iRIC Software

Changing River Science

Nays2D Flood Examples

Directed by ***Yasuyuki Shimizu***

Edited by ***Takuya Inoue***

Eiichi Suzuki

Satomi Kawamura

Toshiki Iwasaki

Michihiro Hamaki

Kensuke Omura

Eriko Kakegawa

Last Update: 2013.10.1
Release Date: 2012.05.25

Contents

Chapter 1
Using Nays2D Flood

This manual explains the basic operation and startup procedures of Nays2D Flood, which is compiled with iRIC. (Nays2D Flood 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/downloads
Software: iRIC version2.0beta

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

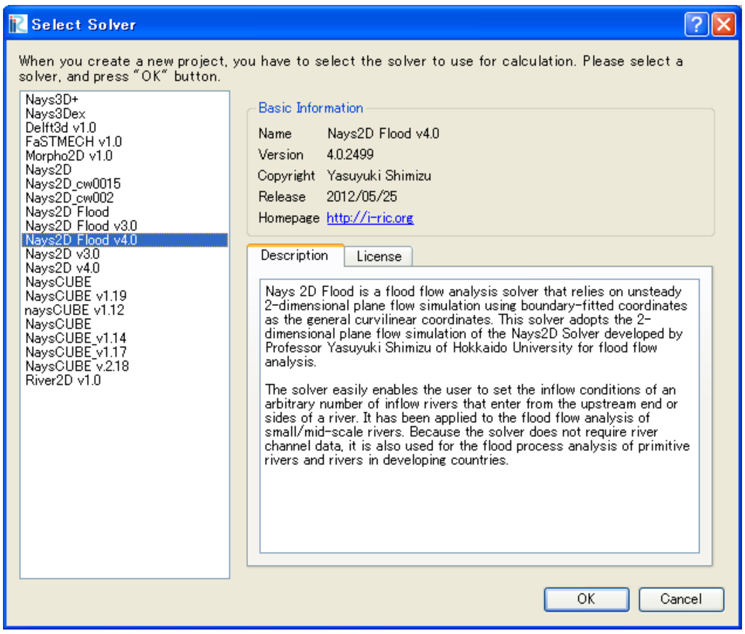
The following is the procedure to launch Nays2D Flood on iRIC.

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

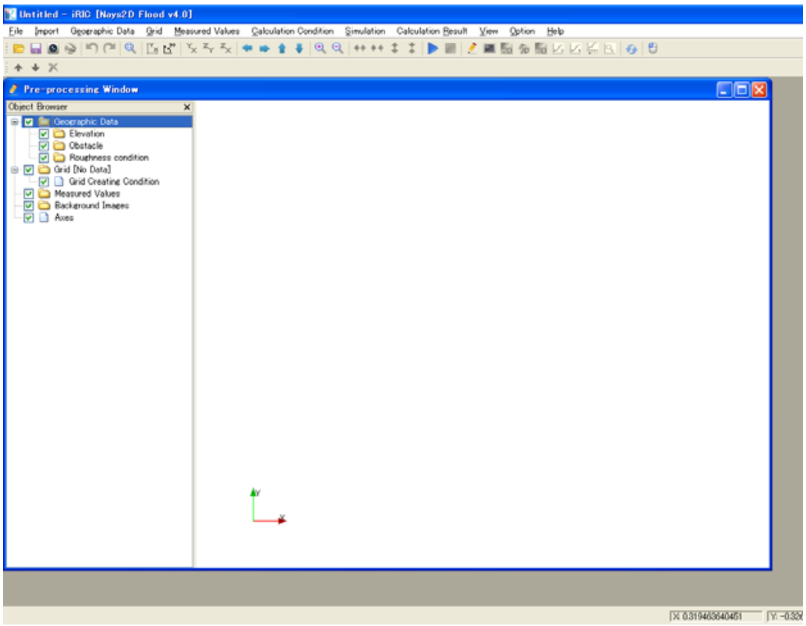
➢ Click on [New Project] in the [iRIC Start Page] window.

The [Select Solver] window will open.

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



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



Nays2D Flood 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/downloads
Examples of simulation: Nays2D Flood

To run Nays2D Flood according to this manual, data should be downloaded from the above site.
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 condition and so on, following this manual enables calculation.

Solver: Nays2D Flood 4.0

4. Simulation conditions of Nays2D Flood

This manual teaches how Nays2D Flood is used to simulate river flow and riverbed deformation.
Hence, 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 functions (setting conditions) additional to those explained by this manual. For details, please refer to the Nays2D Flood Solver Manual.

Chapter 2
Examples of flood calculation for an actual river section

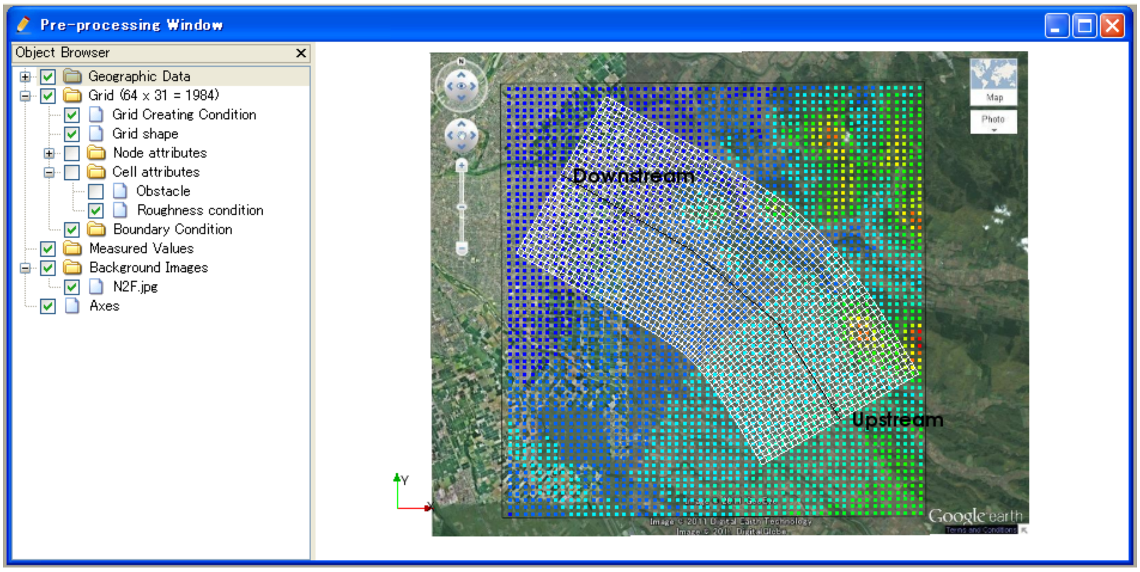
Objectives

Simulate the flow regime (water depth and flow velocity) by using Nays2D Flood for an actual river section with a flood discharge, and 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: 31 divisions in the transverse direction, and 64 divisions in the longitudinal direction.



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

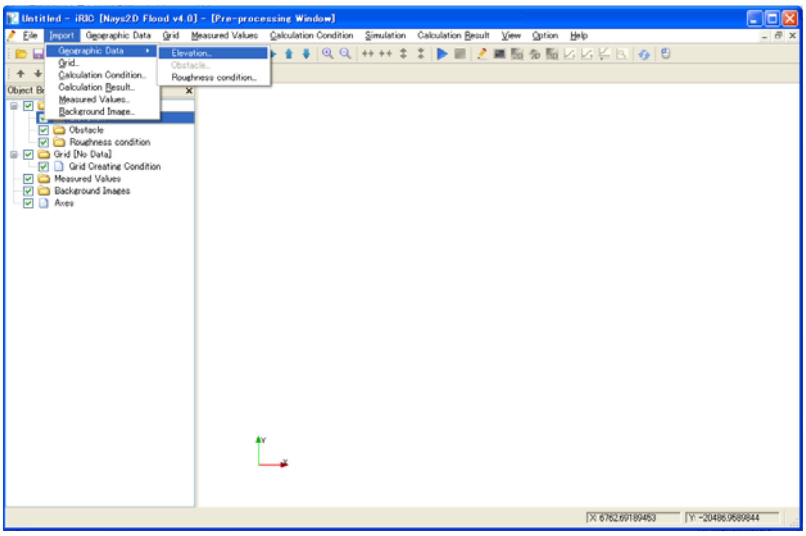
Here, we introduce how to display a water depth contour map and a flow velocity vector map.

