L-113-02-64

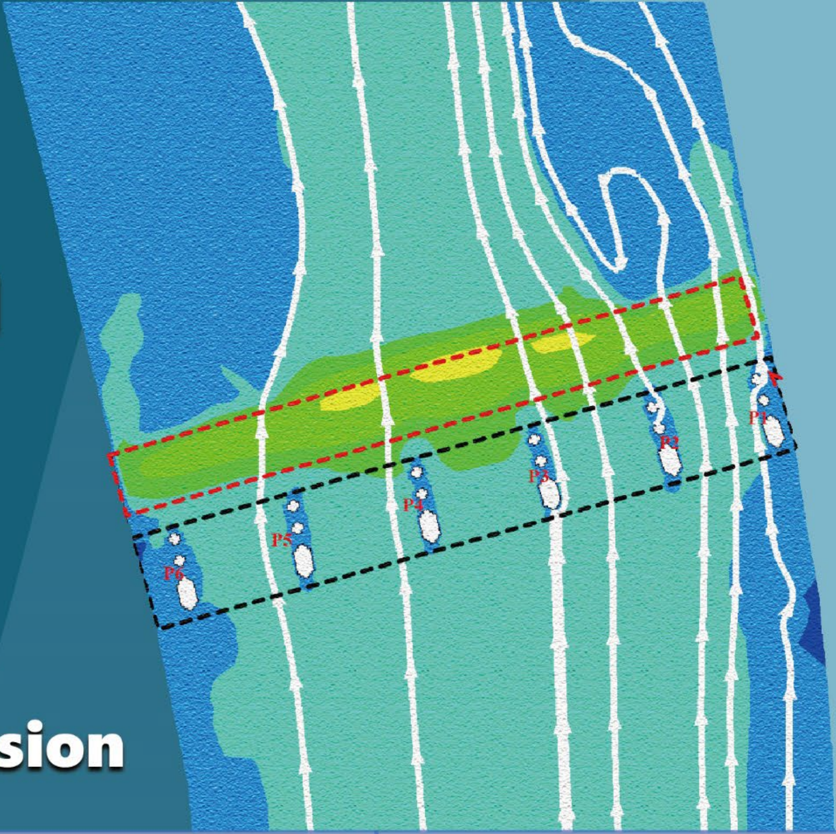
International River Interface Cooperative

Training Session

Hosting:
Sinotech Foundation for R&D of Engineering Sciences & Technologies
Hydrotech Research Institute, NTU
Ecological Engineering Research Center, NTU
Sinotech Engineering Consultants, Ltd.

iRIC
international River Interface Cooperative

Training Session



Time	Session / Content	Host / Speaker
09:20~09:30	Registration	
09:30~09:45	Open Ceremony & General Overview of the iRIC Software	Dr. Jihn-Sung Lai Prof. Yasuyuki Shimizu
09:45~10:45	Basic flood Analysis and Its Utilization Using iRIC-Nays2D Flood : Training Session (1)	Dr. Takaaki Abe Civil Engineering Research Institute for Cold Region
10:45~11:00	Tea break	
11:00~12:00	Basic flood Analysis and Its Utilization Using iRIC-Nays2D Flood : Training Session (2)	Dr. Takaaki Abe Civil Engineering Research Institute for Cold Region

Audio-Visual Classroom 406 Hydrotech Research Institute, NTU Apr-27-2024

Hosting:
Sinotech Foundation for R&D of Engineering Sciences & Technologies
Hydrotech Research Institute, NTU
Ecological Engineering Research Center, NTU
Sinotech Engineering Consultants, Ltd.

Co-sponsor:
Water Resources Committee, /
Sustainable Development Committee
of the Chinese Institute of Civil
and Hydraulic Engineering

Co-organizer:
Dept. of Bioenvironmental Systems Engineering, NTU
Dept. of Civil Engineering, NTU
Dept. of Civil Engineering, NCHU
Sinotech Engineering Consultants, Inc.

臺日「河川預警與模擬技術」交流講席會

Taiwan-Japan River Flood Warning and Simulation Technology Workshop

In today's world, water resource management and disaster prevention have become crucial issues that countries worldwide must collectively face. This is especially true in the context of research and application of river flood warning and simulation technologies, which are indispensable. Taiwan and Japan are two island nations that are prone to frequent earthquakes and often affected by typhoons. The river systems in Taiwan and Japan play vital roles, impacting various aspects such as agriculture, urban development, and ecological conservation. Thus, through the “Taiwan-Japan River Flood Warning and Simulation Technology Workshop", we explore and strengthen collaboration and communication between the two countries in this field.

Rivers are not only vital sources of water resources but also significant areas for human life and production. However, rivers also serve as important pathways for natural disasters, including disasters like floods and landslides, which frequently cause severe losses and disruptions. Consequently, the development of effective river flood warning systems and simulation technologies is crucial for ensuring the safety of people's lives and properties. Taiwan and Japan, being countries prone to earthquakes and typhoons, have profound needs and experiences in river flood warning and simulation technologies. While both countries have relatively well-developed hydraulic engineering and disaster prevention systems, they must continuously innovate and improve to confront new challenges like climate change. Through this workshop, we hope to collectively discuss the latest developments, research findings, and application cases of river flood warning and simulation technologies, seeking better ways to address future challenges. Furthermore, both countries possess abundant resources and advantages in technology and talent. Taiwan is home to many outstanding tech professionals and research institutions, while Japan has numerous globally recognized universities and research organizations. Through this workshop, we can promote technological exchange and cooperation between the two countries, jointly promoting innovation and application in river flood warning and simulation technologies. Therefore, this workshop is organized by the Sinotech Foundation for Research & Development of Engineering Sciences & Technologies, the Ecological Engineering Research Center of National Taiwan University, the Hydrotech Research Institute of National Taiwan University, and Sinotech Engineering Consultants, Ltd. Also co-sponsored by the Sustainable Development Committee of the Chinese Institute of Civil and Hydraulic Engineering and the Water Resources Committee of the Chinese Institute of Civil and Hydraulic Engineering. and The Department of Bioenvironmental Systems Engineering of National Taiwan University, the Department of Civil Engineering of National Taiwan University, and the Department of Civil Engineering of National Chung Hsing University , Sinotech Engineering Consultants, Inc. co-organizes the event.

The Sinotech Foundation for Research & Development of Engineering Sciences & Technologies is dedicated to elevating the domestic standards of hydraulic and civil engineering technology. In addition to actively gathering relevant domestic literature on hydraulic and civil engineering, the Foundation actively introduces advanced technologies from abroad. Hence, through this workshop, we invite academic institutions with substantial experience and expertise, such as Hokkai-Gakuen University in Japan, National

Taiwan University, and National Chung Hsing University, to participate. Particularly in the fields of hydrology, water resource management, and geographic information systems, their participation will facilitate technical exchange and knowledge sharing, driving continuous innovation and improvement in river simulation technology. Additionally, the Foundation has invited the Japan River Information Center to participate in the workshop, facilitating the exchange of river hydrological data, meteorological information, and hydrological observation data. This is crucial for Taiwan's research and development of river flood warning systems. This workshop can promote cooperation between Taiwan and the Center, fully utilizing its rich data resources to ensure the reliability and accuracy of river flood warning systems.

We also hope that this workshop will provide an opportunity to promote cultural exchange and deepen friendship. Despite the geographical distance between Taiwan and Japan, there are many commonalities in culture, history, and values. Through the Taiwan-Japan River Flood Warning and Simulation Technology Workshop, we can deepen mutual understanding and friendship, collectively contributing more to global water resource management and disaster prevention. It will also strengthen cooperation and communication between the two countries in the field of water resource management and disaster prevention, promoting technological innovation and talent development, and making positive contributions to the sustainable development of the region and the world.

Basic Flood Analysis and Its Utilization Using iRIC-Nays2D Flood

阿部　孝章

Takaaki Abe

国立研究開発法人土木研究所（寒地土木研究所）　博士

Doctor, Civil Engineering Research Institute for Cold Region

iRIC Software

Changing River Science

Nays2D Flood Examples

Directed by ***Yasuyuki Shimizu***
Edited by ***Takuya Inoue***
Eiichi Suzuki
Satomi Kawamura
Toshiki Iwasaki
Michihiro Hamaki
Tomohiko Yoshida
Jagriti Mishra
Kensuke Omura
Eriko Kakegawa

iRIC version 4.0

Last Update: 2023.6.28
Release Date: 2023.6.28

Contents

Chapter 1
Using Nays2D Flood

This manual explains the basic operation and startup procedures of Nays2D Flood, which is compiled with iRIC. TheNays2D Flood solver simulates two-dimensional plane river flow and riverbed deformation. It was developed by Professor Yasuyuki Shimizu of Hokkaido University. The following explanation is based on the assumption that you have installed the iRIC software on your computer. If you have not installed the iRIC software, download it from the following URL and install it on your computer.

URL:	http://i-ric.org/download
Software:	iRIC version4.0 or later

1. Nays2D Flood basic operating procedures

The following are the basic procedures for operating Nays2D Flood with iRIC:

Launching Nays2D Flood
Prepare to use Nays2D Flood on iRIC.

↓

Creating the calculation grids
Create a grid for calculation using Digital Elevation Model (DEM) data or the like.

↓

Setting the calculation conditions
Set simulation discharge, boundary conditions, roughness and other items.

↓

Making simulations
Run Nays2D Flood for the simulation.

↓

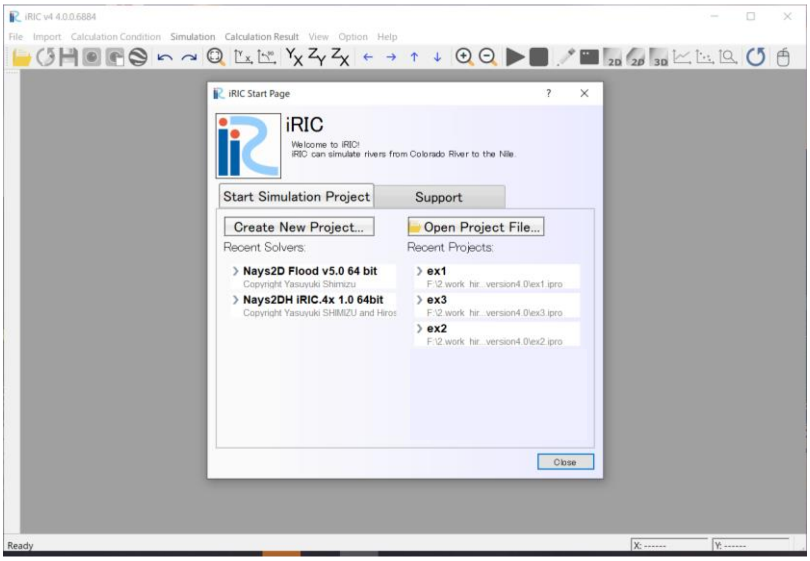
Visualizing the calculation results
Visualize the simulation results, such as flow velocity, water depth and riverbed elevation, by means of a contour map/vector map to see whether the simulation has successfully run.

2. Launching Nays2D Flood

The following is the procedure to launch Nays2D Flood on iRIC.

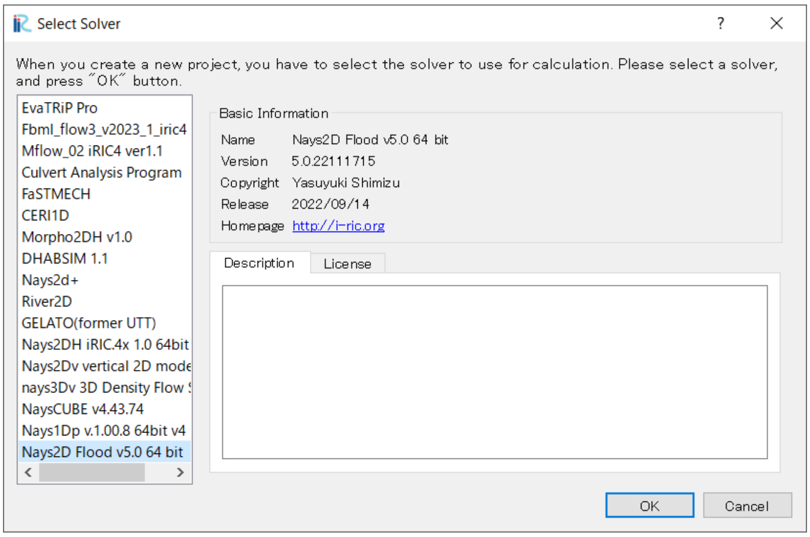
When launching iRIC, the [iRIC Start Page] window will open.

Click on [Create New Project] in the [iRIC Start Page] window.

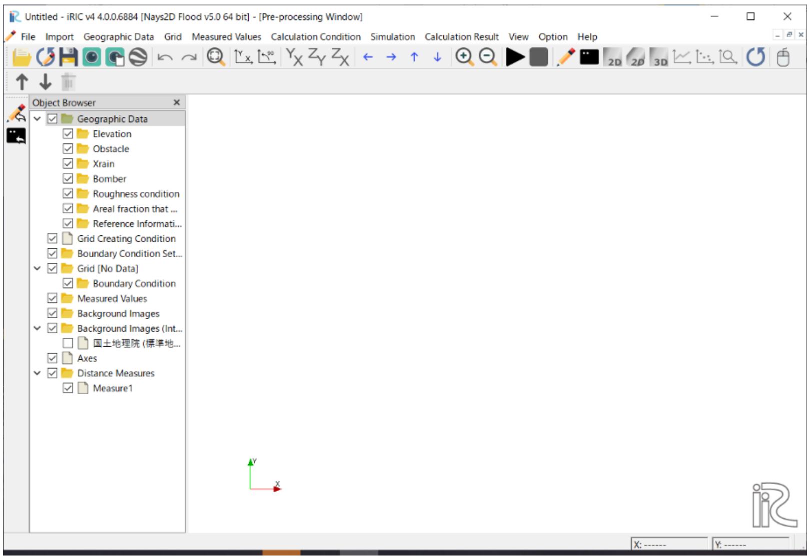


The [Select Solver] window will open.

Select [Nays2D Flood v5.0] in the [Select Solver] window, and click on [OK].



A window with the title bar "Untitled-iRIC v4 [Nays2D Flood v5.0]" will appear.



Nays2D Flood v5.0 is ready for use.

3. The sample data

The sample data used for the sample simulations explained in this manual are available at:

URL: http://i-ric.org/en/download
Examples of simulation: Nays2D Flood

To run Nays2D Flood according to this manual, data should be downloaded from the above site or provided by the staff in an iRIC training course.
The project files (*.ipro) in the description and sample data of each chapter in this manual is created by following solver. Although the project file cannot be used without change if the solver version is different, resetting calculation conditions and so on, following this manual enables calculation with other versions.

Solver: Nays2D Flood v5.0

Sample data and corresponding solver version

Data name	Contents	version
N2F	Examples of flood calculation for an actual river section	Nays2D Flood 5.0
N2F_2	Examples of tsunami runup analysis using SRTM data	Nays2D Flood 5.0
N2F_3	Examples of rainfall induced flooding in actual river	Nays2D Flood 5.0

4. Simulation conditions of Nays2D Flood

This manual teaches how Nays2D Flood is used to simulate river flow during flood events for two relatively simple examples. There are some omissions in the explanations of the physical and numerical aspects of the simulation conditions that are to be set. Nays2D Flood has other functions (setting conditions) in addition to those explained by this manual and these two specific examples. For more complete details, please refer to the Nays2D Flood Solver Manual.

Chapter 2
Examples of flood calculation for an actual river section

Objectives

Simulate the flow regime (water depth and flow velocity) using Nays2D Flood for an actual river section with a flood discharge, and check to see whether the simulation is successfully run.

Outline

1. Creating the calculation grid

Using elevation data of an actual river section, create a calculation grid.

2.Setting the calculation conditions

Set flood discharge for unsteady flow. Set various other conditions necessary for simulation.

3. Making a simulation

4. Visualizing the calculation results

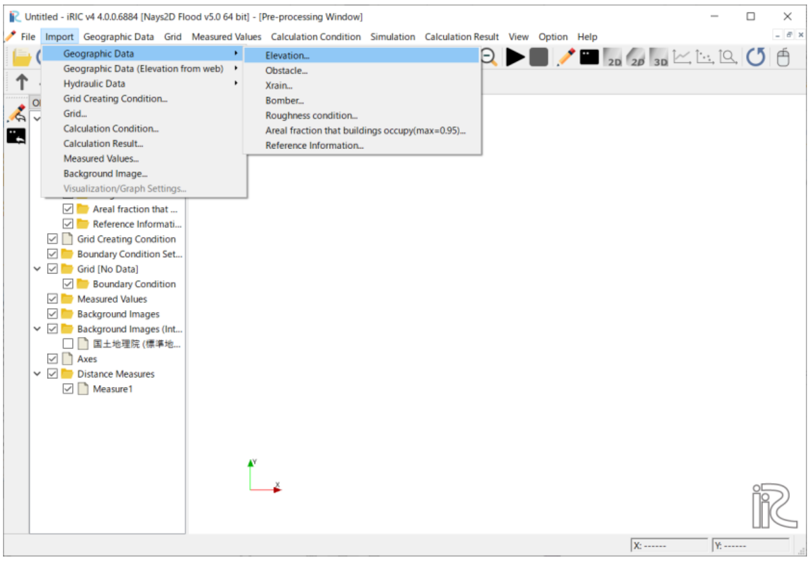
Here, we introduce how to display a water-depth contour map and a flow velocity vector map.

1. Creating the calculation grid

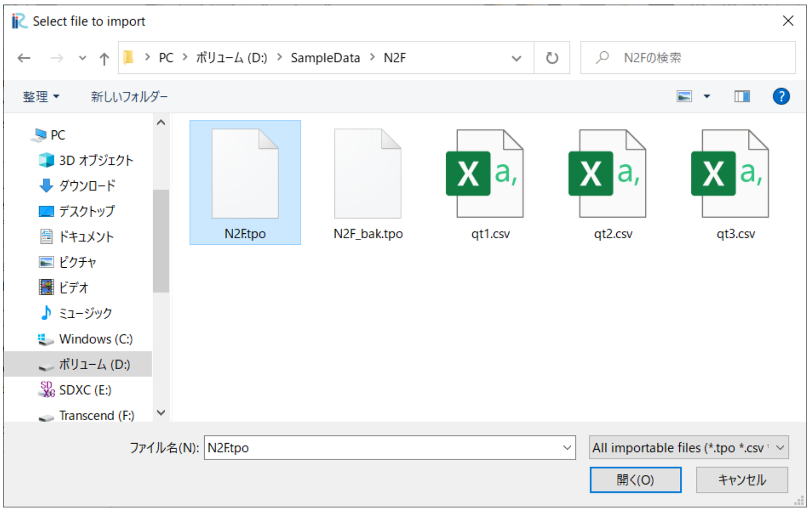
1 Importing cross-sectional river survey data

Importing geographic data

On the menu bar, select [Import] - [Geographic Data] - [Elevation].

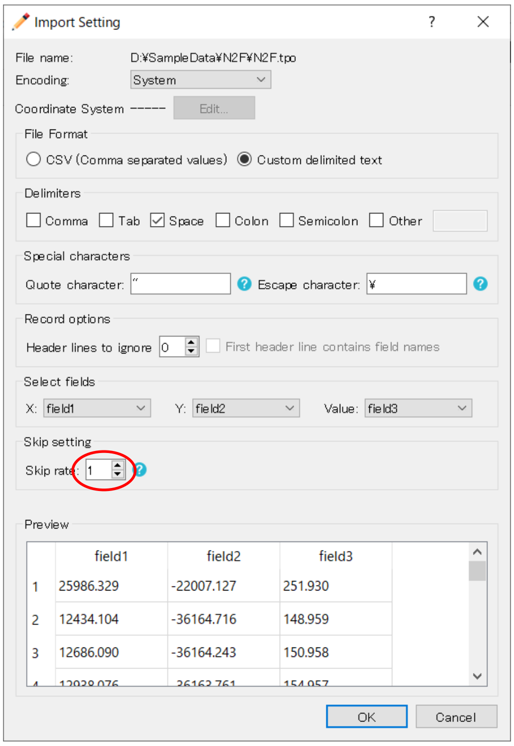


Open [¥¥SampleData¥¥N2F], select [N2F.tpo] and click on [Open].

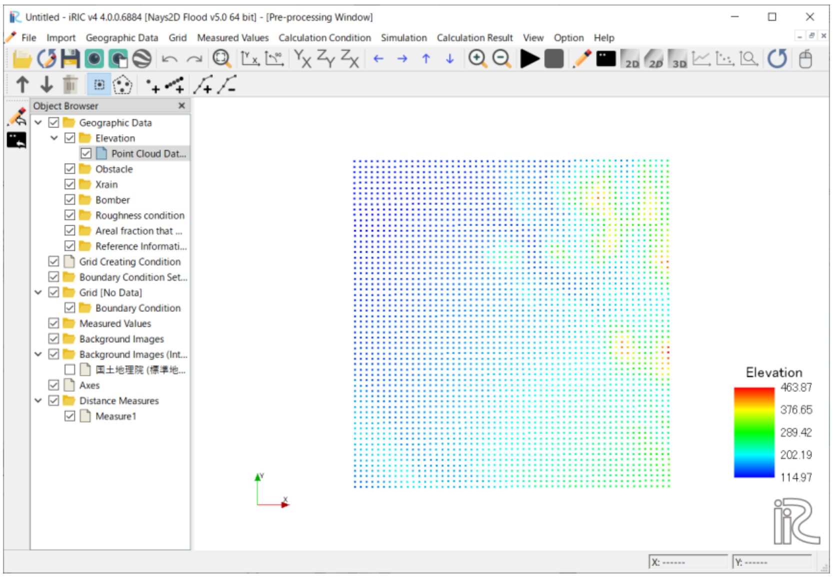


On the dialog [Filtering Setting], input [1] for [Filter] value and select [OK].

Set a larger number for the filter value and filter the dataset if the operation is slow because of the high number of data points. The filter simply skips over the data; i.e., if the filter is one, each data point is read into iRIC, but if the value is two, only every other data point is read in, and so forth.

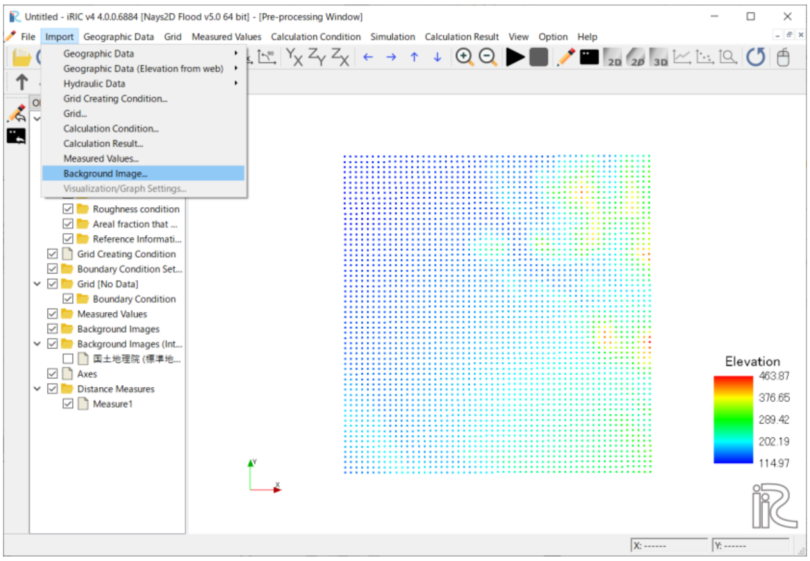


If the [Pre-Processing] window shows the shape of the river section that you are simulating, the data have been successfully imported.



Importing a background image

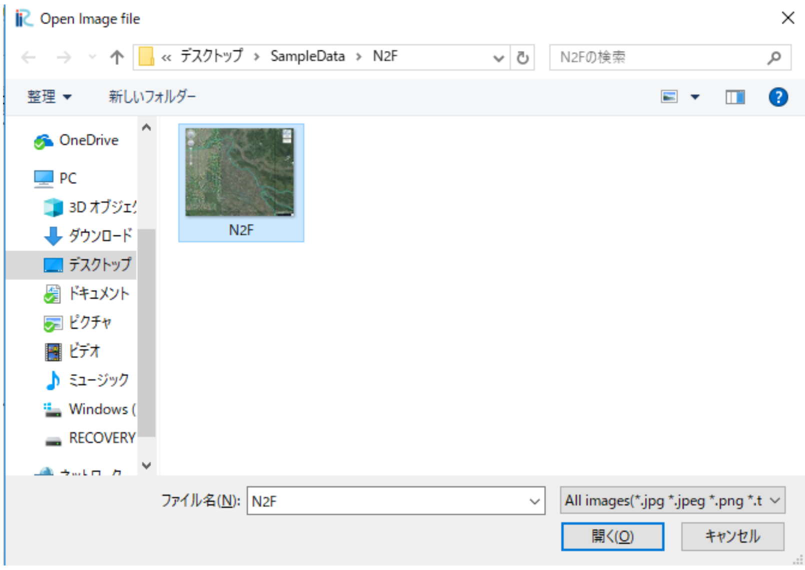
On the menu bar, select [Import] - [Background Image].



Background image

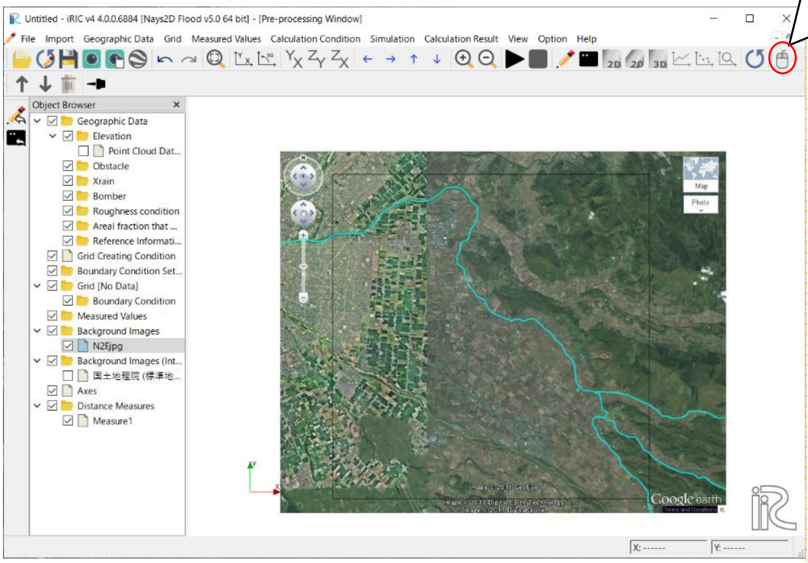
When creating grids for calculation, importing background images such as maps and aerial photos makes it possible to create grids that incorporate riverbanks and land use. Obstacle cells and roughness cells mentioned below can be set in reference to the background image.

Open [¥¥SampleData¥¥N2F] and select [N2F.jpg].



Using the [Move], [Rotate] and [Zoom] functions, match the background image with the elevation data.

[Move] [Rotate] [Zoom] refer to mouse hint

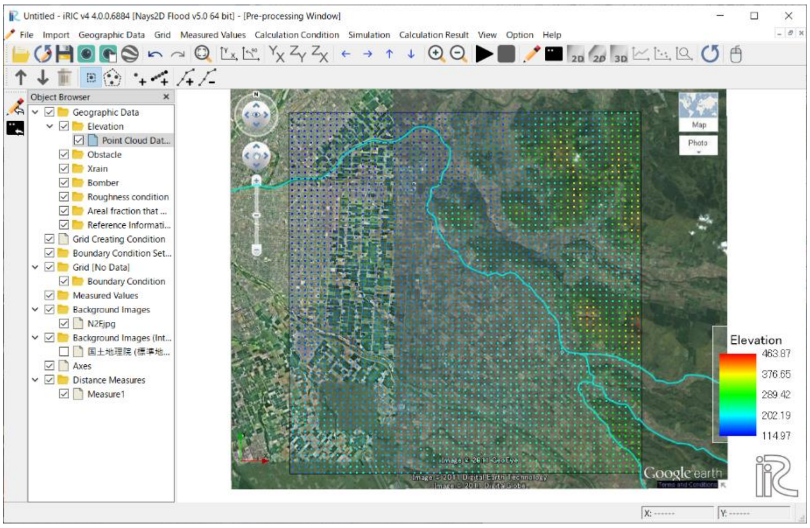


Hint

It is convenient to find the geographic features and mark on the background image when you match the background image with elevation data.

