



iRIC Software

Changing River Science

*Nays2D Flood
Solver Manual*

Directed by Yasuyuki Shimizu

Edited by Takuya Inoue

Eiichi Suzuki

Satomi Kawamura

Toshiki Iwasaki

Michihiro Hamaki

Kensuke Omura

Eriko Kakegawa

Tomohiko Yoshida

Last Update:2015.06.23

Release Date: 2015. 06.23

Copyright 2011 iRIC Project All Rights Reserved

Contents

I. OUTLINE	1
I.1 Nays2D Flood is	1
I.2 Features of the flow field simulation model.....	2
II. BASIC EQUATIONS.....	3
II.1 Basic flow equations	3
II.1.1 Basic equations in a rectangular coordinate system.....	3
II.1.2 Mapping to a general curvilinear coordinate system.....	4
II.1.3 Basic equations in a general curvilinear coordinate system.....	6
II.2 Simulation method applied to turbulent flow fields	8
II.2.1 Zero-equation model.....	8
II.3 Simulation method applied to bottom friction.....	9
II.4 Simulation method applied to a box culvert	10
II.4.1 Roles of a box culvert.....	10
II.4.2 Simulation method	10
II.4.3 Equations for calculating the discharge through the culvert	14
II.5 Simulation method for a sluice pipe	15
II.5.1 Roles of a sluice pipe	15
II.5.2 Simulation method	15
II.5.3 Equations for calculating the discharge through a sluice pipe	21
II.6 Simulation methods for pumps	22
II.6.1 Roles of pumps.....	22
II.6.2 The simulation method.....	22
III. CALCULATION CONDITION.....	24
III.1 Explanation of names of items in [Boundary Conditions]	24
III.2 Setting inflow/outflow boundary conditions	25
III.3 Setting initial water surface profile	27
III.4 Setting time condition.....	28
III.5 Other settings.....	29
III.6 Setting the inflow river	31
III.7 Setting riverbed roughness coefficient	33
III.8 Setting obstacles	34
III.9 Setting building occupancy ratio by area	35
III.10 Setting for a box culvert	37
III.11 Setting for a sluice pipe	40
III.12 Setting for a pump	43
IV. INPUT/OUTPUT FILES.....	46
IV.1 Input file for grid cell numbers of inflow.....	46
IV.2 Discharge file (Inflow boundary conditions)	47
IV.3 Discharge file (boundary conditions for banks).....	48

I. Outline

I.1 Nays2D Flood is

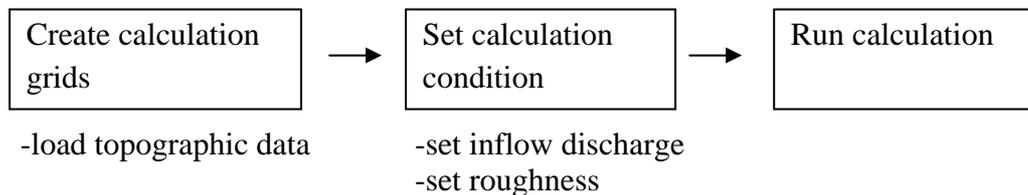
Nays 2D Flood is a flood flow analysis solver that relies on unsteady 2-dimensional plane flow simulation using boundary-fitted coordinates¹ as the general curvilinear coordinates. This solver adopts the 2-dimensional plane flow simulation of the Nays2D Solver developed by Professor Yasuyuki Shimizu of Hokkaido University for flood flow analysis.

The solver easily enables the user to set the inflow conditions of an arbitrary number of inflow rivers that enter from the upstream end or sides of a river. It has been applied to the flood flow analysis of small/mid-scale rivers. Because the solver does not require river channel data, it is also used for the flood process analysis of primitive rivers and rivers in developing countries.

¹ Boundary-fitted coordinates are set along the boundaries because a rectangular Cartesian coordinate system has difficulty reproducing complicated, winding boundaries. The governing equations in the rectangular Cartesian coordinate system are mapped to the general curvilinear coordinate system and are calculated. Because of its features, the system is also called a boundary-fitted coordinate system.

1.2 Features of the flow field simulation model

The data needed for an overflow calculation by Nays 2D Flood are a topographic data, and data of inflow discharge and roughness of each river or each inflow point. The basic process to begin the overflow calculation is the following;



And in this model, as necessary, it is able to consider or set as followings;

- ① The use of a general curvilinear coordinate system enables complicated boundaries and riverbed topography to be incorporated directly.
- ② As the difference method of the advection terms of equations of motion, the user can choose between upwind scheme (first-order accuracy) and constrained interpolation profile (CIP)².
- ③ It enables various parameters to be set for the boundary conditions of the upstream/downstream ends, such as the water surface elevation of the downstream end and the flow velocity of the upstream end.
- ④ As the boundary condition for a bank, the user can choose between [Inflow] and [Outflow]. By selecting [Inflow], it is possible to reproduce complicated flooding situations, such as bank overtopping on several rivers.
- ⑤ As [Initial Water Surface Profile], the user can choose between [Water depth = 0] and [Constant slope].
- ⑥ The bottom friction is set using Manning's roughness coefficient. The Manning's roughness coefficient can be set for each grid cell.
- ⑦ Obstacles in the area subject to simulation can be incorporated into the simulation by grid cell. A flag is set for each grid cell to define the obstacle. This enables the easy incorporation of bridge piers and other river structures into the simulation.
- ⑧ You can incorporate discharge by a box culvert, a sluice pipe or a pump into the simulation.

² A high-order accurate finite difference method. It interpolates a third-order polynomial, reduces numerical diffusion and enables local, high-accuracy interpolation.

II. Basic Equations

II.1 Basic flow equations

II.1.1 Basic equations in a rectangular coordinate system

The following are the basic equations in a rectangular coordinate system (x , y) before those equations are mapped to a general curvilinear coordinate system.

[Equation of continuity]

$$\frac{\partial h}{\partial t} + \frac{\partial(hu)}{\partial x} + \frac{\partial(hv)}{\partial y} = q + r \quad (1)$$

[Equations of motion]

$$\frac{\partial(uh)}{\partial t} + \frac{\partial(hu^2)}{\partial x} + \frac{\partial(huv)}{\partial y} = -hg \frac{\partial H}{\partial x} - \frac{\tau_x}{\rho} + D^x \quad (2)$$

$$\frac{\partial(vh)}{\partial t} + \frac{\partial(huv)}{\partial x} + \frac{\partial(hv^2)}{\partial y} = -hg \frac{\partial H}{\partial y} - \frac{\tau_y}{\rho} + D^y \quad (3)$$

where,

$$\frac{\tau_x}{\rho} = C_f u \sqrt{u^2 + v^2} \quad \frac{\tau_y}{\rho} = C_f v \sqrt{u^2 + v^2} \quad (4)$$

$$D^x = \frac{\partial}{\partial x} \left[v_t \frac{\partial(uh)}{\partial x} \right] + \frac{\partial}{\partial y} \left[v_t \frac{\partial(uh)}{\partial y} \right] \quad (5)$$

$$D^y = \frac{\partial}{\partial x} \left[v_t \frac{\partial(vh)}{\partial x} \right] + \frac{\partial}{\partial y} \left[v_t \frac{\partial(vh)}{\partial y} \right] \quad (6)$$

where, h is water depth, t is time, u is flow velocity in the x direction, v is flow velocity in the y direction, g is gravitational acceleration, H is water surface elevation, τ_x is riverbed shear stress in the x direction, τ_y is riverbed shear stress in the y direction, C_f is riverbed friction coefficient, v_t is eddy viscosity coefficient, ρ is the density of water, q is inflow through a box culvert, a sluice pipe or a pump per unit area and r is rainfall.

II.1.2 Mapping to a general curvilinear coordinate system

Next is the mapping of basic equations of 2-dimensional plane flow in a rectangular coordinate system to a general coordinate system (ξ, η) . By mapping the equations to a general coordinate system, the simulation mesh can be set to any shape according to the boundary conditions. The following is the method of mapping from a rectangular coordinate system to a general curvilinear coordinate system.

$$\frac{\partial}{\partial x} = \frac{\partial \xi}{\partial x} \frac{\partial}{\partial \xi} + \frac{\partial \eta}{\partial x} \frac{\partial}{\partial \eta} \quad (7)$$

$$\frac{\partial}{\partial y} = \frac{\partial \xi}{\partial y} \frac{\partial}{\partial \xi} + \frac{\partial \eta}{\partial y} \frac{\partial}{\partial \eta} \quad (8)$$

or

$$\begin{pmatrix} \frac{\partial}{\partial x} \\ \frac{\partial}{\partial y} \end{pmatrix} = \begin{pmatrix} \xi_x & \eta_x \\ \xi_y & \eta_y \end{pmatrix} \begin{pmatrix} \frac{\partial}{\partial \xi} \\ \frac{\partial}{\partial \eta} \end{pmatrix} \quad (9)$$

where,

$$\xi_x = \frac{\partial \xi}{\partial x}, \quad \xi_y = \frac{\partial \xi}{\partial y}, \quad \eta_x = \frac{\partial \eta}{\partial x}, \quad \eta_y = \frac{\partial \eta}{\partial y} \quad (10)$$

As well as,

$$\frac{\partial}{\partial \xi} = \frac{\partial x}{\partial \xi} \frac{\partial}{\partial x} + \frac{\partial y}{\partial \xi} \frac{\partial}{\partial y} \quad (11)$$

$$\frac{\partial}{\partial \eta} = \frac{\partial x}{\partial \eta} \frac{\partial}{\partial x} + \frac{\partial y}{\partial \eta} \frac{\partial}{\partial y} \quad (12)$$

or

$$\begin{pmatrix} \frac{\partial}{\partial \xi} \\ \frac{\partial}{\partial \eta} \end{pmatrix} = \begin{pmatrix} x_\xi & y_\xi \\ x_\eta & y_\eta \end{pmatrix} \begin{pmatrix} \frac{\partial}{\partial x} \\ \frac{\partial}{\partial y} \end{pmatrix} \quad (13)$$

where,

$$x_\xi = \frac{\partial x}{\partial \xi}, \quad x_\eta = \frac{\partial x}{\partial \eta}, \quad y_\xi = \frac{\partial y}{\partial \xi}, \quad y_\eta = \frac{\partial y}{\partial \eta} \quad (14)$$

Hence,

$$\begin{pmatrix} \frac{\partial}{\partial \xi} \\ \frac{\partial}{\partial \eta} \end{pmatrix} = \frac{1}{\xi_x \eta_y - \xi_y \eta_x} \begin{pmatrix} \eta_y & -\eta_x \\ -\xi_y & \xi_x \end{pmatrix} \begin{pmatrix} \frac{\partial}{\partial x} \\ \frac{\partial}{\partial y} \end{pmatrix} = \begin{pmatrix} x_\xi & y_\xi \\ x_\eta & y_\eta \end{pmatrix} \begin{pmatrix} \frac{\partial}{\partial x} \\ \frac{\partial}{\partial y} \end{pmatrix} \quad (15)$$

Assuming $J = \xi_x \eta_y - \xi_y \eta_x$, then,

$$\frac{1}{J} \begin{pmatrix} \eta_y & -\eta_x \\ -\xi_y & \xi_x \end{pmatrix} = \begin{pmatrix} x_\xi & y_\xi \\ x_\eta & y_\eta \end{pmatrix} \quad (16)$$

Consequently,

$$x_\xi = \frac{1}{J} \eta_y, \quad y_\xi = -\frac{1}{J} \eta_x, \quad x_\eta = -\frac{1}{J} \xi_y, \quad y_\eta = \frac{1}{J} \xi_x \quad (17)$$

or

$$\eta_y = J x_\xi, \quad \eta_x = -J y_\xi, \quad \xi_y = -J x_\eta, \quad \xi_x = J y_\eta \quad (18)$$

$$J = \xi_x \eta_y - \xi_y \eta_x = J^2 (x_\xi y_\eta - x_\eta y_\xi) \quad (19)$$

Thus,

$$J = \frac{1}{x_\xi y_\eta - x_\eta y_\xi} \quad (20)$$

Assuming (u^ξ, u^η) are (ζ, η) components of flow velocity, then

$$u^\xi = \xi_x u + \xi_y v \quad (21)$$

$$u^\eta = \eta_x u + \eta_y v \quad (22)$$

or

$$\begin{pmatrix} u^\xi \\ u^\eta \end{pmatrix} = \begin{pmatrix} \xi_x & \xi_y \\ \eta_x & \eta_y \end{pmatrix} \begin{pmatrix} u \\ v \end{pmatrix} \quad (23)$$

$$\begin{pmatrix} u \\ v \end{pmatrix} = \frac{1}{J} \begin{pmatrix} \eta_y & -\xi_y \\ -\eta_x & \xi_x \end{pmatrix} \begin{pmatrix} u^\xi \\ u^\eta \end{pmatrix} \quad (24)$$

II.1.3 Basic equations in a general curvilinear coordinate system

The following are the basic equations after the equations in a rectangular coordinate system (x, y) are mapped to a general curvilinear coordinate system.

[Equation of continuity]

$$\frac{\partial}{\partial t} \left(\frac{h}{J} \right) + \frac{\partial}{\partial \xi} \left(\frac{hu^\xi}{J} \right) + \frac{\partial}{\partial \eta} \left(\frac{hu^\eta}{J} \right) = \frac{q+r}{J} \quad (25)$$

[Equations of motion]

$$\begin{aligned} \frac{\partial u^\xi}{\partial t} + u^\xi \frac{\partial u^\xi}{\partial \xi} + u^\eta \frac{\partial u^\xi}{\partial \eta} + \alpha_1 u^\xi u^\xi + \alpha_2 u^\xi u^\eta + \alpha_3 u^\eta u^\eta = \\ -g \left[(\xi_x^2 + \xi_y^2) \frac{\partial H}{\partial \xi} + (\xi_x \eta_x + \xi_y \eta_y) \frac{\partial H}{\partial \eta} \right] \\ - C_f \frac{u^\xi}{hJ} \sqrt{(\eta_y u^\xi - \xi_y u^\eta)^2 + (-\eta_x u^\xi + \xi_x u^\eta)^2} + D^\xi \end{aligned} \quad (26)$$

$$\begin{aligned} \frac{\partial u^\eta}{\partial t} + u^\xi \frac{\partial u^\eta}{\partial \xi} + u^\eta \frac{\partial u^\eta}{\partial \eta} + \alpha_4 u^\xi u^\xi + \alpha_5 u^\xi u^\eta + \alpha_6 u^\eta u^\eta = \\ -g \left[(\eta_x \xi_x + \eta_y \xi_y) \frac{\partial H}{\partial \xi} + (\eta_x^2 + \eta_y^2) \frac{\partial H}{\partial \eta} \right] \\ - C_f \frac{u^\eta}{hJ} \sqrt{(\eta_y u^\xi - \xi_y u^\eta)^2 + (-\eta_x u^\xi + \xi_x u^\eta)^2} + D^\eta \end{aligned} \quad (27)$$

where,

$$\alpha_1 = \xi_x \frac{\partial^2 x}{\partial \xi^2} + \xi_y \frac{\partial^2 y}{\partial \xi^2}, \quad \alpha_2 = 2 \left(\xi_x \frac{\partial^2 x}{\partial \xi \partial \eta} + \xi_y \frac{\partial^2 y}{\partial \xi \partial \eta} \right), \quad \alpha_3 = \xi_x \frac{\partial^2 x}{\partial \eta^2} + \xi_y \frac{\partial^2 y}{\partial \eta^2} \quad (28)$$

$$\alpha_4 = \eta_x \frac{\partial^2 x}{\partial \xi^2} + \eta_y \frac{\partial^2 y}{\partial \xi^2}, \quad \alpha_5 = 2 \left(\eta_x \frac{\partial^2 x}{\partial \xi \partial \eta} + \eta_y \frac{\partial^2 y}{\partial \xi \partial \eta} \right), \quad \alpha_6 = \eta_x \frac{\partial^2 x}{\partial \eta^2} + \eta_y \frac{\partial^2 y}{\partial \eta^2} \quad (29)$$

$D^\xi =$

$$\left(\xi_x \frac{\partial}{\partial \xi} + \eta_x \frac{\partial}{\partial \eta} \right) \left[v_t \left(\xi_x \frac{\partial u^\xi}{\partial \xi} + \eta_x \frac{\partial u^\xi}{\partial \eta} \right) \right] + \left(\xi_y \frac{\partial}{\partial \xi} + \eta_y \frac{\partial}{\partial \eta} \right) \left[v_t \left(\xi_y \frac{\partial u^\xi}{\partial \xi} + \eta_y \frac{\partial u^\xi}{\partial \eta} \right) \right] \quad (30)$$

$D^\eta =$

$$\left(\xi_x \frac{\partial}{\partial \xi} + \eta_x \frac{\partial}{\partial \eta} \right) \left[v_t \left(\xi_x \frac{\partial u^\eta}{\partial \xi} + \eta_x \frac{\partial u^\eta}{\partial \eta} \right) \right] + \left(\xi_y \frac{\partial}{\partial \xi} + \eta_y \frac{\partial}{\partial \eta} \right) \left[v_t \left(\xi_y \frac{\partial u^\eta}{\partial \xi} + \eta_y \frac{\partial u^\eta}{\partial \eta} \right) \right] \quad (31)$$

$$\xi_x = \frac{\partial \xi}{\partial x}, \quad \xi_y = \frac{\partial \xi}{\partial y}, \quad \eta_x = \frac{\partial \eta}{\partial x}, \quad \eta_y = \frac{\partial \eta}{\partial y} \quad (32)$$

$$u^\xi = \xi_x u + \xi_y v, \quad u^\eta = \eta_x u + \eta_y v \quad (33)$$

$$J = \frac{1}{x_\xi y_\eta - x_\eta y_\xi} \quad (34)$$

Because when developed, the numbers of terms are huge, the following conditions are assumed for the advection terms of D^ξ and D^η in the equations of motion in a general curvilinear coordinate system to simplify the calculation.