1. Creating the calculation grid

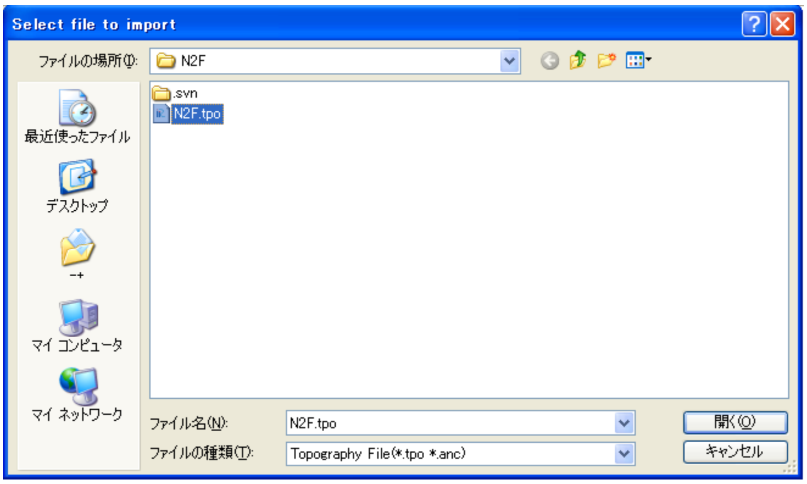
1. Importing cross-sectional river survey data

① Importing geographic data

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

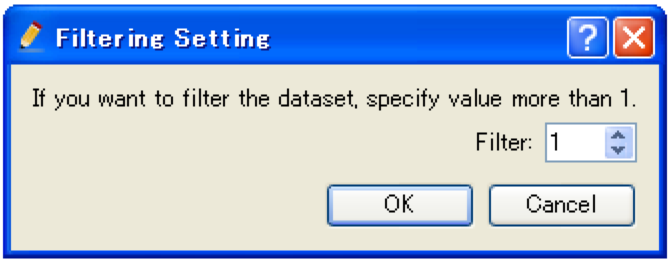


➢ Open [¥SampleData¥N2F], select [N2F.tpo] and click on [Open].

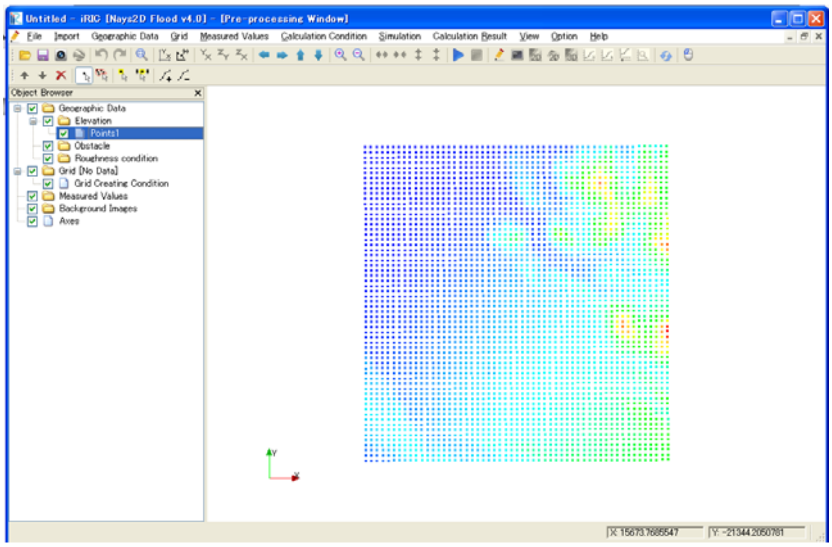


➢ On the dialog [Filtering Setting], input [1] for [Filter] value and select [OK].

Set a larger number for filter value and filter the dataset if the operation is slow because of the number of data.

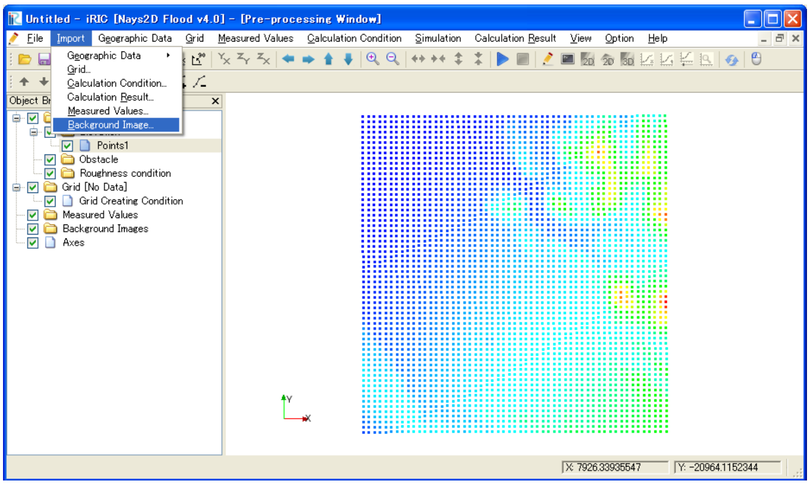


If the [Pre-Processing] window shows the shape of the river section that you are simulating, the data have been successfully imported.



② Importing a background image

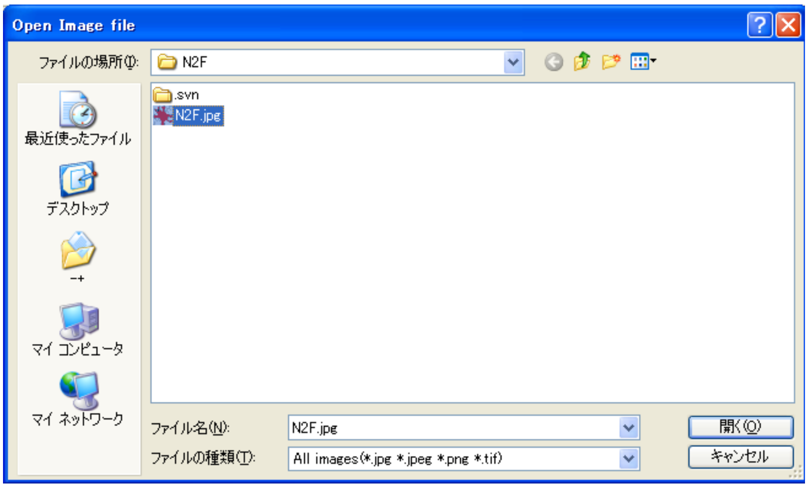
➢ On the menu bar, select [Import] - [Background Image].



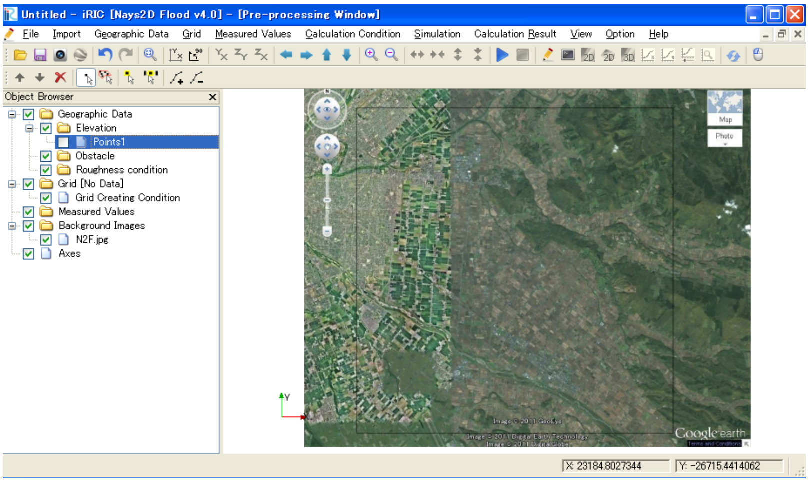
Background image

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 [map2.jpg].

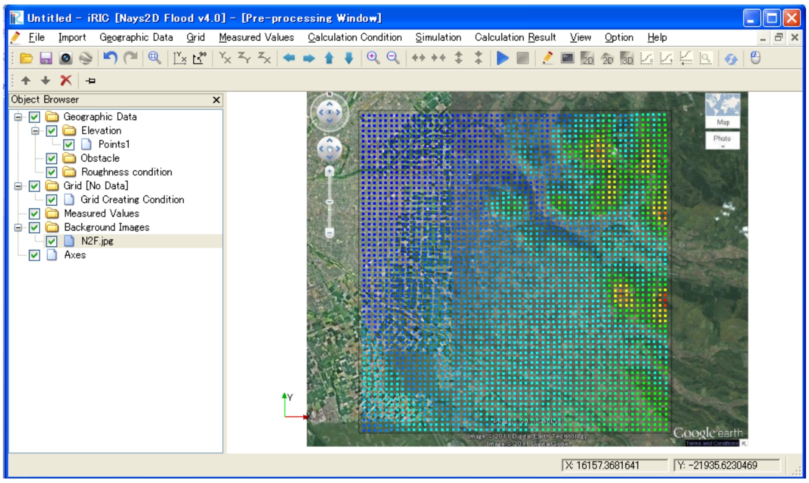


➢ Using the [Move], [Rotate] and [Zoom] functions, match the background image with the elevation data.



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 flame line is drawn and you can use this flame line to match.