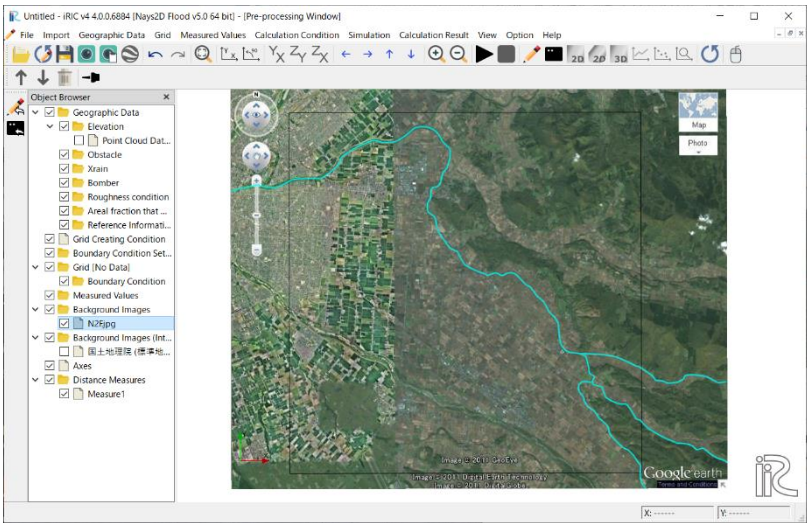
In this case, the frame line is drawn and you can use this frame line to match.

Referring to the hint above, match the background image with the elevation data.

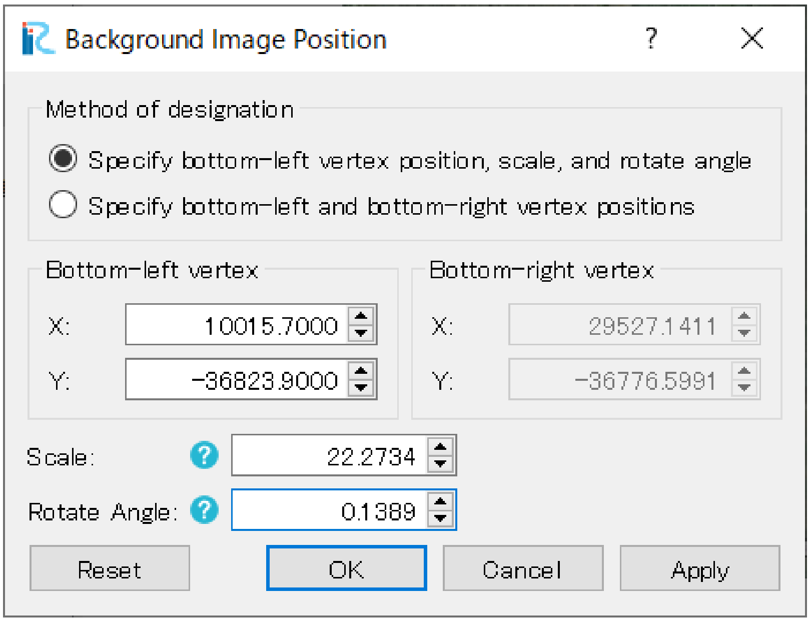


Click on [Background Image]-[N2F.jpg] on the object browser.

Fix the background image with [fix position].



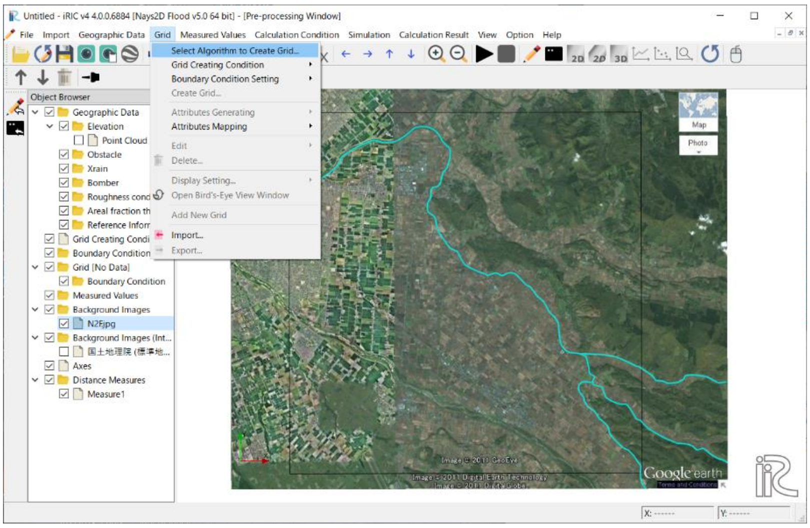
You can also set the position of the background image by right-clicking the object browser [Background Image]-[N2F.jpg] and specifying coordinates from [Properties].



2 Selecting an algorithm for creating a grid

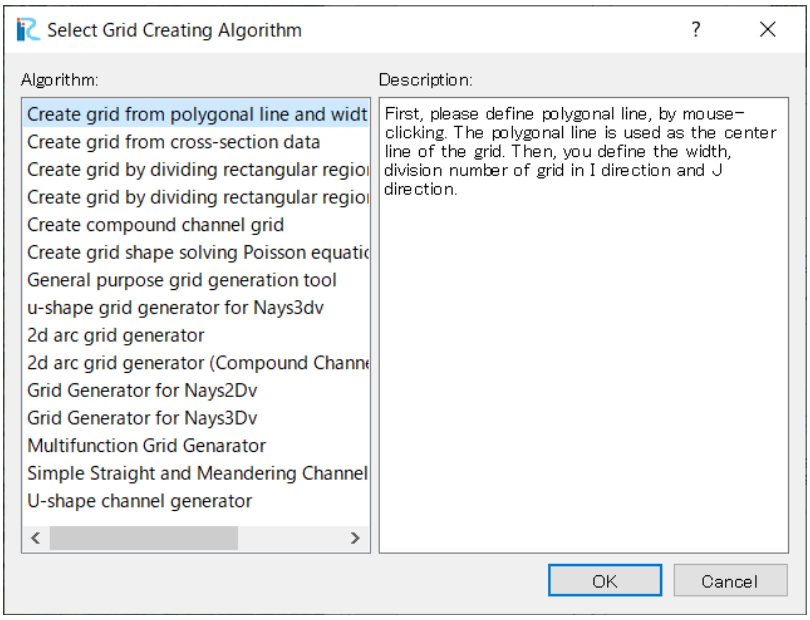
On the menu bar, select [Grid] - [Select algorithm to create grid].

The [Select Grid Creating Algorithm] window will open.



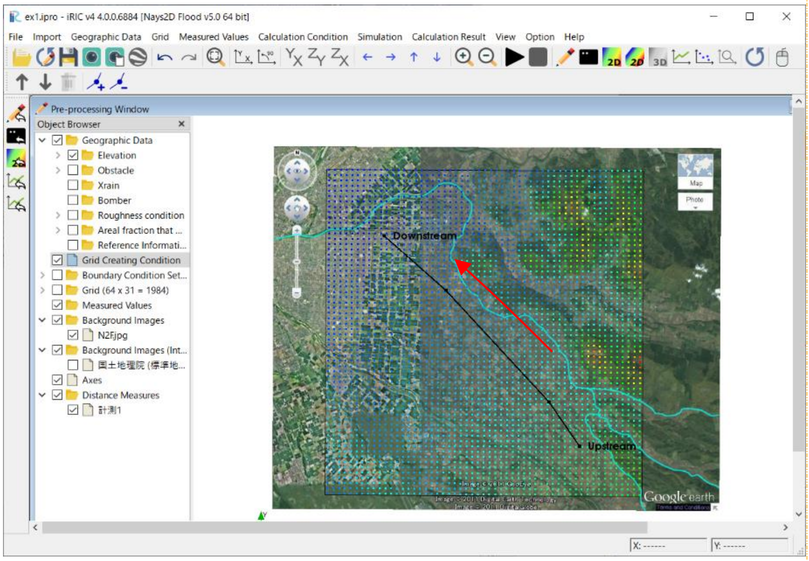
Select [Create grid from polygonal line and width] from the list below the [Select Grid Creating Algorithm] window, and click on [OK].

Nays2D Flood primarily creates a grid from polygonal lines and grid widths.



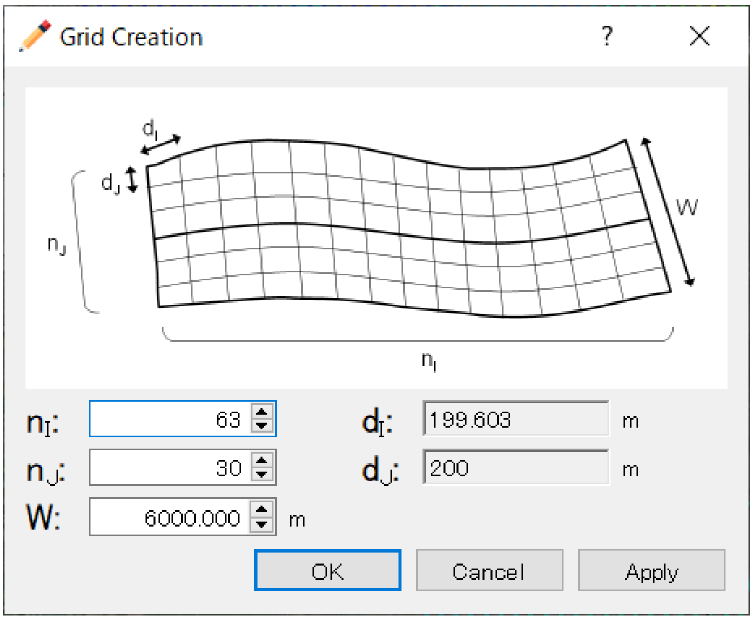
3 Creating a grid

Right-click on several points through which the grid centerline passes, and press the “Enter” key.



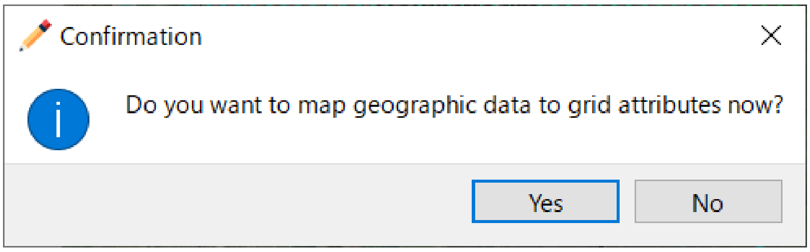
Set the grid centerline from the upstream end, where flood flow enters, to the downstream end, where the flood flow exits. To finish, double-click on the end of the centerline, or hit the “Enter” key.

In the [Grid Creation] window, make the following settings and click on [OK].



Number of divisions in the longitudinal direction : **63**
Number of divisions in the transverse direction : **30**
Grid width in the transverse direction : **6000 m**

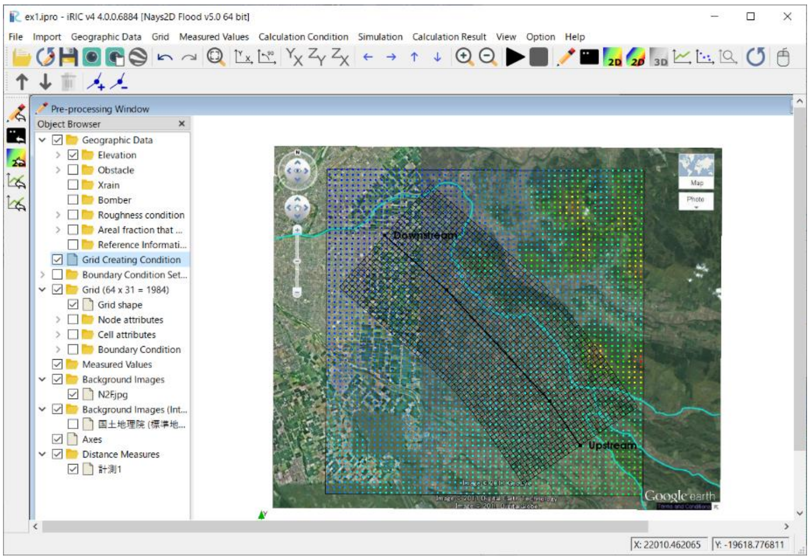
Click on [Yes] in the [Confirmation] window.



Mapping geographic data

Elevation data are applied to the grid.

A grid will be created.



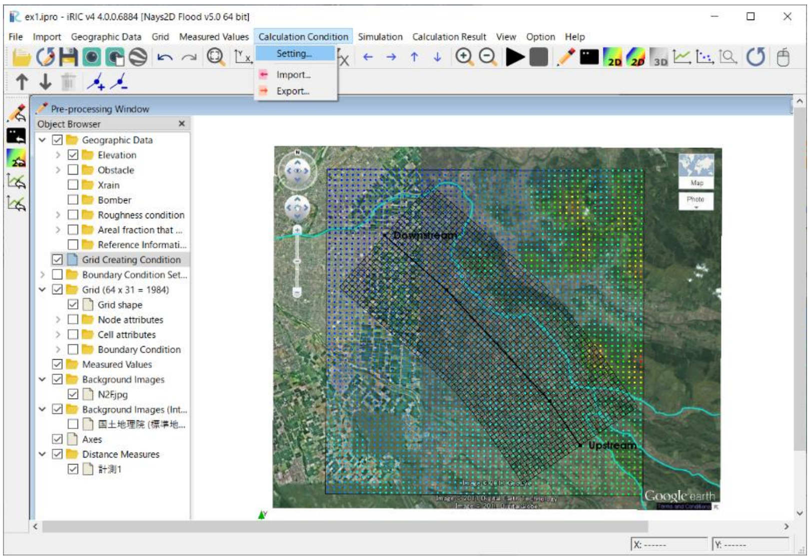
It is possible to move, add or remove centerline points under [Grid Creating Conditions] even after the grid is created.

2. Setting the calculation conditions

8 Open "Calculation Conditions"

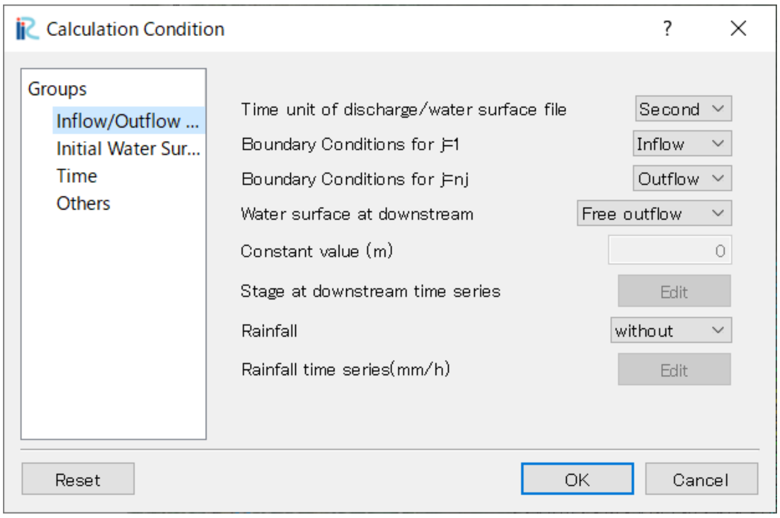
On the menu bar, select [Calculation Conditions] - [Setting].

The [Calculation Conditions] window will open.



9　Setting the inflow boundary conditions

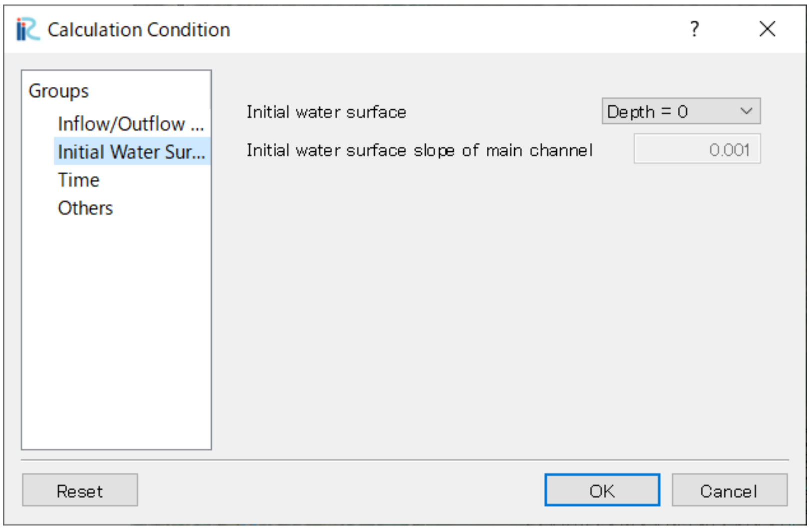
Click on [Inflow/ Outflow Boundary Conditions] from the [Groups] list to make the following settings:



Time unit of discharge/water surface elevation files: Second
Boundary Conditions for j=1: Inflow
Boundary Condition for j=nj: Outflow
Water surface at downstream: Free Outflow
Rainfall: without
Note：Inflow conditions are decided at [6.Setting the inflow river conditions]
Use constant value or read from file when the stage at downstream is effected by sea level or overflow level at the outflow.

10 Setting the initial water surface profile

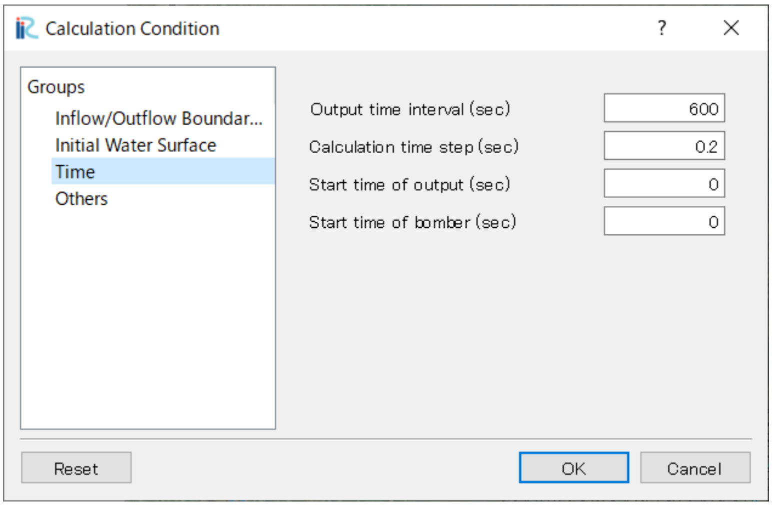
Click on [Initial Water Surface Elevation] from the [Group] list to make the following settings:



Initial water surface: Depth = 0
Note: When the water surface elevation of the downstream end may be affected by the sea level or the downstream flood level, use a constant value (a line).

11 Setting the time

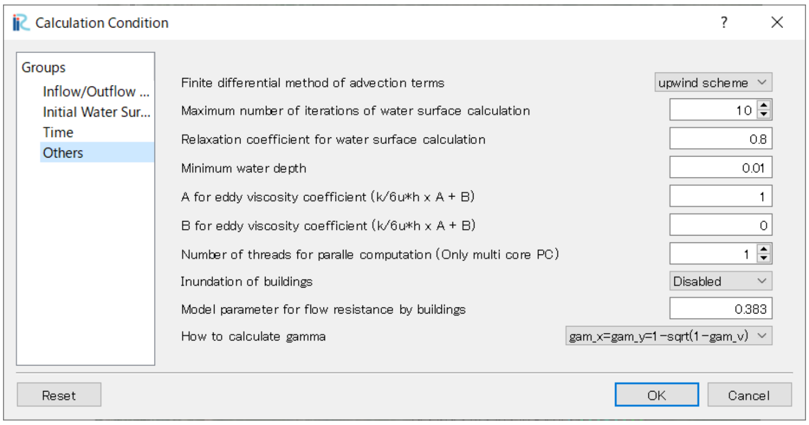
Click on [Time] from the [Group] list to make the following settings:



Output time interval (sec): 600
Calculation time step (sec): 0.2
Start time of output (sec): 0
Start time of bomber (sec): 0

12 Other settings

Click on [Other] from the [Group] list to make the following settings:

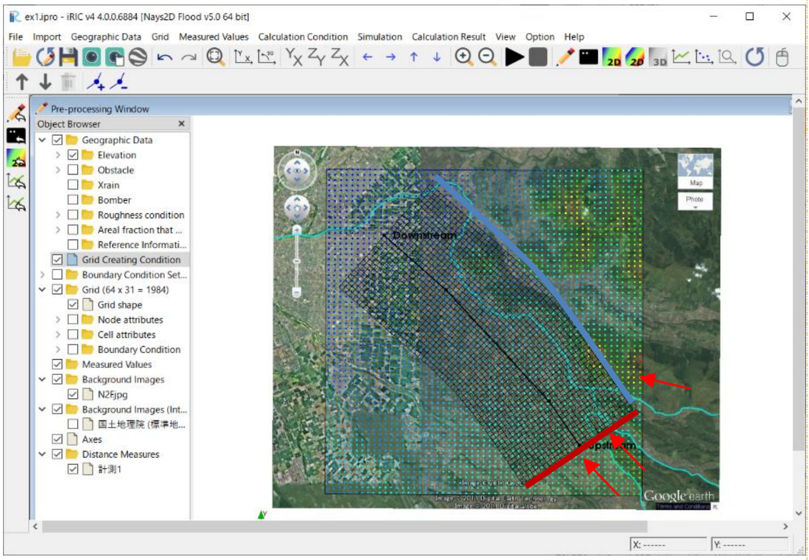


Finite difference method of advection terms: upwind scheme
Maximum number of iterations of water surface calculation: 10
Relaxation coefficient for water surface calculation: 0.8
Minimum water depth: 0.01
A for eddy viscosity….: 1
B for eddy viscosity….: 0
Number of threads ….: 1

Many other conditions can be set; however, they do not need to be set for this simulation as the default values are acceptable. After making the settings above, click on [Save and Close] to close the window.

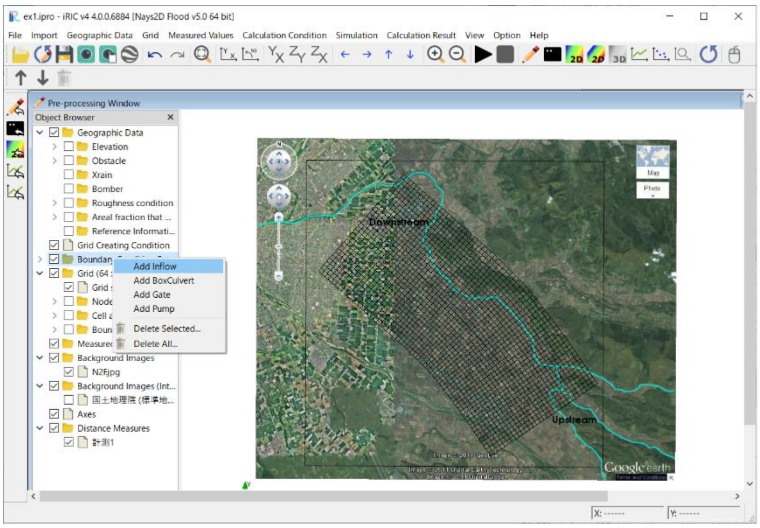
13 Setting inflow rivers

Set the inflow rivers (or the bank opening point) at the upstream (red line) and right side (blue line) since you set the [boundary condition for j = 1] as [inflow] in the inflow/outflow boundary conditions.



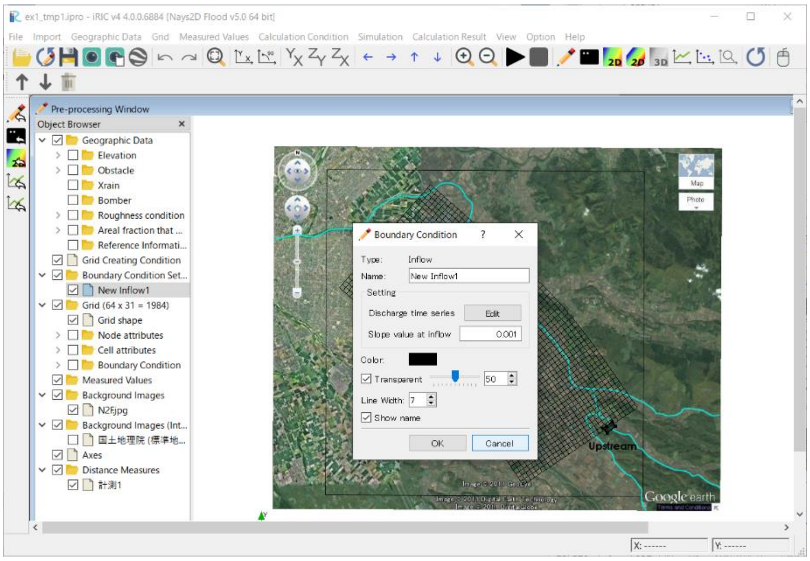
In this river section, two rivers (including agricultural canals) flow across the red line and one river flows across the blue line, as indicated by the red arrows.
No river comes in the area from the outlet i=ni or the other bank j=nj and these boundary conditions are outflow only.

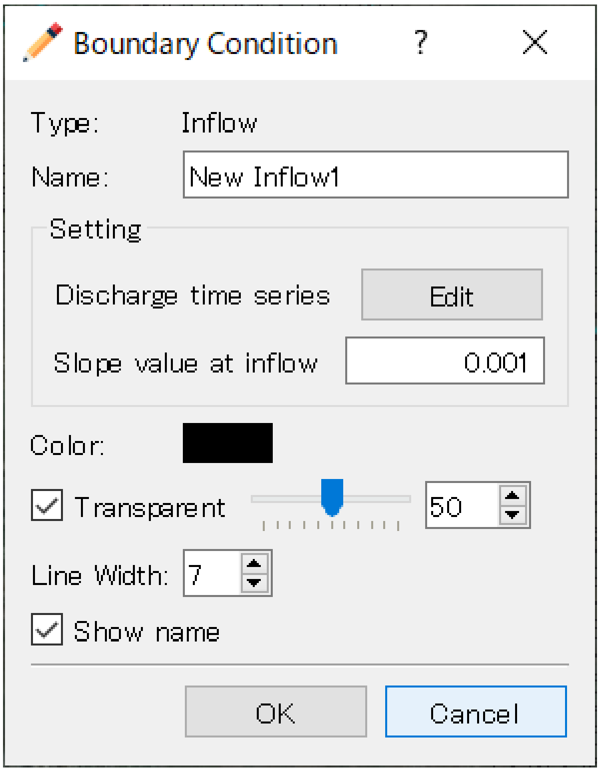
In the Object Browser, select [Boundary condition setting]-[Add inflow]



While selecting the [Inflow], enclose the sides of “Inflow” locations with polygons.

When the location is confirmed, the [Boundary condition] setting dialog opens.