- 1) The second derivative of a metric parameter is locally set as zero.
- 2) A quasi-general curvilinear coordinate system is adopted locally as for the advection terms.

Consequently, the advection terms can be expressed by approximation as follows:

$$D^\xi \simeq \frac{\partial}{\partial \xi} \left(v_i \xi_r^2 \frac{\partial u^\xi}{\partial \xi} \right) + \frac{\partial}{\partial \eta} \left(v_i \eta_r^2 \frac{\partial u^\xi}{\partial \eta} \right) \quad (35)$$

$$D^\eta \simeq \frac{\partial}{\partial \xi} \left(v_i \xi_r^2 \frac{\partial u^\eta}{\partial \xi} \right) + \frac{\partial}{\partial \eta} \left(v_i \eta_r^2 \frac{\partial u^\eta}{\partial \eta} \right) \quad (36)$$

where, ξ_r and η_r are parameters expressing the local ratio of grid size in a general curvilinear coordinate system over the corresponding distance on actual topography, and are defined by the following equation:

$$\frac{\Delta \xi}{\Delta \tilde{\xi}} = \xi_r, \quad \frac{\Delta \eta}{\Delta \tilde{\eta}} = \eta_r \quad (37)$$

To derive the approximate equations of D^ξ and D^η above are based on the following relationships by assuming a local orthogonal relationship between axes.

$$\xi_x^2 + \xi_y^2 = \xi_r^2 (\tilde{\xi}_x^2 + \tilde{\xi}_y^2) = \xi_r^2 (\sin^2 \theta + \cos^2 \theta) = \xi_r^2 \quad (38)$$

$$\xi_x \eta_x + \xi_y \eta_y = \xi_r \eta_r (\tilde{\xi}_x \tilde{\eta}_x + \tilde{\xi}_y \tilde{\eta}_y) = \xi_r \eta_r (-\cos \theta \sin \theta + \cos \theta \sin \theta) = 0 \quad (39)$$

$$\eta_x^2 + \eta_y^2 = \eta_r^2 (\tilde{\eta}_x^2 + \tilde{\eta}_y^2) = \eta_r^2 (\sin^2 \theta + \cos^2 \theta) = \eta_r^2 \quad (40)$$

$$J = \xi_x \eta_y - \xi_y \eta_x = \xi_r \eta_r (\tilde{\xi}_x \tilde{\eta}_y - \tilde{\xi}_y \tilde{\eta}_x) = \xi_r \eta_r (\sin^2 \theta + \cos^2 \theta) = \xi_r \eta_r \quad (41)$$

Where, θ is the angle between x-axis and ξ -axis (or y-axis and η -axis).

II.2 Simulation method applied to turbulent flow fields

Turbulent flow is irregular flow involving many small and large eddies. Nays2D Flood employs a zero-equation model for turbulent flow field simulation.

II.2.1 Zero-equation model

The eddy viscosity coefficient ν_t is generally expressed by the multiplying the representative velocity of turbulent flow ν and the representative length l of the turbulent flow. Therefore,

$$\nu_t = \nu l \quad (42)$$

In a flow field where water depth and riverbed roughness gently change in the cross-sectional direction, if we assume that eddy viscosity coefficients in the vertical and the horizontal directions have the same order and that bottom friction velocity and water depth are the dominant factors in momentum transfer, then the eddy viscosity coefficient ν_t is given by the following equation.

$$\nu_t = a u_* h \quad (43)$$

where, a is a proportional constant.

Fisher¹⁾ and Webel and Schatzmann²⁾ found in their experiments that the value of a in relation to momentum transfer in the vertical direction is about 0.07. Here, eddy viscosity coefficient ν_t is given by the following equation using the Karman constant ($\kappa=0.4$):

$$\nu_t = A \frac{\kappa}{6} u_* h + B \quad (44)$$

Since this modeling does not require any transportation equation for turbulence statistics, it is called the "zero-equation model." A and B are a user defined parameters for an eddy viscosity coefficient. Default value of A and B is 1 and 0, respectively. In horizontal two dimensional flow calculations, since three dimensional flow structures are completely neglected, unrealistic horizontal large vortices tend to be generated. The user can adjust A and B for modelling such vortices.

II.3 Simulation method applied to bottom friction

Nays-2D Flood sets the bottom friction by using the Manning's roughness coefficient. In Equation (4), riverbed shear stress τ_x and τ_y are expressed by riverbed friction coefficient C_f . The relationship between riverbed friction coefficient C_f and the Manning's roughness coefficient n_m is as follows:

$$C_f = \frac{gn_m^2}{h^{1/3}} \quad (45)$$

n_m can be given to each grid cell by setting [Cell attributes].

<References>

- 1) Fisher, H.B.: Longitudinal Dispersion and Turbulent Mixing in Open-Channel Flow, Annual Review of Fluid Mechanics, Vol. 5, pp.59-78, 1973.
- 2) Webel, G., Schatzmann, M.: Transverse Mixing in Open Channel Flow, Journal of Hydraulic Engineering, Vol. 110, No. 4, pp. 423-435, 1984.

II.4 Simulation method applied to a box culvert

II.4.1 Roles of a box culvert

A box culvert is a closed conduit buried in a road or bridge embankment. When more than two box culverts with the same dimensions are applied, to obtain the total drainage flow rate, first calculate the drainage discharge of one unit of box culvert, and then multiply that calculation result by the number of box culverts.

II.4.2 Simulation method

For simulation, the entrance and exit of the box culvert are regarded as a unit, and different cells are set for entrance and for the exit. The drainage flow rate of the box culvert is determined by the difference in water level between the entrance cell and the exit cell. Refer to the following schematic.

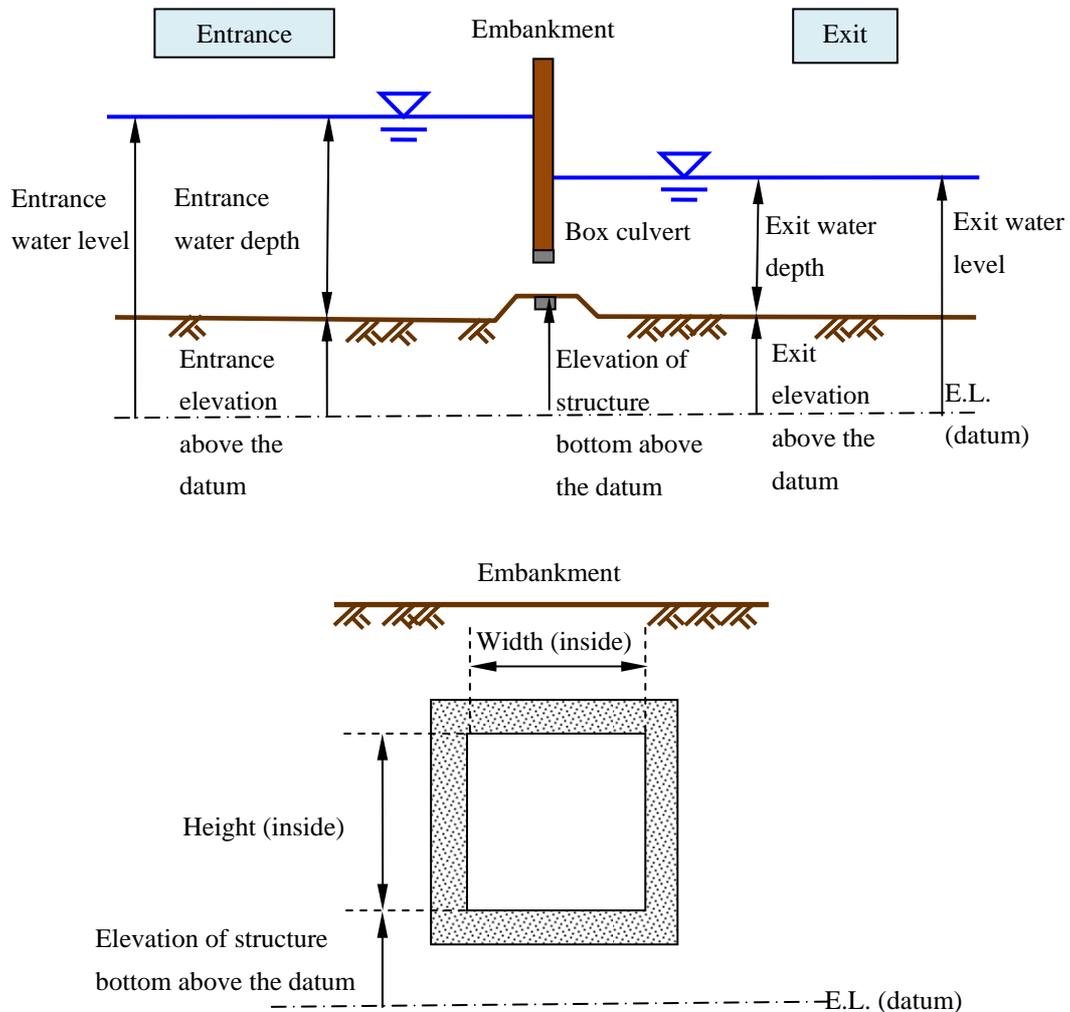


Figure II—1 Schematic of a box culvert (side view and section view)

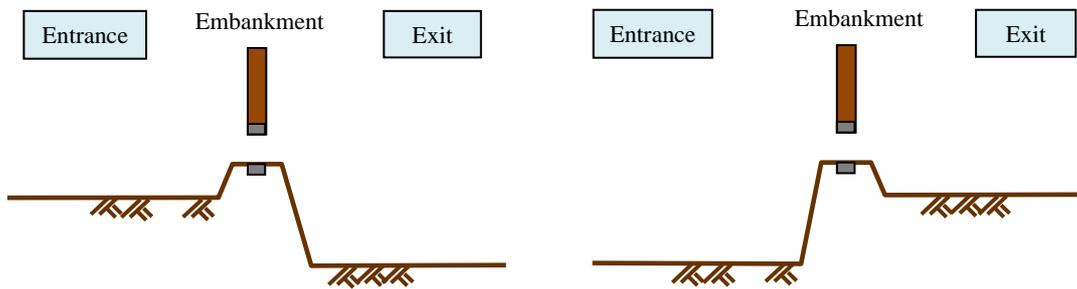
The water levels at the entrance and the exit of a box culvert are used in simulations as follows.

Entrance water level
 Entrance water level
 = Entrance elevation above the datum + Entrance water depth

Exit water level
 Exit water level = Exit elevation above the datum + Exit water depth

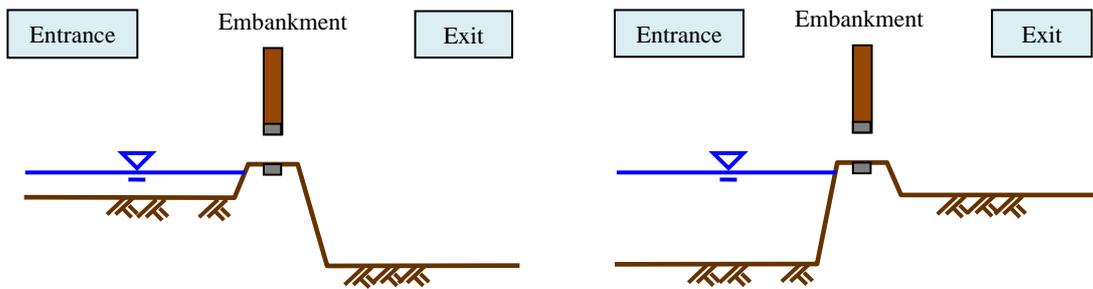
The simulation method for discharge of a culvert is as follows.

① When there is no water



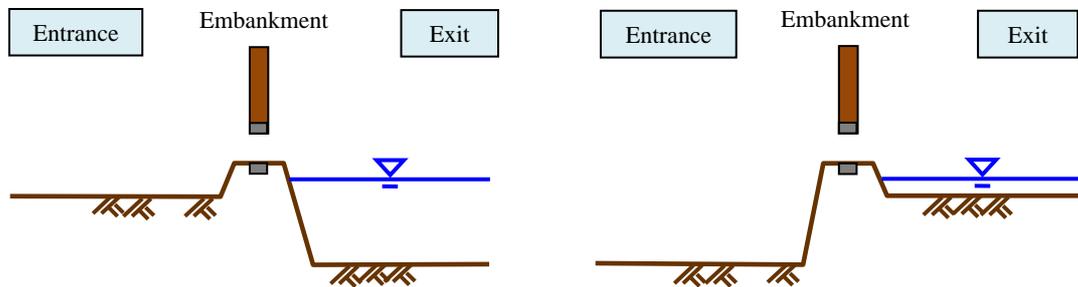
⇒ No water flow

② When “Entrance water level > Exit water level” and “Entrance water level \leq Elevation of structure bottom above the datum”



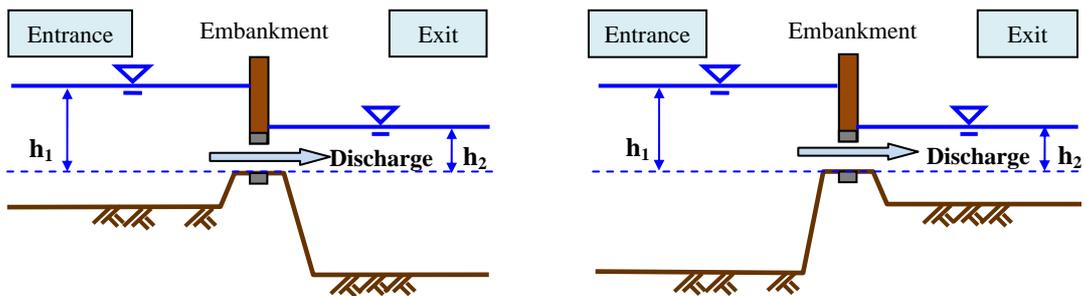
⇒ No water flow

- ③ When “Entrance water level < Exit water level” and “Exit water level \leq Elevation of structure bottom above the datum”



⇒ No water flow

- ④ When “Entrance water level > Exit water level” and “Entrance water level > Elevation of structure bottom above the datum”



The higher water depth between entrance and exit (h_1)

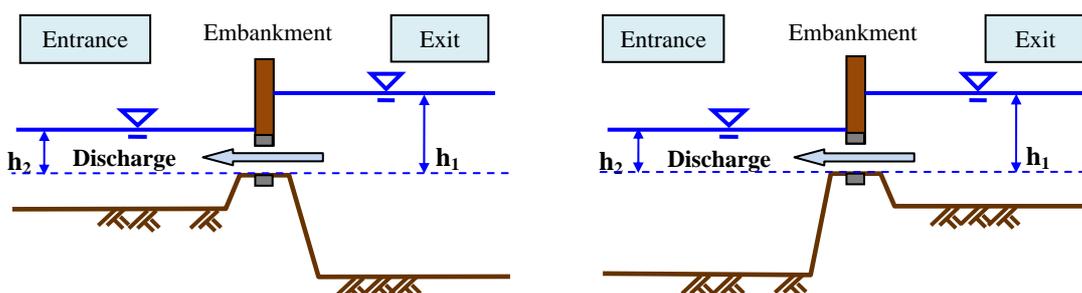
$$= \text{Entrance water level} - \text{Elevation of structure bottom above the datum}$$

The lower water depth between entrance and exit (h_2)

$$= \text{Exit water level} - \text{Elevation of structure bottom above the datum}$$

⇒ Water flows from entrance to exit. Discharge through a sluice pipe is calculated by equations (50) and (51) in II.4.3.