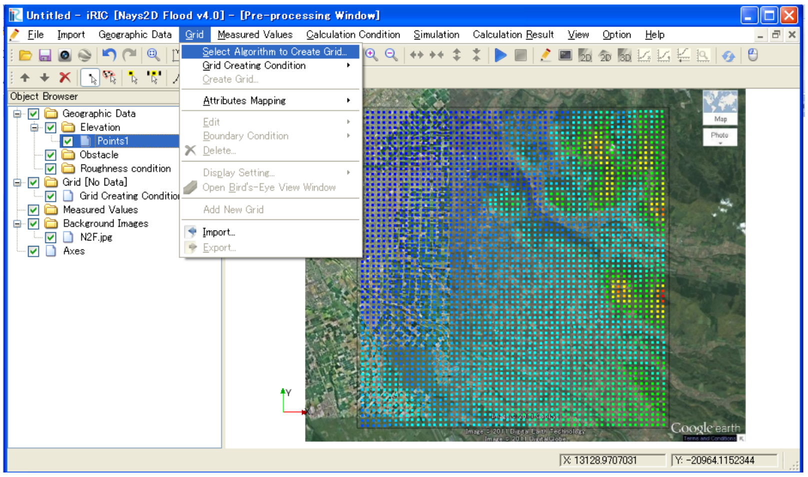
➢ Refering to the hint above, match the background image with the evaluation data.



2. Selecting an algorithm for creating a grid

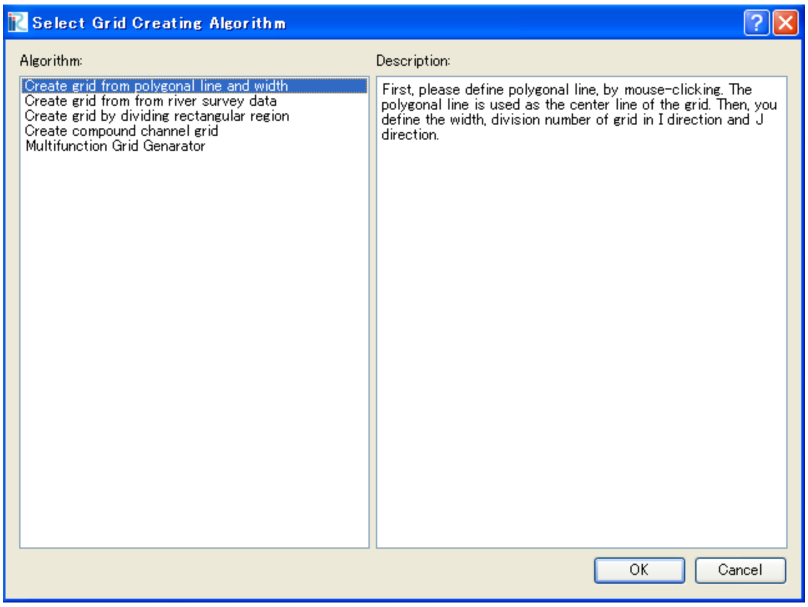
➢ On the menu bar, select [Grid] - [Select algorithm to create grid].

The [Select Grid Creating Algorithm] window will open.



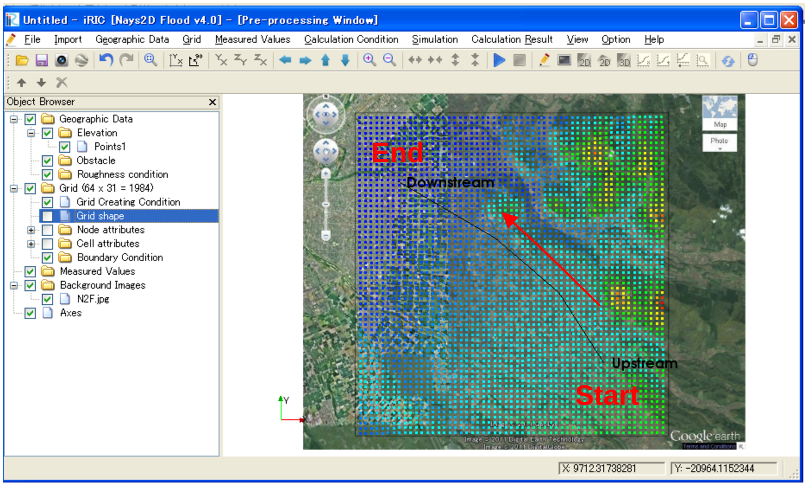
➢ Select [Create grid from polygonal line and width] from the list below the [Select Grid Creating Algorithm] window, and click on [OK].

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



3. Creating a grid

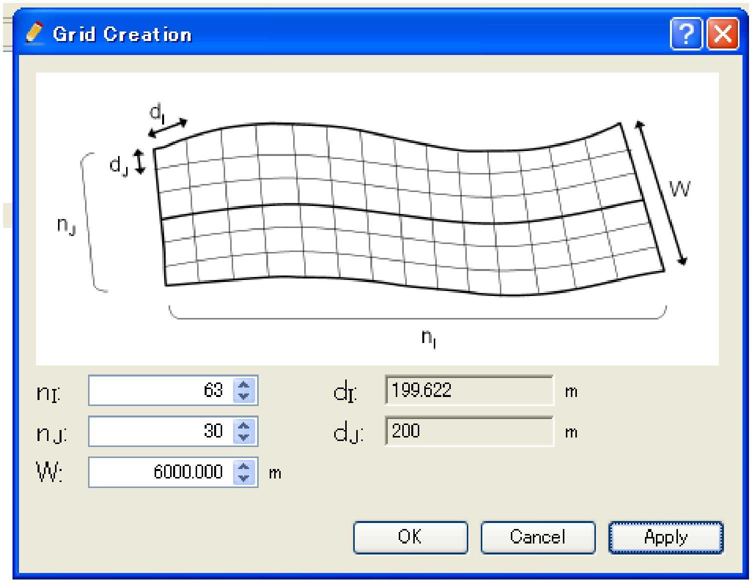
➢ Right-click on several points through which the grid centerline passes, and press the "Enter" key.



Setting the grid centerline

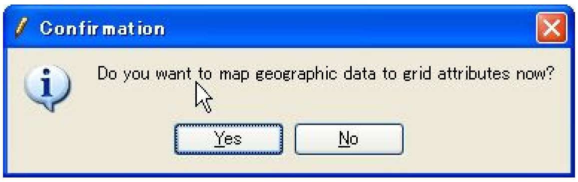
● Set the grid centerline from the upstream end, where flood flow enters, to the downstream end, where the flood flow exits. To finish, double-click on the end of the centerline, or hit the "Enter" key.

➢ 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

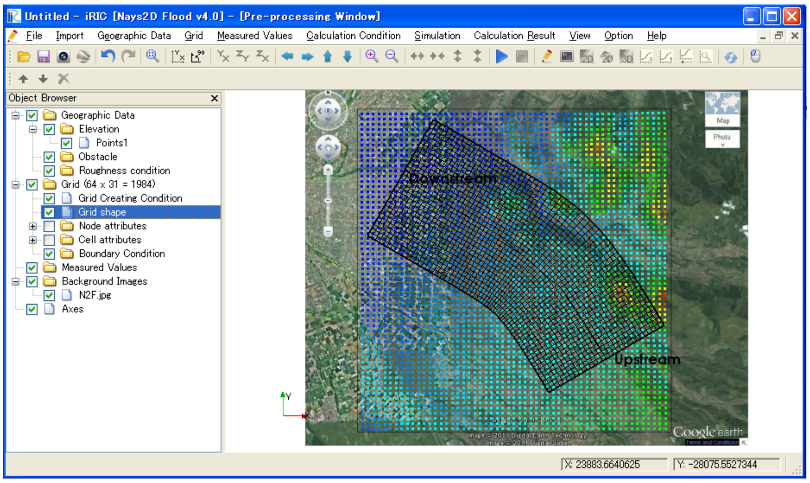
➢ Click on [Yes] in the [Confirmation] window.



Mapping geographic data

- Elevation data are applied to the grid.

A grid will be created.



Adjusting the calculation grid

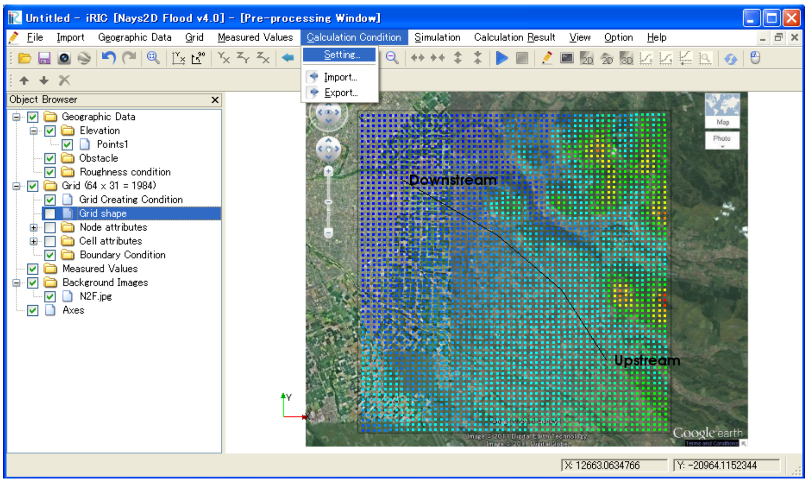
- It is possible to move, add or remove centerlines under [Grid Creating Conditions] even after the grid is created.