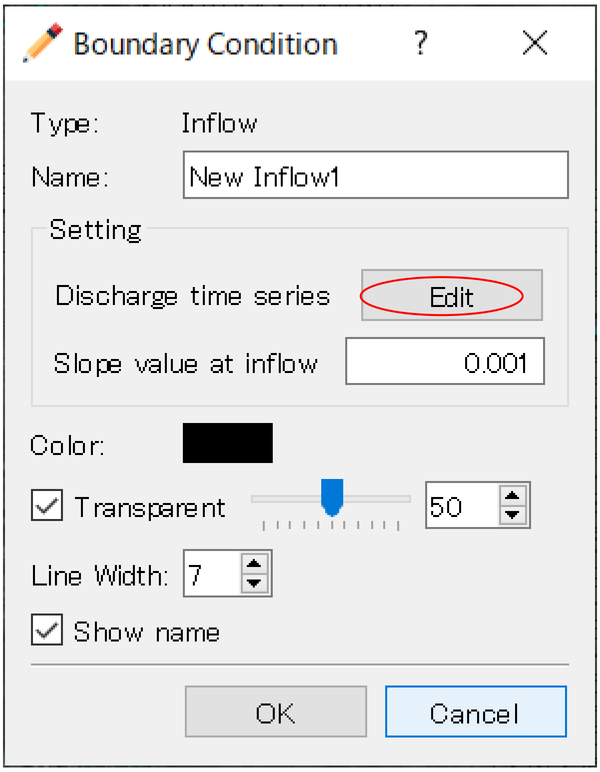




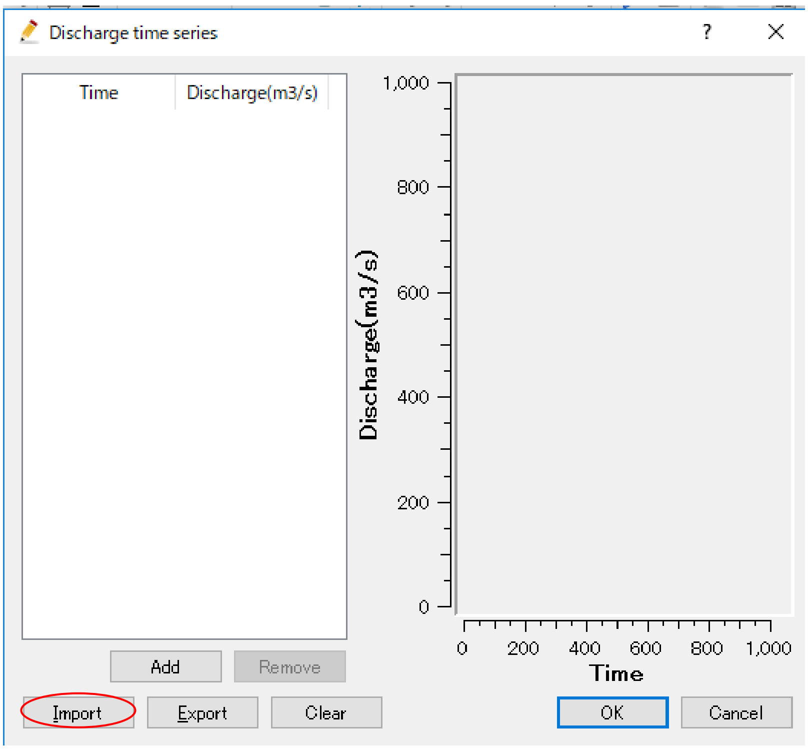
Name: arbitrary name, we use New Inflow 1

Slope value at inflow: 0.001

In the [Boundary Condition], click on [Discharge time series]

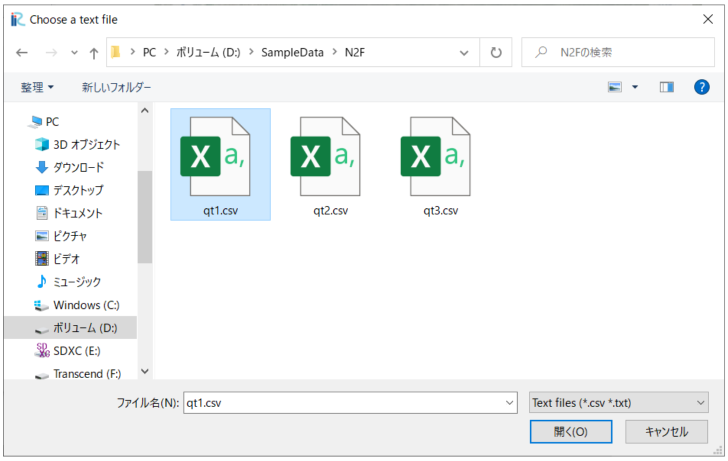


In the [Calculation Condition], click on [Import]



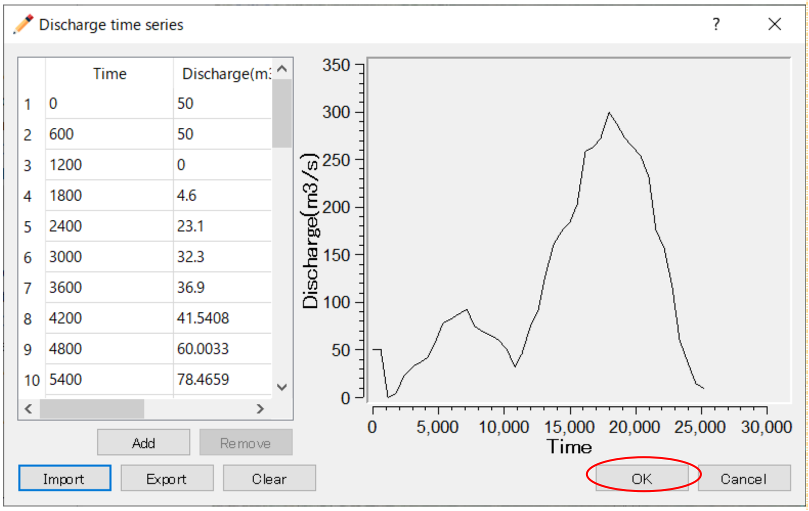
In [Choose a text file] click on [qt1.csv] and then [open].

Open the discharge data.



Click on [OK].

Repeat above operations for New Inflow 2 to New Inflow 3.



Correspondence of the inflow river and the discharge data are as follows;

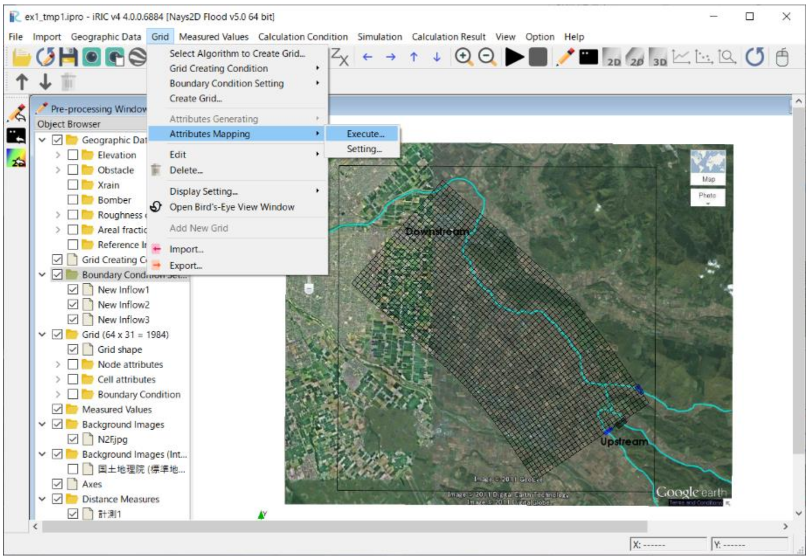
Inflow1: qt1.csv

Inflow2: qt2.csv

Inflow3: qt3.csv

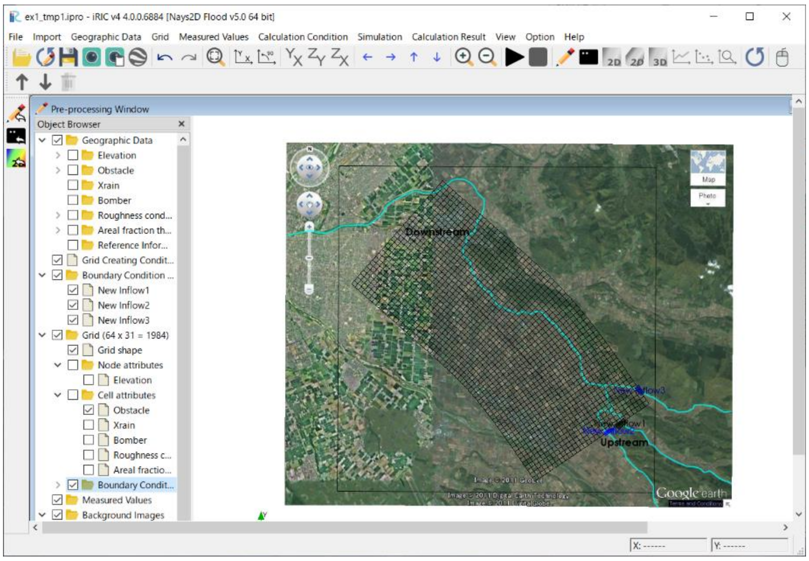
Note: The time step of the discharge files need to be set the same.

Click on [Grid] on the menu bar [Attribute Mapping]-[Execute].



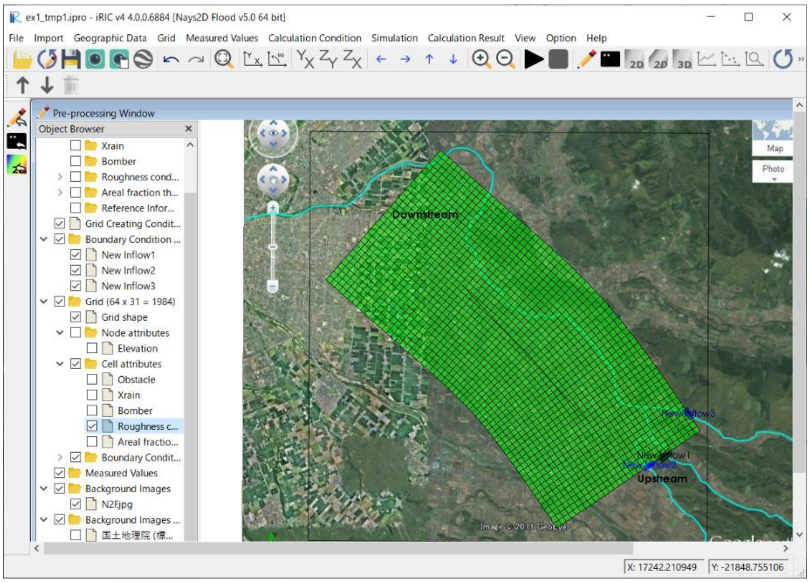
Position setting of inflow boundary condition

Inflow rivers are set on the sides.

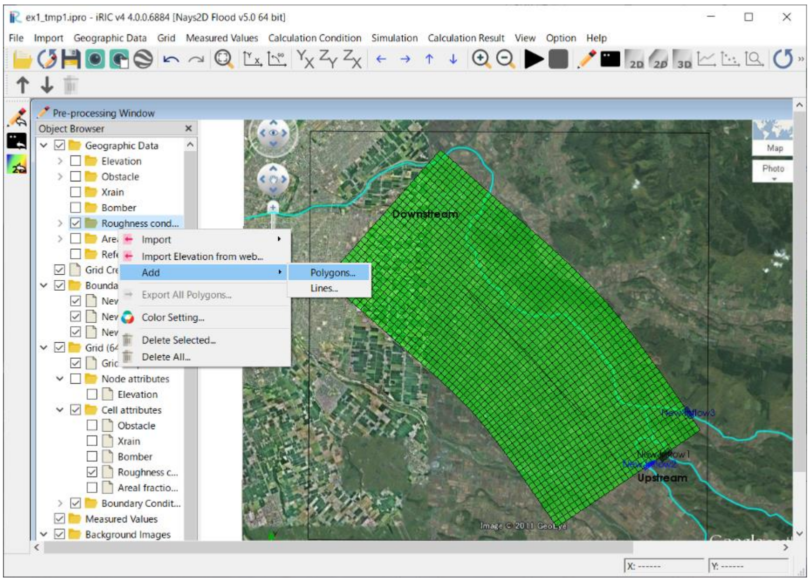


14 Setting roughness condition

In the Object Browser, check [Grid] – [Cell attributes]-[Roughness condition].



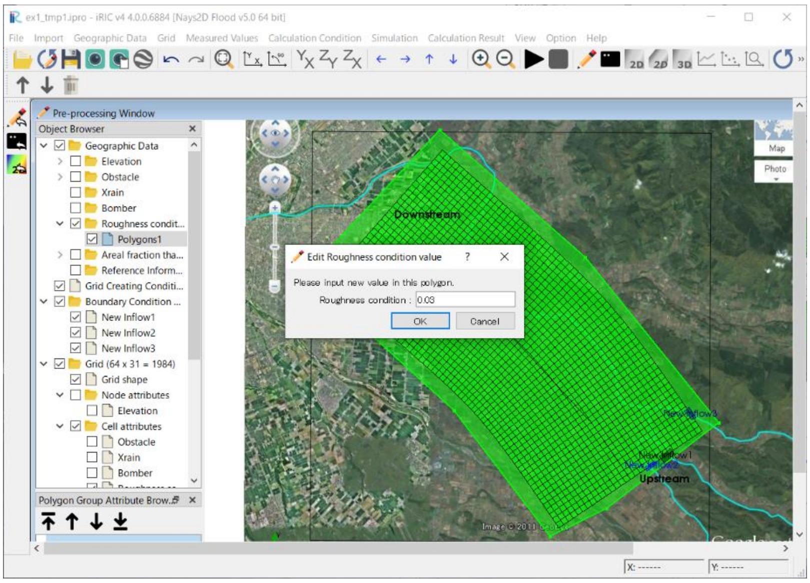
Right-click on the object browser [Geographic Data]-[Roughness condition] and click [Add]-[polygon].



Click on Object Browser[Geographic Data]-[Roughness condition]－[Polygon1].

Double-click the cell with the same roughness surrounded by polygons.

In [Edit Roughness condition value], directly input the desired Manning roughness.

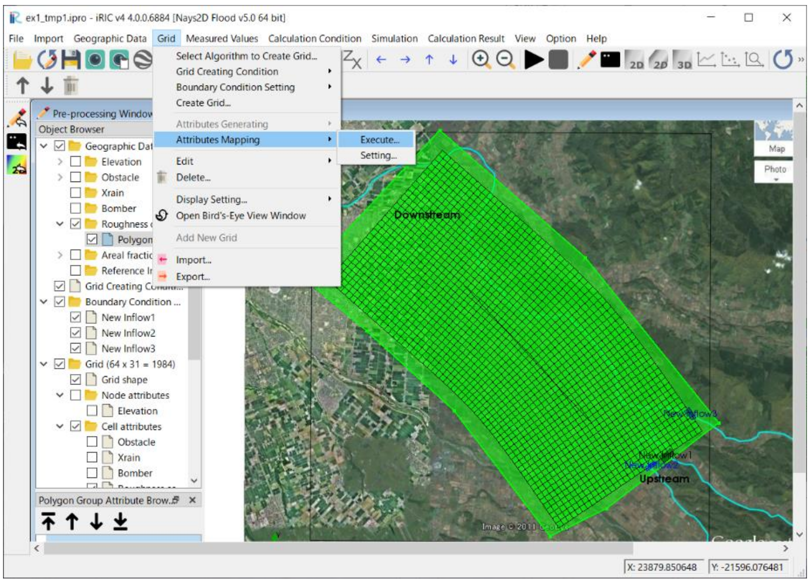


Roughness condition:0.03

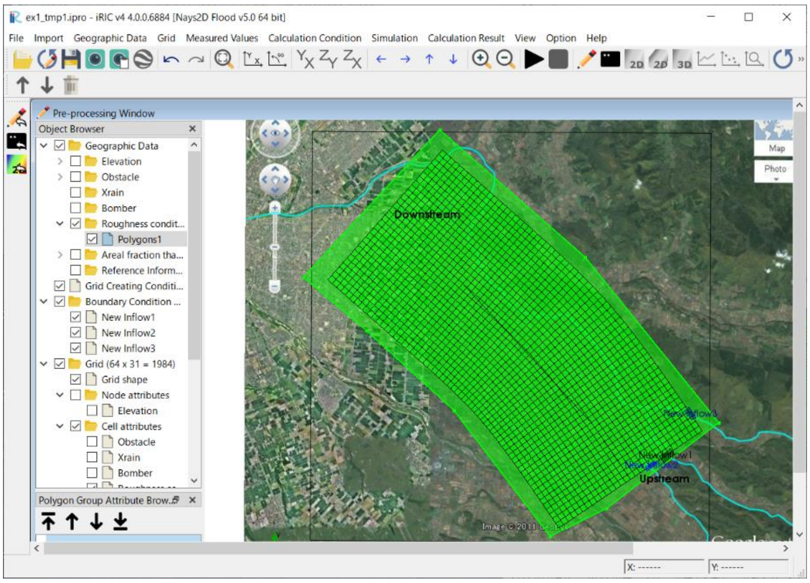
Note: Set Manning's roughness coefficient by taking comprehensive consideration of calculation model, land use and past flooding data.

In setting the roughness coefficient, refer to page 33 of Flood Simulation Manual (draft) and page 89 of Hydraulic Formula 1999 Edition (both in Japanese).

Click on [Grid] on the menu bar – [Attribute Mapping]-[Execute].



Roughness is reflected in the cell.



References:

Extract from pp. 33 of

Flood Simulation Manual (Draft) – Guide for simulation and verification of new model, Urban River Research Lab, River Section, Public Works Research Institute (Japan), February 1996

1) Find the area of each land use within each mesh. Land use: building; farmland A_1, road A_2 and other land use A_3. Here, "farmland" means rice paddy, upland field or orchard. "Road" includes sidewalks along the road. Wilderness, grass fields and wetlands are included in "other land use."
2) The roughness coefficients by land use are set as follows: The bottom roughness coefficient other than that of buildings is calculated from the following weighted average:

   エラー! 編集中のフィールド コードからは、オブジェクトを作成できません。

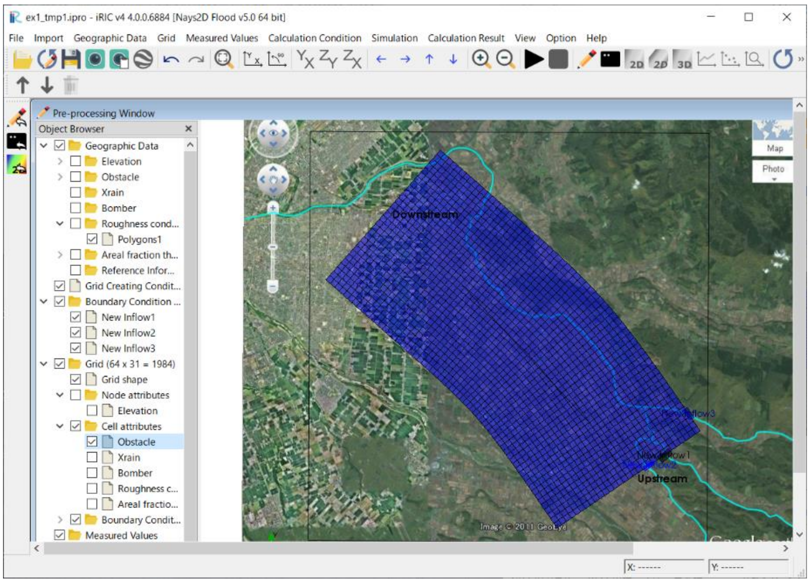
   where, n_1=0.060, n_2=0.047 and n_3=0.050.
3) The composite equivalent roughness coefficient of the land and buildings is given by the following equation:

   エラー! 編集中のフィールド コードからは、オブジェクトを作成できません。

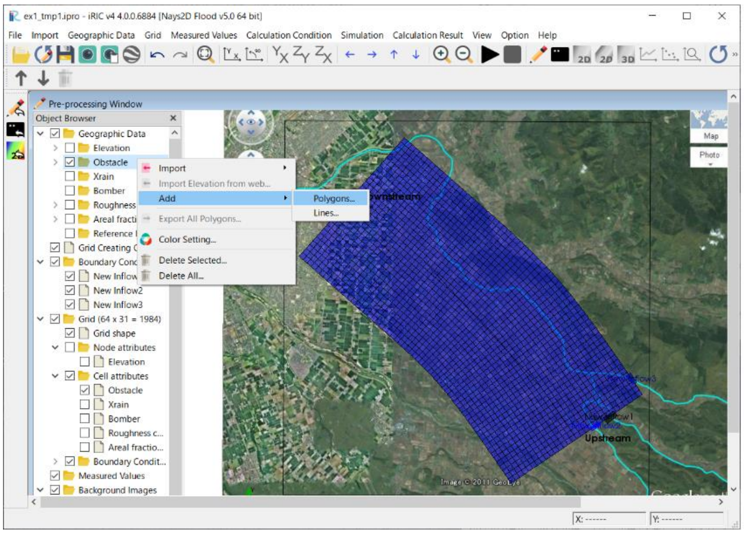
   where, θ is the proportion of land occupied by buildings and h is water depth.

15 Setting obstacle cells

In the Object Browser, check [Grid]-[Cell attributes]-[Obstacle].



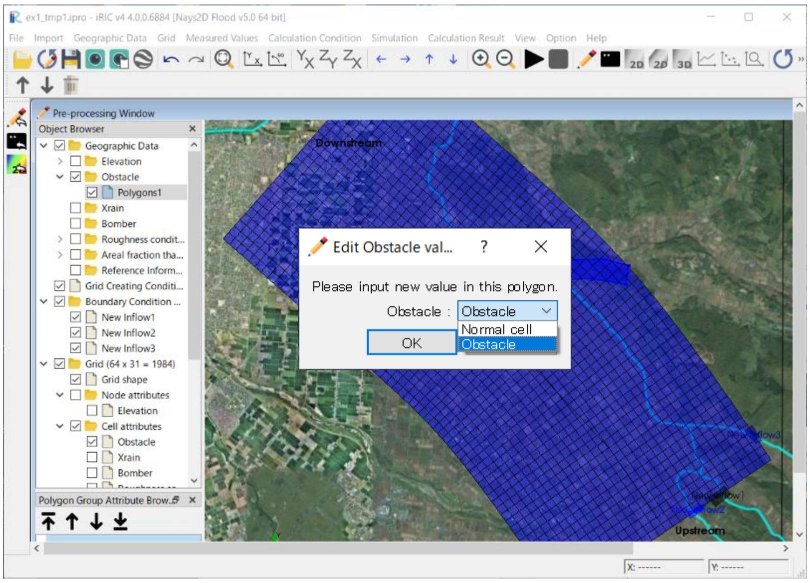
Right-click the object browser [Geographic Data]-[Obstacle] and click [Add]-[Polygon].



Click the object browser [Geographic Data]-[Obstacle]-[Polygon1].

Double-click a cell that will become an obstacle cell with polygons.

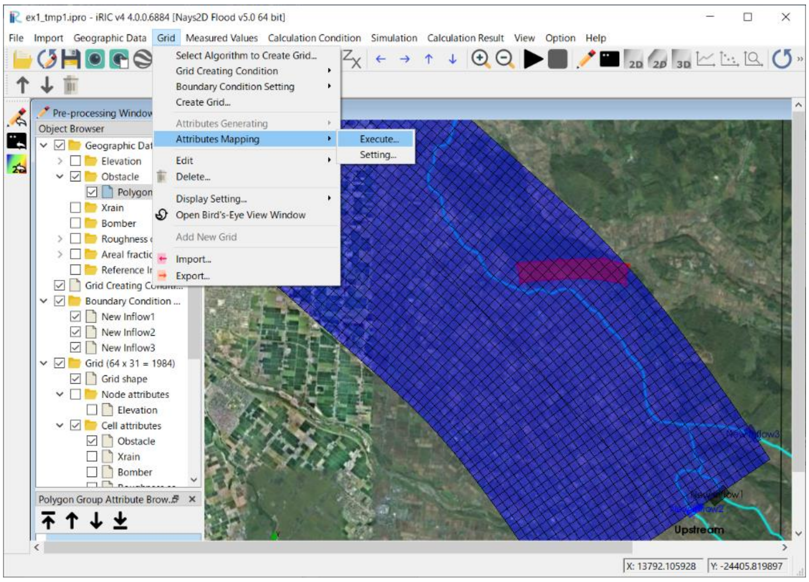
In [Edit Obstacle value], select Obstacle.



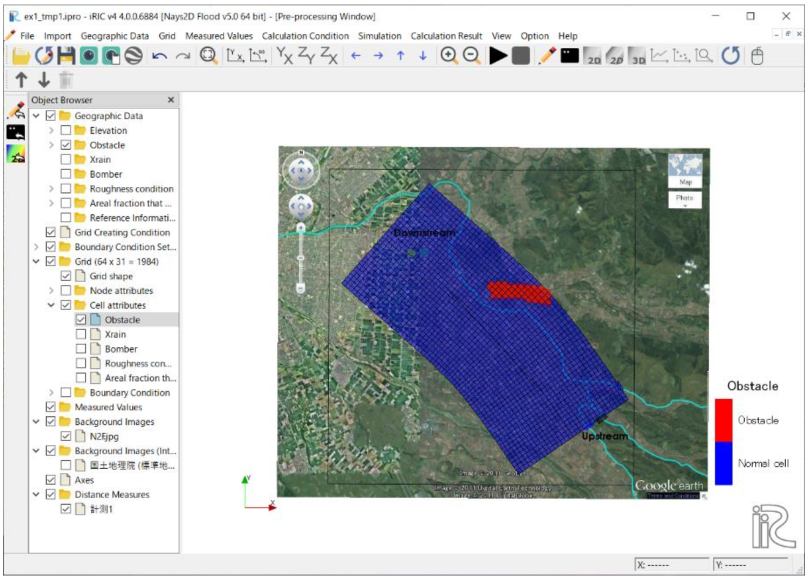
Set obs tacles such as roads, banks and embankments.

Note: Unless the calculation grid cells are small enough to depict roads, banks and embankments as a part of the topography, set roads, banks and embankments as obstacles.

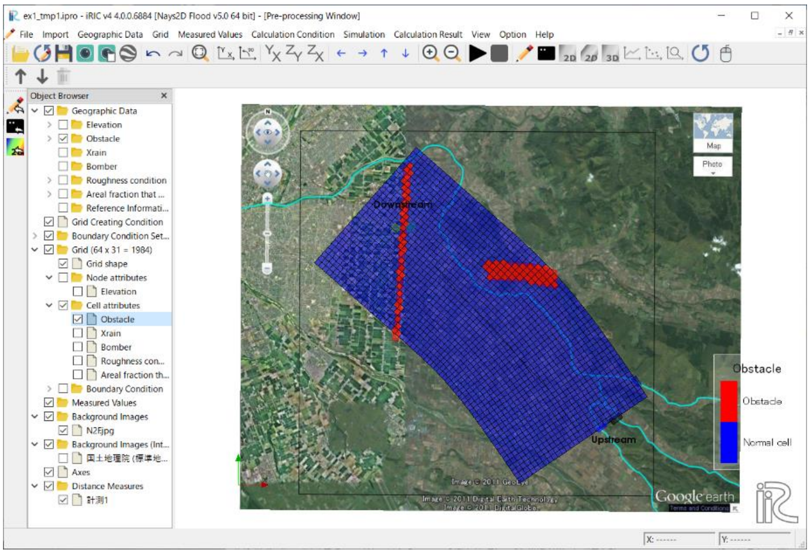
Click on [Grid]on the menu bar –[Attribute Mapping]-[Execute].



Obstacle is reflected in the cell.

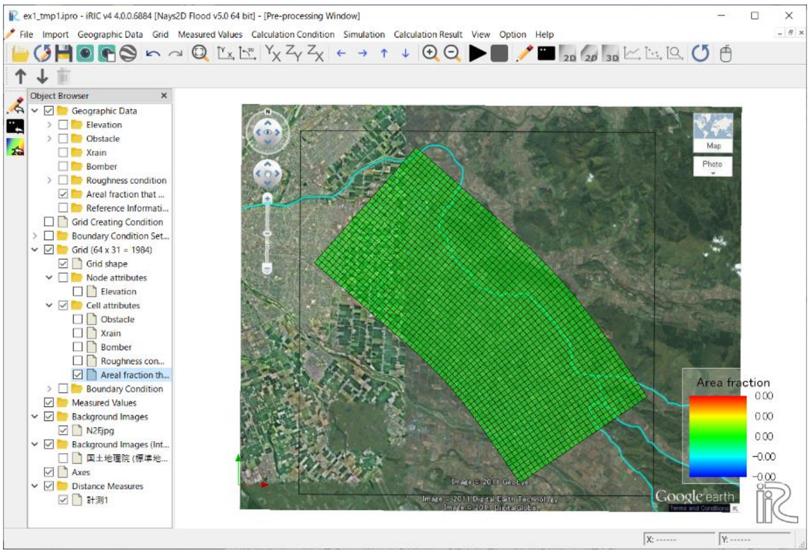


Repeat above operations from Polygon1 to Polygon2.