- ⑤ When “Entrance water level < Exit water level” and “Exit water level > Elevation of structure bottom above the datum”



The higher water depth between entrance and exit (h_1)

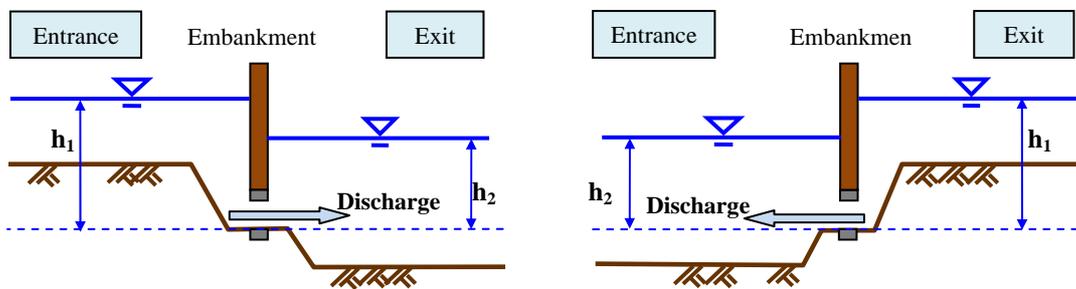
= Exit water level - Elevation of structure bottom above the datum

The lower water depth between entrance and exit (h_2)

= Entrance water level - Elevation of structure bottom above the datum

⇒ Water flows from exit to entrance. Discharge through a sluice pipe is calculated by equations (50) and (51) II.4.3.

Note: Even when the elevation of structure bottom is lower than the entrance or the exit elevation, the higher water depth between entrance and exit (h_1) or the lower water depth between entrance and exit (h_2) is obtained as the elevation difference from the elevation of the structure bottom.



II.4.3 Equations for calculating the discharge through the culvert

Discharge through the culvert is calculated by the equations below. The initial setting is submerged flow and free outflow. Any values can be set for the discharge coefficients C_1 and C_3 . Their initial values are set as $C_1=0.75$ and $C_3=0.79$.

$$\text{When } h_2 \geq H, Q = nC_1BH\sqrt{2g(h_1 - h_2)} \text{ is submerged flow} \quad (46)$$

$$\text{When } h_2 < H, Q = nC_3Bh_2\sqrt{2g(h_1 - h_2)} \text{ is free outflow} \quad (47)$$

However, when $h_1/h_2 \geq 3/2, h_2 = 2/3h_1$

Here, Q is discharge (m^3/s), n is the number of box culverts, B is width (m), H is height (m), h_1 is the water depth at the entrance/exit whose elevation is higher than the elevation of the structure bottom above the datum (m), h_2 is the water depth at the entrance/exit whose elevation is lower than the elevation of structure bottom above the datum (m) and g is the gravitational acceleration.

Note: As an option, you can add a case of subsurface flow. Any values can be set for the discharge coefficients C_1, C_2 and C_3 . Their initial values are set as $C_1=0.75, C_2=0.51$ and $C_3=0.79$.

$$\text{When } h_2 \geq H, \quad Q = nC_1BH\sqrt{2g(h_1 - h_2)} \quad \text{submerged flow}$$

$$\text{When } h_2 < H \quad h_1/H \geq 3/2, \quad Q = nC_2Bh_2\sqrt{2gh_1} \quad \text{interflow}$$

$$\text{When } h_2 < H \quad h_1/H < 3/2, \quad Q = nC_3Bh_2\sqrt{2g(h_1 - h_2)} \quad \text{free outflow}$$

However, when $h_1/h_2 \geq 3/2, h_2 = 2/3h_1$

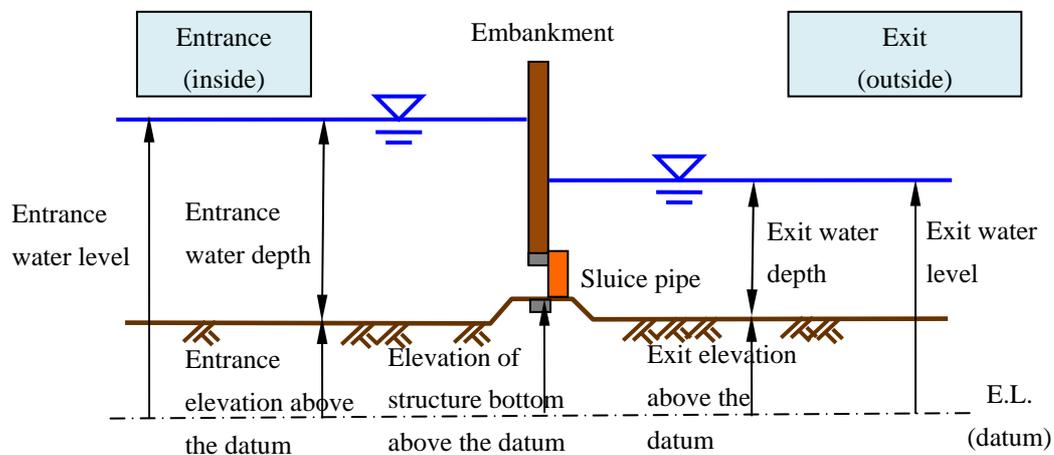
II.5 Simulation method for a sluice pipe

II.5.1 Roles of a sluice pipe

A sluice pipe is a closed conduit that is buried transverse to the embankment. It has a gate on the river channel side. At normal times, the gate is open to drain water from the area protected by the embankment. At times of flooding, the gate is closed to prevent river water from flowing back into the area protected by the embankment. When more than two sluice pipes of the same size are applied, to obtain the total drainage discharge, first calculate the drainage discharge of one unit of sluice pipe, and then multiply that calculation result by the number of sluice pipes.

II.5.2 Simulation method

For simulation, the sluice pipe entrance is set on the protected side of the embankment (hereinafter: “the inside”) and the sluice pipe exit is set on the river channel side of the embankment (hereinafter: “the outside”). Different cells are set for entrance and exit. The discharge through a sluice pipe is determined by the differences in water levels between the entrance cell and the exit cell, or the open or closed status of the gate. Refer to the following sluice pipe schematic.



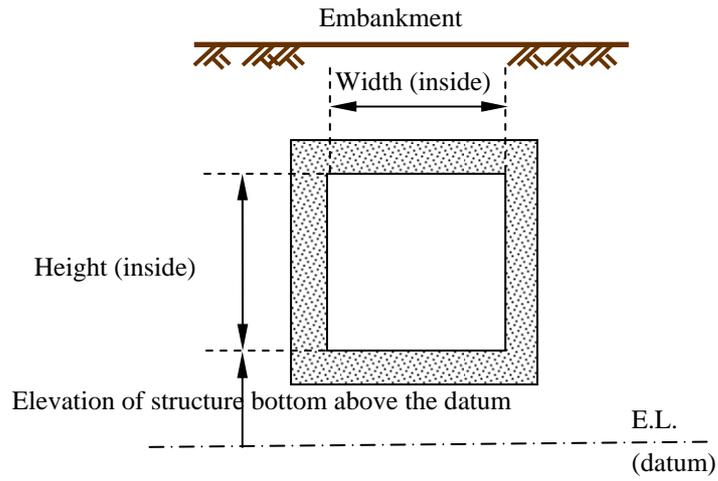


Figure II—2 Schematic of a sluice pipe (side view and section view)

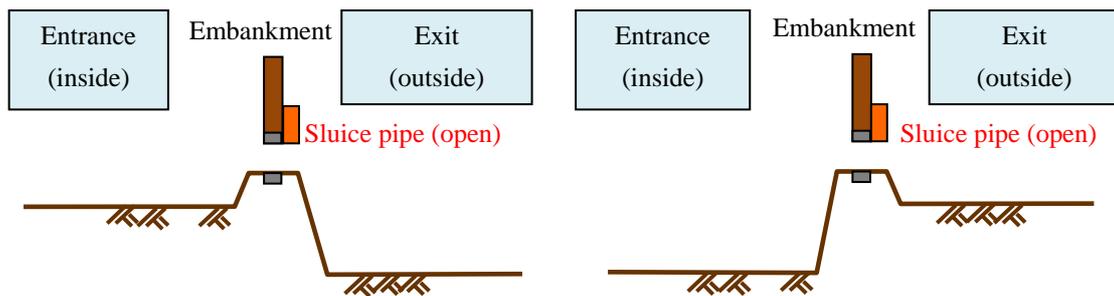
The water levels at the entrance and the exit of the sluice pipe are used for calculation as follows.

- Entrance water level (inside)
 - Entrance water level
 - = Entrance elevation above the datum + Entrance water depth
- Water level at the exit (outside)
 - 【When the water level at exit is not set】
 - Exit water level = Exit elevation above the datum + Exit water depth
 - 【When the water level at the exit is set】
 - Exit water level = The water level input by the time series

Simulation methods for discharge through a sluice pipe when the sluice pipe “automatically opens/closes” and when the “open/close of the sluice pipe is set by the data” are as follows.

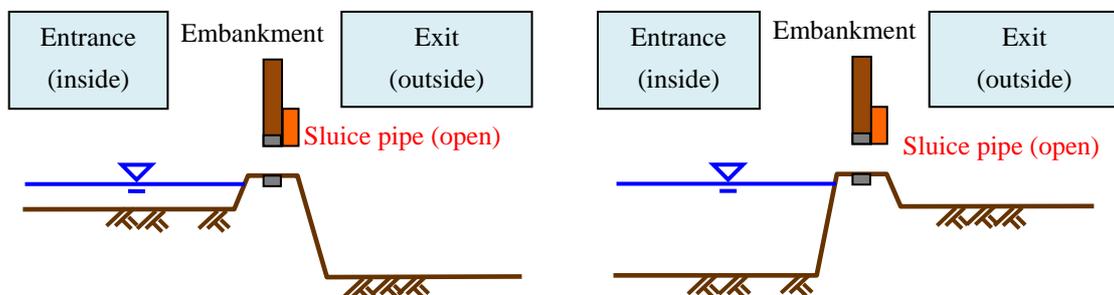
【When the sluice pipe automatically opens/closes】

① When there is no water



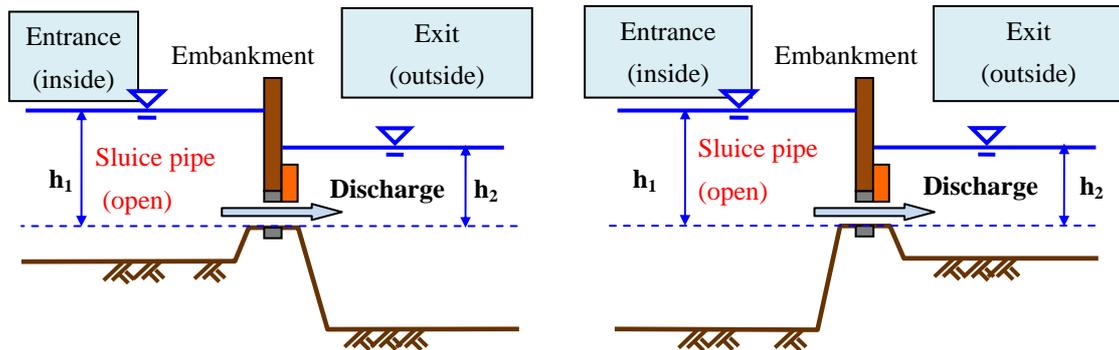
⇒ No water flow

② When “Entrance water level > Exit water level” and “Entrance water level ≤ Elevation of the structure bottom above the datum of the sluice pipe”



⇒ No water flow

- ③ When “Entrance water level > Exit water level” and “Entrance water level > Elevation of the structure bottom above the datum of the sluice pipe”



The higher water depth between entrance and exit (h_1)

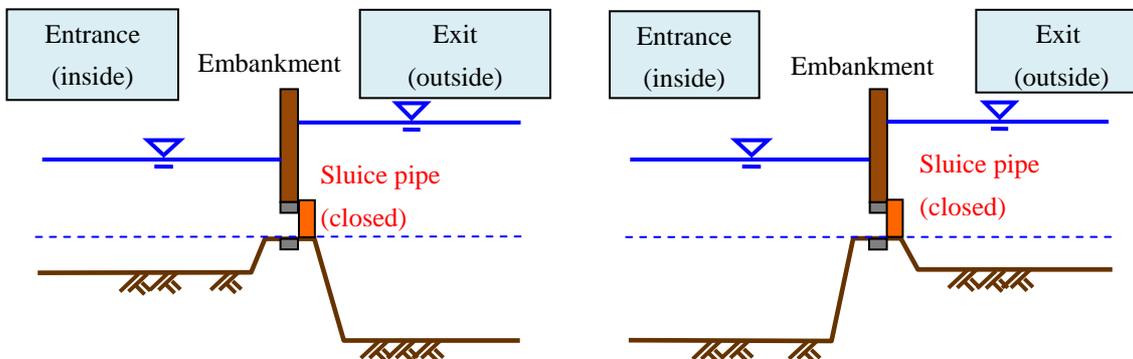
$$= \text{Entrance water level} - \text{Elevation of the structure bottom above the datum}$$

The lower water depth between entrance and exit (h_2)

$$= \text{Exit water level} - \text{Elevation of the structure bottom above the datum}$$

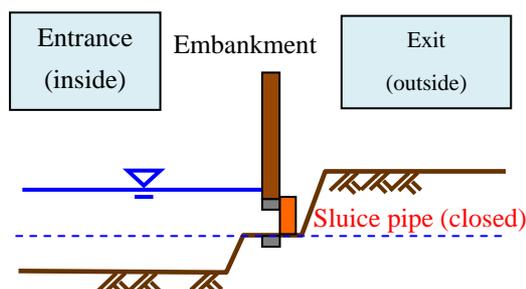
⇒ Water that flows from the entrance to the exit discharge through a sluice pipe is calculated by equations (50) and (51) II.4.3.

- ④ When “Entrance water level < Exit water level”



⇒ No water flows, because the sluice pipe gate is closed to prevent backflow from the river.

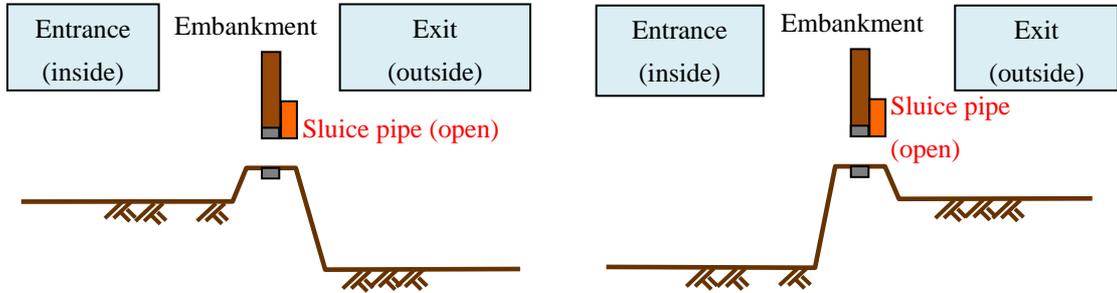
- ⑤ When “Entrance water level < Exit water level” and “Entrance water level > Elevation of the structure bottom above the datum of the sluice pipe”



⇒ No water flows when there is certain entrance water depth and the entrance water level is lower than the exit elevation above the datum.