2. Setting the calculation conditions

1. Open "Calculation Conditions"

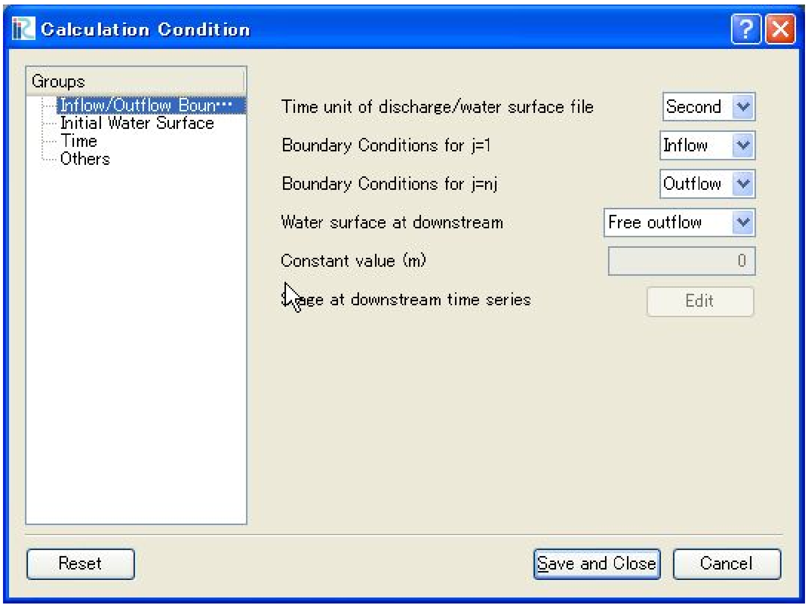
➢ On the menu bar, select [Calculation Conditions] - [Setting].

The [Calculation Conditions] window will open.



2. Setting the inflow boundary conditions

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



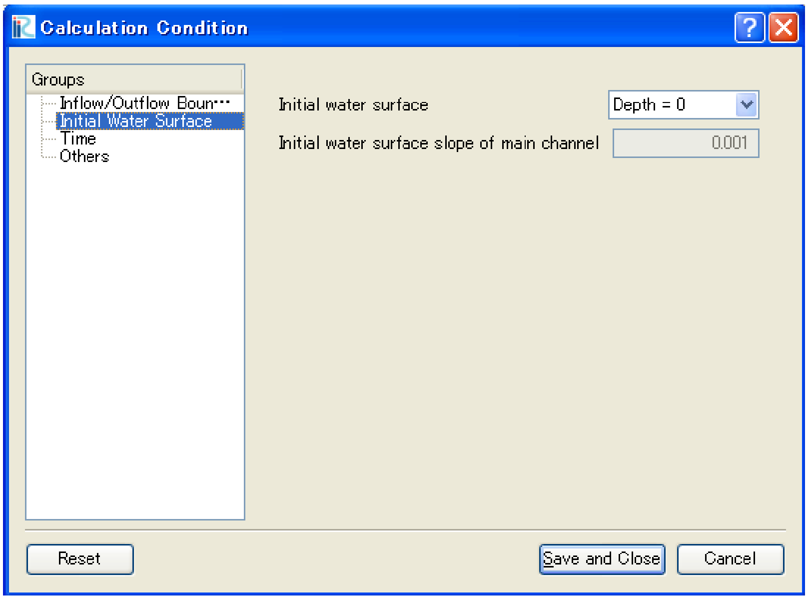
- Time unit of discharge/water surface elevation files: second
- Boundary Conditions for j=1: Inflow
- Boundary Condition for j=nj: Outflow
- Water surface at downstream: Free Outflow

※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 outflow.

3. Setting the initial water surface profile

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

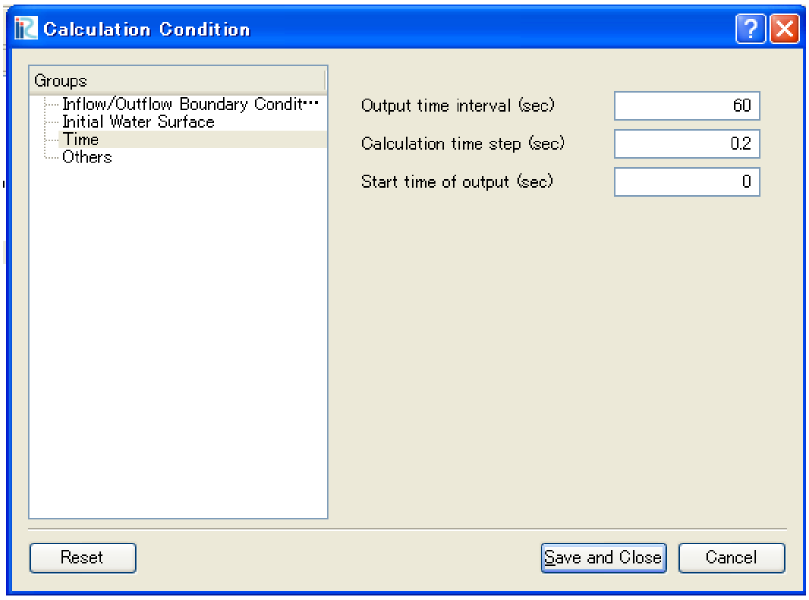


●"Initial water surface": Depth = 0

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).

4. Setting the time

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



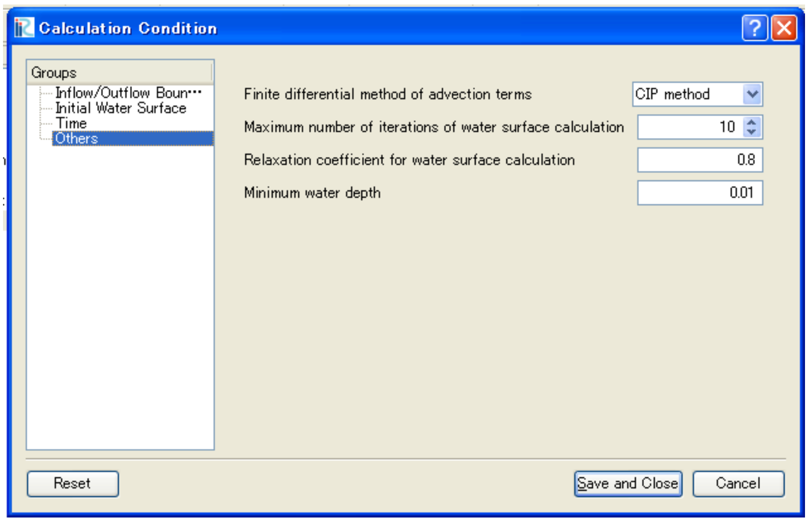
- Output time interval (sec): 60
- Calculation time step (sec): 0.2
- Start time of output (sec): 0

Many other conditions can be set; however, they do not need to be set for this simulation.

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

5. Other settings

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



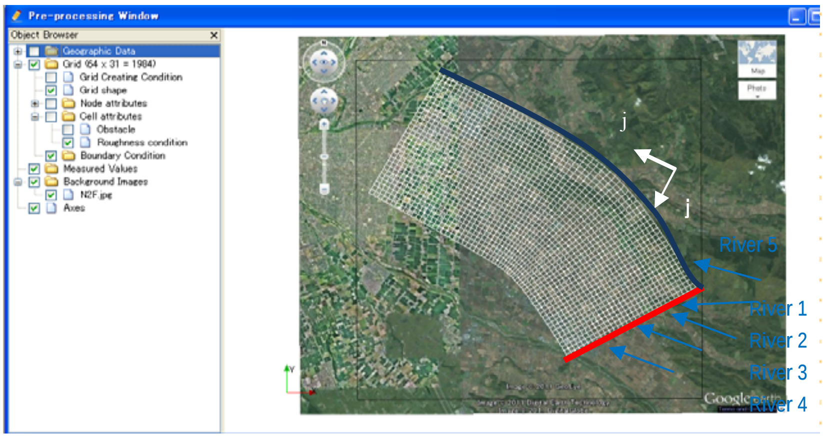
- Finite difference method of advection terms: CIP method
- Maximum number of iterations of water surface calculation: 10
- Relaxation coefficient for water surface calculation: 0.8
- Minimum water depth: 0.01

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

6. Setting inflow rivers

① The number of inflow rivers

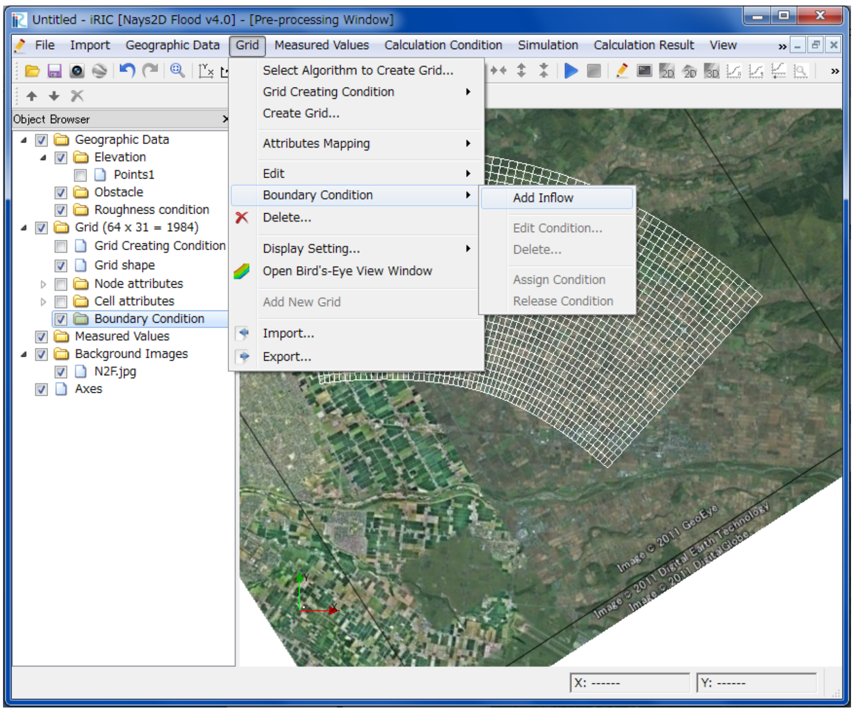
Set the inflow rivers (or the bank opening point) at inflow side (red line) and the side(j=1) you set as [inflow] (blue line) at setting inflow condition.
※The maximum number of rivers could be added is up to five.