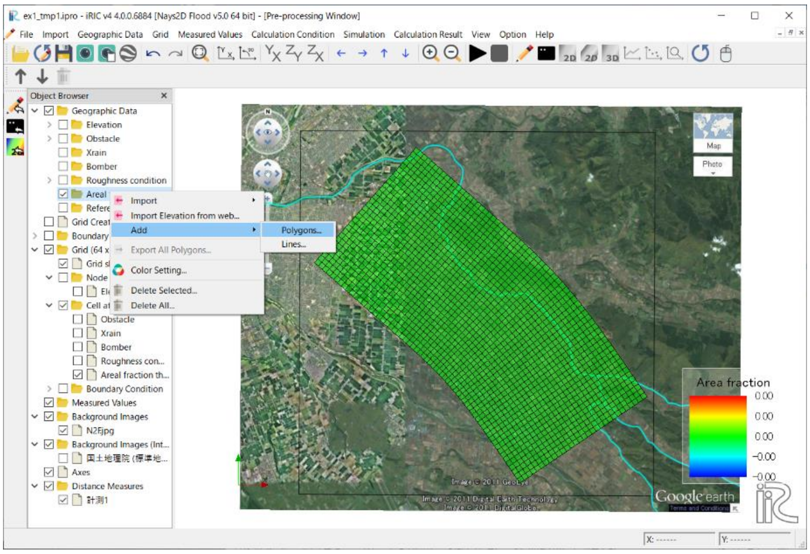


16 Setting Areal fraction that buildings occupy

Check the Object Browser [Grid] - [Cell attributes]-[Areal fraction that buildings occupy].



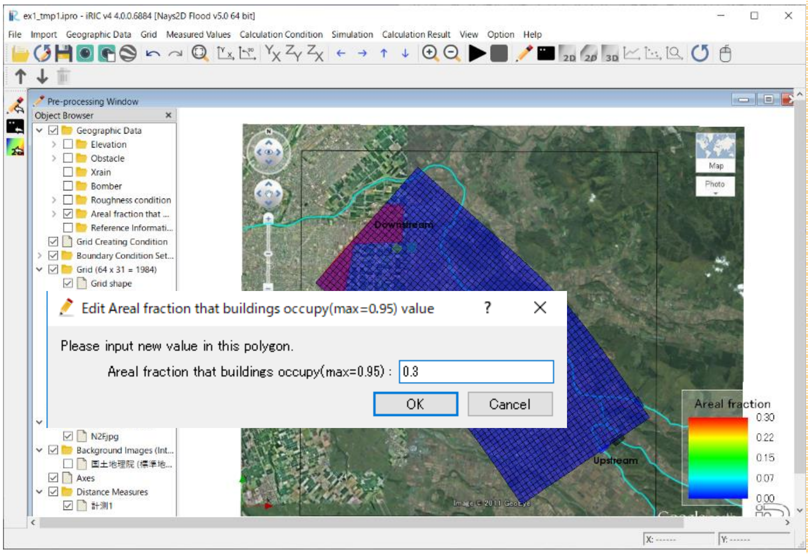
Right-click the Object Browser [Geographic Data] - [Areal fraction that buildings occupy] and click [Add] - [Polygon].



Click the Object Browser [Geographic Data]-[Areal fraction that buildings occupy]-[Polygon1].

Double-click the cell with the same building occupancy surrounded by polygons.

In [Edit Areal fraction that buildings occupy value], enter directly the building occupancy fraction.



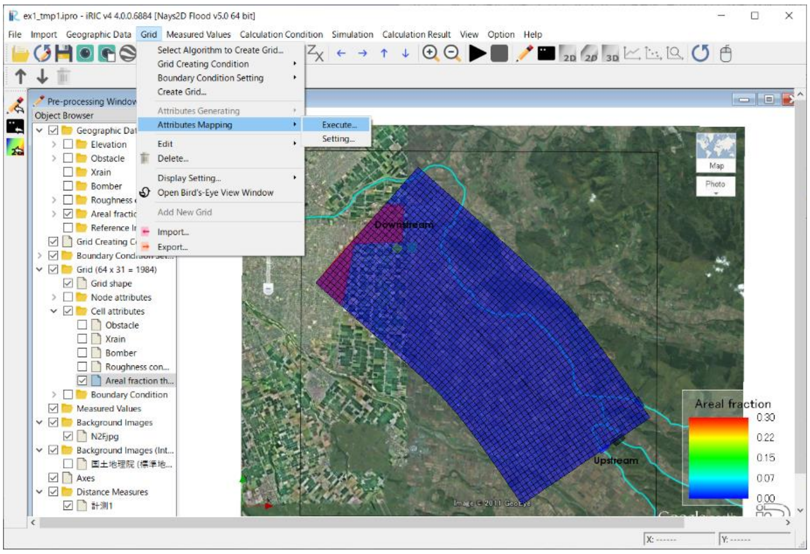
Building occupancy ratio: 0.3

Note: Resistance by buildings is based on the phenomenon that flow is affected by buildings and flooding occurs in buildings. For this reason, the premise differs from the above-mentioned obstacle setting, so be careful.

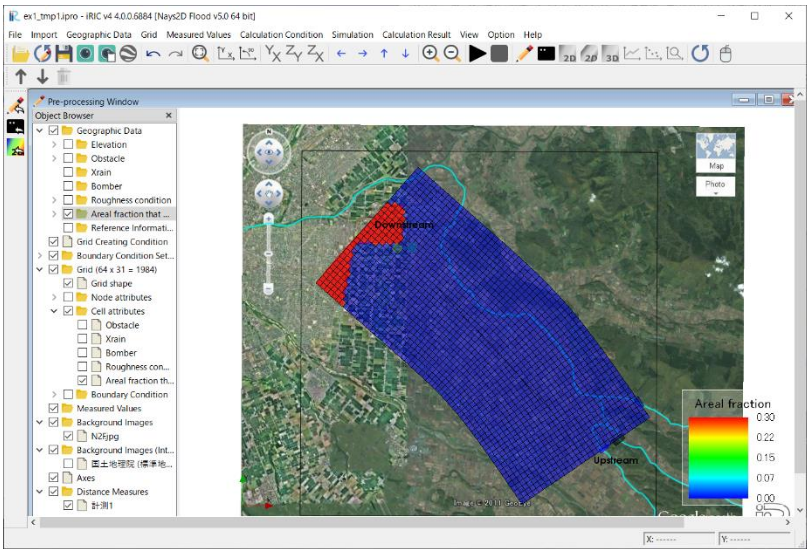
If the areal fraction that buildings occupy is larger than 0.95, the model set the value to 0.95. Because this model is a model for permeable obstacles and can not consider completely impermeable obstacles.

When you want to put impermeable obstacles, please use [Obstacle cell].

Click the menu bar [Grid]-[Attribute Mapping]-[Execute].



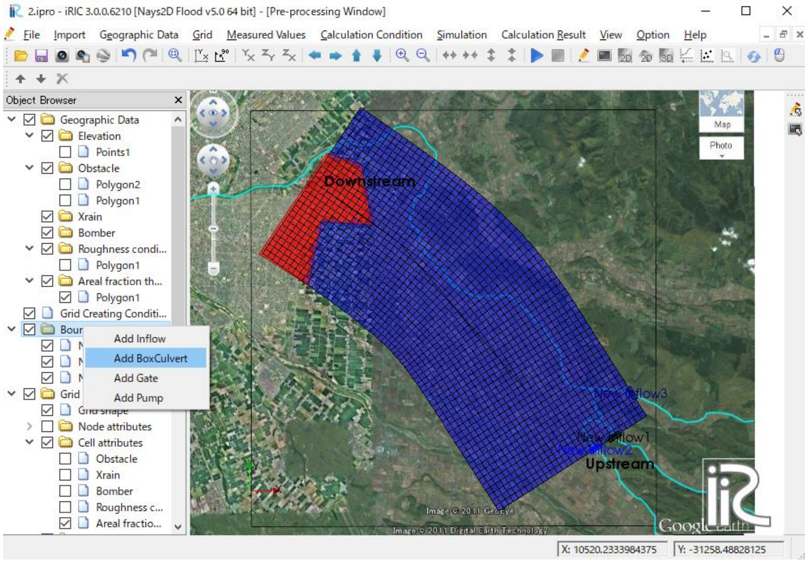
The building occupancy rate is reflected in the cell.



17 Setting for the box culvert

Set “inside” geometry of box culvert.

Click the Object Browser [Boundary Condition Setting]-[Add Box culvert].

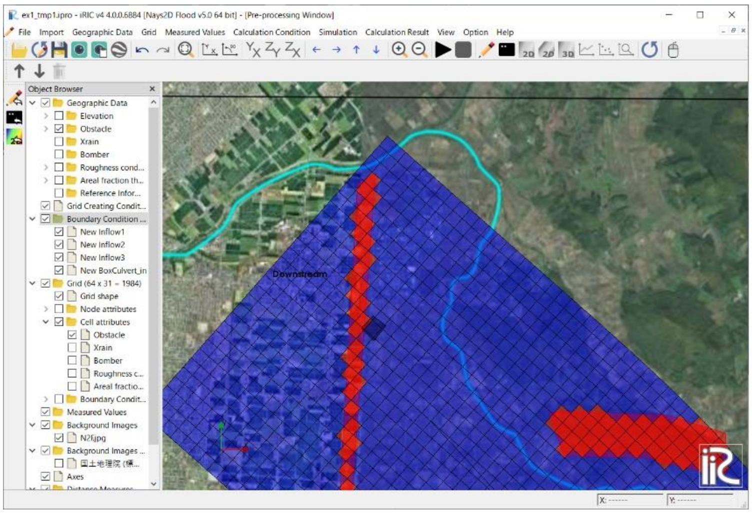


Pass the water through the box culvert to the road set for the obstacle cell.

Note: The setting method of gate, pump is the same as the setting method of box culvert.

Click the Object Browser [Grid] – [Boundary Condition Setting] – [New Box Culvert].

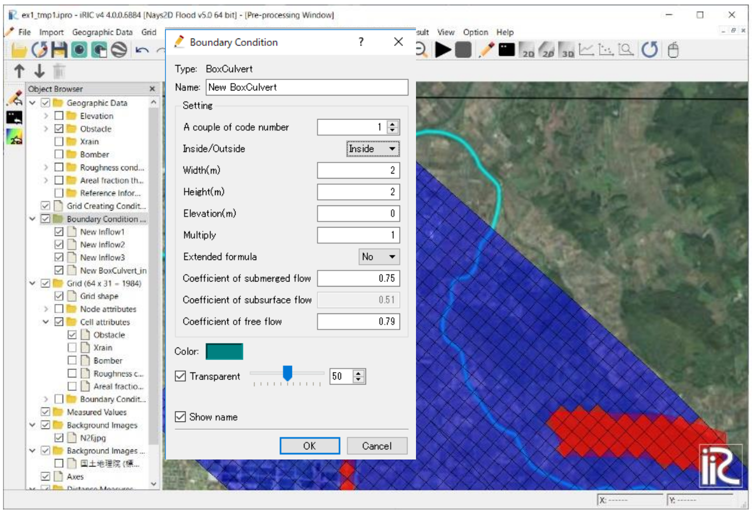
Surround the cell at the "inlet" position of the box culvert with polygons.



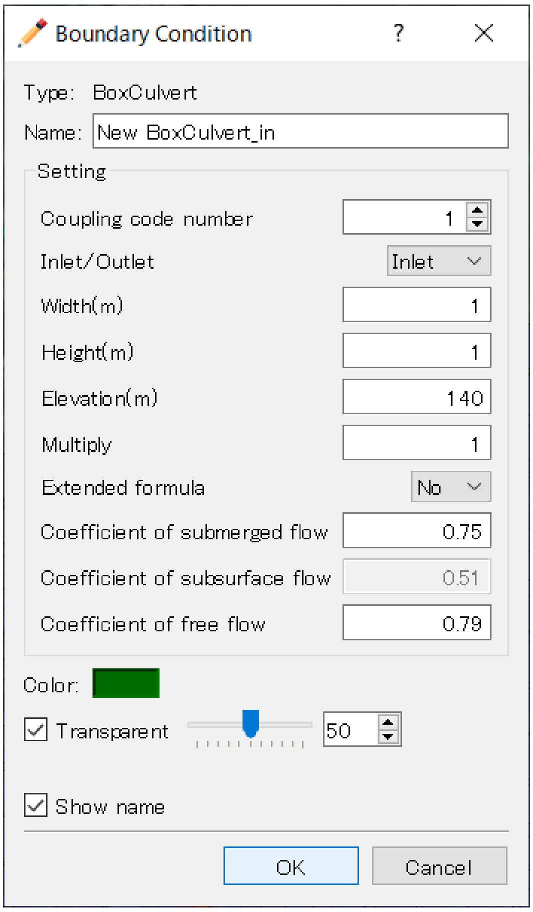
In the figure on the left, the object browser [grid] - [cell attribute] - [obstacle cell] is checked so that the location of the obstacle cell can be seen on the figure.

Double-click to confirm the position of “inside”.

The boundary Condition screen is displayed.



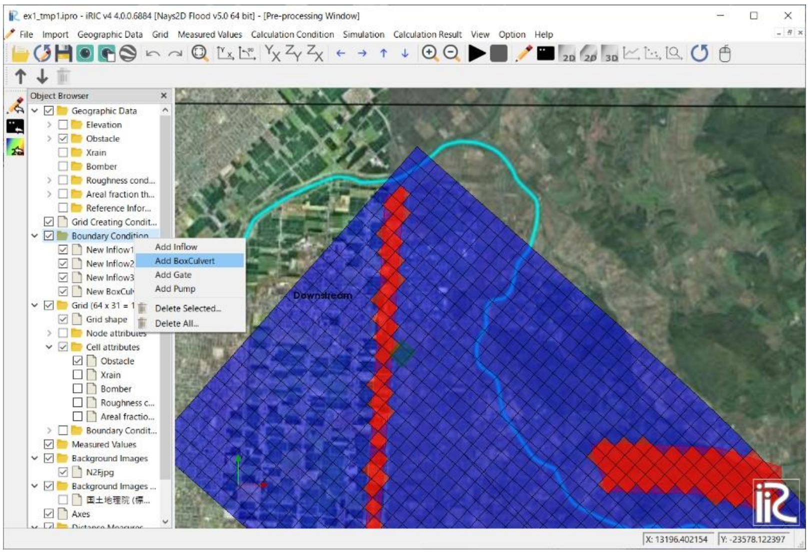
Change the name on [Boundary Condition] screen to [new Box Culvert_in] and set it as an inside.



Name: arbitrary, we use New Box Culvert_in
A couple of code number: 1
Inlet/Outlet: Inlet
Width(m): 1
Height(m): 1
Elevation(m): 140
Multiply: 1
Extended formula : No
Coefficient of submerged flow: 0.75
Coefficient of free flow: 0.79
Note: Combine the code number at inlet/outlet.

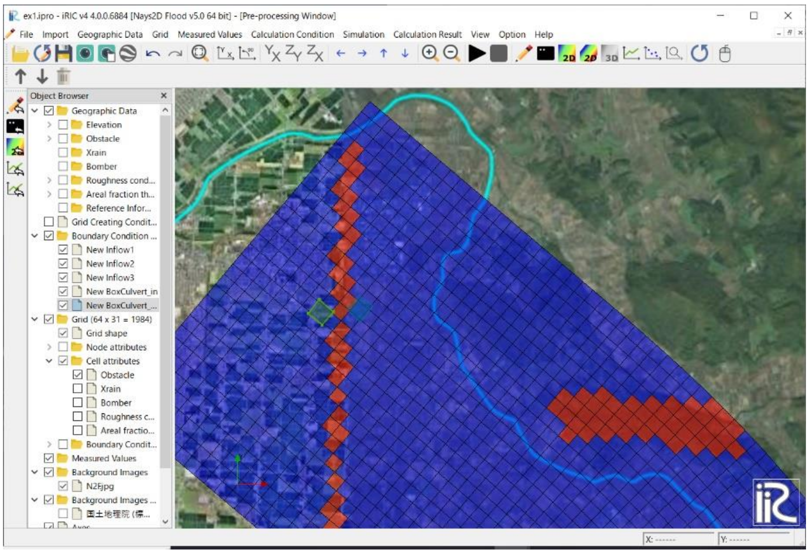
Set “Outside” of the Box culvert.

In the Object Browser, select [Boundary condition setting]-[Add Box Culvert]



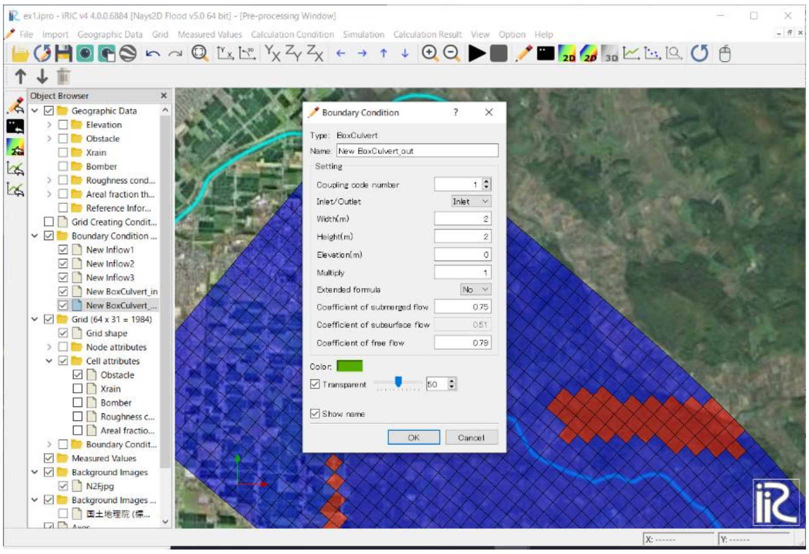
Click the Object Browser [Boundary Condition Setting]-[New Box Culvert].

Enclose the cell at “outside” position of the box culvert with polygons.

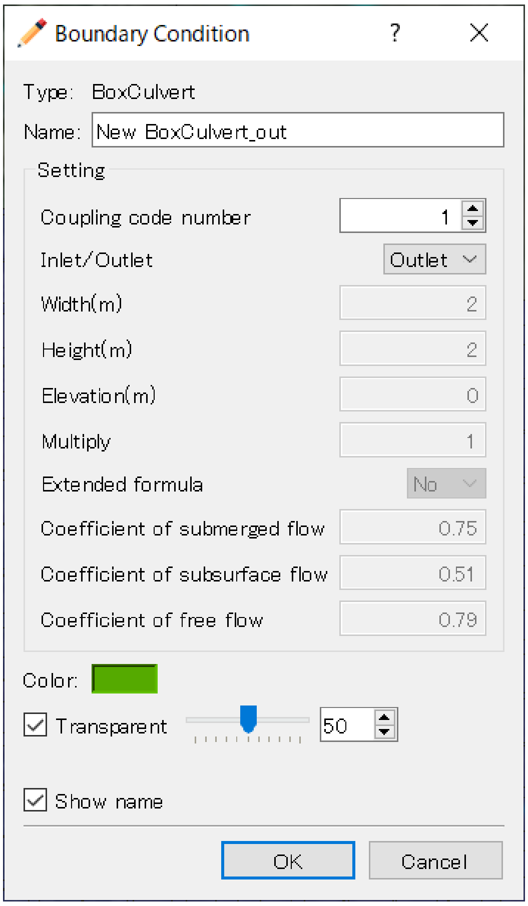


Double-click to set the position of "outside".

The Boundary Condition screen is displayed.

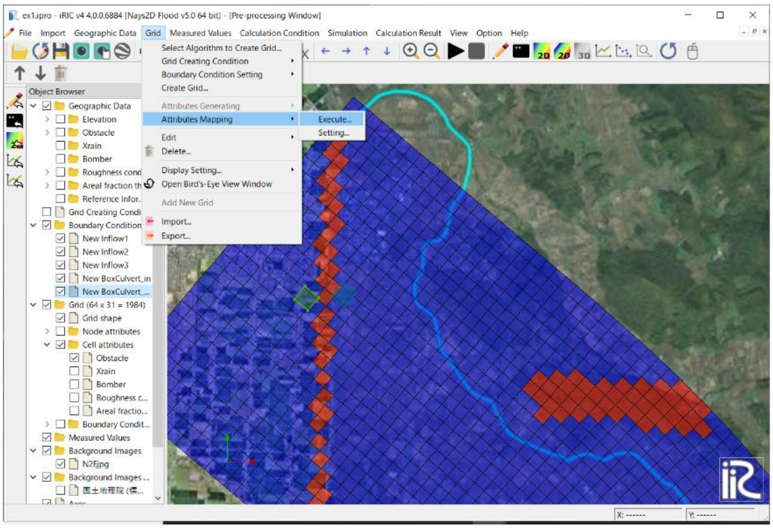


Set the conditions of the box culvert from the dialog.

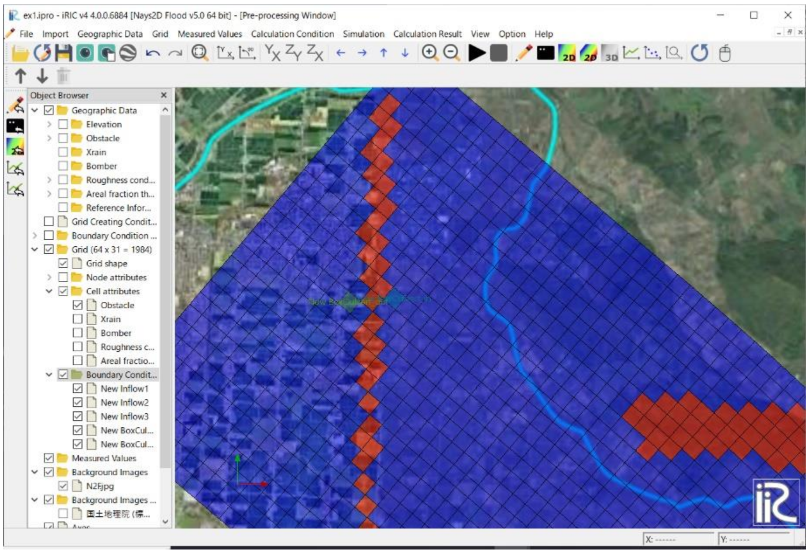


Name: New Box Culvert_out
Coupling code number: 1
Inlet/Outlet : Outlet

On the menu bar, select [Grid] - [Attributes Mapping] – [Execute…]



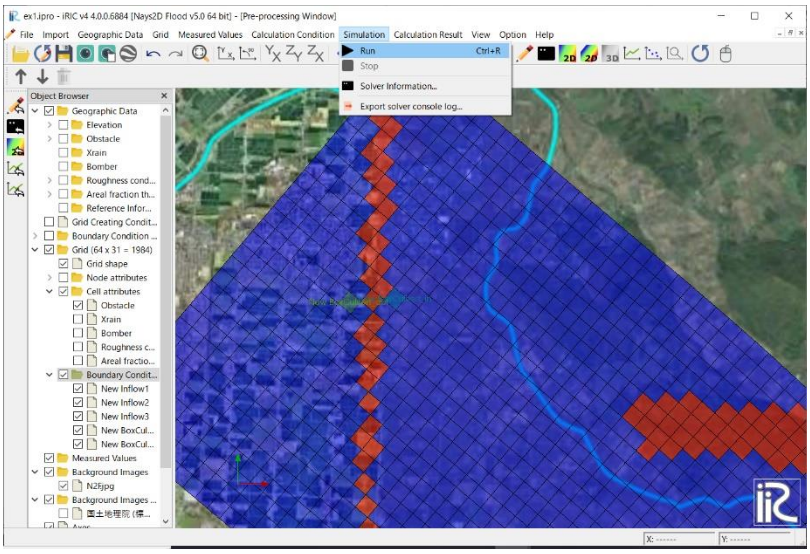
The box culvert is set in the cell.

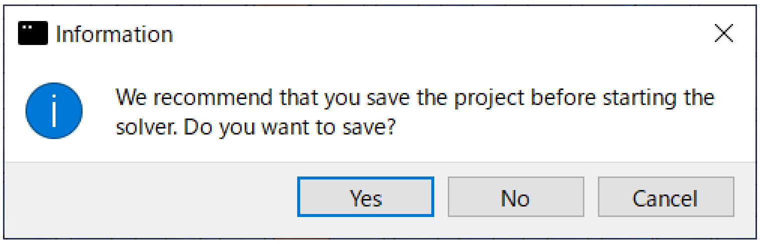


3. Making a simulation

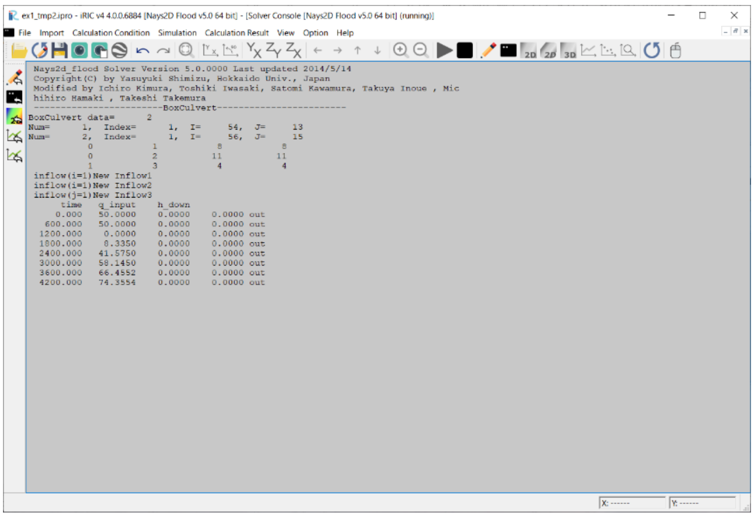
On the menu bar, select [Simulation] - [Run].

Save the project.





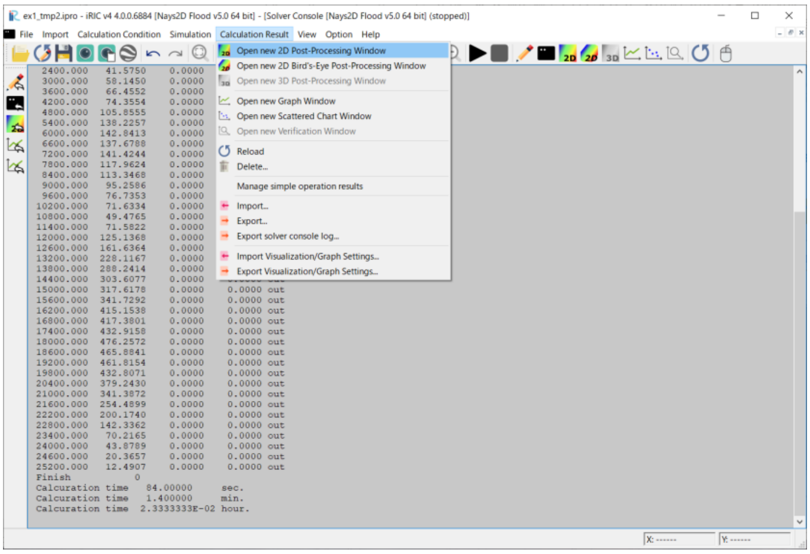
The [Solver Console [Nays2D Flood] (running)] window will open to start the simulation.