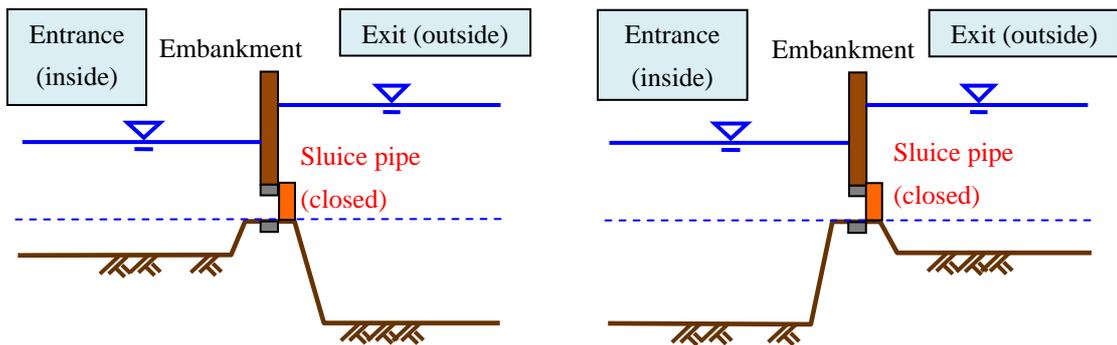
【When open/close of a sluice pipe is set by data】

① When there is no water



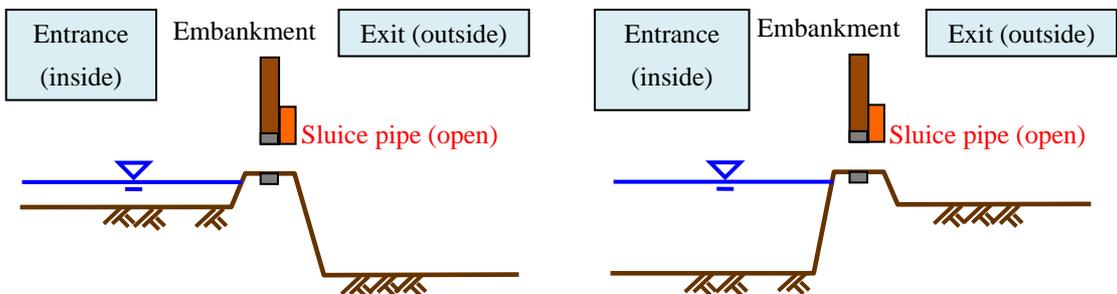
⇒ No water flow

② When “Sluice pipe is closed”



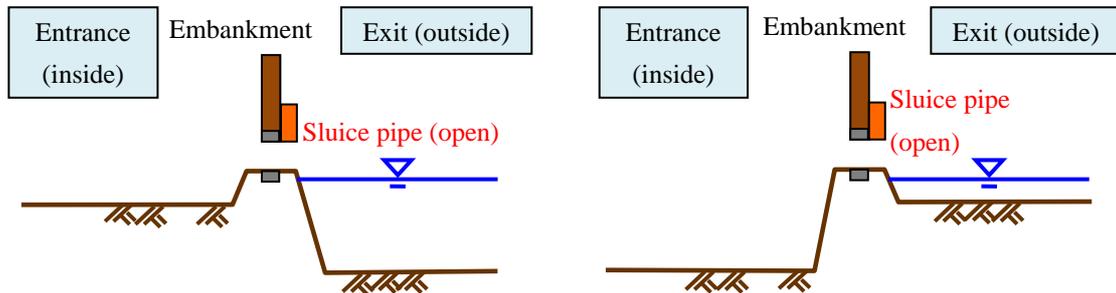
⇒ No water flow

③ When “Sluice pipe open”, “Entrance water level > Exit water level” and “Entrance water level ≤ Elevation of the sluice pipe bottom above the datum”



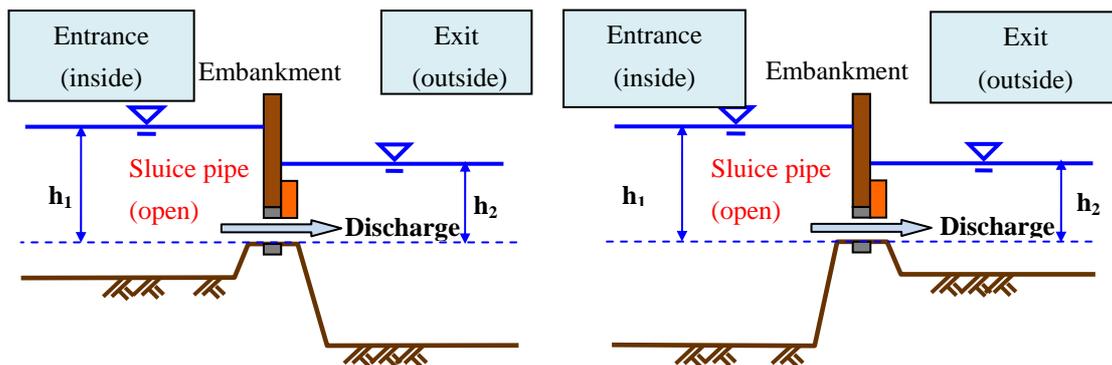
⇒ No water flow

- ④ When “Sluice pipe open”, “Entrance water level < Exit water level” and “Exit water level \leq Elevation of the sluice pipe bottom above the datum”



⇒ No water flow

- ⑤ When “Sluice pipe open”, “Entrance water level > Exit water level” and “Entrance water level \leq Elevation of the sluice pipe bottom above the datum”



The higher water depth between entrance and exit (h_1)

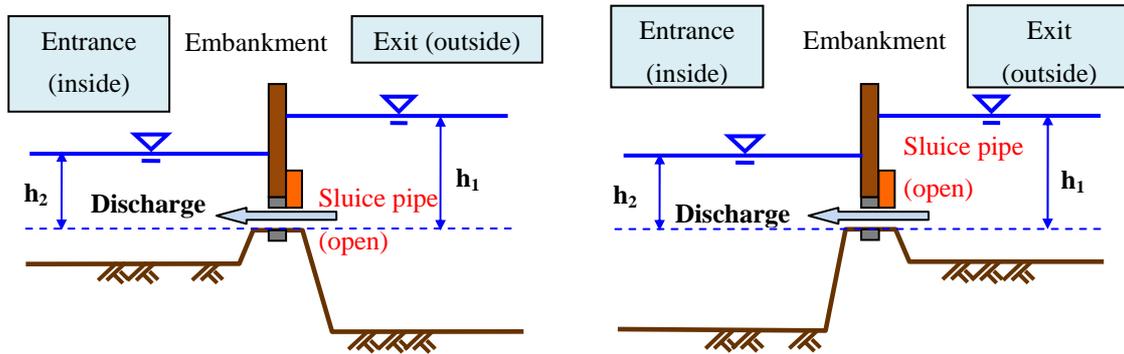
= Entrance water level – Elevation of structure bottom above the datum

The lower water depth between entrance and exit (h_2)

= Exit water level – Elevation of the structure bottom above the datum

⇒ Water that flows from the entrance to the exit discharge through a sluice pipe is calculated by equations (50) and (51) II.4.3

- ⑥ When “Sluice pipe open”, “Entrance water level < Exit water level” and “Exit water level > Elevation of the sluice pipe bottom above the datum”



The higher water depth between entrance and exit (h_1)

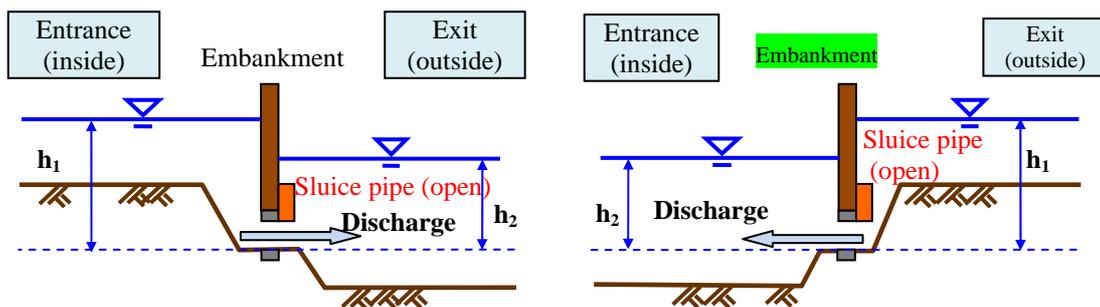
= Exit water level – Elevation of the structure bottom above the datum

The lower water depth between entrance and exit (h_2)

= Entrance water level – Elevation of the structure bottom above the datum

⇒ Water that flows from the exit to the entrance discharge through a sluice pipe is calculated by equations (50) and (51) II.4.3.

Note: When the elevation of the structure bottom above the datum is lower than the entrance elevation or the exit elevation, the higher water depth between entrance and exit (h_1) and the lower water depth between entrance and exit (h_2) are calculated as the differences from the elevation of the sluice pipe bottom above the datum.



II.5.3 Equations for calculating the discharge through a sluice pipe

The same equations used for calculating the discharge of a box culvert are used. (Refer to II.4.3.)

II.6 Simulation methods for pumps

II.6.1 Roles of pumps

Pumps drain water in the area protected by the embankment to the river channel. Pumps are installed at drainage pumping stations and on pumping vehicles. A pump starts operating when the water level inside the area protected by the embankment reaches the preset operation start water level, and it stops operating when the water level reaches the preset operation stop water level that is determined in consideration of submergence and other factors that prevent the pump from operating.

II.6.2 The simulation method

For the simulation, the side of the pump in the area protected by the embankment is set as the “entrance” and the side of the pump on the river channel as set as the “exit”; different cells are set for entrance versus exit. A pump starts operating when the “entrance water level” on the side of the pump in the area protected by the embankment reaches the operation start level, and it stops operating when the water level reaches the preset operation stop level. The latter is set in consideration of situations when the pump is inoperable, such as due to submergence. Pump displacement is based on the maximum displacement; however, no more water than exists at the entrance cell is discharged. A pump schematic is shown below.

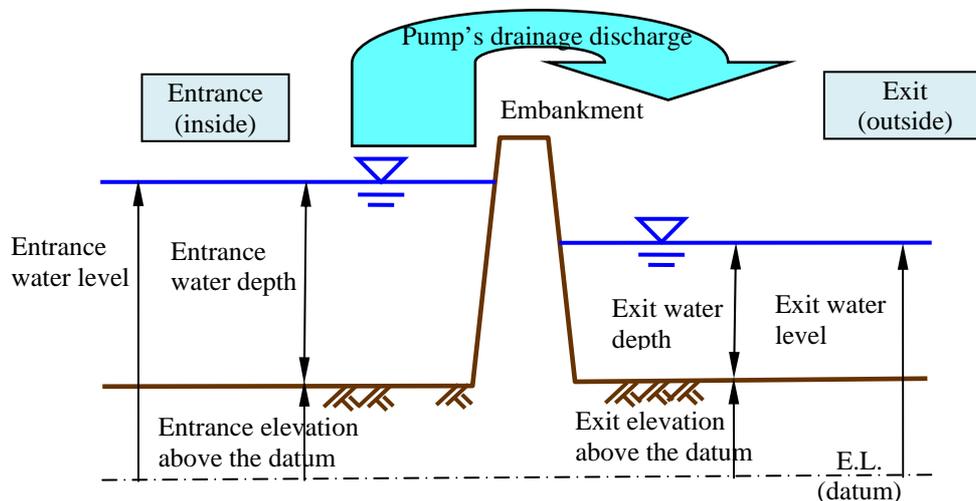


Figure II—3 Schematic of a pump (side view)

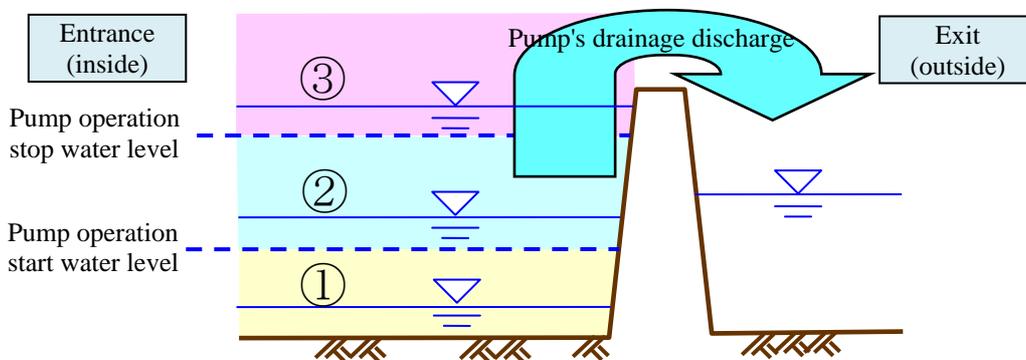
The calculation methods of pump drainage discharge for “when there is an entrance” and “when there is only exit” are shown below.

【When there is an entrance】

Instances of “when there is an entrance” can be with or without an exit. When there is an entrance and an exit, the pump’s drainage discharge at the entrance is given to the exit. “When there is only an entrance” means that water is discharged other than in the calculation domain. The pump’s drainage discharge at the entrance cell decreases, but water is not discharged into any other cells. The entrance pump discharge can be set by either of two methods: “automatic setting of the pump’s drainage discharge” or “data setting of the pump’s drainage discharge”.

• When the pump’s drainage discharge is set automatically

- ① Entrance water level < Pump operation start water level
⇒ No pump flow
- ② Pump operation start water level \leq Entrance water level < Pump operation stop water level
⇒ Pump drainage discharge is the maximum displacement of the pump.
(However, no more water than exists at the entrance cell is discharged.)
- ③ Pump operation stop water level \leq Entrance water level
⇒ No pump flow



• When the pump drainage discharge is set by data

The pump drainage discharge is given by the time-series data of pump displacement. (However, no more water than exists at the entrance cell is discharged.)

【When there is only exit】

The time-series data of pump displacement are given to the exit.

III. Calculation condition

In this chapter, the calculation conditions of Nays2D Flood are explained with setting dialogs of iRIC calculation conditions.

III.1 Explanation of names of items in [Boundary Conditions]

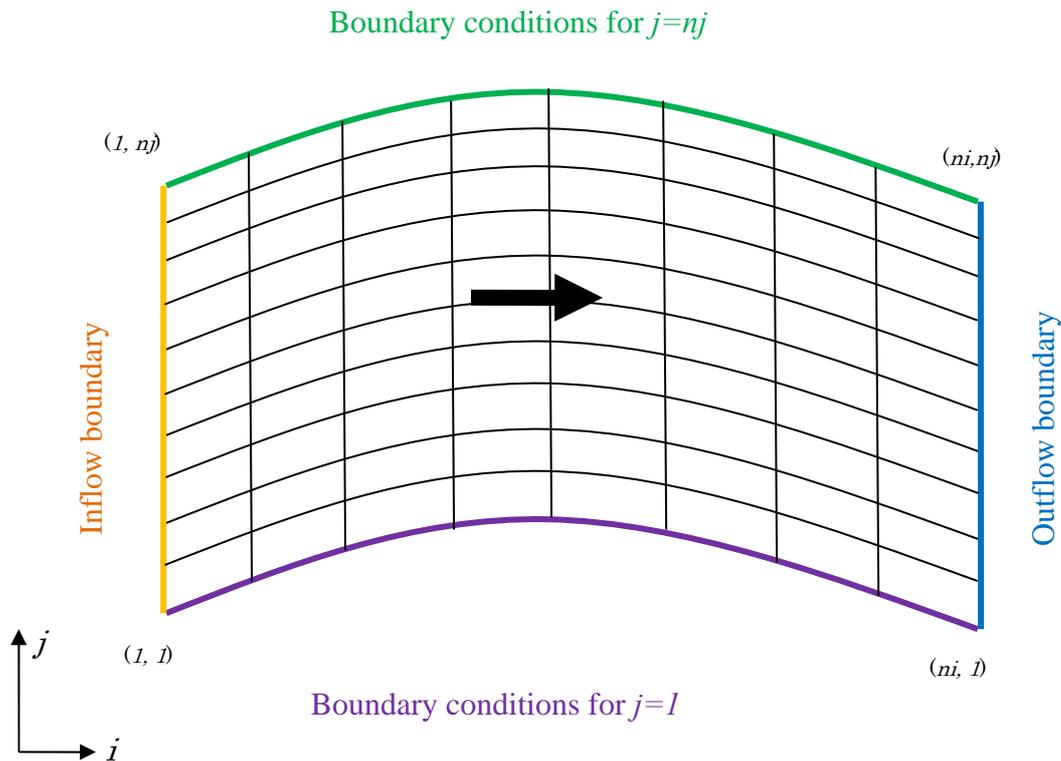


Figure III-1 Grid cell modeling

Boundary conditions need to be set for each four lines of a grid.

- "[Inflow Boundary Conditions]" mean the boundary conditions of the line between $(1, l)$ and $(1, nj)$ of the grid. The line between $(1, l)$ and $(1, nj)$ must be set as [Inflow]. Here, "discharge," which is necessary for calculation, is given.
- "Boundary conditions for $j=l$ " means the boundary conditions of the line between $(1, l)$ and (ni, l) of the grid. Either [Inflow] or [Outflow] can be set for the line between $(1, l)$ and (ni, l) .
- "Boundary conditions for $j=nj$ " means the boundary conditions of the line between $(1, nj)$ and (ni, nj) of the grid. Either [Inflow] or [Outflow] can be set for the line between $(1, nj)$ and (ni, nj) .
- "Outflow boundary conditions" means the boundary conditions of the line between (ni, l) and (ni, nj) of the grid. The line between (ni, l) and (ni, nj) must be set as [Outflow]. Here, "water surface elevation," which is necessary for calculation, is given.