- In this river section, four rivers (including agricultural canals) flow across the red line and one river flows across the blue line, as indicated by the blue arrows.
- No river comes in the area from i=ni and j=nj and these boundary conditions are outflow.
- The number of rivers could be up to five.

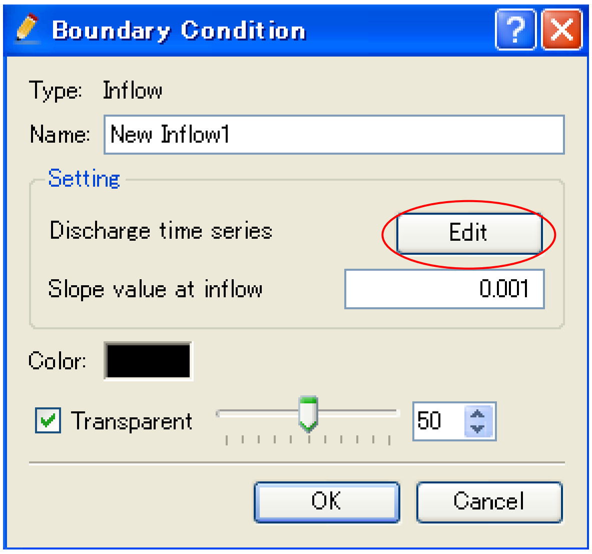
② Additional setting of inflow condition

➢ In the menu bar, select [Grid]-[Boundary condition]-[Add inflow]



➢ In the [Boundary Condition]-[Name], write [New inflow1].

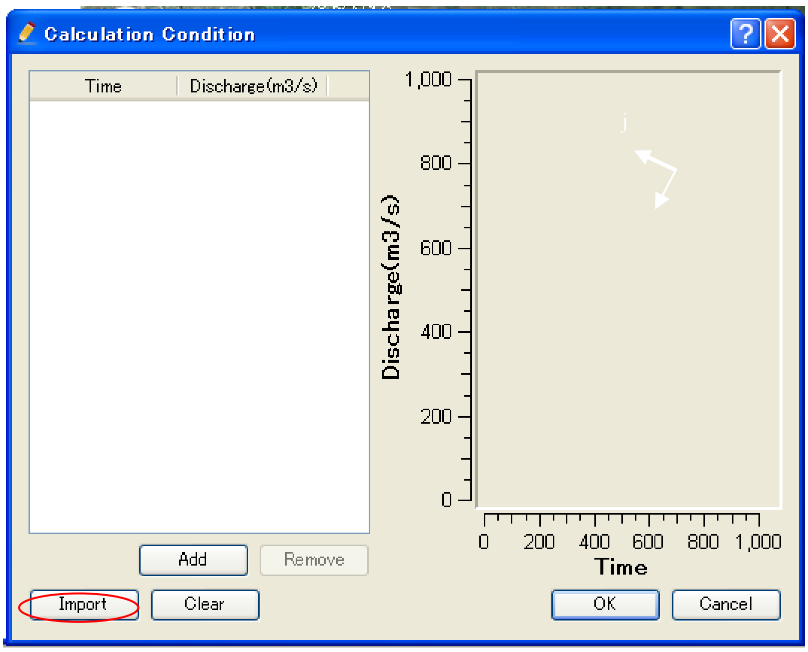
➢ In the [Boundary Condition], click on [edit] from the [Setting]



●Name: arbitrary name
●Slope value at inflow: 0.001

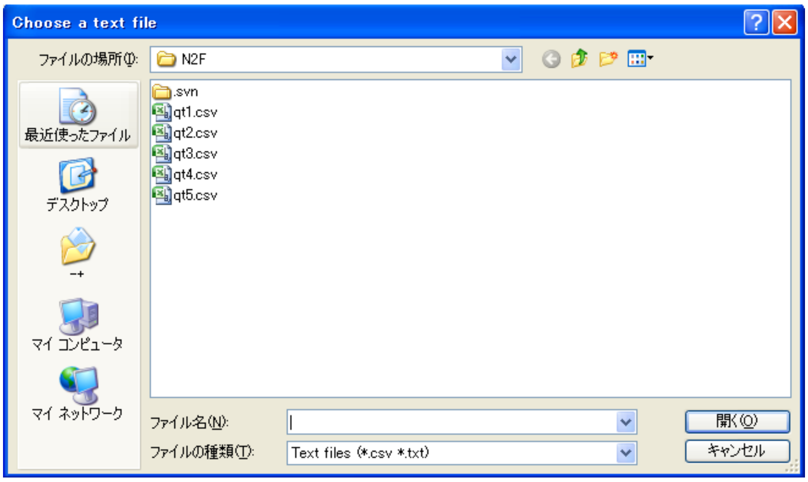
➢ In the [Calculation Condition],click on [Import]

Open the [Choose a text file]



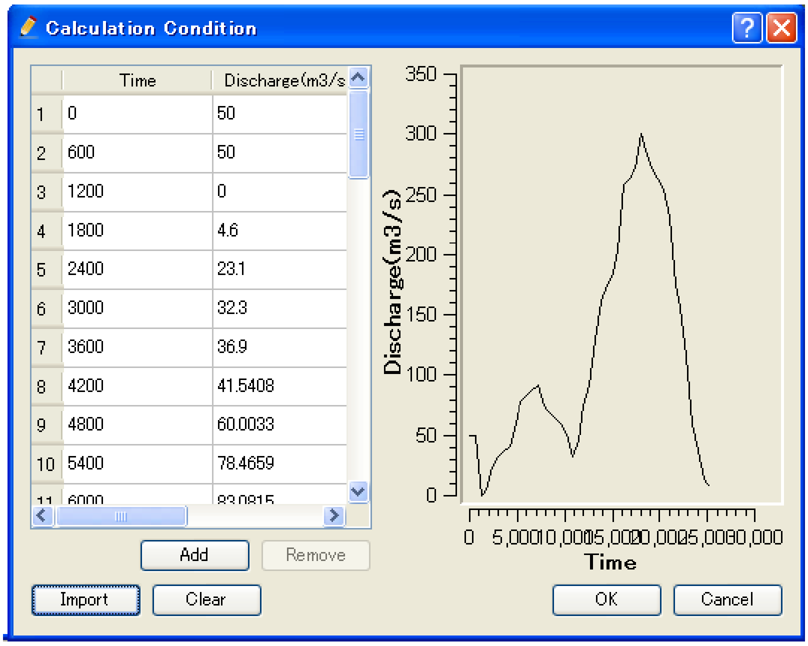
➢ In the [Choose the text file] click on the [qt1.csv] and then [open].

Open the discharge data.



➢ Click on [OK].

➢ Repeat above operations from New Inflow1 to NewInflow5.



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

Inflow1: qt1.csv
Inflow2: qt2.csv
Inflow3: qt3.csv
Inflow4: qt4.csv
Inflow5: qt5.csv

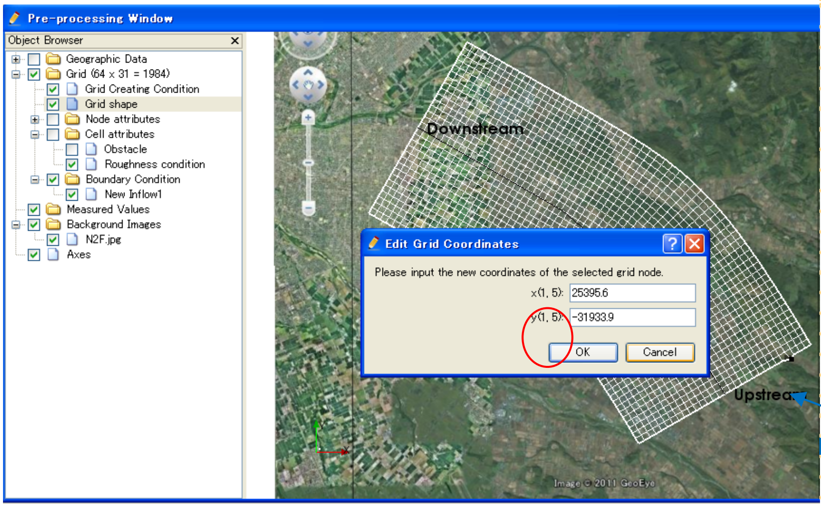
※The time step of discharge file need to be set all same.