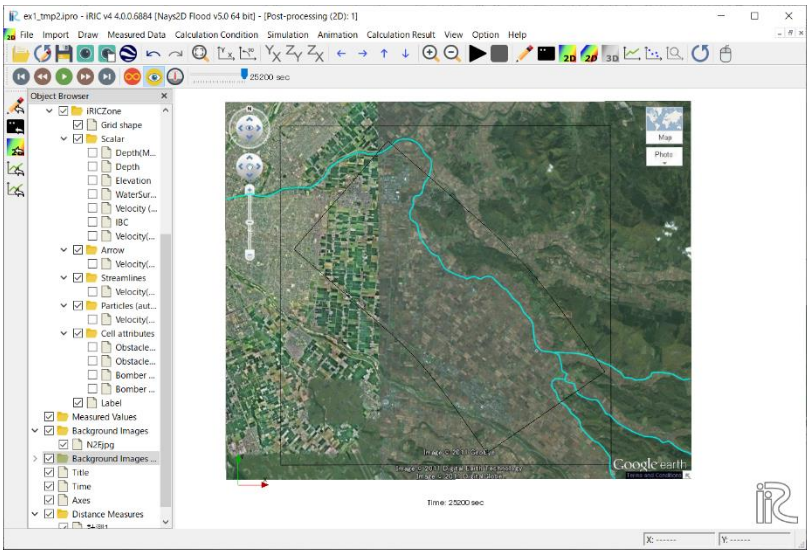
4. Visualizing the calculation results

8 Open the "2D Post-Processing" window

On the menu bar, select [Calculation Result] - [Open new 2D Post-Processing Window].



The "Post-Processing (2D)" window will open.



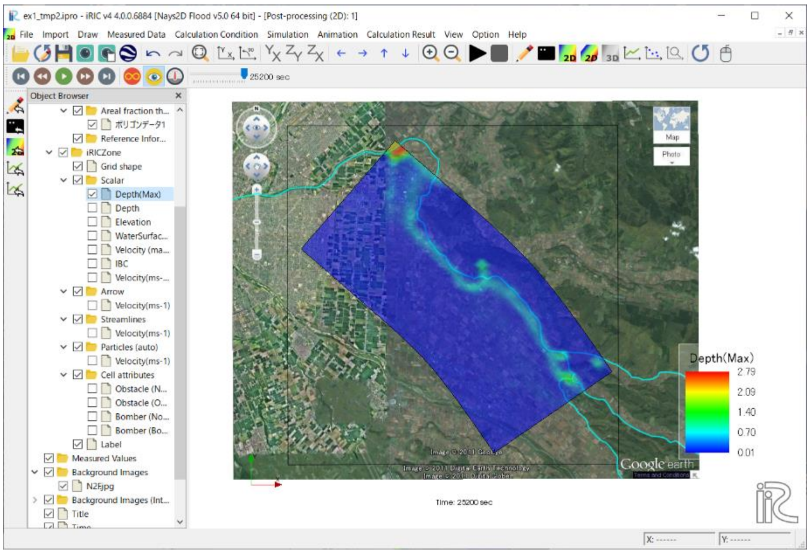
9 Visualizable quantities

DESCRIPTIONS IN THE OBJECT BROWSER	DESCRIPTION OF EACH QUANTITY
● CONTOUR	
DEPTH(MAX)	The max. water depth by the time of visualization (m).
DEPTH	The water depth at the time of visualization (m)
ELEVATION	The ground height of calculation grid (m)
WATERSURFACEELEVATION	The water surface elevation at the time of visualization (m)
VELOCITY (MAGNITUDE MAX)	The max. flow velocity by the time of visualization (m/s)
VELOCITY (MAGNITUDE)	Flow velocity at the time of visualization (m/s)
● VECTOR	
VELOCITY	Vector of flow velocity (m/s) at the time of visualization
● STREAMLINE	
VELOCITY	Displays a streamline.
● PARTICLES	
VELOCITY	Displays particles.
● CELL ATTRIBUTES	
OBSTACLE CELL (A NORMAL CELL)	Displays a normal cell.
OBSTACLE CELL (A CELL WITH AN OBSTACLE)	Displays an obstacle cell.
BOMBER CELL (A NORMAL CELL)	Displays a normal cell.
BOMBER CELL (A CELL WITH ANBOMBER)	Displays an bomber cell.

10 Visualizing the max. water depth

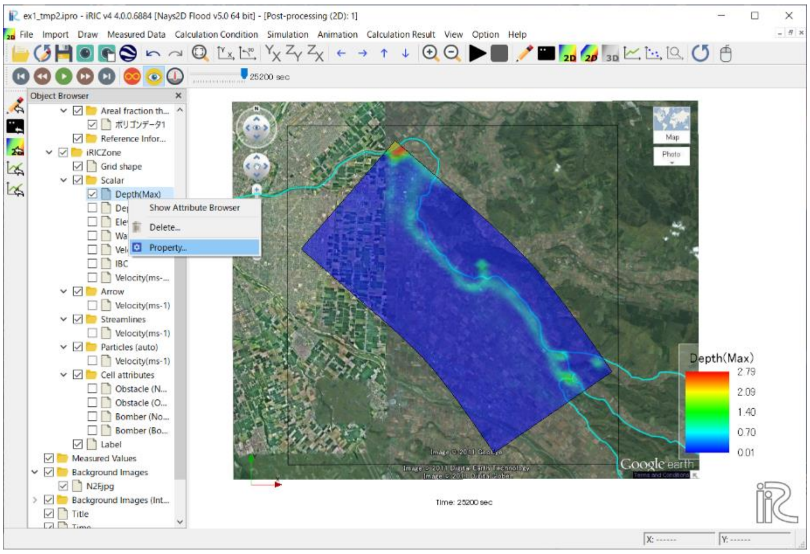
In the Object Browser, select [Nays2D Flood Grids] - [iRICZone] – [Scalar] - [Depth (Max)].

A contour map of water depth will open.

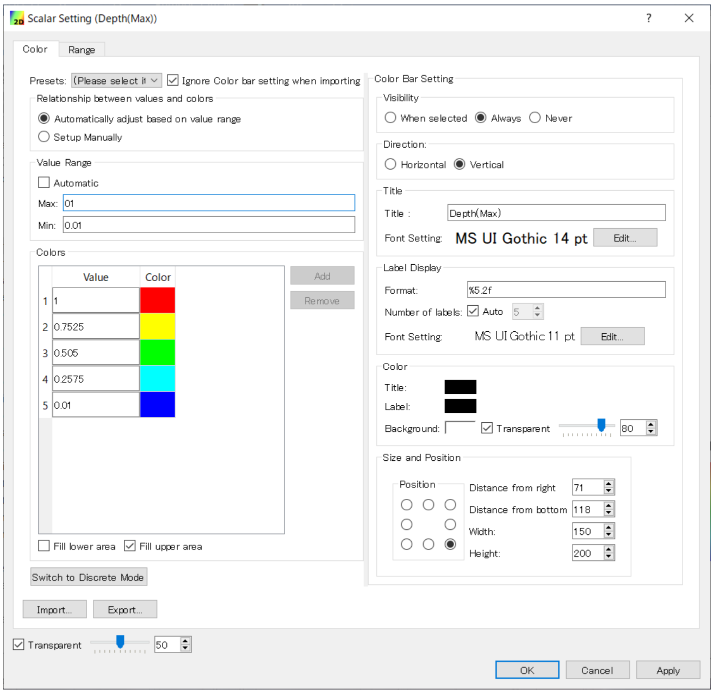


In the Object Browser, select [Nays2D Flood Grids] - [iRIC Zone] - [Scalar]. Right-click on [Depth(max)] to select [Property].

The [Contour Setting] window will open.

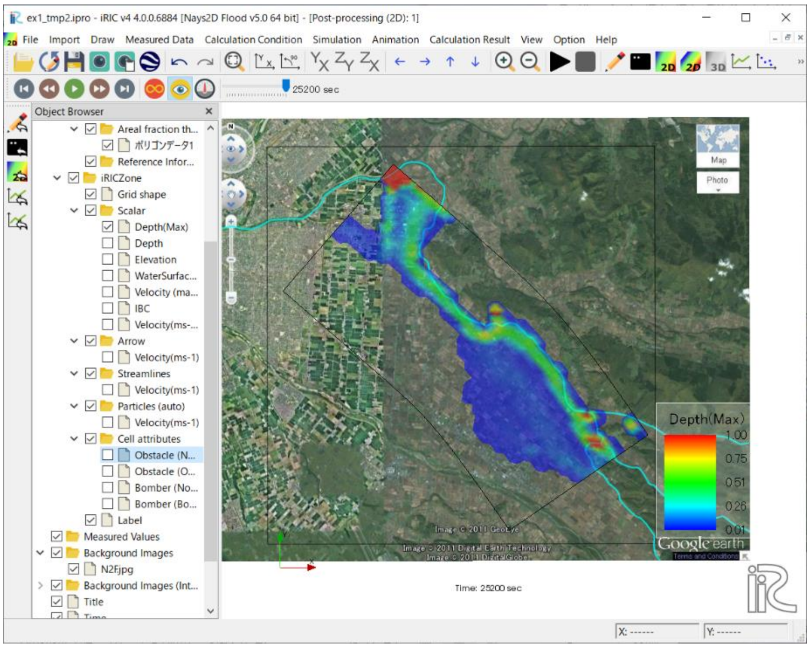


In the [Contour Setting] window, make the following settings and click on [OK].



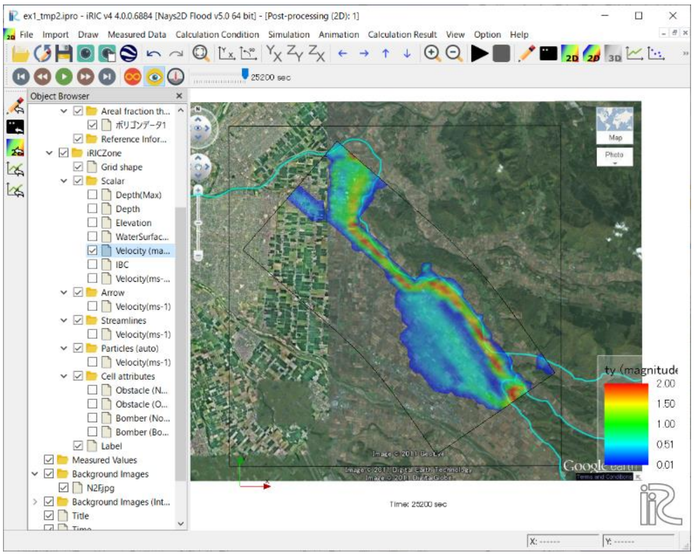
Value range：
Remove ☑ from Automatic
Max：1
Min：0.01
Colors：No change
Remove ☑ from Fill lower area
Transparent: No change
Color Bar Setting: No change

The contour map is clear.



11 Visualizing the max. flow velocity

In the Object Browser, check [Nays2D Flood Grids] - [iRIC Zone] - [Scalar] - [Velocity (magnitude Max)].

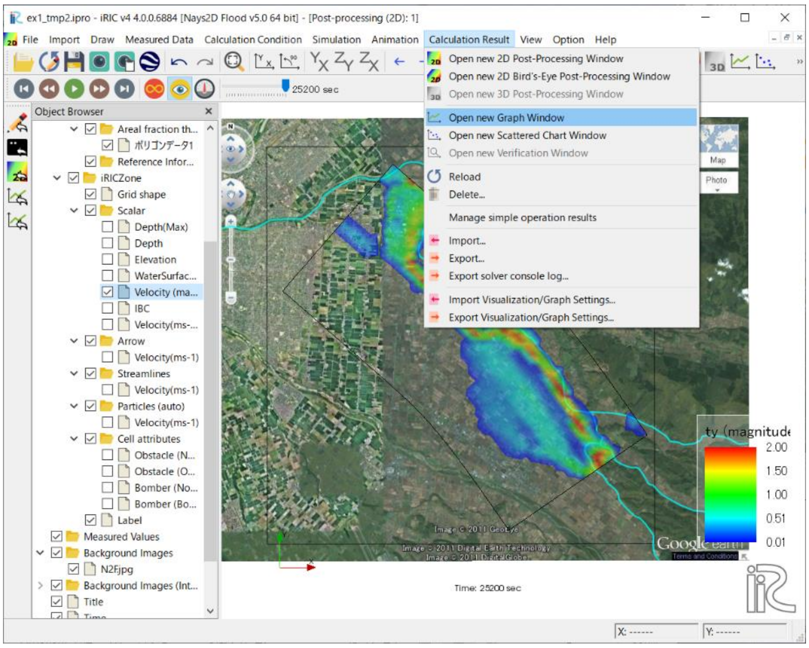


Value range：
Remove ☑ from Automatic
Max：2
Min：0.01
Colors：No change
Remove ☑ from Fill lower area
Transparent: No change

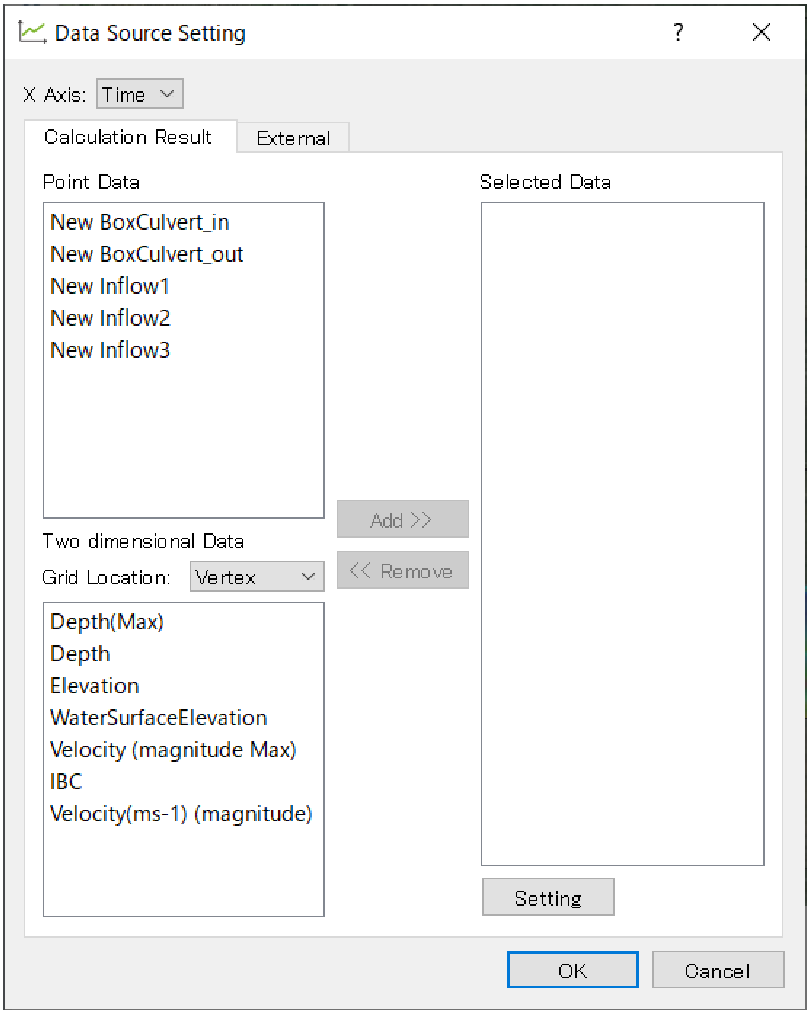
Color Bar Setting: No change

12 Visualizing the inflow discharge hydrograph

Open [New Graph Window].



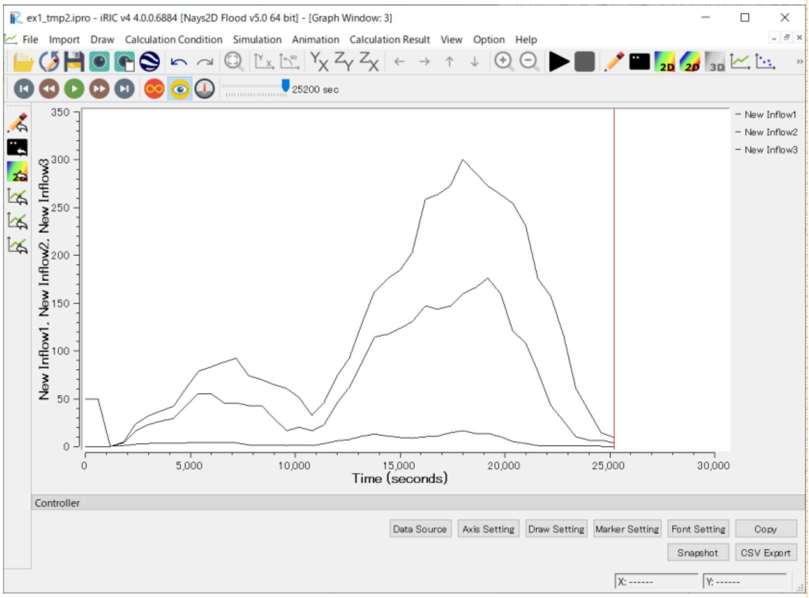
[Data Source Setting] screen is displayed.



Unit: m^3/s

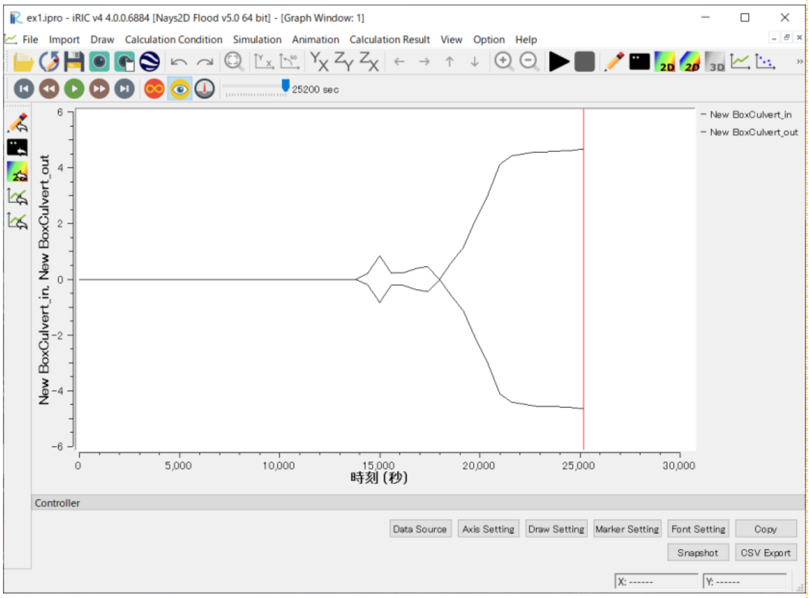
Select [Calculation Result] - [Point data] - [New Inflow1], [New Inflow2] and [New Inflow3] - [Add].

The hydrograph of the incoming river is displayed.



Select [Calculation Result] - [Point data] –[New Box Culvert_in], [New Box Culvert_out] - [Add].

The hydrograph of box culvert is displayed.



Chapter 4
Examples of Rainfall Induced Flooding in Actual River

◆ Objectives

The objectives are to use Shuttle Radar Topography Mission (SRTM) data, to simulate and visualize the simulated flow regime, such as water surface elevation and flow velocity, at a river when rainfall induced flooding occurs with Nays2D Flood.

◆ Outline

1. Creating the calculation grid

Using elevation data of an actual river basin, create calculation grids: 101 division points in the transverse direction, and 131 division points in the longitudinal direction.

2. Setting the calculation conditions

Set a value for Elevation, Rainfall and Inflow Discharge. Set various other conditions necessary for simulation.

3. Making a simulation

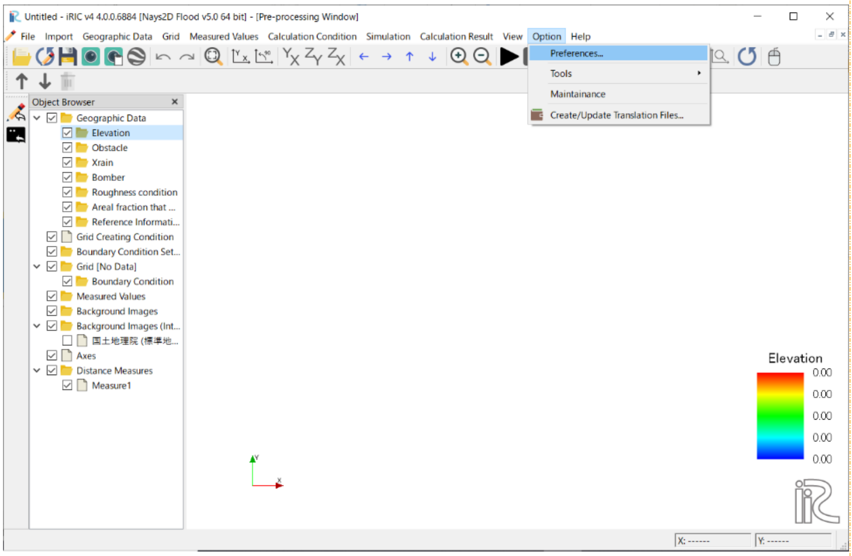
4. Visualizing the calculation results

Here, we introduce how to display a water depth contour map and a flow velocity vector map.

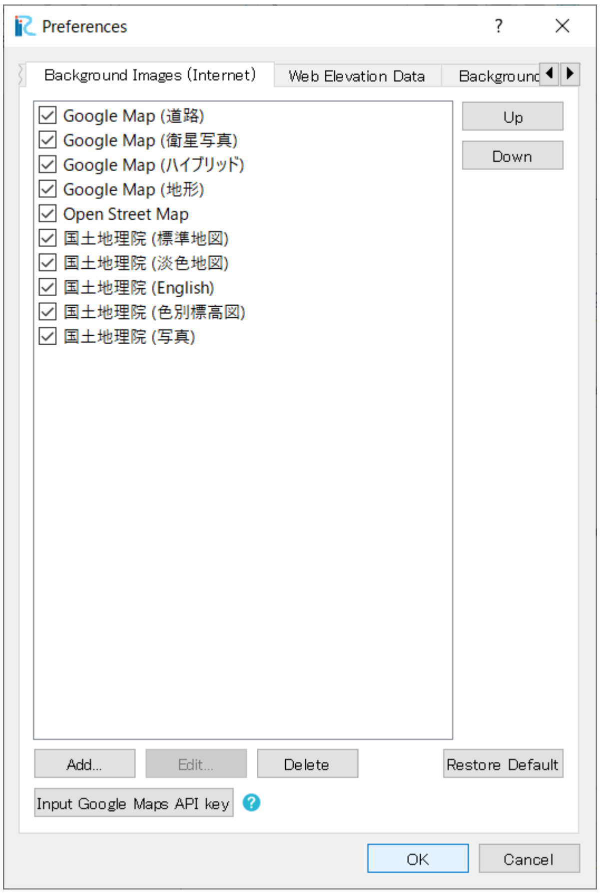
1. Creating the calculation grid

1 Importing geographic data

In the Object Browser, click [Option]-[Preferences]



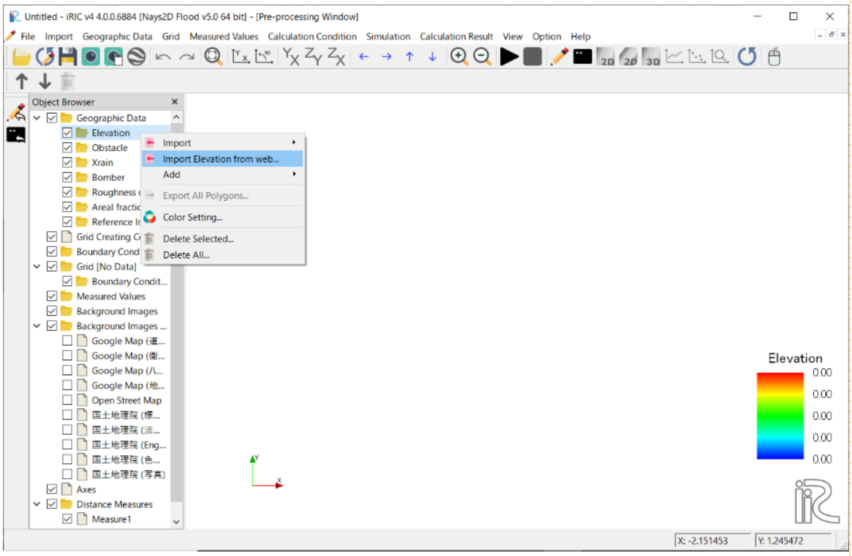
In [Preferences] scroll to [Background Images (Internet)]



Google Map can also be used instead of Ortho-images but it requires registration with Google.

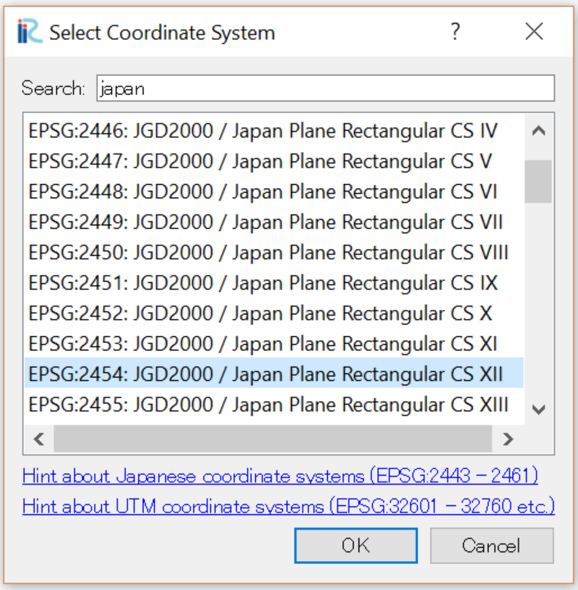
Select [GSI (English) (Japan only)] and [GSI (Ortho images) (Japan Only)]

In the Object Browser, right-click [Geographic Data]-[Elevation].Click [Import elevation from web].



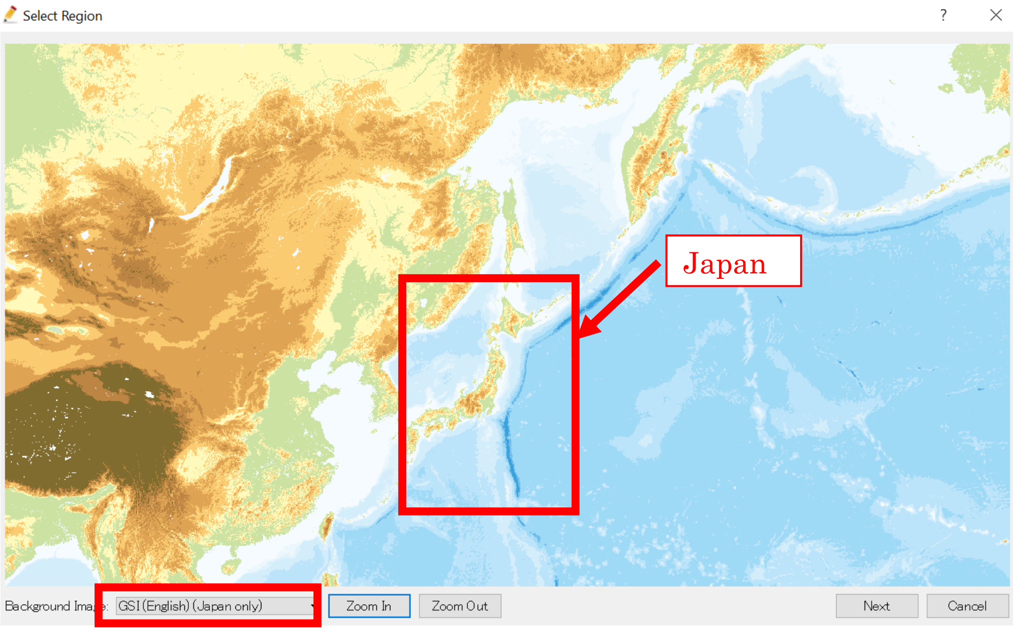
[Select Region] screen is displayed. Drag the target part and click [Next].