III.2 Setting inflow/outflow boundary conditions

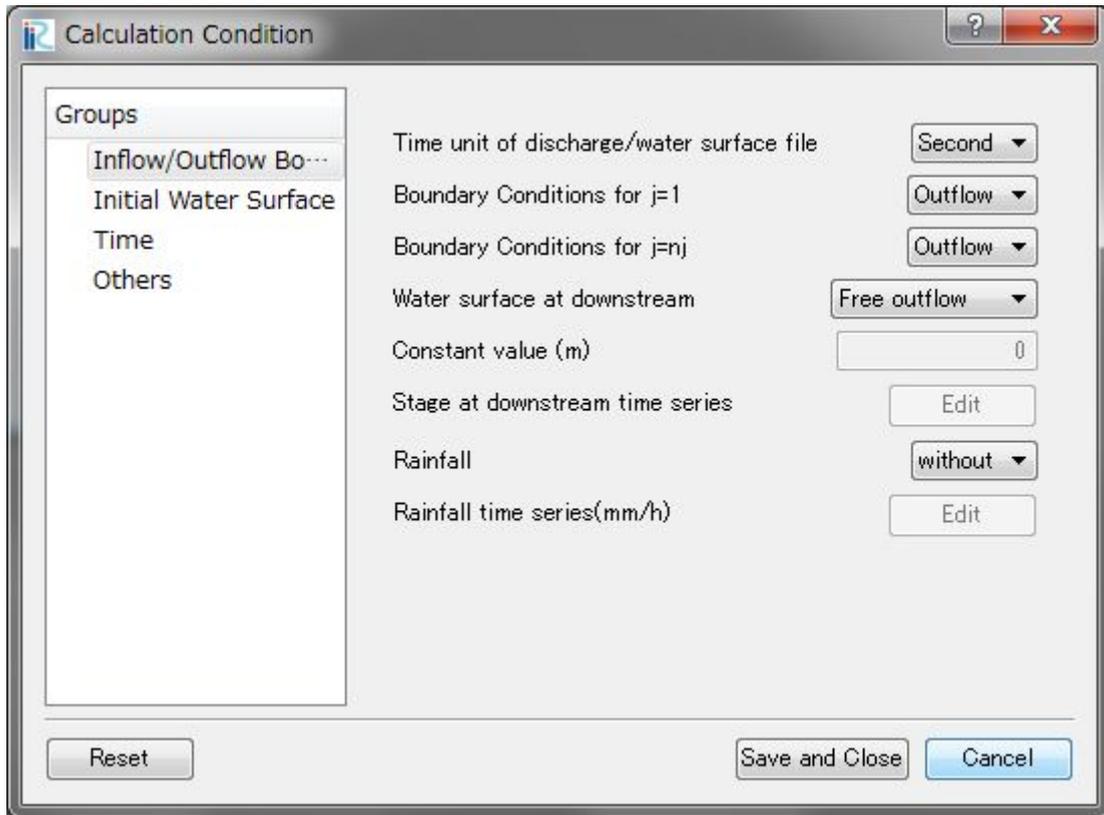


Figure III-2 Dialog for setting [Inflow/ Outflow Boundary Conditions]

Table III-1 Items of [Inflow/ Outflow Boundary Conditions]

#	Item	How to set the item	Notice
1	Time unit of discharge/water surface file	Input time unit for the time series of discharge and of water surface elevation of downstream end. [Second] or [Hour] can be selected.	The same time unit is applied to time series of inflow discharges from the bank sides and the water surface elevation at the downstream end.
2	Boundary Conditions for j=1	Select from [Wall], [Free outflow] and [Inflow].	The data format is given in Chapter III-6 Setting the inflow river files.
3	Boundary Conditions for j=nj	Select from [Wall], [Free outflow] and [Inflow].)	
4	Water surface at the downstream	Set water surface at the downstream. Select between [Constant value], [Read from a file] and [Free outflow].	

5	Constant value (m)	Input it when [Constant value] is selected for the water surface elevation at the downstream end..	
6	Stage at downstream time series	Input it when [Read from a file] is selected for the water surface elevation at the downstream end.	Input water surface elevation (m) for the column of [Water Surface Elevation]. The [Time] column should be set the same as the time column of the discharge file.
7	Rainfall	Set with/without rainfall. Select between [with] and [without.]	
8	Rainfall time series (mm/h) (i.e., time series file of rainfall)	When [with] is selected for [Rainfall], click on [Edit] to set [Rainfall hyetograph].	

Notes:

- 1) Grid cells not specified as "Inflow" cells are regarded as "Wall."
- 2) "Wall" means a boundary condition where no inflow/outflow occurs.
- 3) Free outflow means the simulation results for the grid cell adjacent to the boundary grid cell are given to the boundary grid cell as its boundary condition.

III.3 Setting initial water surface profile

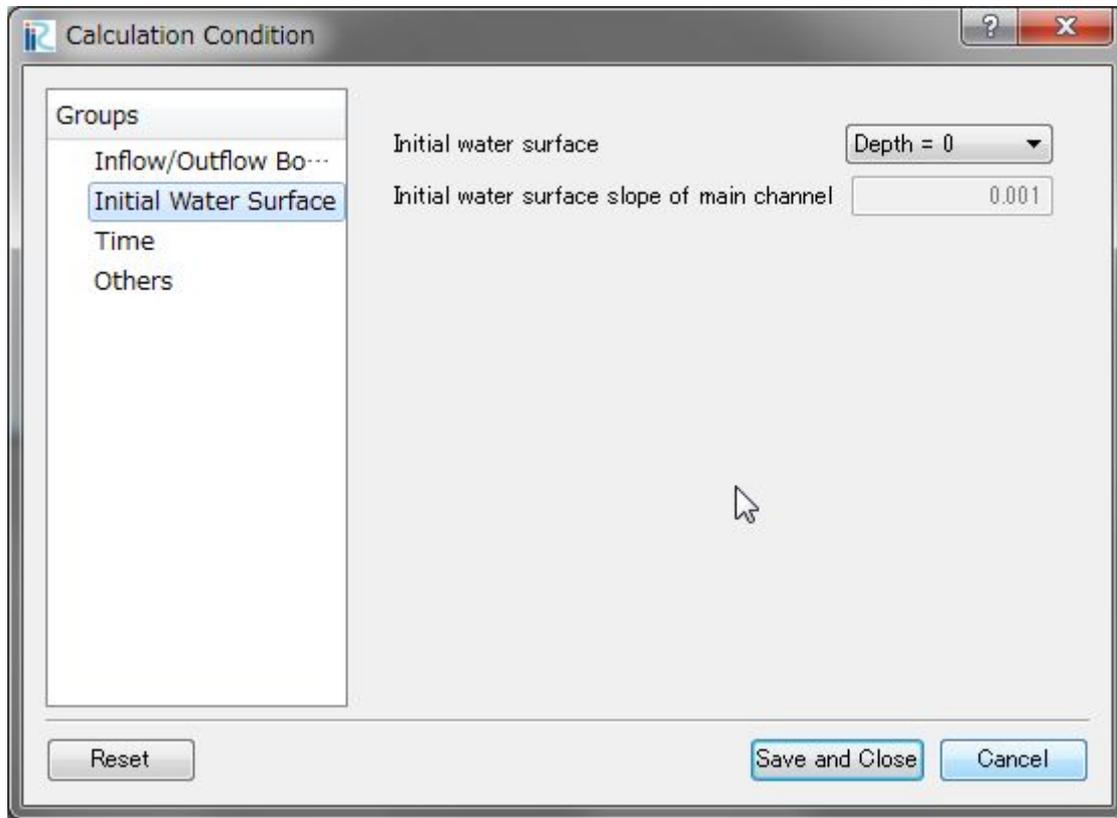


Figure III-3 Dialog for setting [Initial water surface]

Table III-2 Items for setting [Initial water surface]

#	Item	How to set the item	Notice
1	Initial water surface profile	Select between [Constant slope (line)] and [Depth =0].	
2	Initial water surface slope of main channel	Input it when [Constant slope (line)] is selected for [Initial water surface profile]. The water surface slope is set as a constant slope whose origin is the water surface elevation of the downstream end.	A flat surface elevation can be set by setting the slope gradient as zero. The location where the ground elevation is higher than the initial water surface profile has a water depth of zero.

III.4 Setting time condition

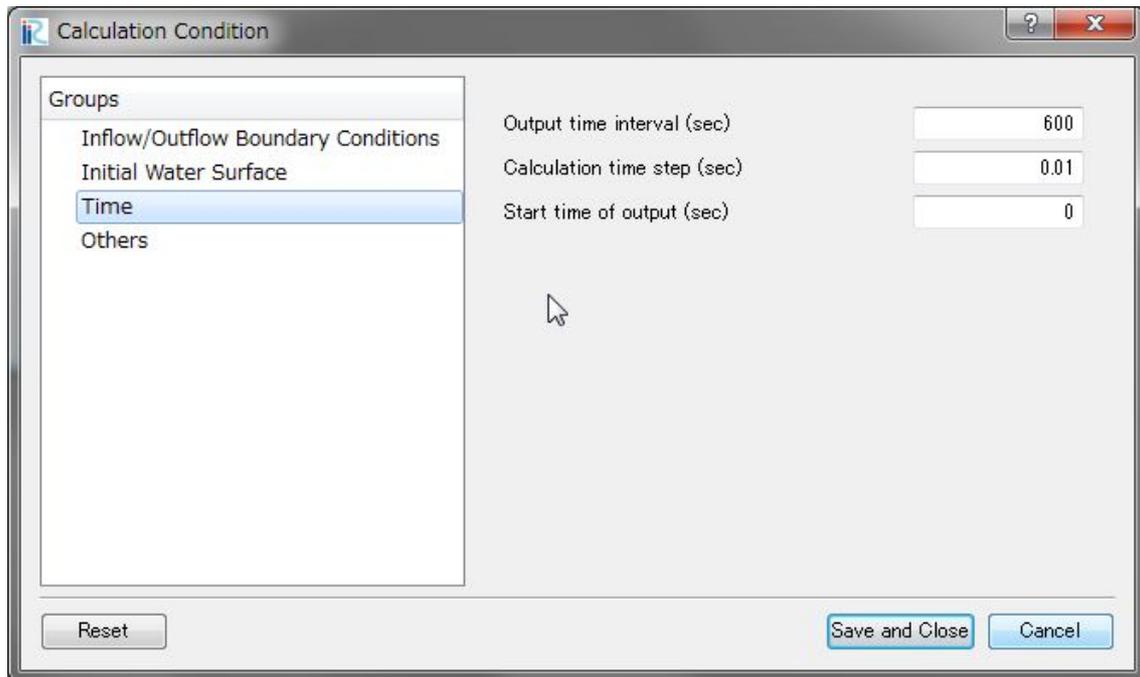


Figure III-4 Dialog for setting [Time Settings]

Table III-3 Items for setting [Time settings]

#	Item	How to set the item	Notice
1	Output time interval (sec)	Input the output time interval.	
2	Calculation time step (sec)	Set the time interval of calculation step.	Important parameter to determine the efficiency and stability of calculation.
3	Start time of output (sec)	Input the start time of the output.	

III.5 Other settings

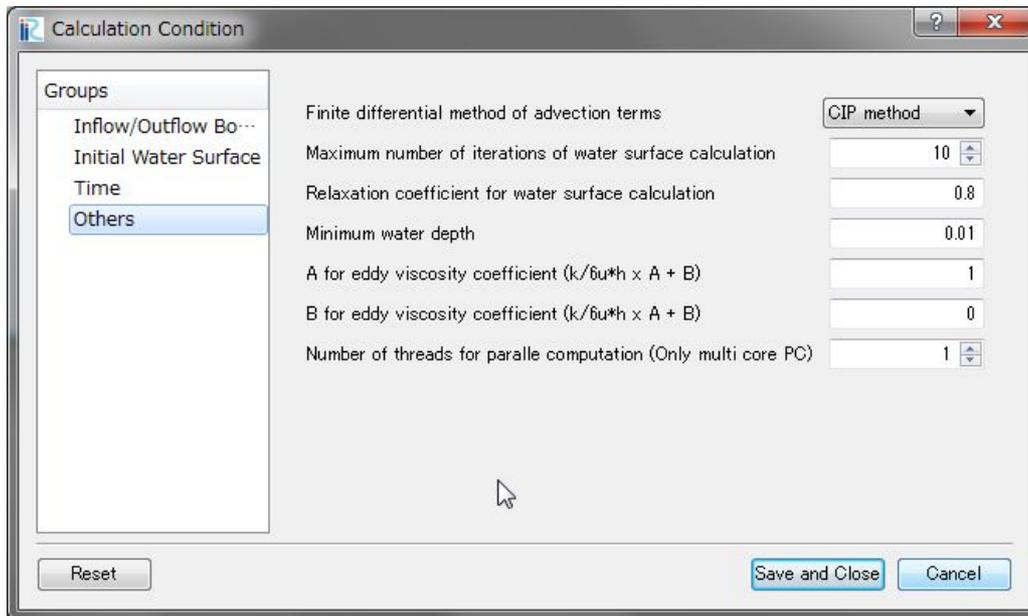


Figure III-5 Dialog for setting [Other information]

Table III-4 Items for setting [Other information]

#	Item	How to set the item	Notice
1	Finite differential method of advection terms	Select between [Upwind scheme] and [CIP method] ³⁾ .	
2	Maximum number of iterations of water surface calculation	Number of iterations of water surface calculation.	
3	Relaxation coefficient for water surface calculation	Relaxation coefficient for water surface calculation.	Adjust when the water surface calculation is not stabilized.
4	Minimum water depth	Minimum water depth for water surface calculation.	
5	A for eddy viscosity coefficient ($k/6u \cdot h \times A+B$)	Set parameter “A” of eddy viscosity coefficient ($k/6u \cdot h \times A+B$).	
6	B for eddy viscosity coefficient ($k/6u \cdot h \times A+B$)	Set parameter “B” of eddy viscosity coefficient ($k/6u \cdot h \times A+B$).	

7	Number of threads for parallel computation (only for multi-core PC)	Set the number of threads for parallel computation.	
---	---	---	--

<References>

- 3) Yabe, T., Ishikawa, T.: A Numerical Cubic-Interpolated Pseudoparticle (CIP) Method without Time Splitting Technique for Hyperbolic Equations, Journal of the Physical Society of Japan, Col.59, No.7, pp.2301-2304, 1990.

III.6 Setting the inflow river

Set inflow rivers (or flood levee breaches) directly to the calculation cell: upper end, side ($j=1$) and side ($j=n_j$). After making a grid for calculation, right click on [Boundary Condition Setting] under [Object Browser] to select [Add Inflow]. Select [Object Browser] - [Boundary Condition Setting] - [New Inflow] to add a river [Inflow]. While selecting the [Inflow], enclose the sides of “Inflow” locations with polygons. When the location is confirmed, the [Boundary condition] setting dialog opens. Set changes in discharge by time and profile of inflow. Run [Attribute mapping] under [Grid] on the menu bar to set the inflow as a side of a grid.

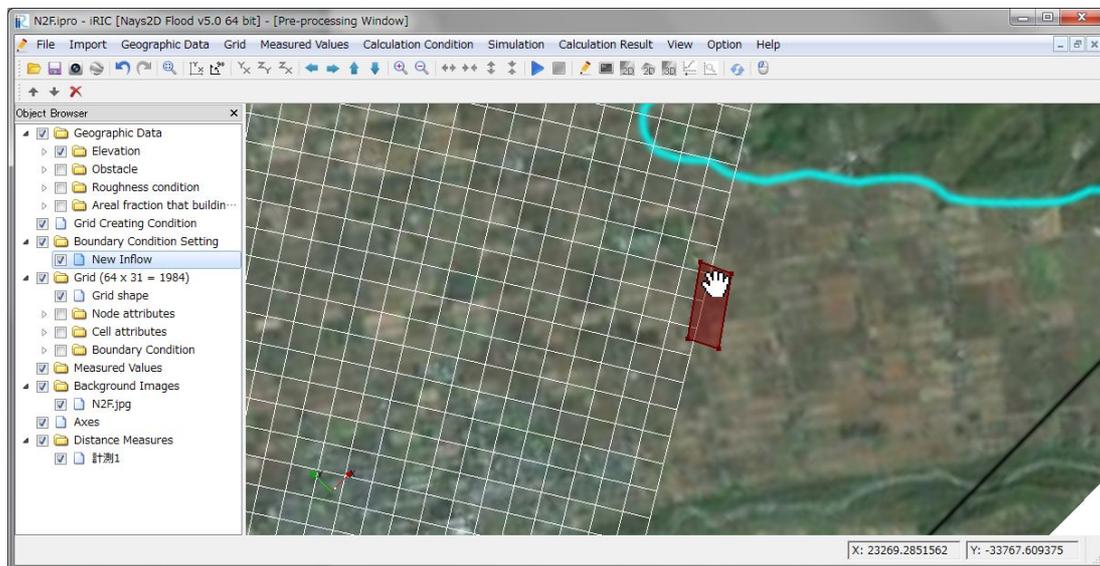
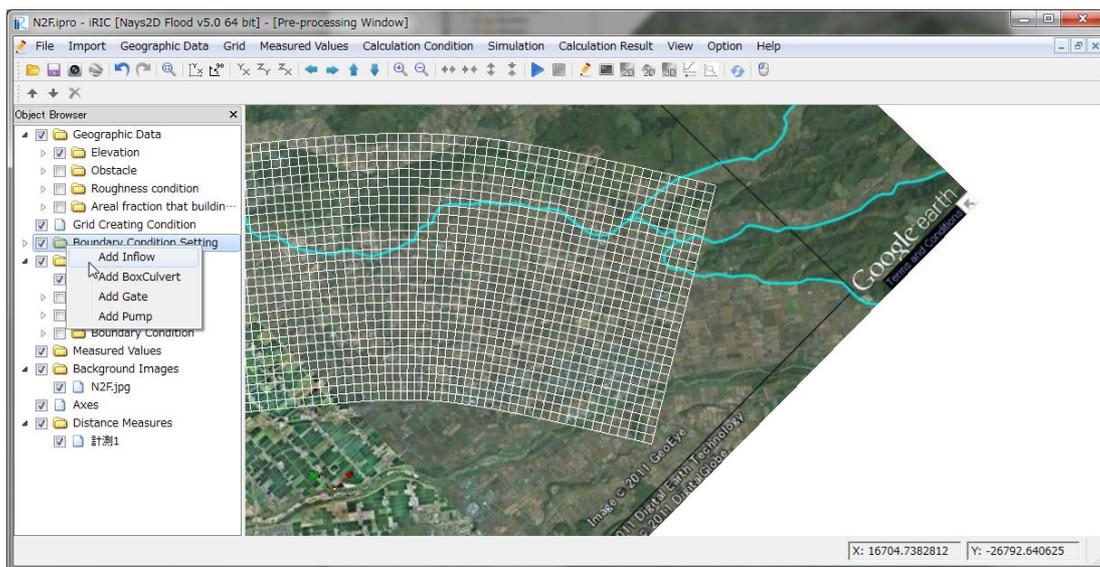