③ Creating and selecting an "Input file for grid cell number of inflow"

➢ In the Object Browser, select [Grid] - [Grid shape].

➢ Select and right-click on a grid node.

Check the grid cell number as the [Grid coordinate edit dialog] launches



● The left figure confirms the inflow grid cell number of inflow river 1.

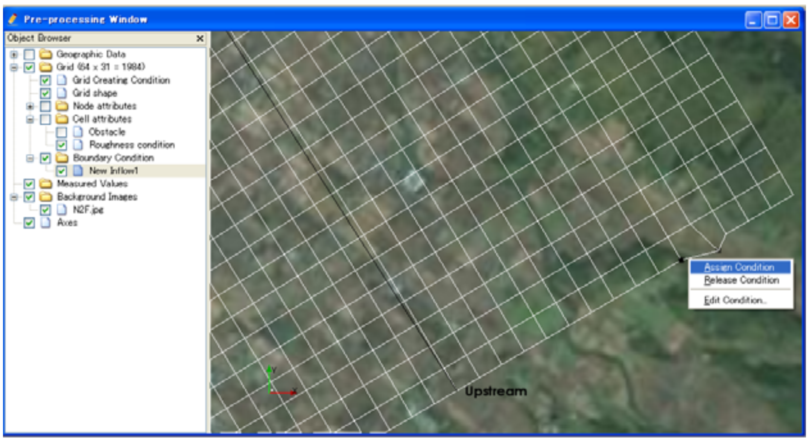
●Confirmation of the grid cell number is not necessary when the inflow point could be assigned from the background image.

➢ In the Object Browser, select [Grid] - [Boundary Condition] - [New Inflow1].

➢ Select a grid point to set as inflow point and right-click it.

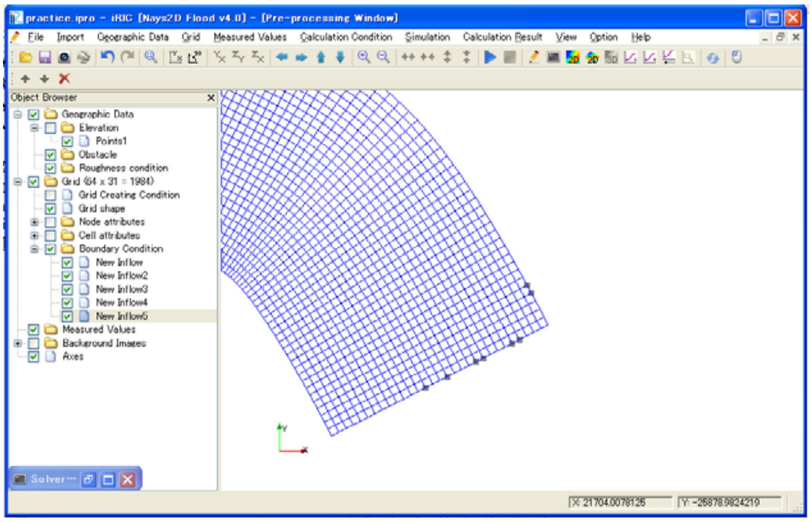
➢ Click on [Assign Condition]

➢ Repeat above operations from New Inflow1 to NewInflow5.



Grid cell numbers of inflow points on the sample data are below;

Inflow river	Grid cell number (i, j)	Discharge	Note
Inflow1	(1,5),(1,6)	qt1.csv	
Inflow2	(1,9),(1,10)	qt2.csv	
Inflow3	(1,14)	qt3.csv	
Inflow4	(1,17)	qt4.csv	
Inflow5	(5,1),(6,1)	qt5.csv	side(j=1)



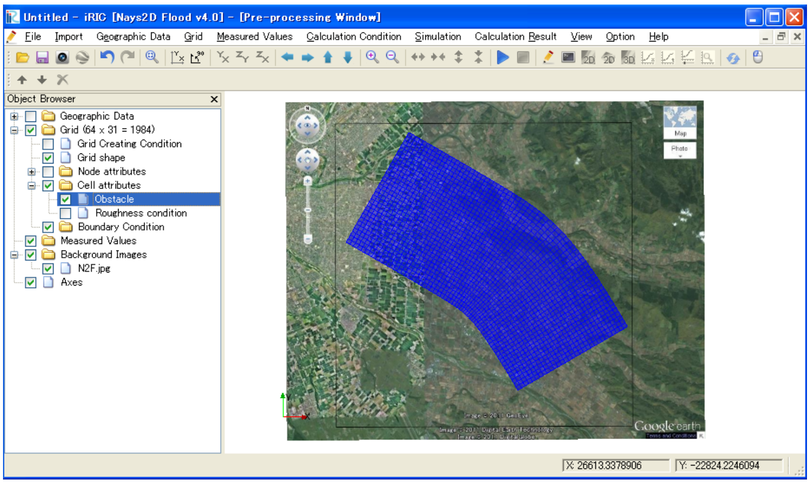
Inflow rivers are not able to be set to other than i =1, j =1 or j =nj

An error occurs the calculation stops when the calculation is run with wrong settings.

In that case, it is deletable right-clicking on [release condition] while selecting the point.

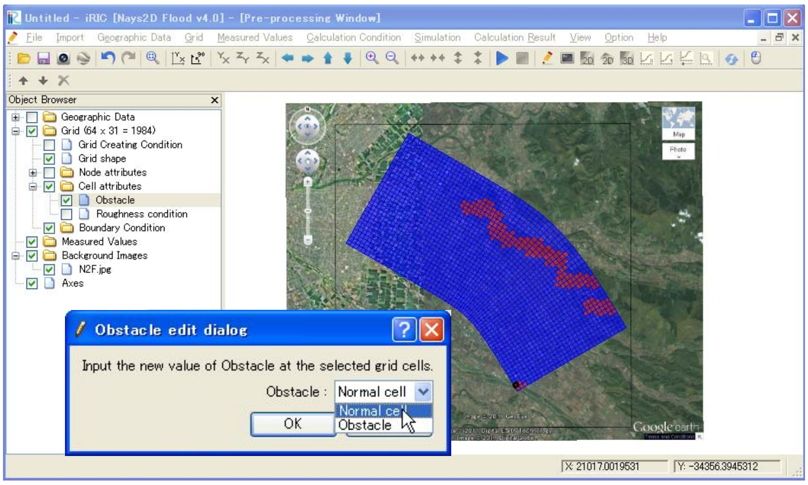
7 Setting obstacle cells

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



➢ Select and right-click on a calculation cell.

➢ Set the obstacle from [Obstacle edit dialog].

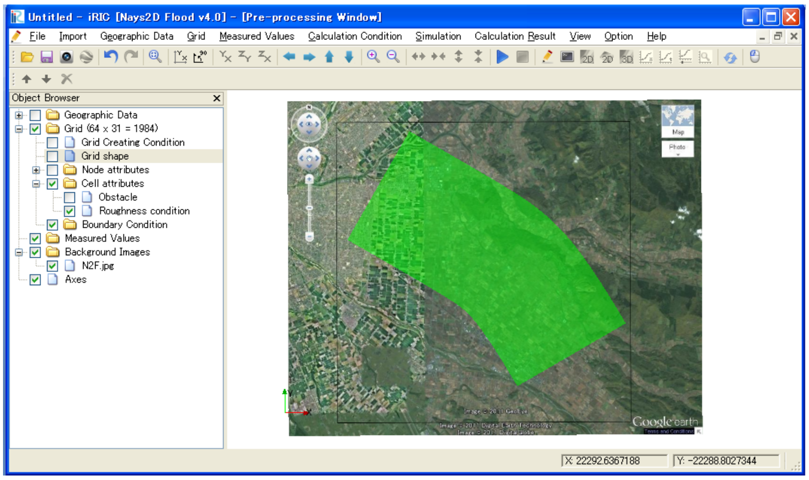


●Set obstacles such as roads, banks and embankments.

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.