[Select Coordinate System] screen is displayed.
Search with JGD and select [EPSG:2454:JGD2000/Japan Plane Rectangular CS XII].

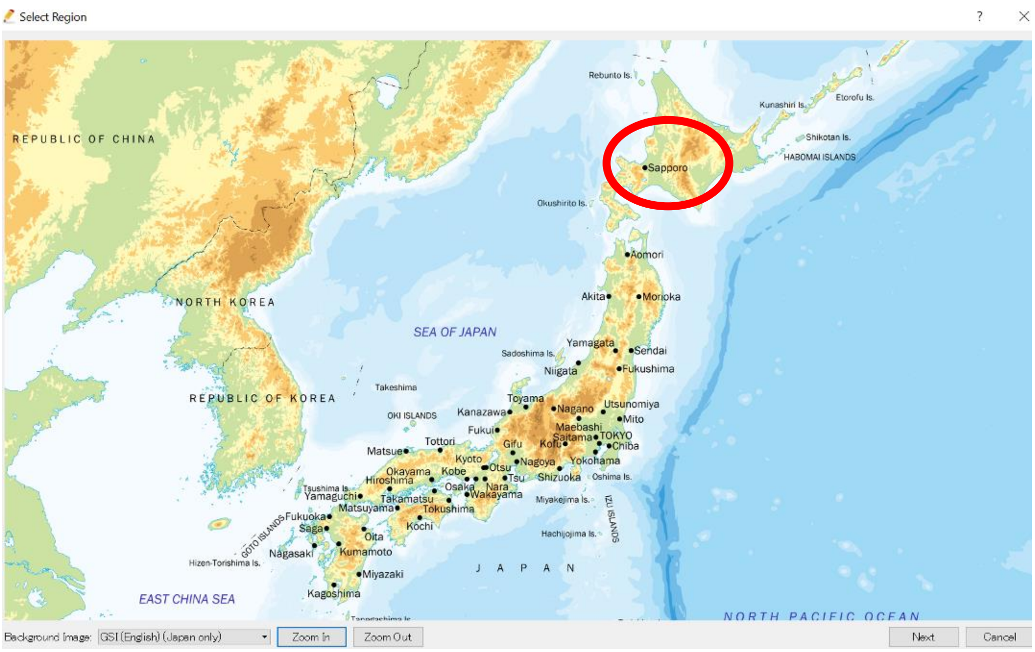


Select [GSI (English)(Japan Only)] in Background Images

[Zoom In] towards ‘Japan’

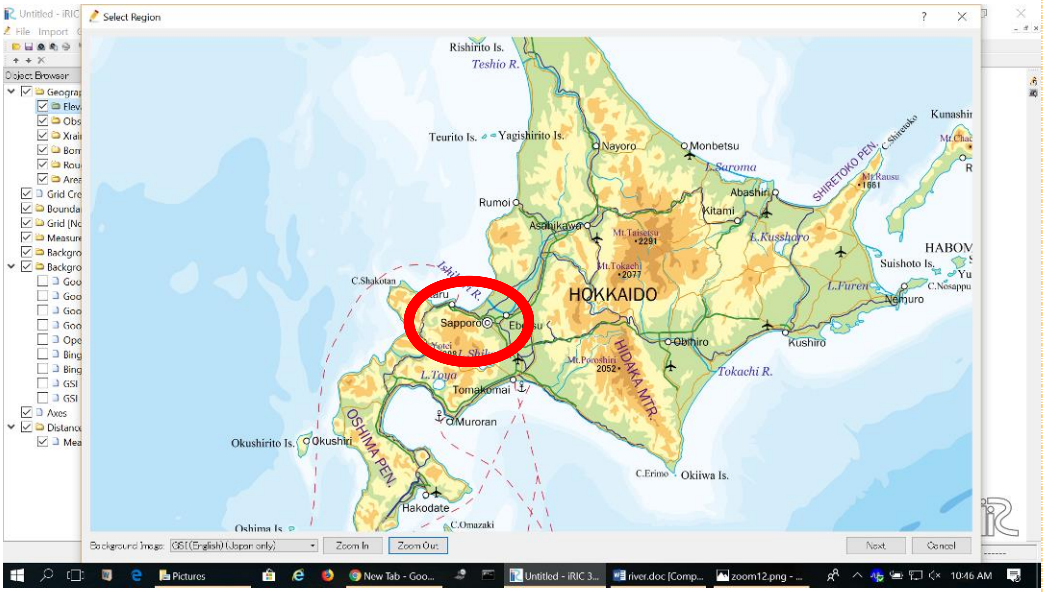


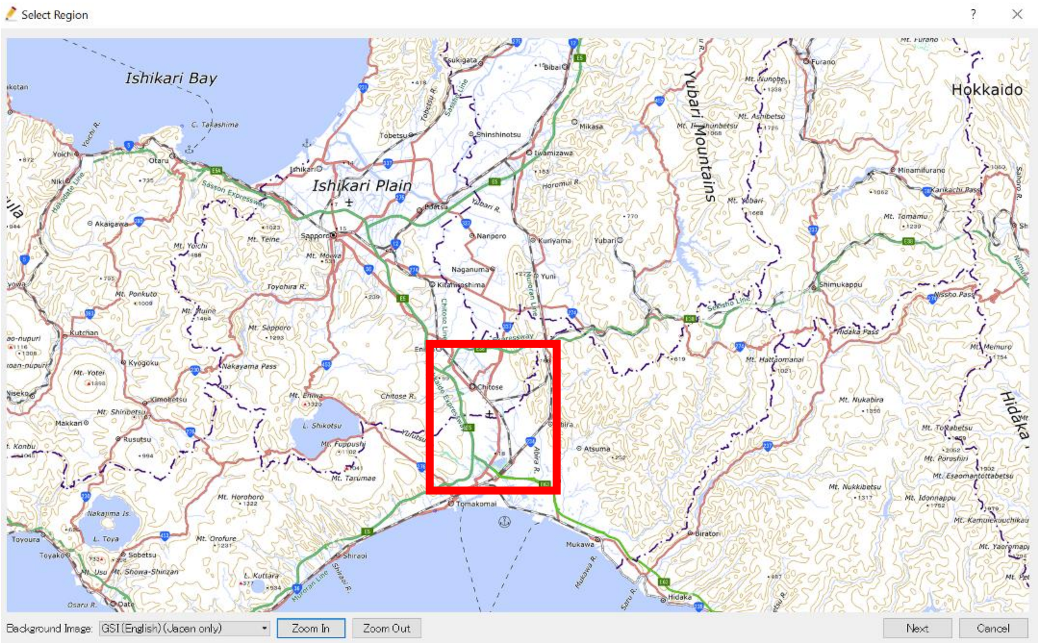
[Zoom In] towards ‘Japan’ so that you can spot ‘Sapporo’



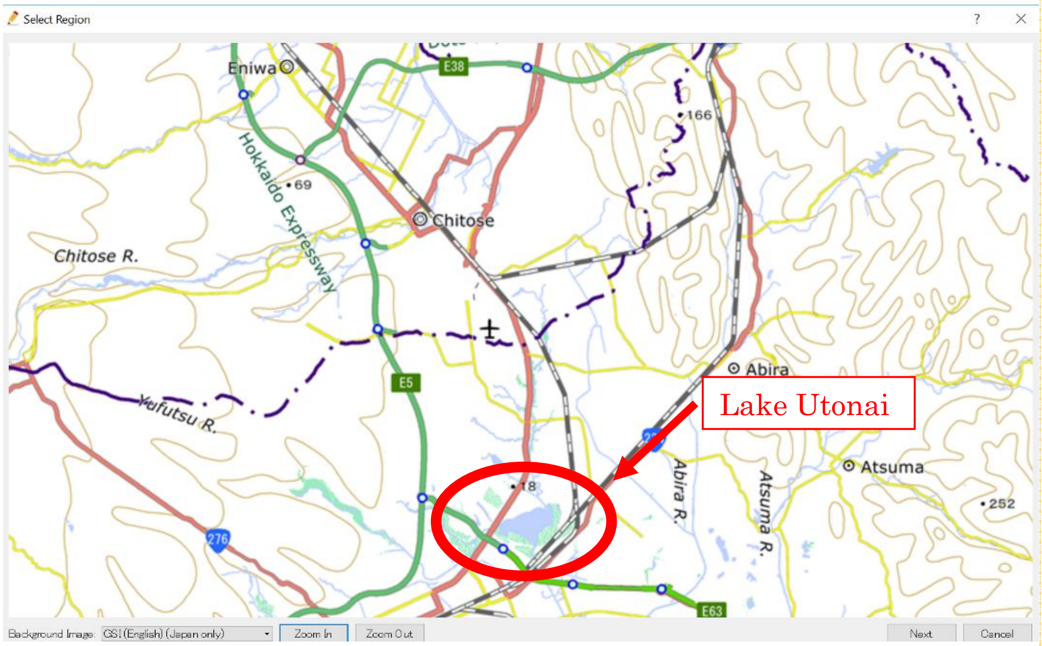
[Zoom In] towards 'Sapporo'

Try to Spot 'Chitose' and [Zoom In] towards 'Chitose'

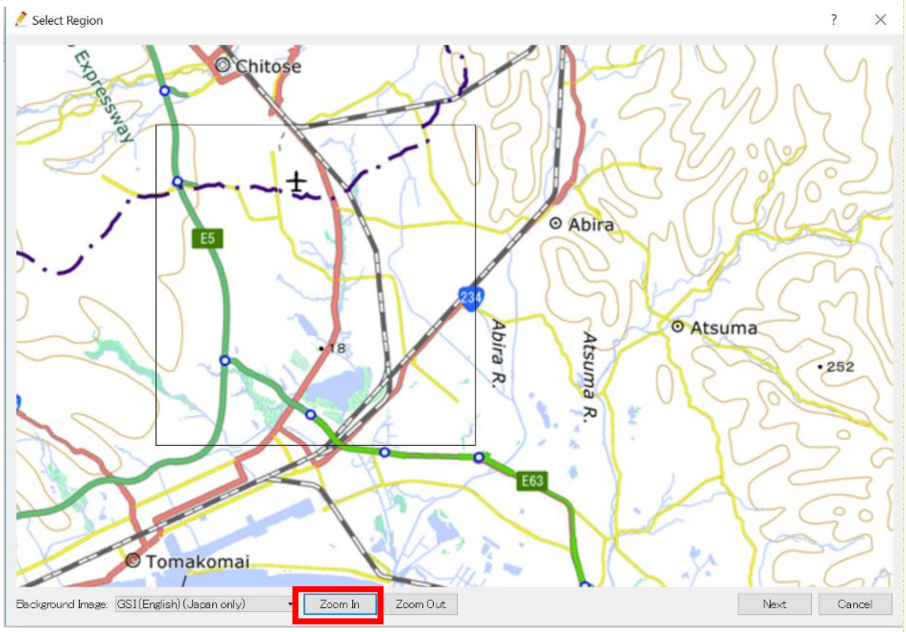




[Zoom In] towards ‘Chitose’ so that you can spot ‘Lake Utonai’



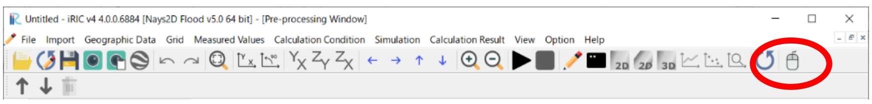
Drag the target part and click [Next].



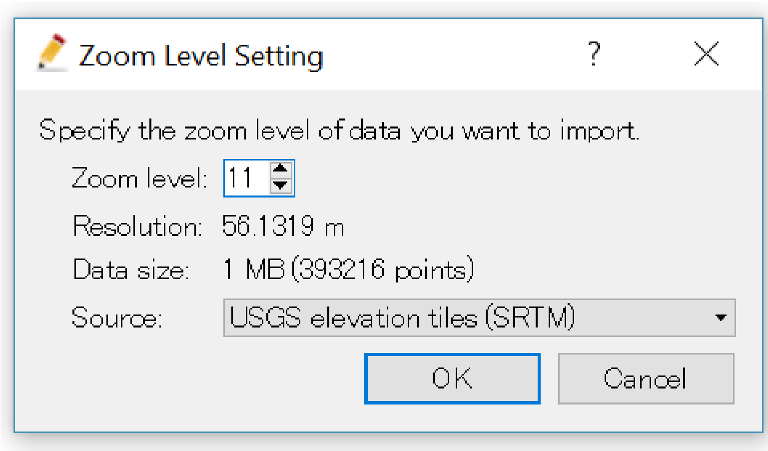
To enlarge or reduce the map, click [Zoom In] / [Zoom Out] at the bottom of the screen.

Mouse Hint:

Ctrl+ Right Click = Pan

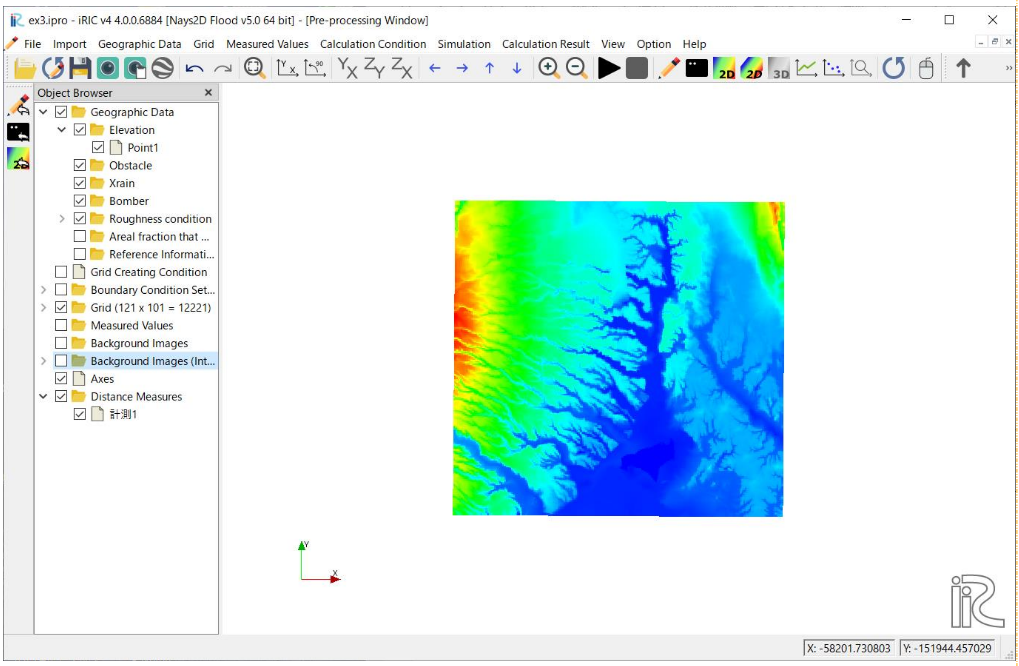


[Zoom Level Setting] screen will be displayed.
Specify the zoom level and select the source

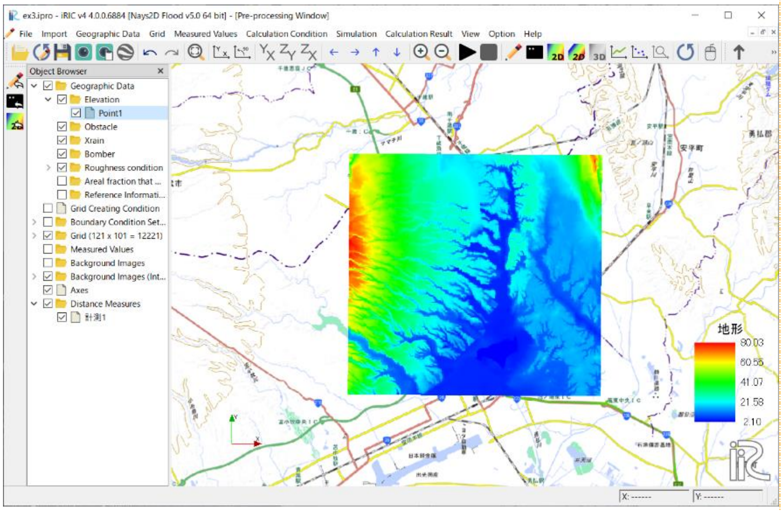


Zoom level : 11
Source : USGS (SRTM)
Note: Since the resolution of the SRTM data is 90 m mesh, even if the zoom level is set to 11 or more here, the data does not change because the maximum resolution has been reached. Therefore, the zoom level is set to 11.

Reading is successful if the shape of the watershed to be calculated is displayed on [Pre-processing Window] screen.



When one of the items in the background image of the object browser is checked, the background image is displayed.



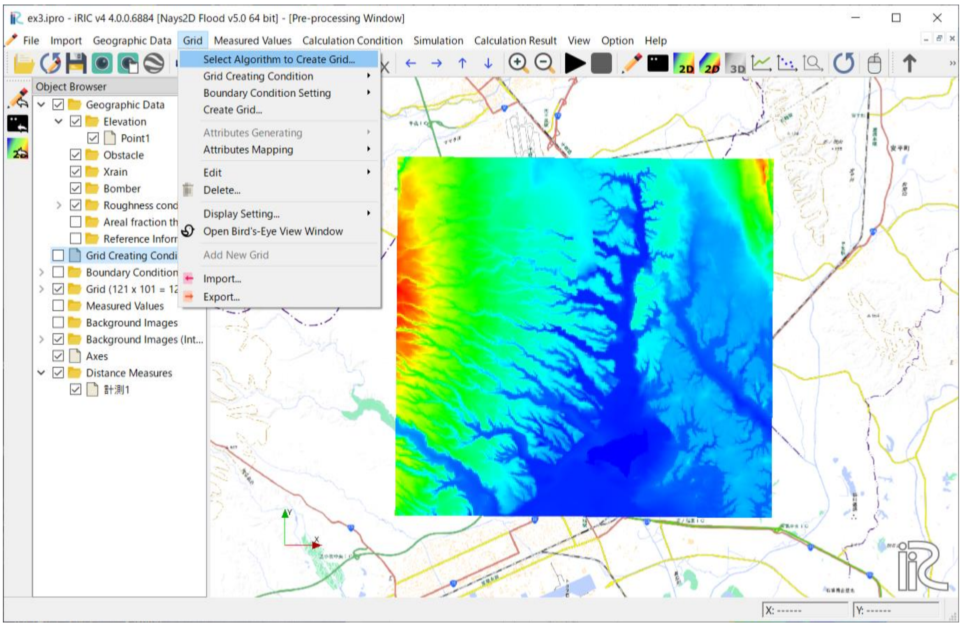
When creating grids for calculation, importing background images such as maps and aerial photos makes it possible to create grids that incorporate riverbanks and land use. Obstacle cells and roughness cells mentioned below can be set in reference to the background image.

Since the coordinate system is selected, it is set automatically

2 Selecting an algorithm for creating the grid

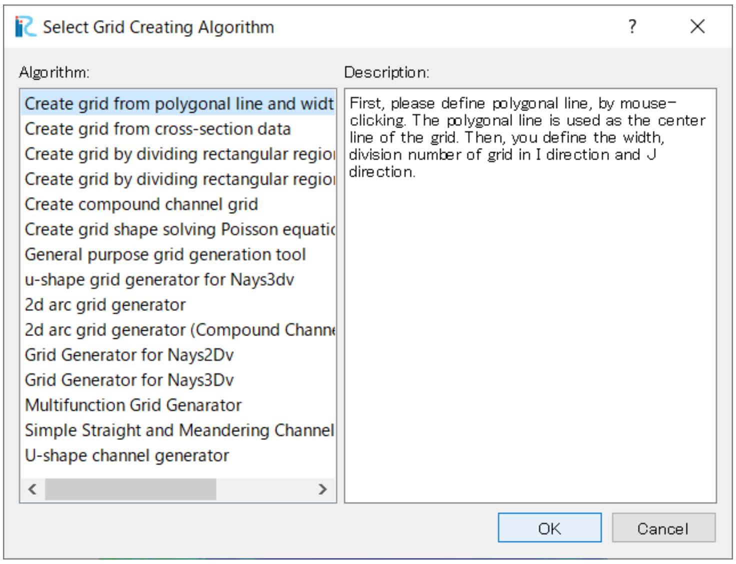
On the menu bar, select [Grid] - [Select algorithm to create grid].

The [Select Grid Creating Algorithm] window will be displayed.



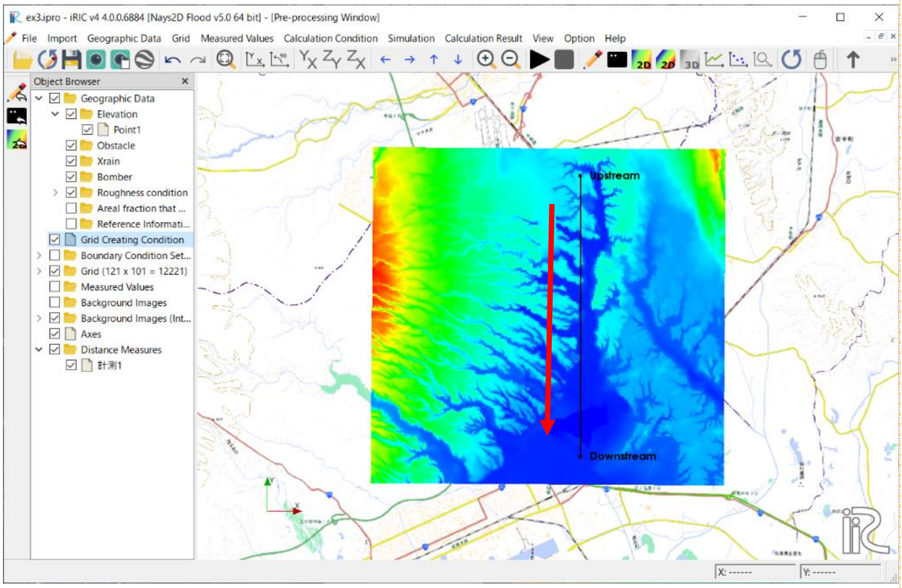
Select [Create grid from polygonal line and width] from the list under the [Select Grid Creating Algorithm] window, and click on [OK].

Nays2D Flood generates a grid from polygonal lines and widths.



3 Creating a grid

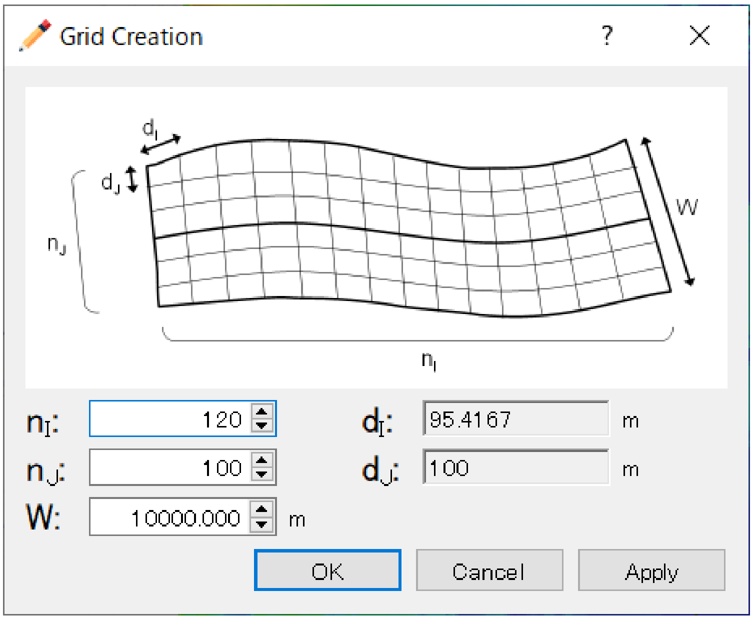
Click on several points through which the grid centerline passes and press the Enter key on your keyboard to draw a centerline.



Setting the grid centerline

Set the grid centerline from the upper reaches of the river (upstream) to the sea (downstream). To finish, press the Enter key or double click.

In the [Grid Creation] window, make the following settings and click on [OK].



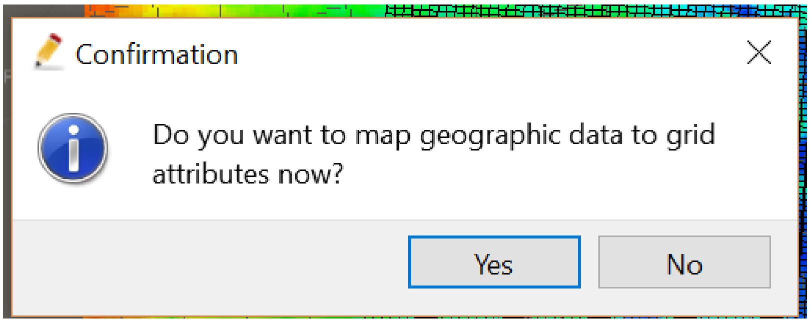
[nj] (Number of divisions in the longitudinal direction): 120

[ni] (Number of divisions in the transverse direction): 100

[W] (Grid width): 10000 m

Note: In this example, grid cell intervals di, dj are set at around 70 and 100 m respectively.

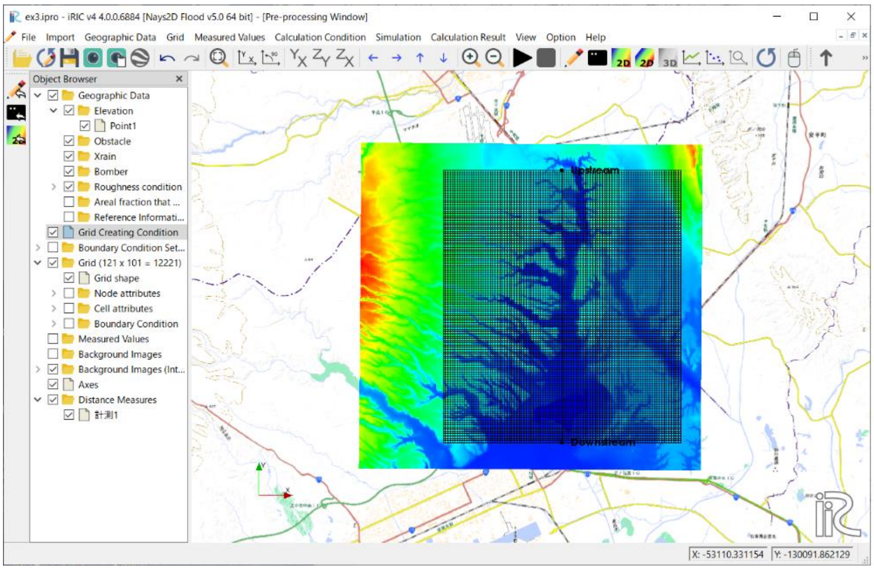
Click on [Yes] in the [Confirmation] window.



Mapping geographic data

Elevation data are applied to the grid.

A grid will be created.



Adjusting the calculation grid

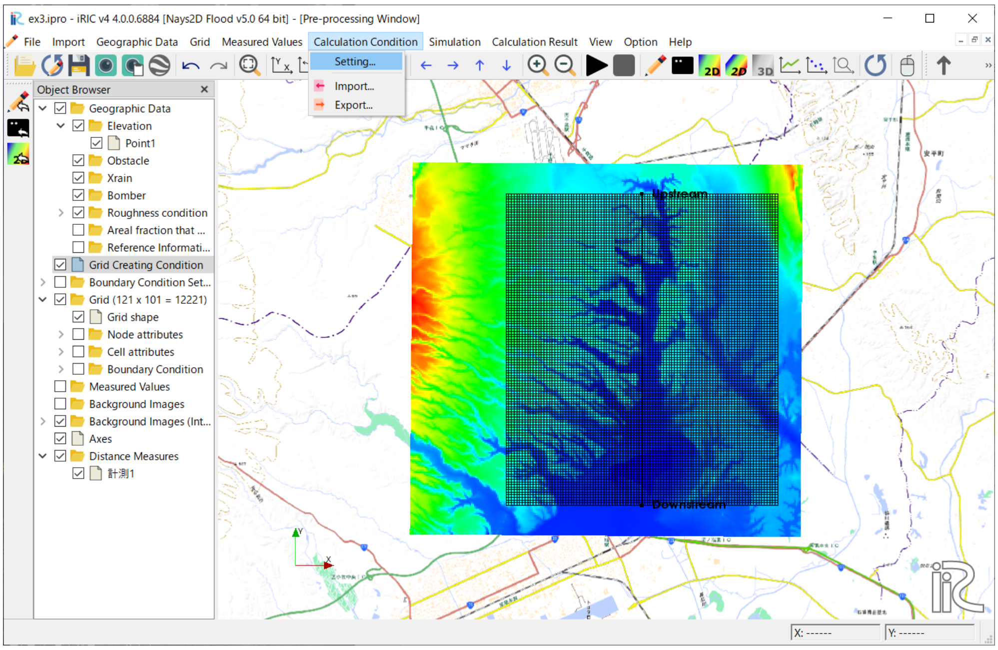
It is possible to move, add or remove any vertexes by selecting the [Grid Creating Condition] even after the grid is created.