Figure III-6 Dialog for setting [River Inflow]

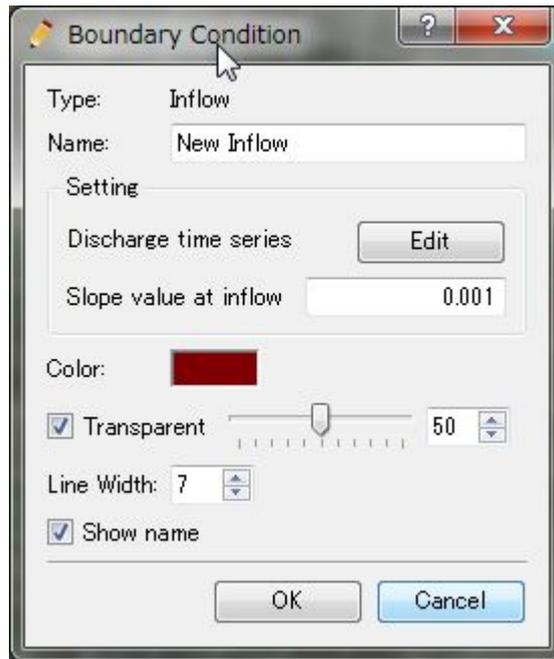


Figure III-7 Dialog for setting [Boundary Condition]

Table III-5 Items setting for [Boundary Condition]

#	Item	How to set the item	Notice
1	Name	Input names of inflow river (or flood levee breach) arbitrary.	
2	Discharge time series	Click Edit and set the discharge hydro-graph.	[Second] or [Hour] can be selected as time unit and [m ³ /s] as discharge.
3	Slope value at inflow	Input the slope value at inflow.	
4	Color	The color of inflow point.	
5	Transparent	The transparent of inflow point.	

III.7 Setting riverbed roughness coefficient

Set the riverbed roughness coefficient for each grid cell. After making a grid for calculation, select [Object Browser] - [Grid] - [Cell Attributes] and check [Roughness]. In the Object Browser, select [Geographic Data] - [Roughness] and right click to select [Add] - [Polygon]. While selecting a polygon that has been added to [Roughness condition] under [Geographic Data], enclose cells of the same roughness in the polygon. Upon confirming the polygon, the [Edit Roughness condition value] dialog opens. Then, input the coefficient of roughness. Run the [Attributes mapping] under [Grid] on the menu bar to set the coefficient of roughness for the cells.

The default setting is 0.0 (Manning's roughness coefficient). Set it by adjusting to the land-use conditions of the floodplain.

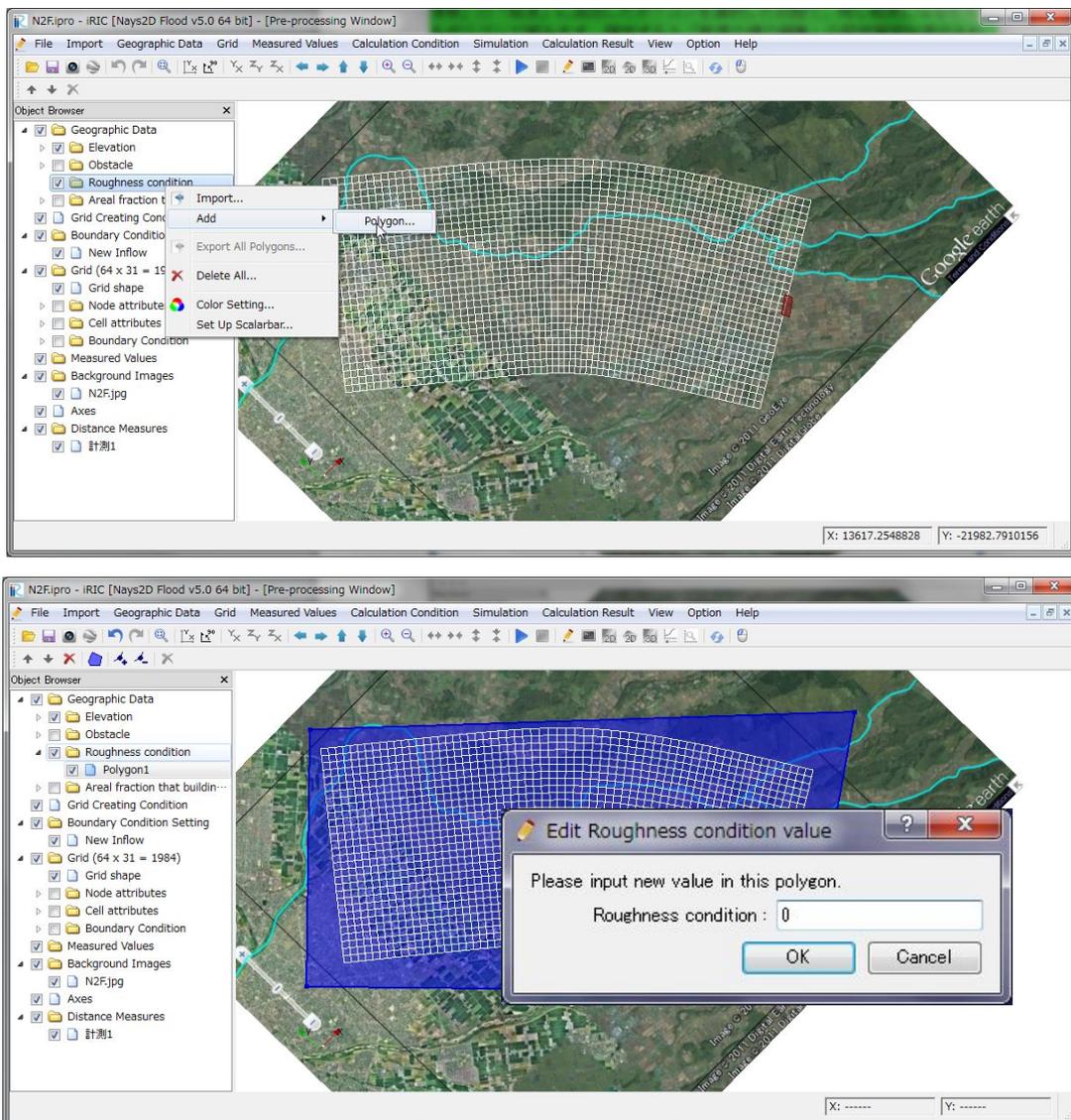


Figure III-8 Dialog for setting [Roughness Coefficient]

III.8 Setting obstacles

Obstacles such as roads, banks and embankments can be set as necessary. After making a grid for calculation, select [Object Browser] - [Cell attributes] and check [Obstacle cell]. Select [Object Browser] - [Geographic Data] - [Obstacle cells]. Right click on [Obstacle cells] to select [Add] - [Polygon]. While selecting a polygon that has been added to [Obstacle cells] under [Geographic Data], enclose the obstacle cells in the polygon. Upon confirming the polygon, the [Edit Obstacle value] dialog opens. Select [Obstacle cells]. Run the [Attributes mapping] under [Grid] on the menu bar to set obstacle cells.

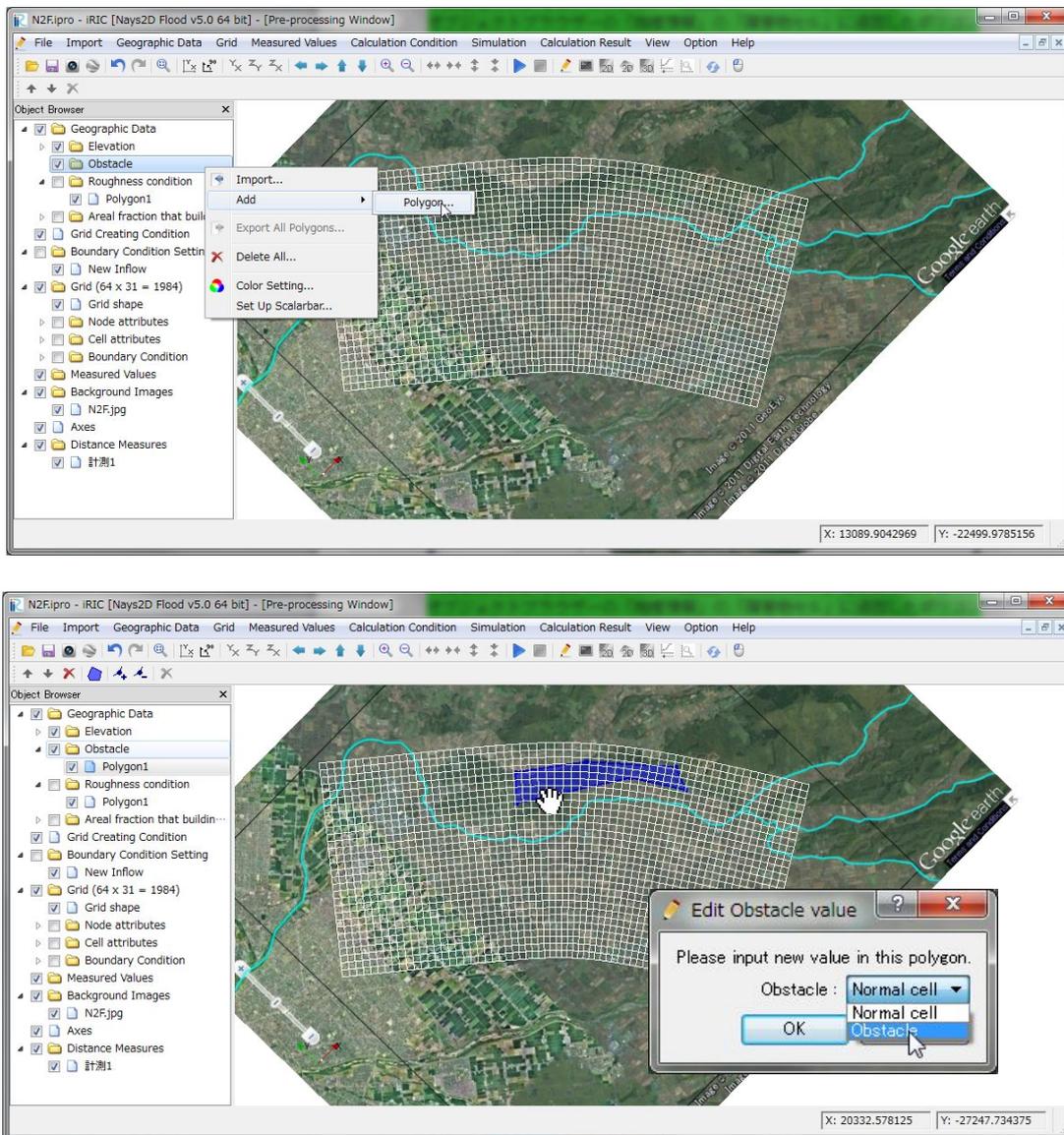


Figure III-9 Dialog for setting [Obstacles]

III.9 Setting building occupancy ratio by area

Set the building occupancy ratio by area for each cell. After making a grid for calculation, select [Object Browser] - [Grid] - [Cell Attributes] and check [Areal fraction that buildings occupy]. Select [Object Browser] - [Geographic Data] - [Areal fraction that building occupy] and right click on [Areal fraction that the building occupy] to select [Add] - [Polygon]. While selecting a polygon that has been added to [Areal fraction that building occupy] under [Object Browser] - [Geographic Data], enclose cells that have the same building occupancy ratio by area in the polygon. Upon confirming the polygon, the [Edit Areal fraction that building occupy] dialog opens. Input the new value. Run [Attributes mapping] under [Grid] on the menu bar to set the “areal fraction that building occupy” or the building occupancy ratio by area for the cells.

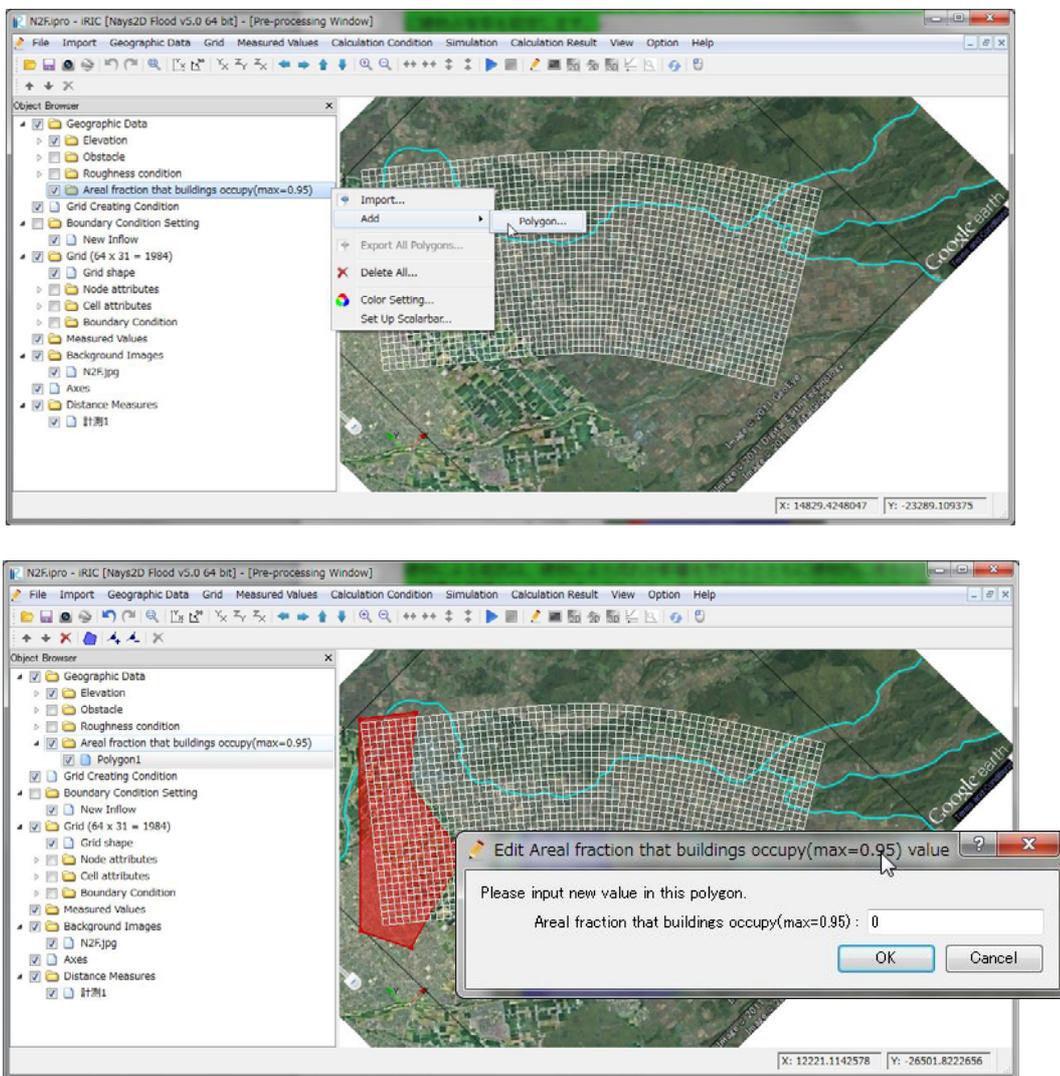


Figure III—7 Windows to set “Areal fraction that building occupy” or building occupancy ratio by area

The building resistance is simulated assuming to have the conditions in which buildings affect the flow and are inundated. Note that the preconditions differ from those for the setting of obstacle cells. Because the calculation tends to diverge when the void ratio is close to 0%, the solver limits values of the void ratio (the structural occupancy ratio is 0.95 or less and the void ratio is 0.05 or more) to converge the simulation.