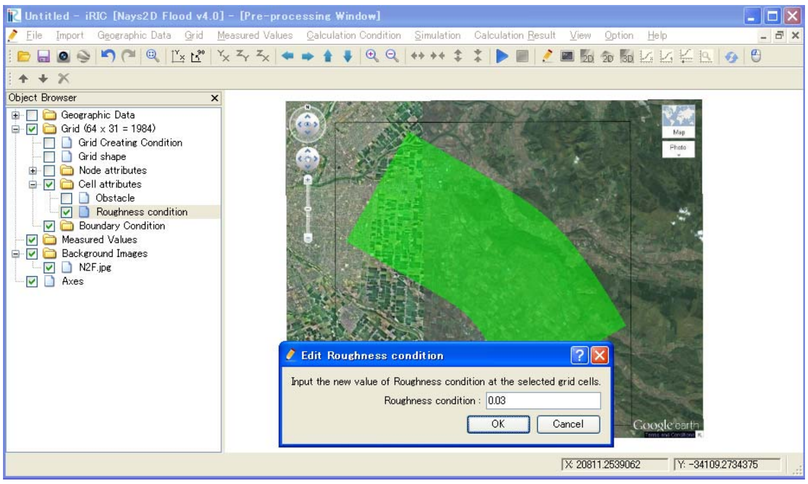
8 Setting the roughness

➢ In the Object Browser, select [Grid] - [Cell attributes] - [Roughness].



➢ Select and right-click on a calculation cell.

➢ Input Manning's roughness coefficient in [Roughness condition edit dialog].



● Roughness coefficient: 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).

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:

$$n_0^{\ 2} = \frac{n_1^{\ 2} A_1 + n_2^{\ 2} A_2 + n_3^{\ 2} A_3}{A_1 + A_2 + A_3}$$

where, n_1=0.060, n_2=0.047 and n_3=0.050.

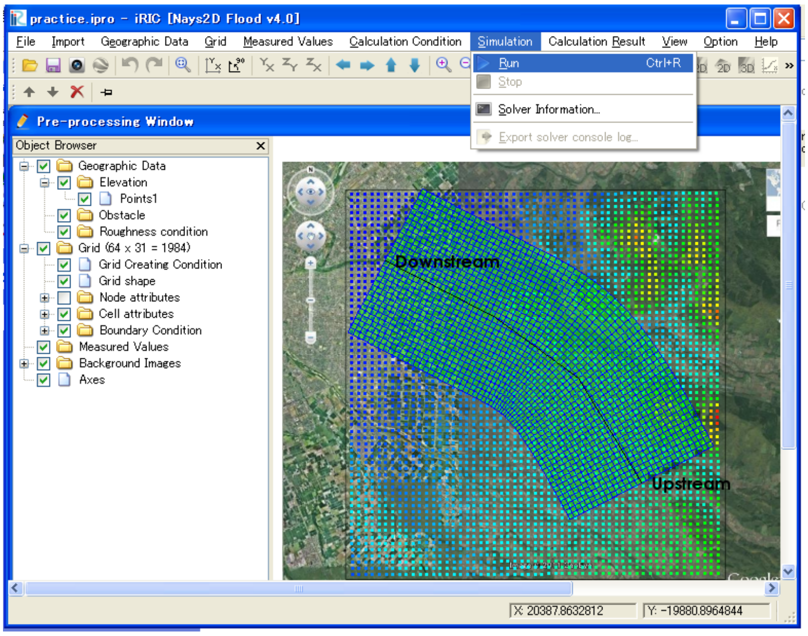
3) The composite equivalent roughness coefficient of the land and buildings is given by the following equation:

$$n^2 = n_0^{\ 2} + 0.020 * \frac{\theta}{100 - \theta} * h^{4/3}$$

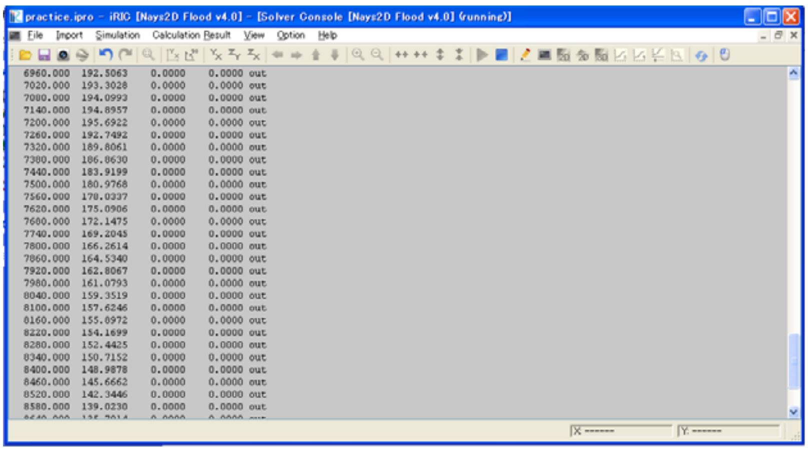
where, θ is the proportion of land occupied by buildings and h is water depth.

3. Making a simulation

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



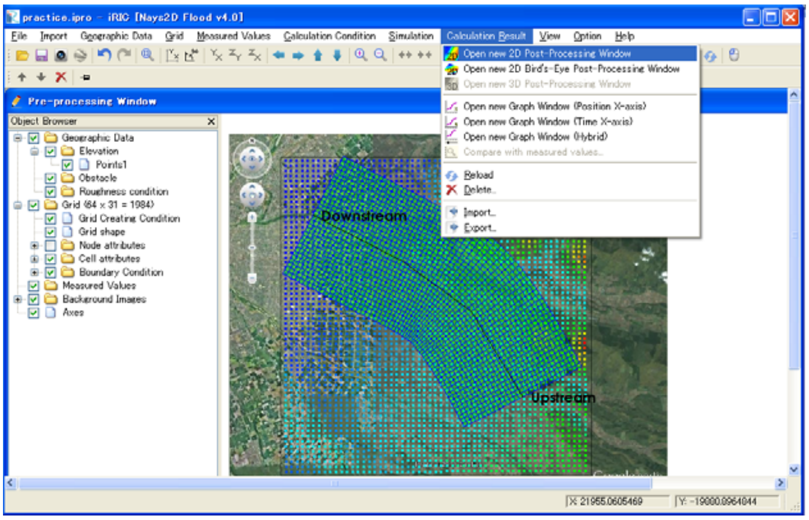
The [Solver Console [Nays2D Flood] (running)] window will open to start the simulation.



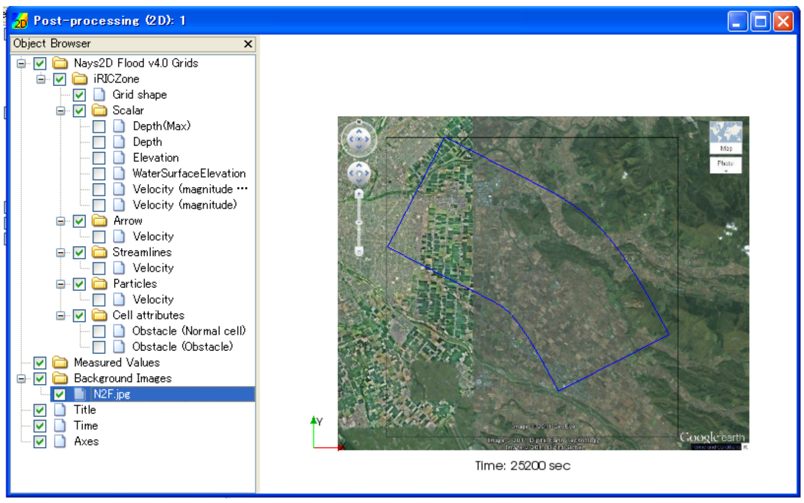
4. Visualizing the calculation results

1. 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.



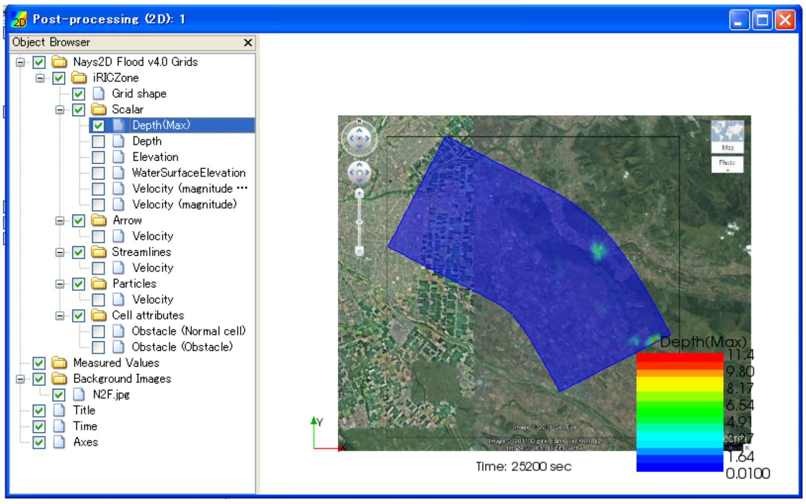
2. Visualizable quantities

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	
Normal cell (a normal cell)	Displays a normal cell.
Obstacle cell (a cell with an obstacle)	Displays an obstacle cell.