2. Setting the calculation conditions

1 Open [Calculation Condition]

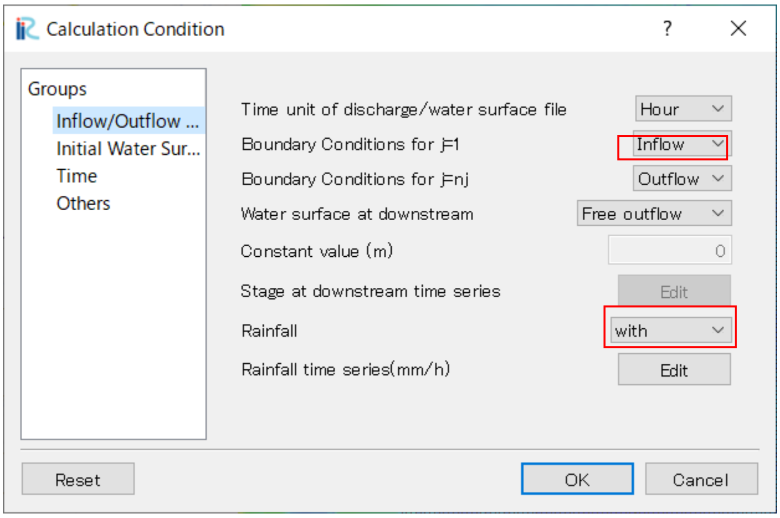
On the menu bar, select [Calculation Condition] - [Setting].

The [Calculation Condition] window will open.



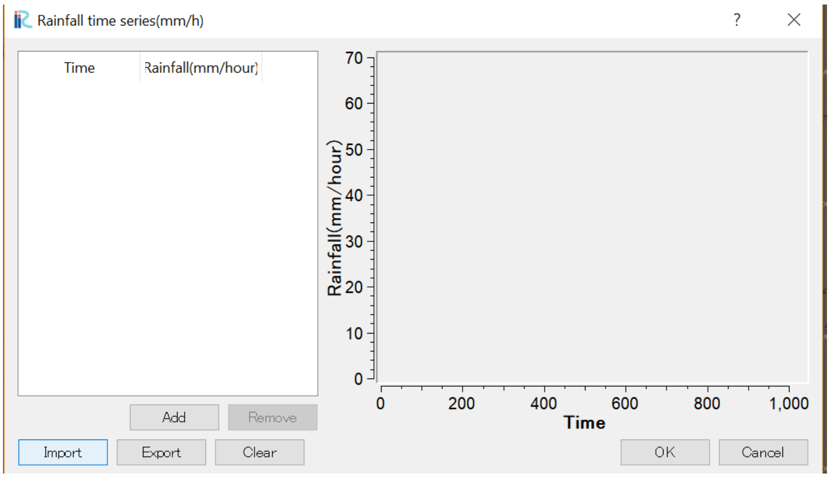
2 Setting the inflow boundary conditions

Click on [Inflow Boundary Condition] from the [Group] list to make the following settings:
Click on [Stage at downstream time series] to edit.
The [Calculation Condition] window will open.



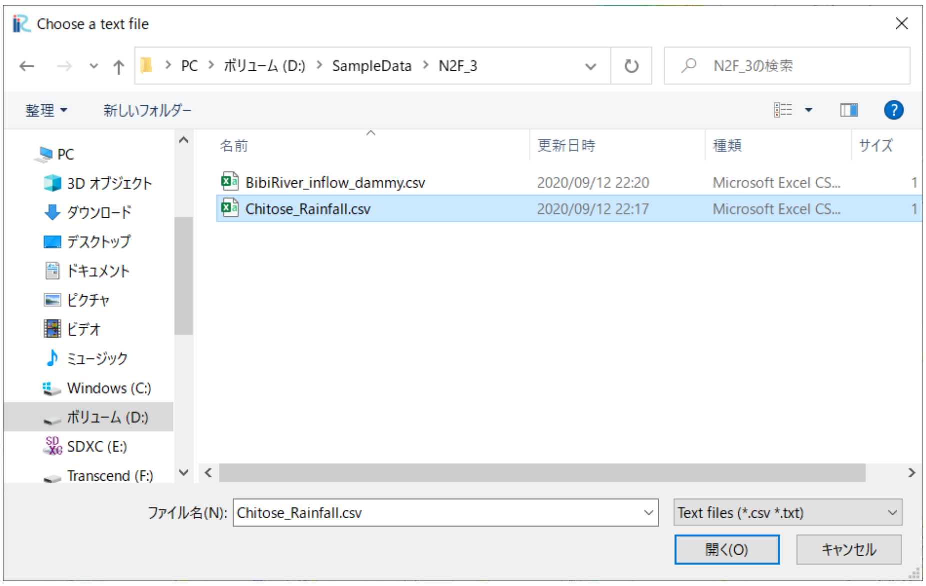
[Time unit of discharge/water surface file]: Hour
[Boundary Conditions for j=1]: Inflow
[Boundary Conditions for j=nj]: Outflow
[Water surface at downstream]: Free Outflow
Rainfall: with
Rainfall time Series: Import Chitose_Rainfall.csv from test file

Click on [Import] on the [Calculation condition] window.
The [Select text file] window will be displayed.



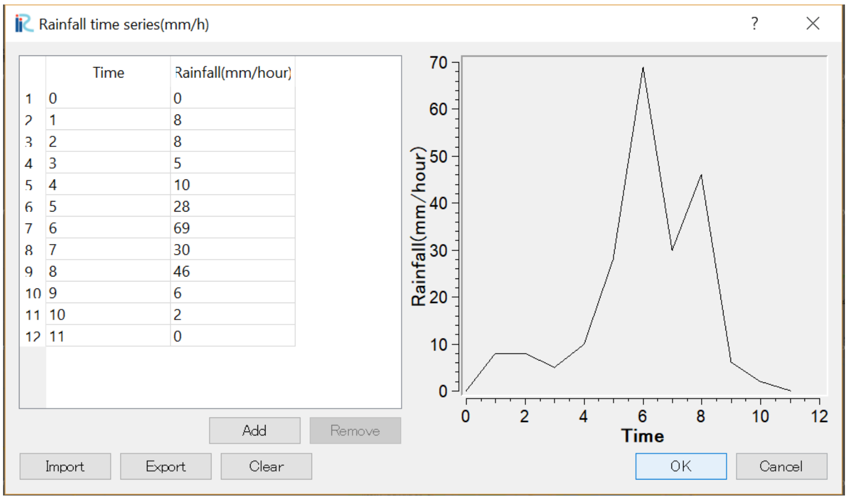
Select [¥¥SampleData¥¥N2F_3] - [Chitose_Rainfall.csv] and click on [Open].

Time series of rainfall data is displayed.



Click on [OK].

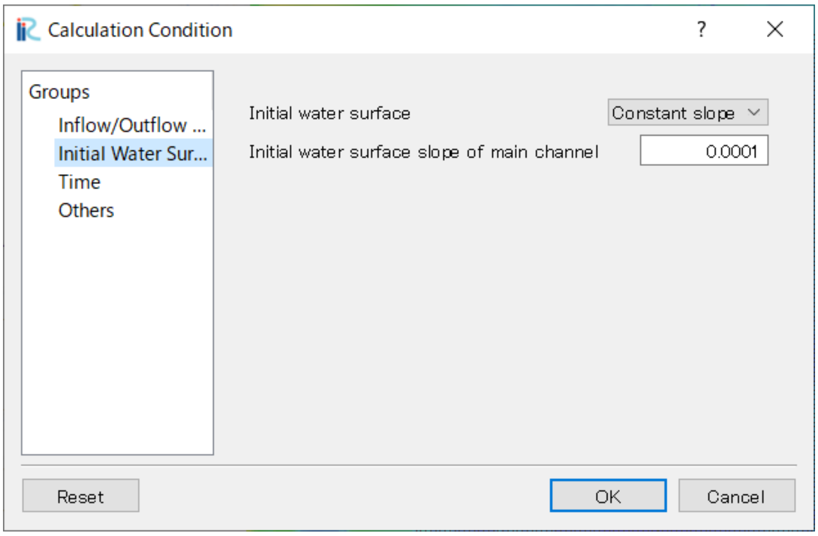
Here, we assume a tsunami whose wave height is 7 m and half-wavelength is 600 sec.



The time steps of the tsunami wave height must be kept at the same value, including at the inflow location.

3 Setting the initial water surface profile

Click on [Initial Water Surface] from the [Group] list to make the following settings:

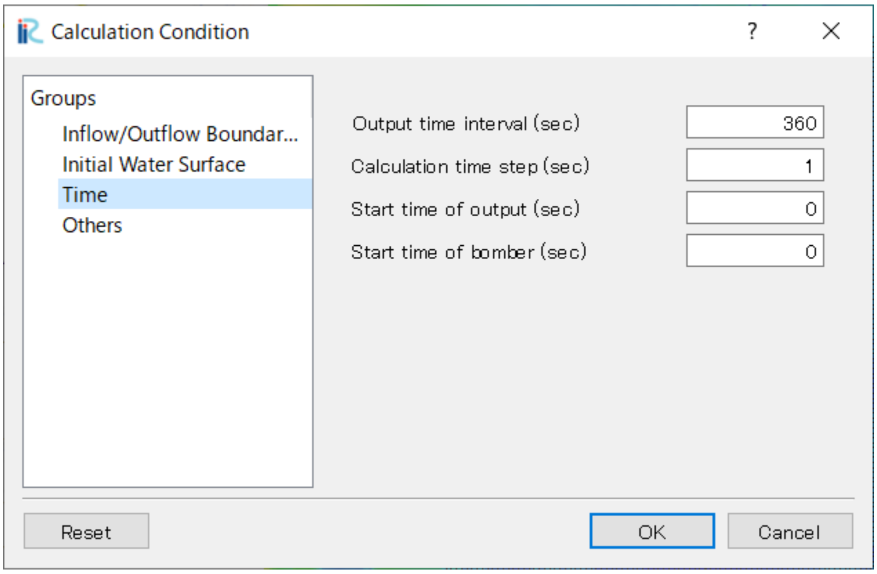


Initial water surface: constant slope

Initial water surface slope of main channel: 0.0001

4 Setting the time

Click on [Time] from the [Group] list to make the following settings:



Output time interval (sec): 360

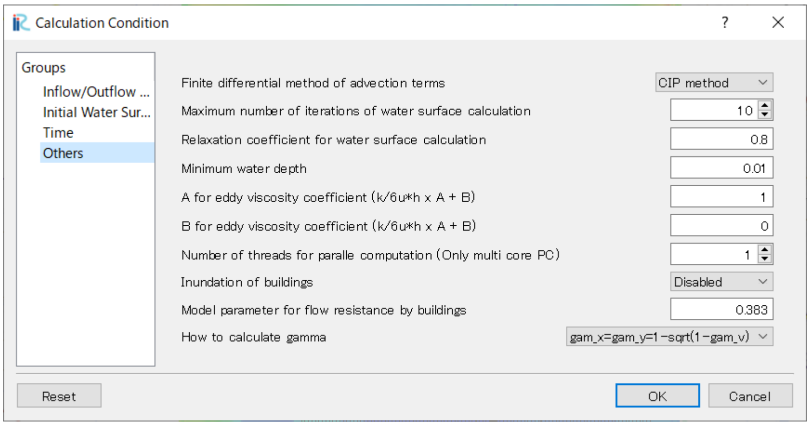
Calculation time step (sec): 1

Start time of output (sec): 0

Start time of bomber (sec): 0

5 Other settings

Click on [Others] from the [Group] list to make the following settings:



Finite differential method of advection terms: CIP method

Maximum number of iterations of water surface calculation: 10

Relaxation coefficient for water surface calculation: 0.8

Minimum water depth: 0.01

A of eddy viscosity coefficient: 1

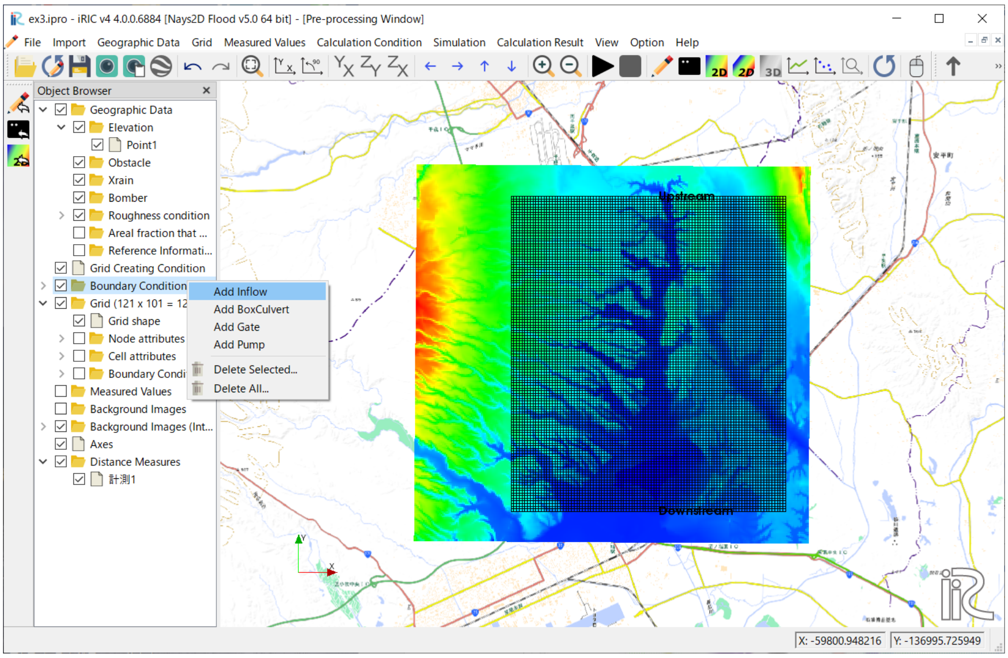
B of eddy viscosity coefficient: 0

After making the settings above, click on [Save and Close] to close the window.

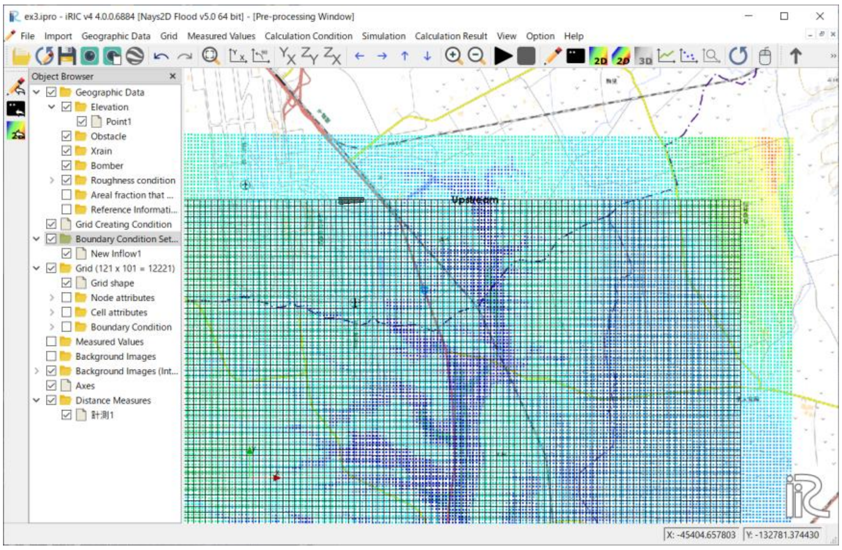
6 Inflow settings

Nays2D Flood requires at least one inflow location to be set on the upstream end.

In the Object Browser, right-click [Boundary Condition Setting] and click [Add Inflow].



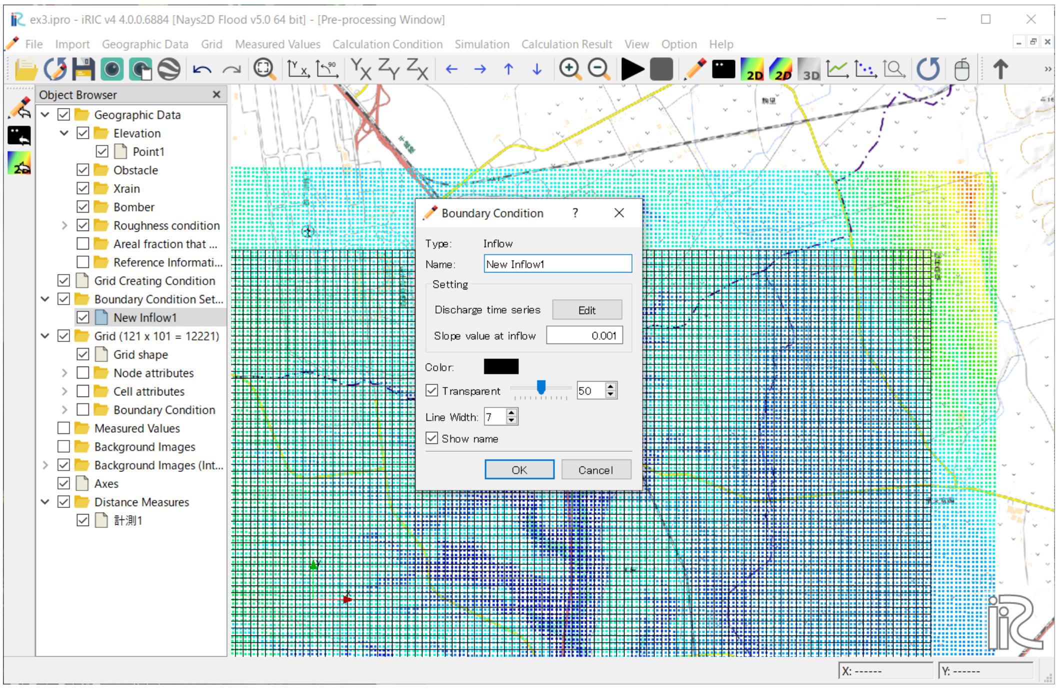
In the Object Browser, click [Boundary Condition Setting]-[Add Inflow].Surround the side of the position of the incoming discharge with polygons.



Zoom the Grid to make it easier to add Inflow points.

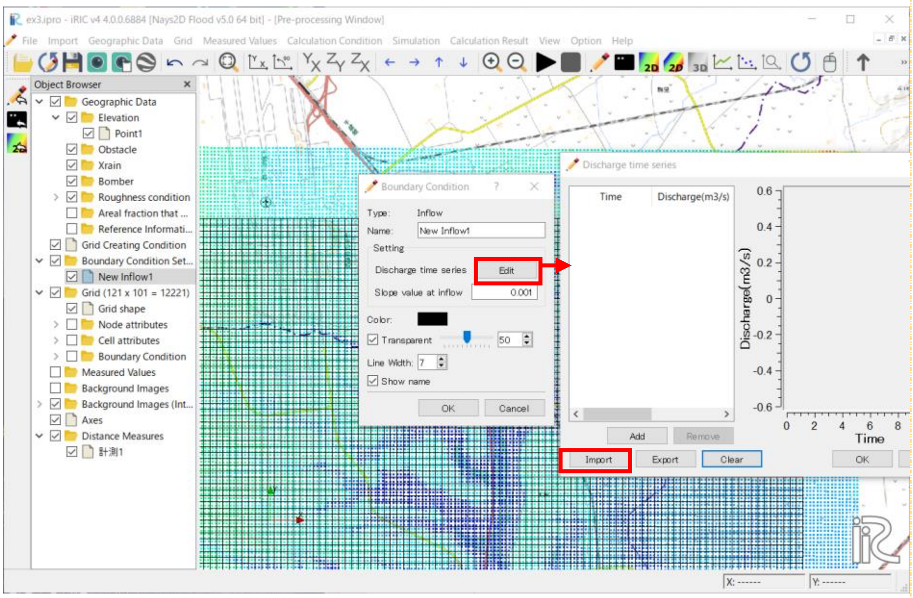
Mouse Hint: Ctrl+Mouse wheel = Zoom IN/Zoom OUT

Double-click to confirm and the [Boundary condition] screen will be displayed.



On the [Boundary Condition] window, input "Inflow 1" to the [Name] box, and select [Edit] for the [Discharge time series] box under [Setting].

The [Calculation Condition] window will be displayed.



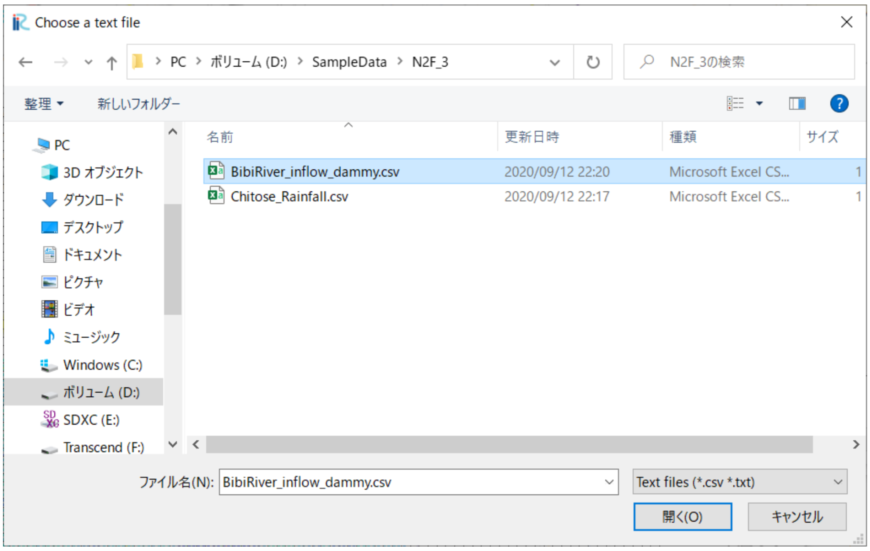
Name: Arbitrary name of inflow

Slope value at inflow: 0.001

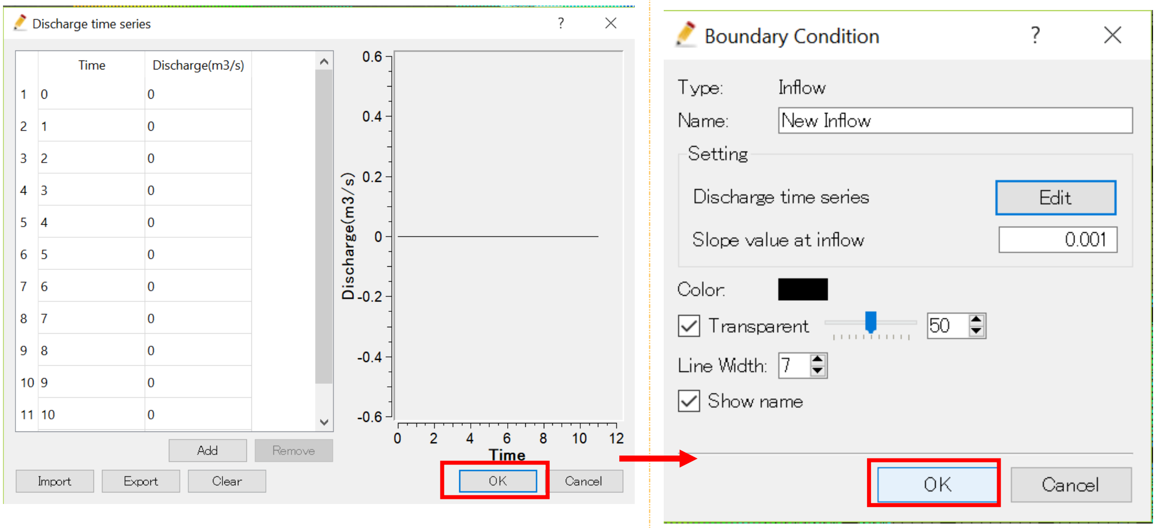
Click on [Import]on the [Calculation Condition] window

Open [¥¥SampleData¥¥N2F_3], select [BibiRIver_Inflow_dummy.csv] and click on [Open].

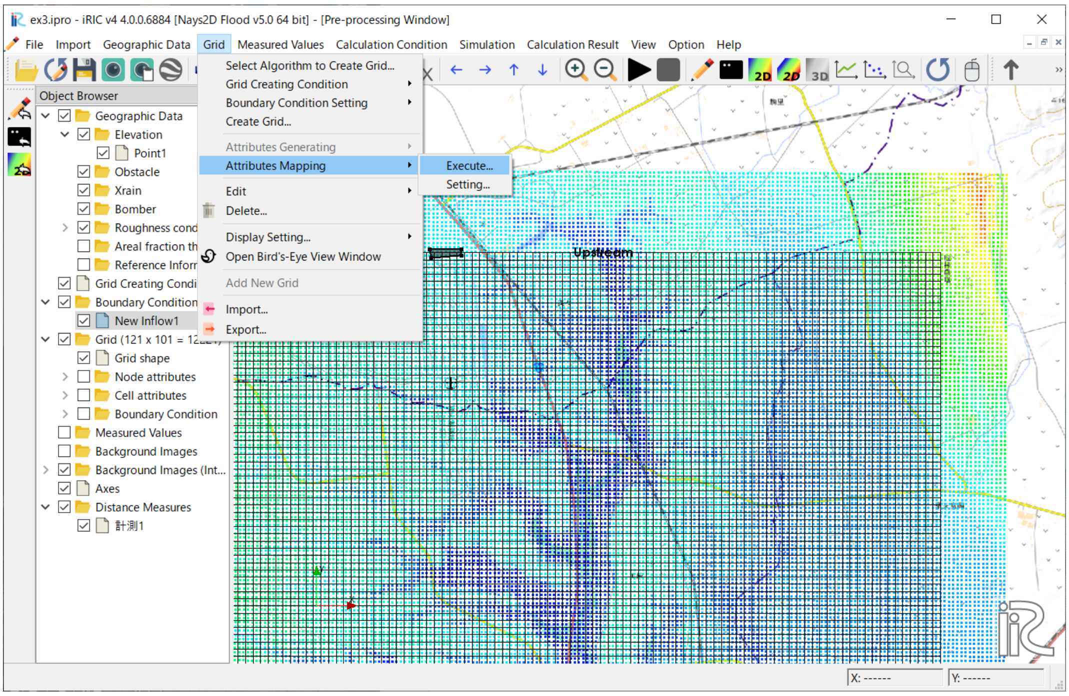
Time series dummy discharge data (all discharges are zero) will be displayed.



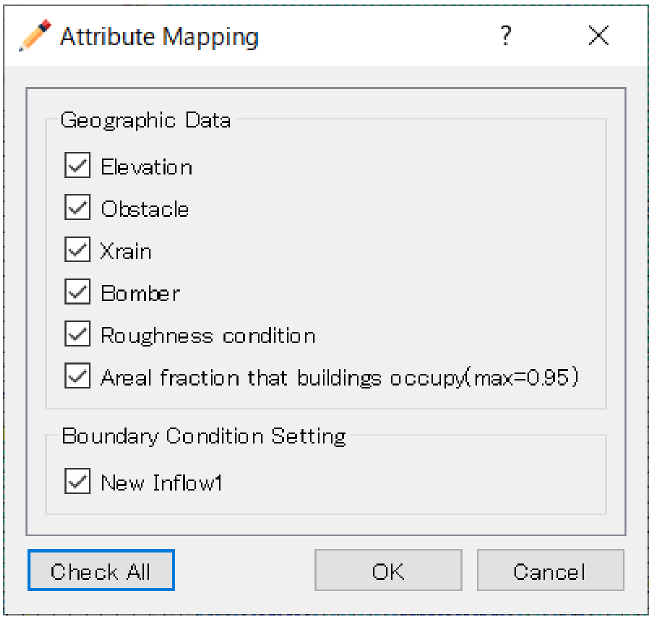
Click on [OK].