Note: Flow velocities between cells are calculated based on “building occupancy ratio by area” for each cell. Because buildings are inundated, there is little water level rise outside the cell.

Note: Flow velocities between cells that face obstacles are not calculated. Because buildings are not inundated, the water level outside the cell tends to rise.

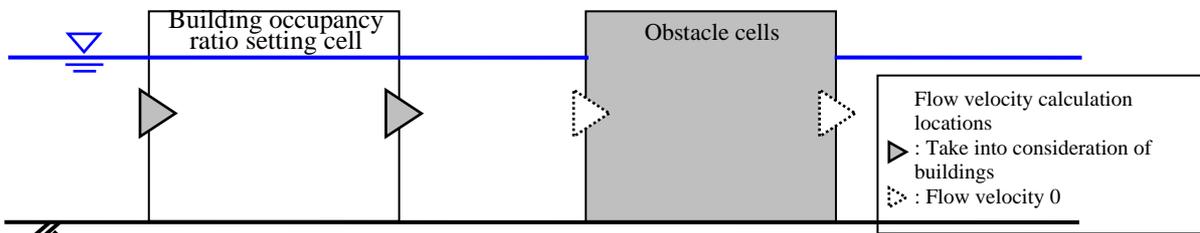


Figure III – 8 Schematic of building resistance (left) and obstacle resistance (right) (side view)

III.10 Setting for a box culvert

For a box culvert, [Inside] (i.e., the entrance) and [Outside] (i.e., the exit) are regarded as a unit, and different cells are set for [Inside] and for [Outside].

After making a grid for calculation, select [Object Browser] - [Boundary Condition Setting], and right click on [Boundary Condition Setting] to select [Add BoxCulvert]. While selecting the added box culvert (the initial setting is [New BoxCulvert]) which has been added to [Boundary Condition Setting] under [Object Browser], enclose the sides of the cells at the location of the box culvert in a polygon. When the polygon is confirmed, the [Boundary Condition] setting dialogue opens. Select or input for fields of “Inside/Outside”, “Width”, “Height”, “Elevation” (i.e., elevation of the structure bottom above the datum) and the number of units and coefficient of discharge of the box culvert. Run [Attributes mapping] under [Grid] on the menu bar to set box culvert on cells.

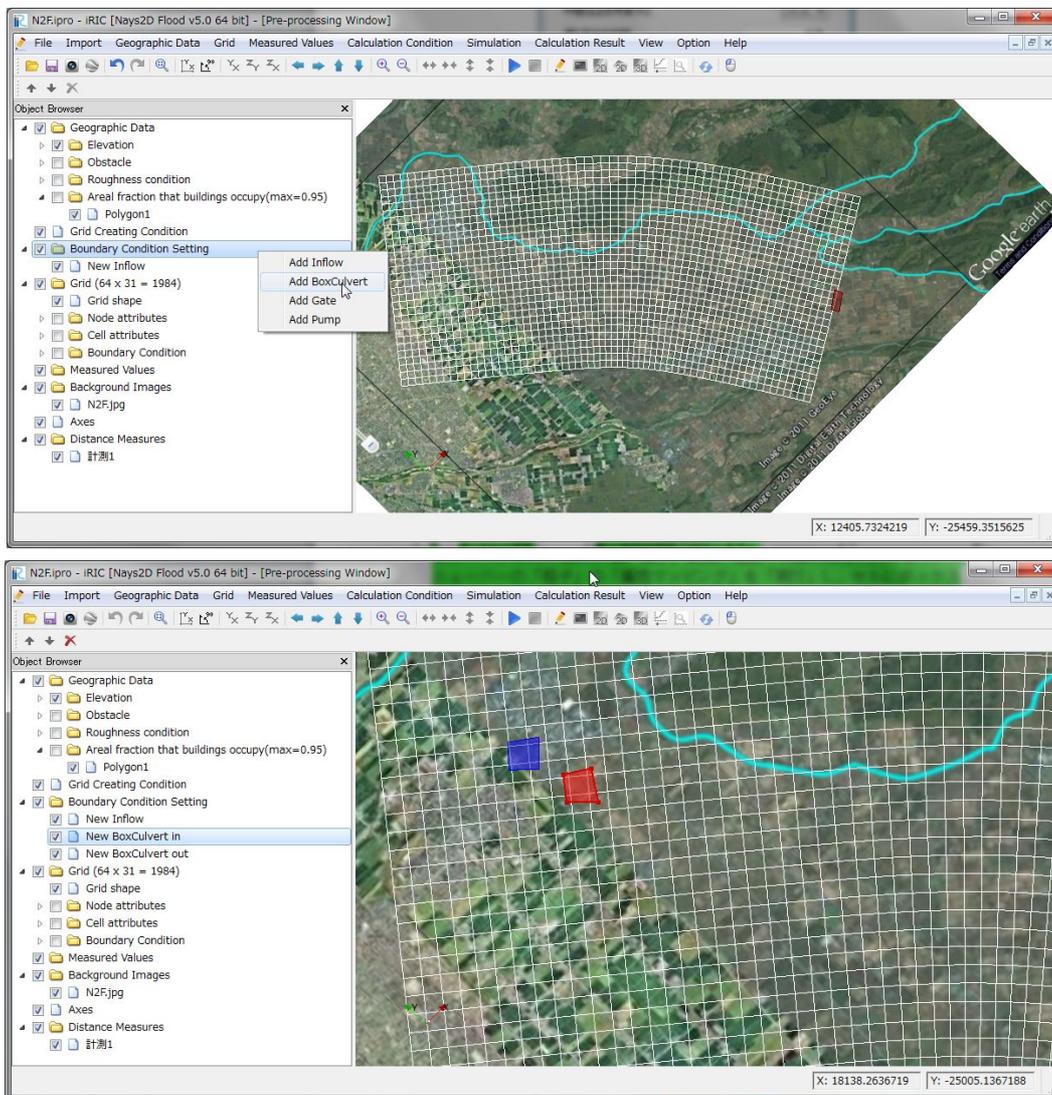


Figure III—9 Box culvert setting window

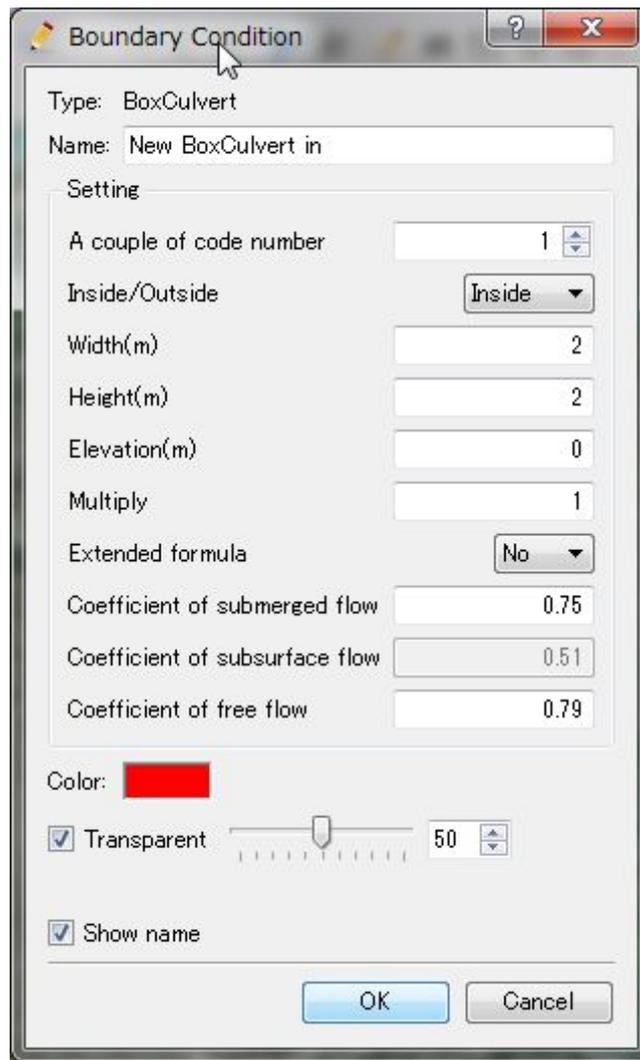


Figure III – 10 Boundary conditions setting window

Table III – 6 Explanation of boundary condition setting

#	Item	Setting method	Important information
1	Name	Input arbitrary name of box culvert	
2	Code number	Specify the code number of the box culvert	The same code number should be used for Inside (i.e., entrance) and Outside (i.e., exit)
3	Inside (i.e., entrance)/Outside (i.e., exit)	Selection can be made between "Inside" (i.e., entrance) and "Outside" (i.e., exit) of the box culvert	
4	Width (m)	Set the inside width (m) of the box culvert.	Refer to Figure II – 1
5	Height (m)	Set the inside height (m) of the box culvert.	Refer to Figure II – 1
6	Elevation (i.e., elevation of the structure bottom above the datum)	Set the elevation of the structure bottom above the datum (m) for the box culvert.	Refer to Figure II – 1
7	Multiply	Set the number of box culverts.	

8	Extend formula (i.e., taking into account the subsurface flow)	Set whether the subsurface flow is taken into account. Select between "Yes" and "No."	
9	Coefficient of submerged flow	Set the coefficient of submerged flow.	
10	Coefficient of subsurface flow	Set the coefficient of subsurface flow.	
11	Coefficient of free flow (i.e., free outflow)	Set the coefficient of free flow (i.e., free outflow).	
12	Color	Set the color to identify the box culvert.	
13	Transparent	Set the transparency of the box culvert.	
14	Show name	Check it if you want the drawings to display the name of the box culvert.	

III.11 Setting for a sluice pipe

For simulation, [Inside] (i.e., entrance) and [Outside] (i.e., exit) of the sluice pipe are set, respectively. Different cells are set for [Inside] and for [Outside].

After making a grid for calculation, select [Object Browser] - [Boundary Condition Setting], and right click on [Boundary Condition Setting] to select [Add Gate]. While selecting the sluice pipe (the initial setting is [New Gate]) that has been added to the [Boundary Condition Setting] under [Object Browser], enclose the cells at the location of the sluice pipe in a polygon. When the polygon is confirmed, the [Boundary Condition] setting dialogue opens. Select or input fields of “Inside/Outside”, “Width”, “Height”, “Elevation” (i.e., elevation of the structure bottom above the datum) and the number of units and coefficients of flow of the sluice pipe. Run [Attributes mapping] under [Grid] on the menu bar to set the sluice pipe for the cells.

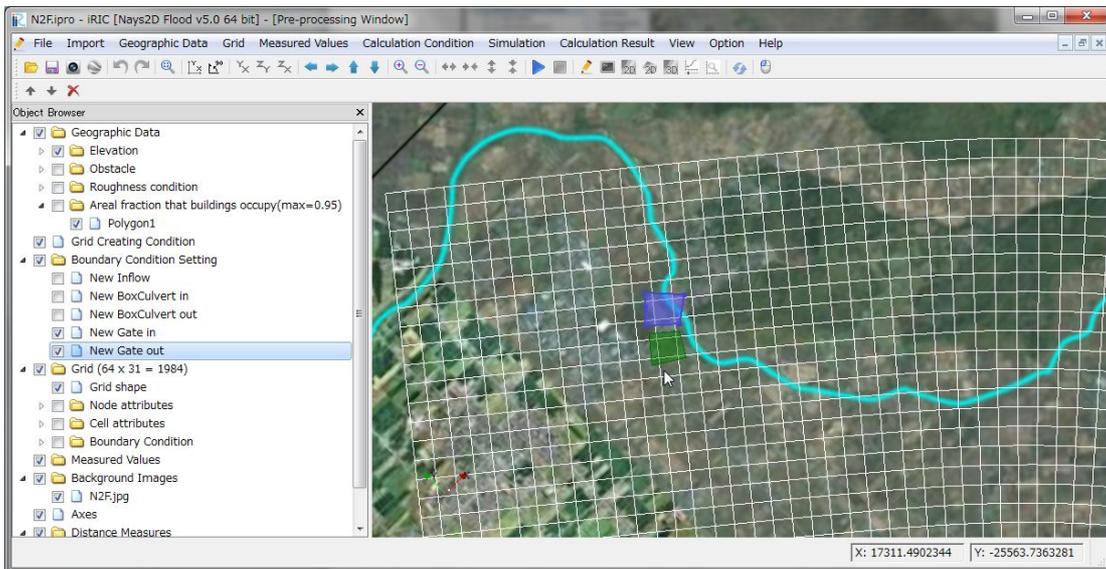
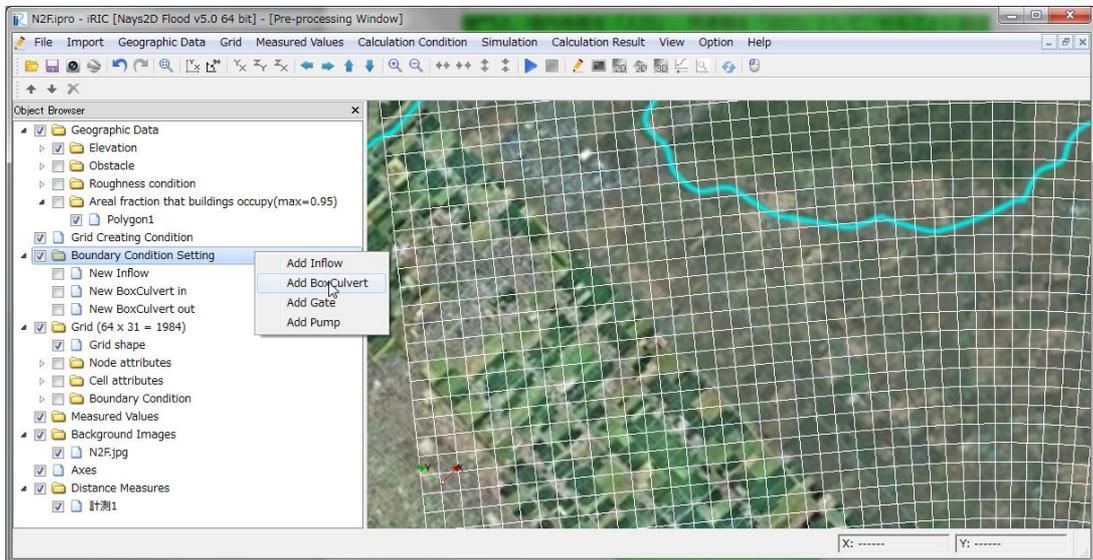


Figure III – 11 Sluice pipe setting window

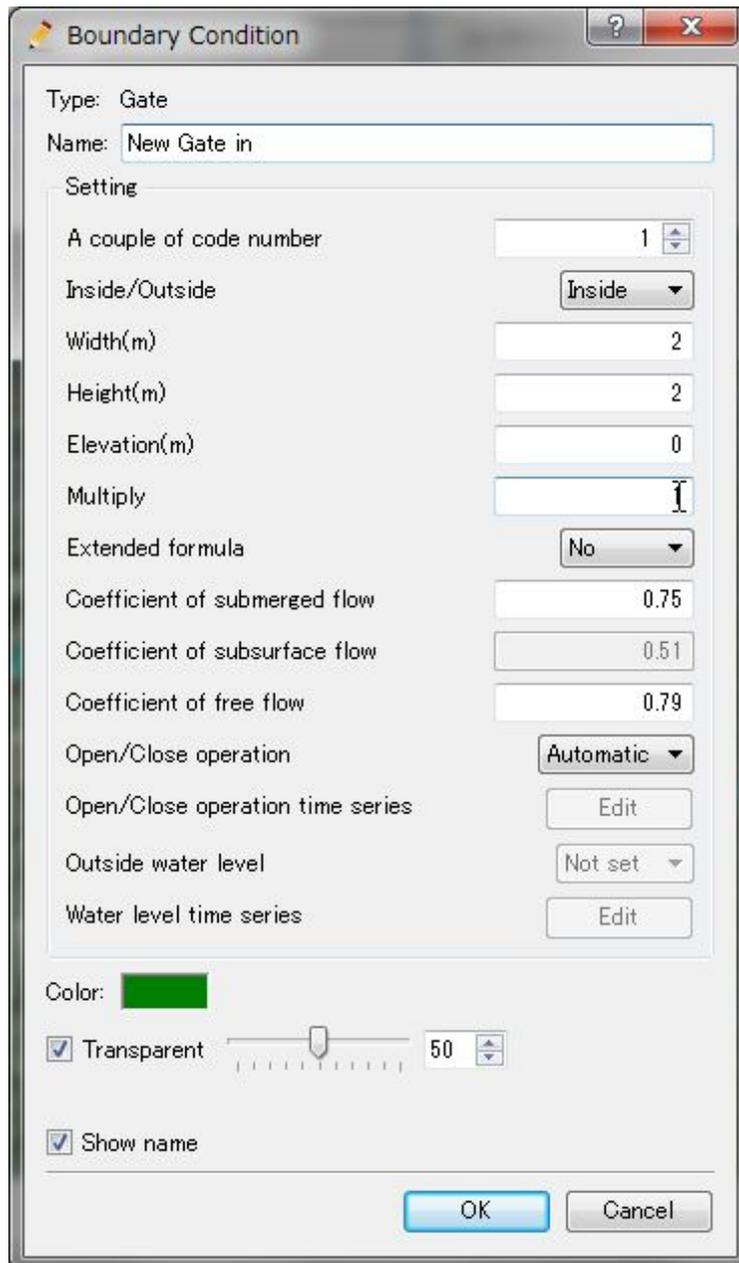


Figure III—12 Boundary condition setting window

Table III—7 Explanation of boundary condition setting

#	Item	Setting method	Important information
1	Name	Input the arbitrary name of the sluice pipe.	
2	Code number	Specify the code number of the sluice pipe.	The same code number should be used for Inside (i.e., entrance) and Outside (i.e., exit)
3	Inside (i.e., entrance)/Outside (i.e., exit)	Selection can be made between “Inside (i.e., entrance)” and “Outside (i.e., exit)” of the sluice pipe.	
4	Width (m)	Set the inside width (m) of the sluice pipe.	Refer to Figure II—2

5	Height (m)	Set the inside width (m) of the sluice pipe.	Refer to Figure II—2
6	Elevation (i.e., elevation of the structure bottom above the datum)	Set the elevation of the structure bottom above the datum (m) for the sluice pipe.	Refer to Figure II—2
7	Multiply	Set the number of sluice pipes.	
8	Extend formula (i.e., taking into account the subsurface flow)	Set whether the subsurface flow is taken into account. Select between "Yes" and "No."	
9	Coefficient of submerged flow	Set the coefficient of submerged flow.	
10	Coefficient of subsurface flow	Set the coefficient of subsurface flow.	
11	Coefficient of free flow (i.e., free outflow)	Set the coefficient of free flow (i.e., free outflow).	
12	Color	Set the color to identify the sluice pipe.	
13	Transparent	Set the transparency of the sluice pipe.	
14	Show name	Check it if you want the drawings to display the name of the box culvert.	

