variables of Integer Type

Array name	Size	Description
INDP	MX,MY,MZ	Index for determining whether the target cell is a computational cell =0 Obstacle or virtual cell =1 Computational cell
INDU	MX,MY,MZ	Index for calculating the X-directional component of flow velocity =-4 Inside of the obstacle =-3 Point on the plane obstacle

		=-2 Wall surface (other than a point on the plane obstacle) =-1 Flow velocity fixed boundary =0 Free inflow/outflow boundary =1 Flow velocity calculation point
INDV	MX,MY,MZ	Index for calculating the Y-directional component of flow velocity *The values of this index have the same meanings as those of INDU.
INDW	MX,MY,MZ	Index for calculating the Z-directional component of flow velocity *The values of this index have the same meanings as those of INDU.
INDK	MX,MY,MZ	Number of wall surfaces that are in contact (Used for calculation with the wall function)
INDP_ML	MX_PARNT, MY_PARNT, MZ_PARNT	INDP for the parent area
INDU_ML	MX_PARNT, MY_PARNT, MZ_PARNT	INDU for the parent area
INDV_ML	MX_PARNT, MY_PARNT, MZ_PARNT	INDV for the parent area
INDW_ML	MX_PARNT, MY_PARNT, MZ_PARNT	INDW for the parent area
KF	MX,MY	Z-directional cell index for the cell that contains the water surface *KF=MZ when all the objects arranged in the Z direction are obstacles.
KG	MX,MY	Z-directional cell index for the cell that touches the seabed. *KG=MZ when all the objects arranged in the Z direction are obstacles.
KP	MX,MY	Z-directional cell index for the cell(s) for which the pressure boundary conditions are set *If the boundary conditions are set for two or more cells, select

		the smallest value. In most cases, KP=KF.
KH	MX,MY	KF for the immediately preceding step
KF_ML	MX_PARNT, MY_PARNT	KF for the parent area
KG_ML	MX_PARNT, MY_PARNT	KG for the parent area
IBUF	NBUFSIZE	MPI communication buffer
I_ML	2,MX_ML	Array for converting cell index I for the parent area into cell index I for the child area located in the same position as that parent area *If the mesh size of the child area is smaller than that of the parent area, there are two or more cell indexes for that child area. In this case, select the greatest value of these indexes as the I_ML value. Specify I_ML=0 for the area that does not contain a child area. *(1, *) and (2, *) include the same value. They are not distinguished.
J_ML	2,MY_ML	Same as I_ML
K_ML	2,MZ_ML	Same as I_ML
I_NS	2,MX	Array for converting cell index I for the child area into cell index I for the parent area located in the same position as that child area. *(1, *) and (2, *) include the same value. They are not distinguished.
J_NS	2,MY	Same as I_NS
K_NS	2,MZ	Same as I_NS
IDST	MX,MY	Flag for destruction processing =-92 Destruction ongoing (destruction by debris) =-91 Destruction ongoing (destruction by fluid) =-12 Destruction completed (destruction by debris) =-11 Destruction completed (destruction by fluid) =-1 Non-wooden building (undestroyed) =0 No building =1 Wooden building (likely to be destroyed) >1 Used when a different destruction standard is introduced. Currently not used.

KBLC	MX,MY	Flag for blocking processing =0 No blocking performed >0 Blocking performed. (The value is Z-directional cell index value for the bottom of debris.)

Table 0-3-22 Array Variables Defined in MOD_LIST

Array name	Size	Description
LLWALL	8,MLWALL	List of wall surface boundary positions (excluding points on plane obstacles) (1,*) X-directional index for the wall surface (2,*) Y-directional index for the wall surface (3,*) Z-directional index for the wall surface (4,*) Normal direction of the wall =0 –X direction =1 +X direction =2 –Y direction =3 +Y direction =4 –Z direction =5 +Z direction (5,*) Pointer to common variable IWALL When the pointer is 0, the default conditions are applied. (6,*) Boundary conditions of flow velocity =0 Slip =1 No slip =2 Wall function =3 Fixed flow velocity in the tangential direction
LLWALP	8,MLWALP	List of wall surface boundary positions on plane obstacles (1,*) X-directional index for the wall surface (2,*) Y-directional index for the wall surface (3,*) Z-directional index for the wall surface (4,*) Normal direction of the wall =0 –X direction =1 +X direction

		=2 –Y direction =3 +Y direction =4 –Z direction =5 +Z direction (5,*) Pointer to common variable IWALL When the pointer is 0, the default conditions are applied. (6,*) Boundary conditions of flow velocity =0 Slip =1 No slip =2 Wall function =3 Fixed flow velocity in the tangential direction
LLWALB	3,MLWALB	List of positions for storm surge barrier processing (1,*) X-directional index for the position for storm surge barrier processing (2,*) Y-directional index for the position for storm surge barrier processing (3,*) Z-directional index for the position for storm surge barrier processing *Data arranged in LLWALB is shown below. (*,1 to MLWALBX) Positions where the X direction is normal (*,MLWALBX+1 to MLWALB) Positions where the Y direction is normal
LLOFL	3,MLOFL	List of positions at which to apply overflow formulas (1,*) X-directional index for the position at which to apply the formulas (2,*) Y-directional index for the position at which to apply the formulas (3,*) Model applied =0 No model applied =1 Breakwater =2 Seawall on the east (north) =3 Seawall on the west (south) *Data arranged in LLOFL is shown below. (*,1 to MLOFLX) Positions where the X direction is normal (*,MLOFLX+1 to MLOFL) Positions where the Y direction is normal

HHOFL	MLOFL	Crown height [m] of the position at which to apply overflow formulas