In the menu bar, click [Grid]-[Attributes Mapping]-[Execute].

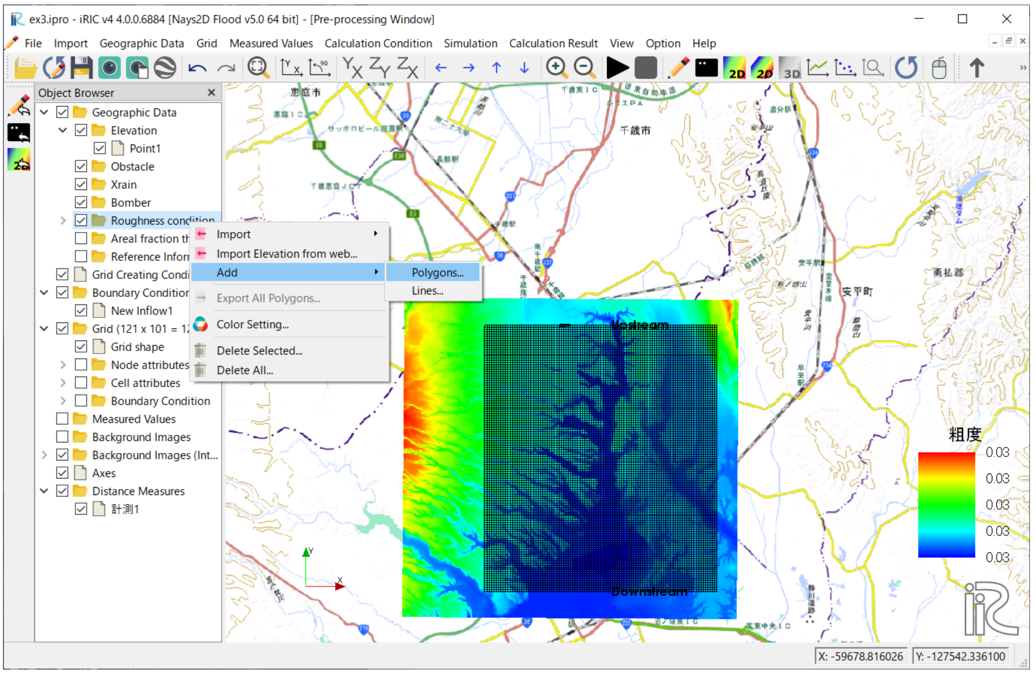


In Attribute Mapping bar -click [Check All]- [OK]

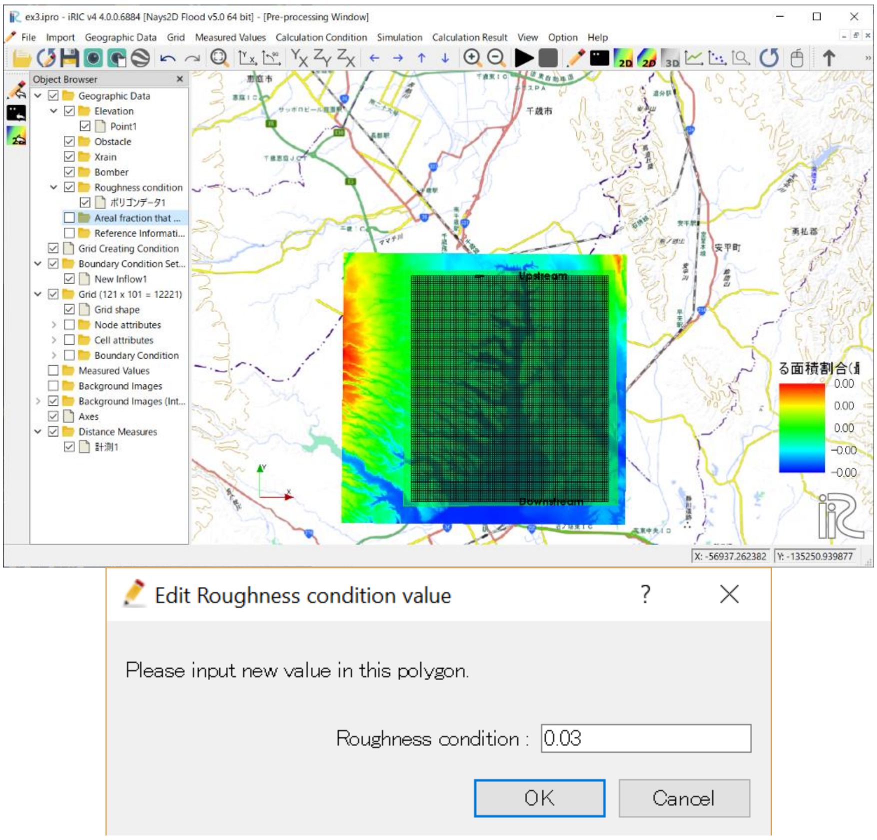


7 Setting the roughness

In the Object Browser, select [Geographic Data] - [Roughness] and right click to select [Add] - [Polygon].



Click on several points to set several vertices such that the vertices enclose the grid area, and press the Enter key on your keyboard to complete the operation.
Then, input a value of Manning's roughness coefficient for [Roughness condition] in the [Edit roughness condition value] window.



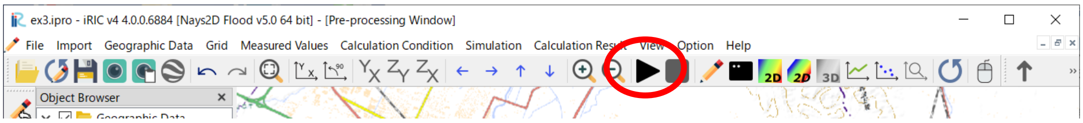
Setting Polygon

A polygon is set by enclosing the area by the line linking vertices made by clicking the location you want to set as a vertex. To finish, press the Enter key or double click.

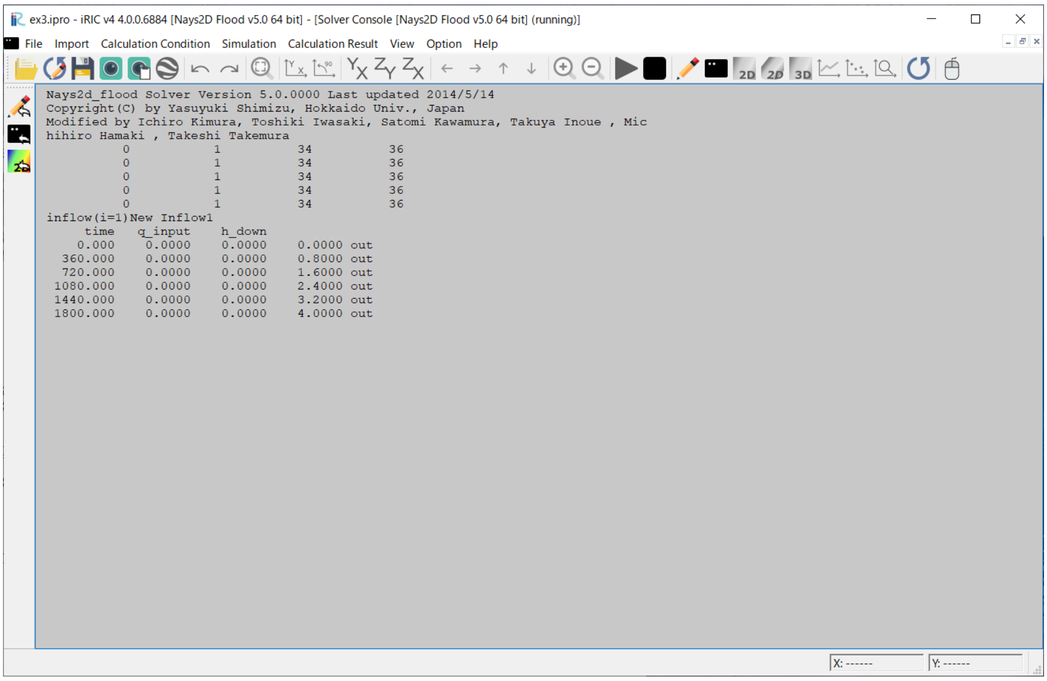
Note: When setting Manning's roughness coefficient, first you make a polygon that encloses all grid nodes. In this example, you want to set the same roughness condition for the entire grid.

3. Making a simulation

On the menu bar, click ▶ symbol to run the simulation



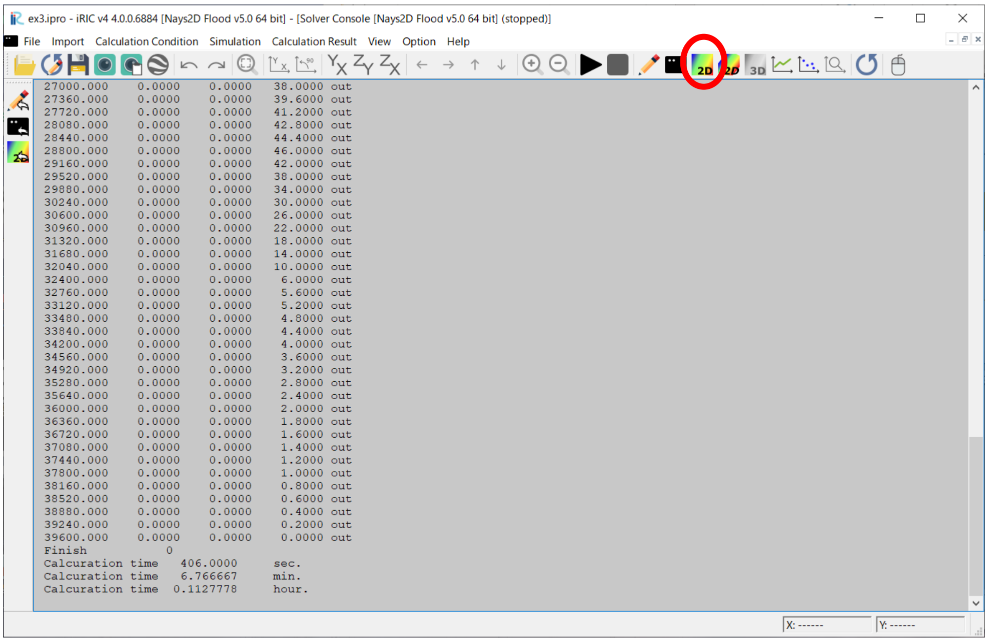
The [Solver Console [Nays2D Flood] (running)] window will open to start the simulation.



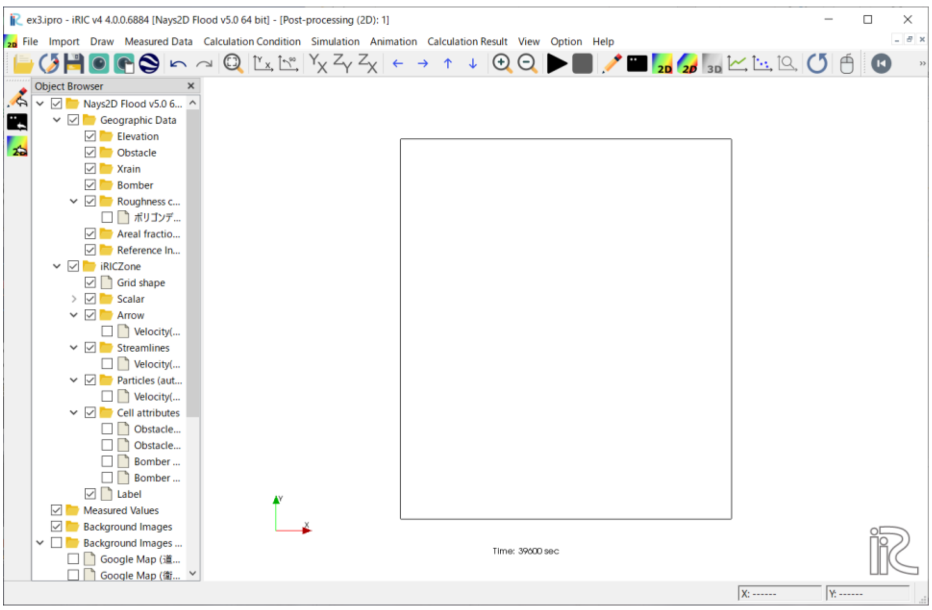
4. Visualization of computational results

1 Open the 2D Post-processing window

On the menu bar, click [Open New 2D Post-Processing Window] icon



The [Post-processing Window (2D)] will open.



Select

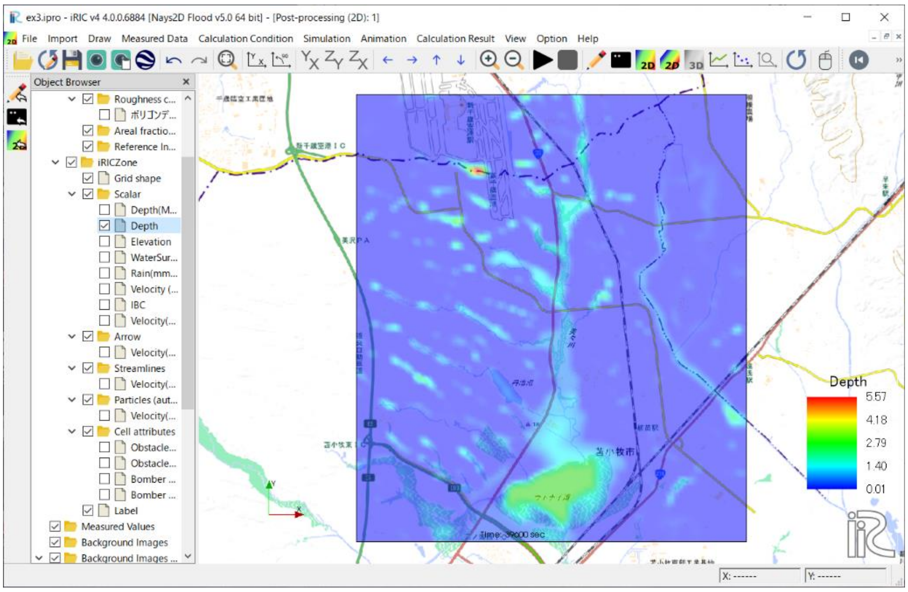
Background

Image

2 Visualizing the water depth

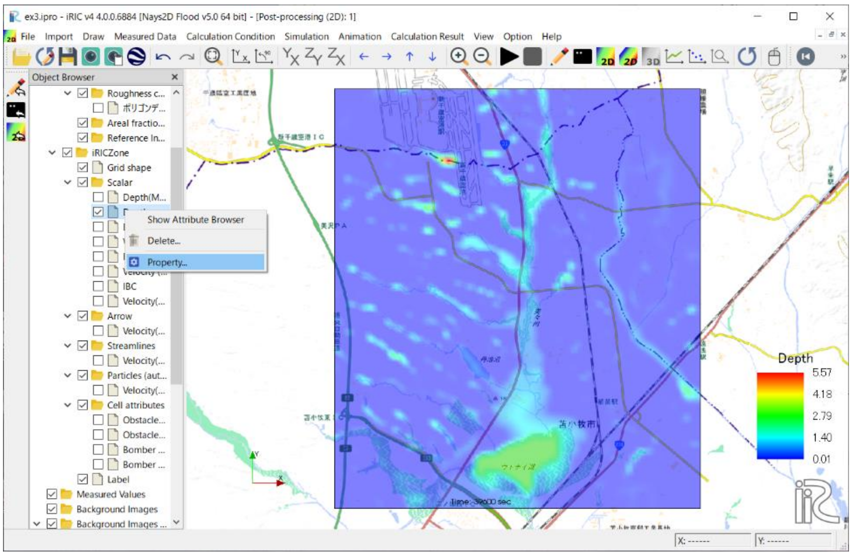
In the Object Browser, select [Nays2D Flood Grids] - [iRICZone] - [Scalar] - [Depth] by making a check mark in each box.

A contour map of water depth will open.

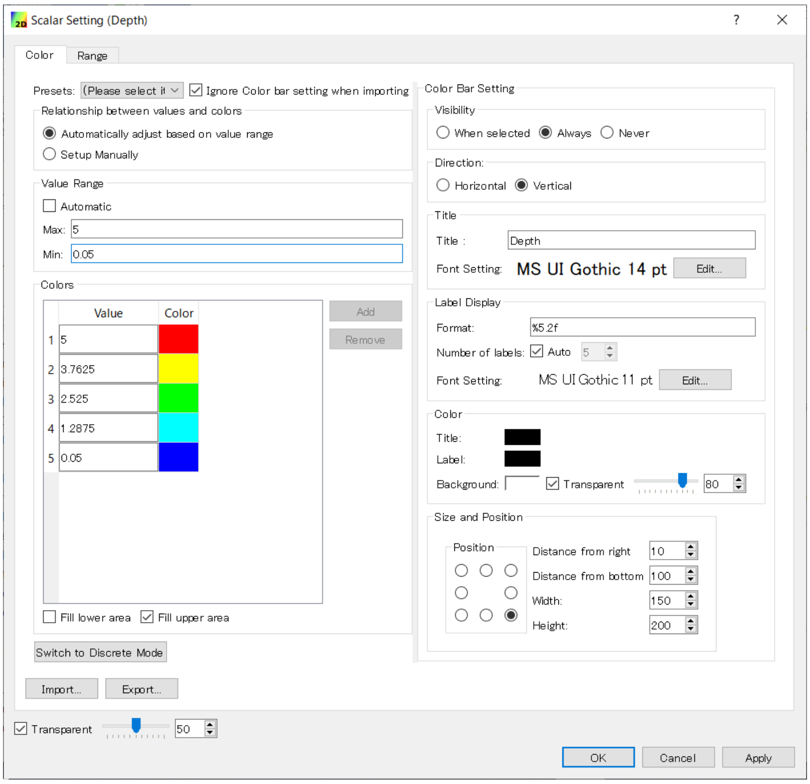


In the Object Browser, select [Nays2D Flood Grids] - [iRICZone] - [Scalar]. Right click on [Depth] to select [Property].

The [Scalar Setting] window will open.



On the [Scalar Setting] window, make the following settings and click on [OK]:



Value range:

Remove ☑ from [Automatic]

Max: 5

Min: 0.05

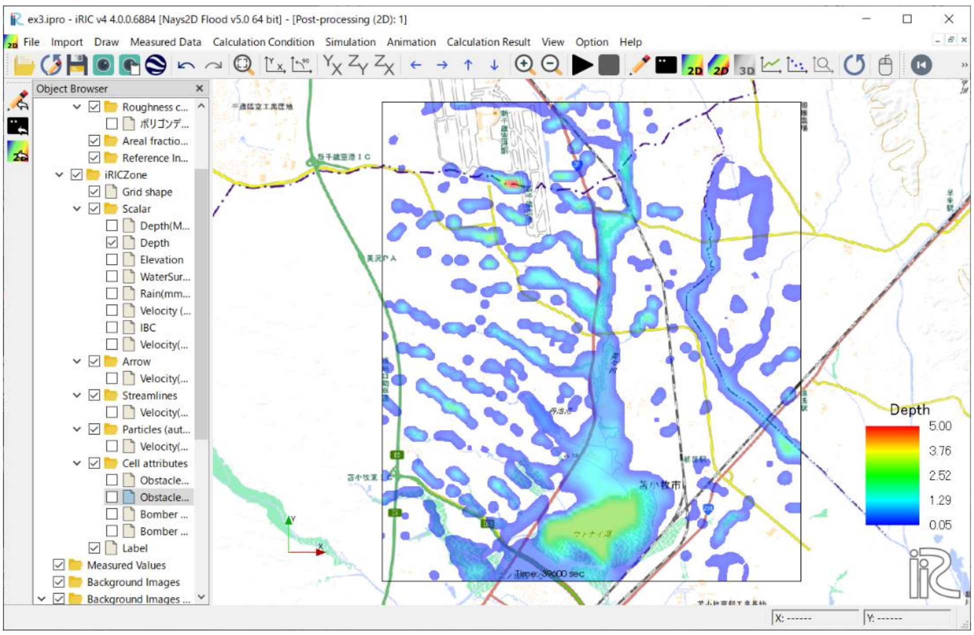
Remove ☑ from [Fill lower area]

Colors: Do not change any settings

Transparent: Do not change any settings.

Color Bar Setting: Do not change any settings.

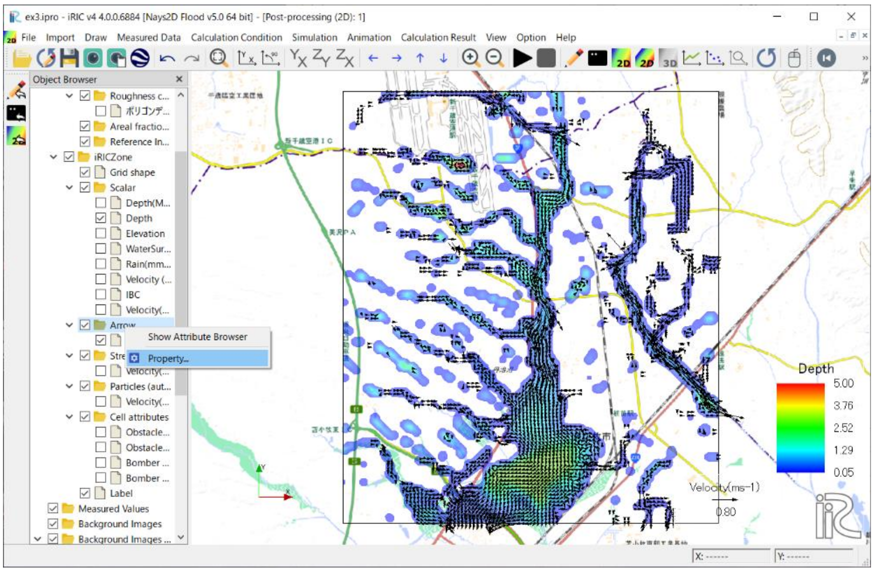
The contour map is easier to see now.



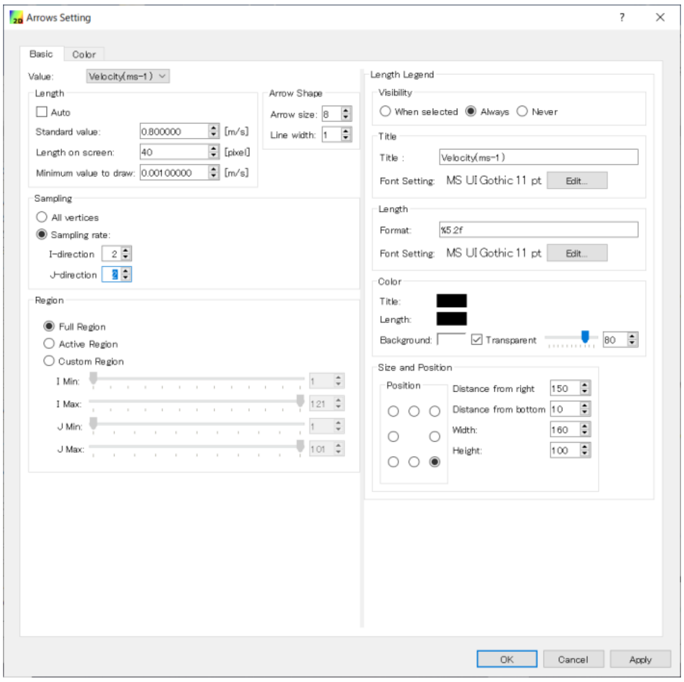
3 Visualizing flow vector

In the Object Browser, select [Nays2D Flood Grids] - [iRICZone] - [Arrow] - [Velocity].
In the Object Browser, select [Nays2D Food Grids] - [iRICZone] - [Arrow]. Right click on [Arrow] to select [Property].

The [Contour Setting] window will open.



On the [Contour Setting] window, make the following settings and click on [OK]:



Length

Remove ☑ from [Auto]

Standard value: 8.00

Length on screen: 40

Minimum value to draw: 0.008

Sampling:

Sampling rate (I-direction): 2

Sampling rate (J-direction): 2

Color:

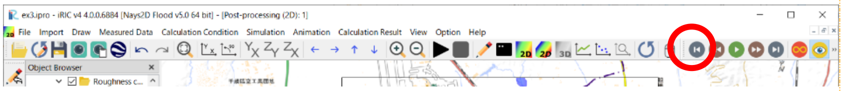
Do not change any settings.

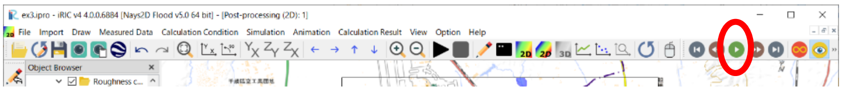
4 Visualizing animated changes

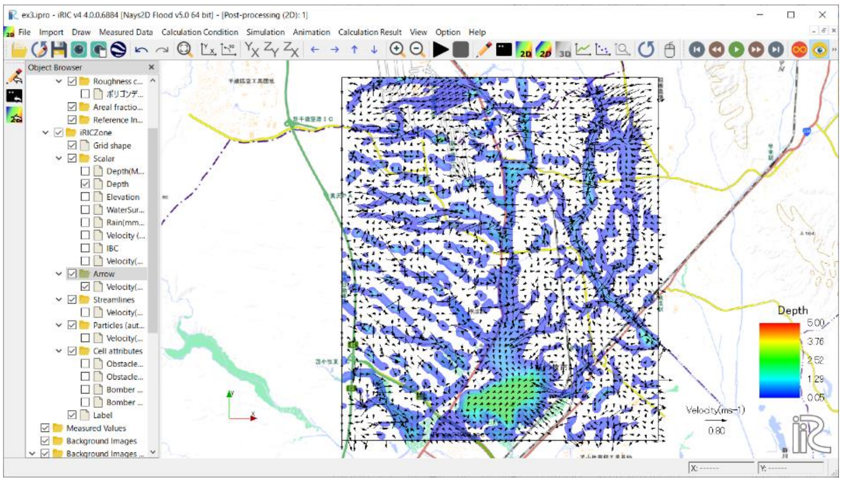
On the menu bar, Click [restart]

On the menu bar, Click [▶]

Animated changes in inundation depth and flow vector will be displayed.







To Reader

- Please reference the iRIC software, if you publish a paper with results from 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

Editors and writers	Yasuyuki SHIMIZU (Hokkaido Univ., Japan)	Editing
	Eiichi SUZUKI (Hokkaido Univ., Japan)	Editing
	Satomi KAWAMURA (Hokkaido Univ., Japan)	Editing
	Takuya INOUE (CERI., Japan)	Editing/Writing
	Toshiki IWASAKI (Hokkaido Univ., Japan)	Writing
	Michihiro HAMAKI(Kaihatsu Koei Co., Ltd., Japan)	Writing
	Tomohiko Yoshida (Docon Co.,Ltd., Japan)	Writing
	Jagriti Mishra (CERI., Japan)	Writing
	Kensuke OMURA (Hokkaido Univ., Japan)	
	Eriko KAKEGAWA (Hokkaido Univ., Japan)	

Supported by River center of Hokkaido, Japan.

Co-sponsored:
Chinese Institute of Civil and Hydraulic Engineering
Sustainable Development Committee

Co-organizing:
Dept. of Bioenvironmental Systems Engineering, NTU
Dept. of Civil Engineering, NTU
Dept. of Civil Engineering, NCHU
Sinotech Engineering Consultants, Inc.