III.12 Setting for a pump

For simulation, [Inside] (i.e., entrance) and [Outside] (i.e., exit) of the pump are set, respectively. Different cells are set for [Inside] and for [Outside].

After making a grid for calculation, select [Object Browser] - [Boundary Condition Setting], and right click on [Boundary Condition Setting] to select [Add Pump]. While selecting the pump (the initial setting is [New Pump]) that has been added to [Boundary Condition Setting] under [Object Browser], enclose the cells at the location of the pump in a polygon. When the polygon is confirmed, the [Boundary Condition] setting dialogue opens. Select or input fields of “Inside/Outside”, “Maximum drainage discharge”, “Begin operating water level” (i.e., water level at which the pump starts operating), “Stop operating water level” (i.e., water level at which the pump stops operating) for the pump. Run [Attributes mapping] under [Grid] on the menu bar to set the pump for the cells.

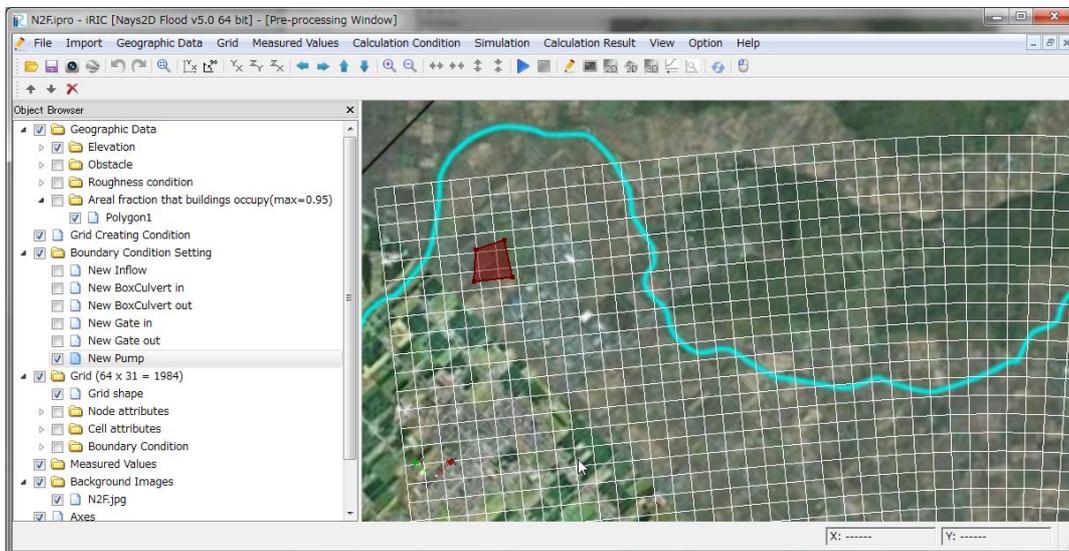
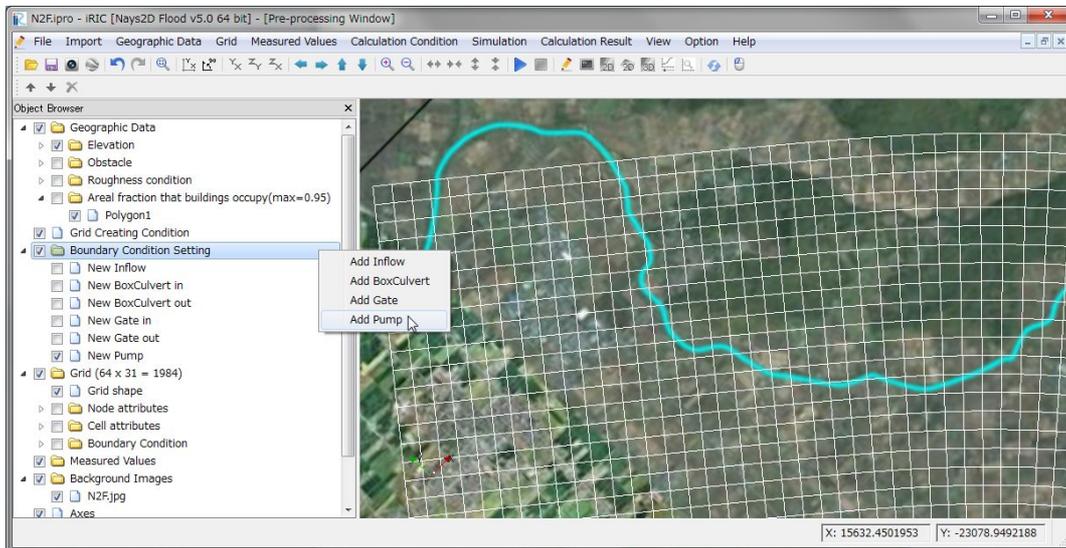


Figure III—13 Pump setting window

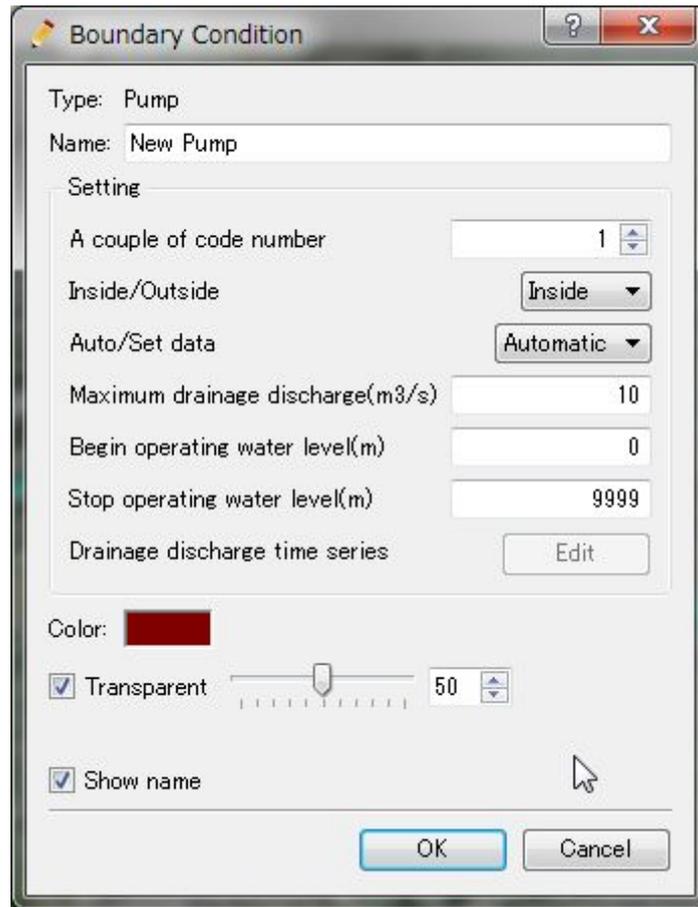


Figure III – 14 Boundary conditions setting window

Table III – 8 Explanation of boundary condition setting

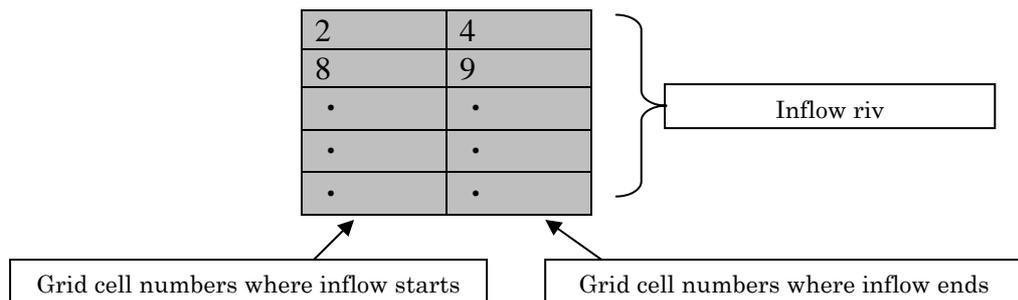
#	Item	Setting method	Important information
1	Name	Input the arbitrary name of the pump.	
2	Code number	Specify the code number of the pump.	The same code number should be used for Inside (i.e., entrance) and Outside (i.e., exit)
3	Inside (i.e., entrance)/Outside (i.e., exit)	Selection can be made between “Inside (i.e., entrance)” and “Outside (i.e., exit)” of the pump.	
4	Auto/Set data	Set the pump operation method. Select between “Auto” and “Set data” (i.e., manual operation.)	
5	Maximum drainage discharge (m3/s)	Set the maximum drainage discharge of the pump.	
6	Begin operating water level (m)	Set the water level at which the pump starts operating.	Refer to Figure II – 3

7	Stop operating water level (m)	Set the water level at which the pump stops operating.	Refer to Figure II—3
8	Drainage discharge time-series	Click on [Edit] to set the time-series data of maximum drainage discharge when [Auto] is selected for the pump operation method.	
9	Color	Set the color to identify the pump.	
10	Transparent	Set the transparency of the sluice pipe.	
11	Show name	Check it if you want the drawings to display the name of the pump.	

IV. Input/Output files

IV.1 Input file for grid cell numbers of inflow

Select a grid cell number of inflow (and/or a bank opening). Numerical values can be separated by a tab, a comma or a space. When the boundary condition of the bank is set as [Inflow], an input file for the grid cell numbers where inflow starts and ends to be made for each river/bank opening.



Notes:

- 1) For [Inflow Boundary Conditions], input the number ? of the grid cell (l , ?).
- 2) For boundary conditions for $j=l$, input the number ? of the grid cell ($?$, l).
- 3) For boundary conditions of $j=nj$, input number ? of grid cell ($?$, nj).

Figure IV-1 Example of input file for grid cell number of inflow

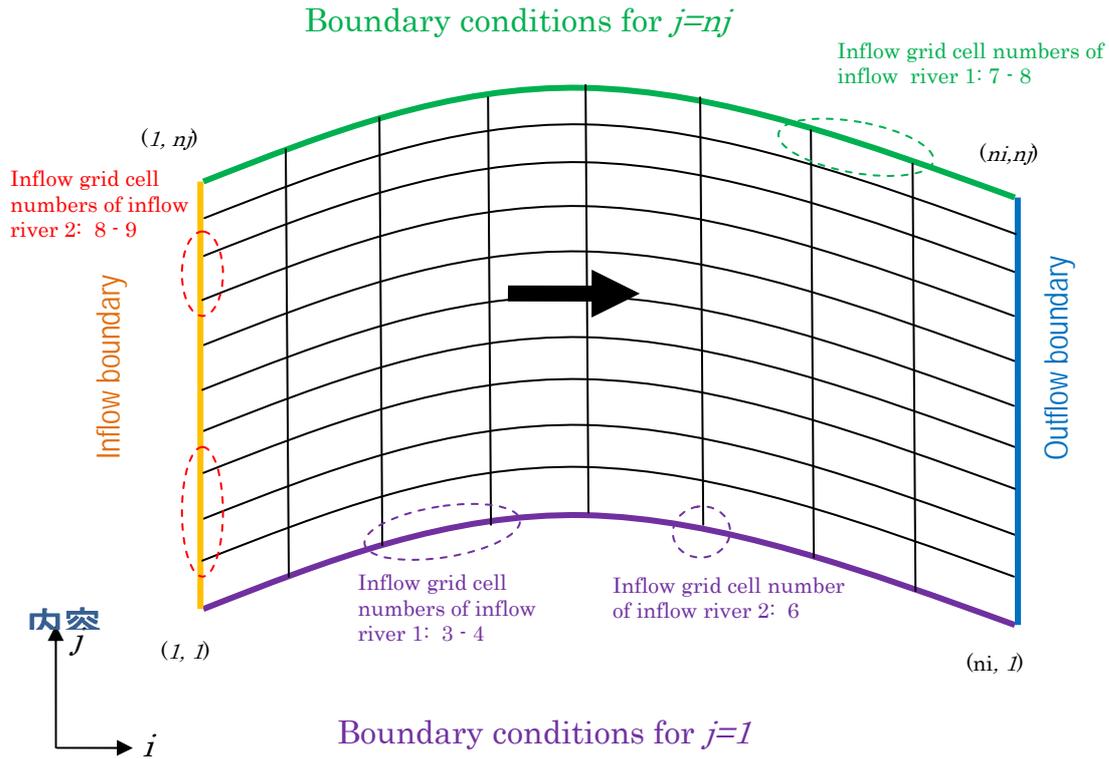


Figure IV-2 Input file for grid cell numbers of inflow

IV.2 Discharge file (Inflow boundary conditions)

Input an inflow river discharge (and/or bank overtopping discharge.) Numerical values can be separated by a tab, a comma or a space.

Time (Sec., hour)	Discharge of inflow river 1 (m ³ /s)	Discharge of inflow river 2 (m ³ /s)	Discharge of inflow river 3 (m ³ /s)	Discharge of inflow river 4 (m ³ /s)	Discharge of inflow river 5 (m ³ /s)
0	100	200	20	200	60
1	140	300	40	250	80
.
.
.

Figure IV-3 Example of discharge file (Inflow boundary conditions)

IV.3 Discharge file (boundary conditions for banks)

When the boundary condition of a bank is [Inflow], set the discharge as that of the inflowing river (or discharge at the bank opening). Numerical values can be separated by a tab, a comma or a space.

Discharge of inflow river 1 (m ³ /s)	Discharge of inflow river 2 (m ³ /s)	Discharge of inflow river 3 (m ³ /s)	Discharge of inflow river 4 (m ³ /s)	Discharge of inflow river 5 (m ³ /s)
100	200	20	200	60
140	300	40	250	80
.
.
.

Note: The time data of [Inflow Boundary Conditions] (IV of Chapter 2) is used for the time column of the discharge file of a bank opening.

Figure IV-4 Example of discharge file (Boundary conditions for banks)

To Reader

- Please indicate that using the iRIC software, if you publish a paper with the results using the iRIC software.
- The datasets provided at the Web site are sample data. Therefore you can use it for a test computation.
- Let us know your suggestions, comments and concerns at <http://i-ric.org>.

iRIC Software Nays2D Flood Solver Manual

Editors and writers	Yasuyuki SHIMIZU (Hokkaido Univ., Japan)	Editing
	Eiichi SUZUKI (Hokkaido Univ., Japan)	Editing
	Satomi KAWAMURA (Hokkaido Univ., Japan)	Editing
	Takuya INOUE (C.E.R.I., Japan)	Writing
	Toshiki IWASAKI (Hokkaido Univ., Japan)	Writing
	Michihiro HAMAKI(Kaihatsu Koei Co., Ltd., Japan)	Writing
	Kensuke OMURA (Hokkaido Univ., Japan)	

Supported by	River center of Hokkaido, Japan.
--------------	----------------------------------