3. Visualizing the max. water depth

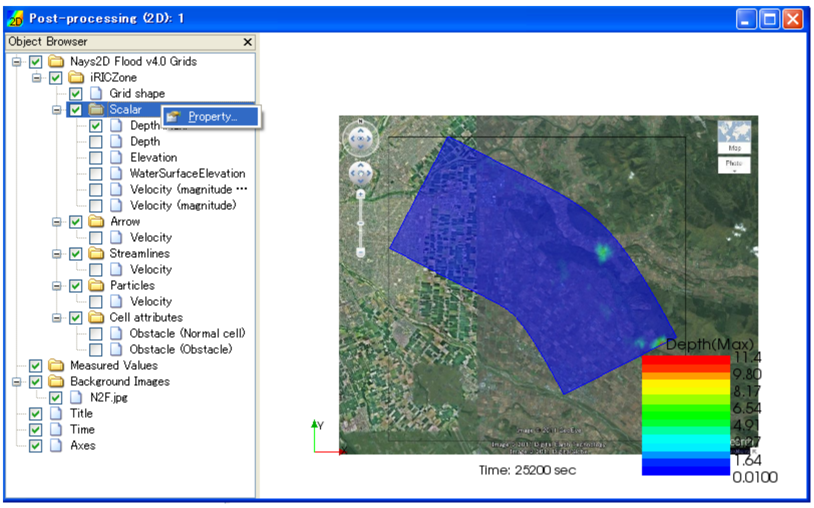
➢ In the Object Browser, select [Nays2D Flood Grids] - [iRICZone] - [Contour] - [Depth (Max)].

A contour map of water depth will open.

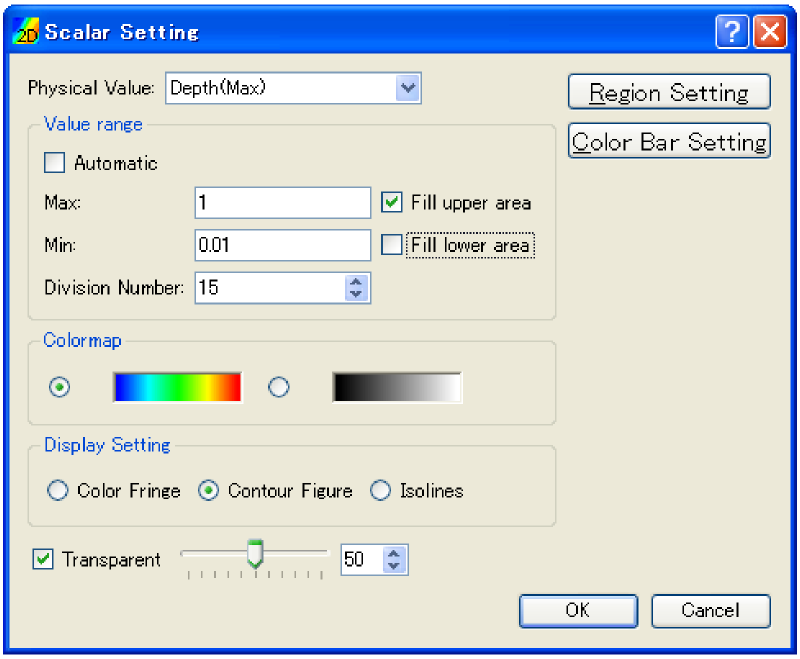


➢ In the Object Browser, select [Nays2D Flood Grids] - [iRIC Zone] - [Contour]. Right-click on [Contour] to select [Property].

The [Contour Setting] window will open.



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

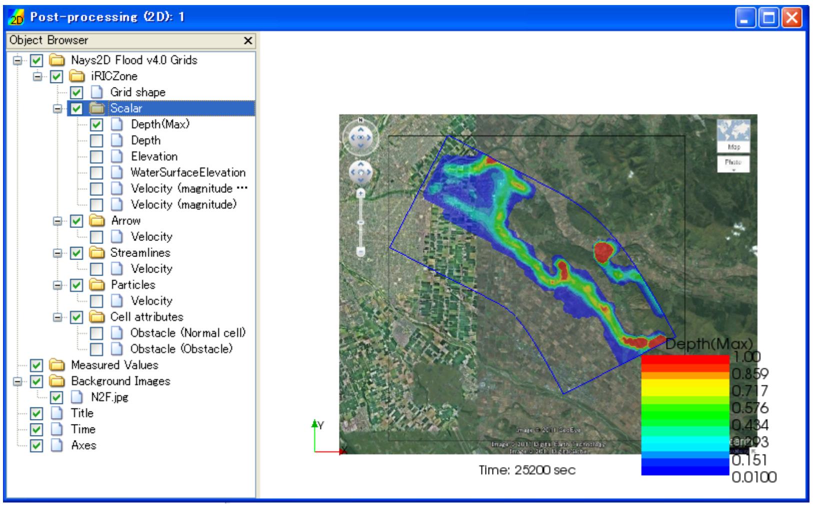


- Value range:
  Remove ☑ from Automatic.
  Max: 1
  Min: 0.01
  Remove ☑ from Fill lower area.
- Color map: Color map
  *no change
- Display Setting:
  Contour Figure
  Division Number:15

Transparent: No change

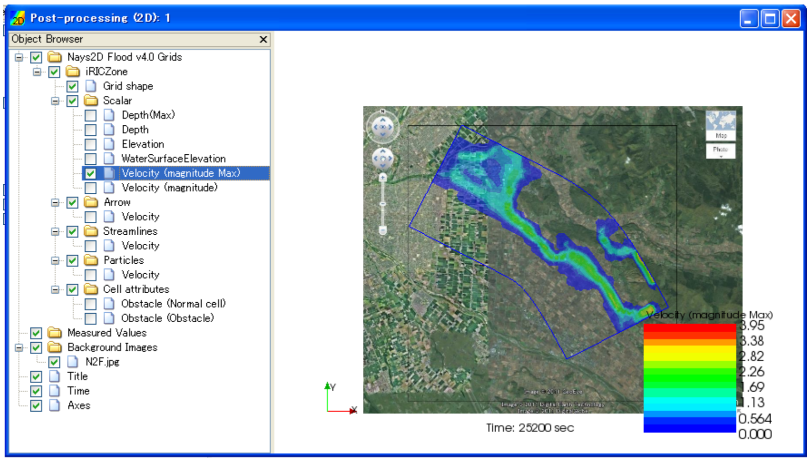
- Region Setting: No change
- Color Bar Setting: No change

The contour map is easier to see now.



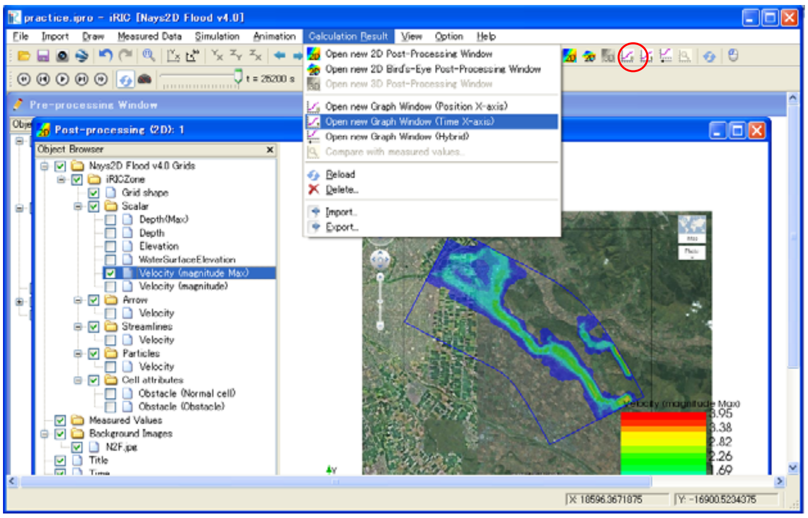
4. Visualizing the max. flow velocity

➢ In the Object Browser, select [Nays2D Flood Grids] - [iRIC Zone] - [Vector] - [Velocity (magnitude Max)].

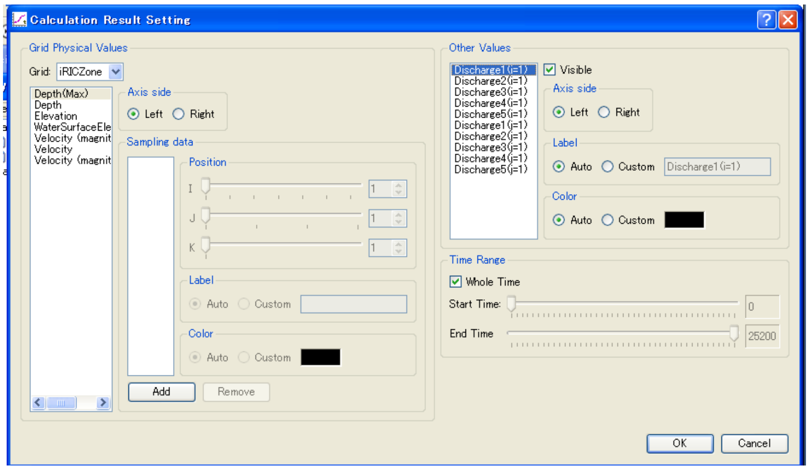


5. Visualizing the inflow discharge hydrograph

➢ Open [New Graph Window (Time X-axis)].



➢ Select [Calculation Result Setting] - [other values] - [Discharge1(i=1)] and [Discharge2(i=1)] - [display].



Note: Other values

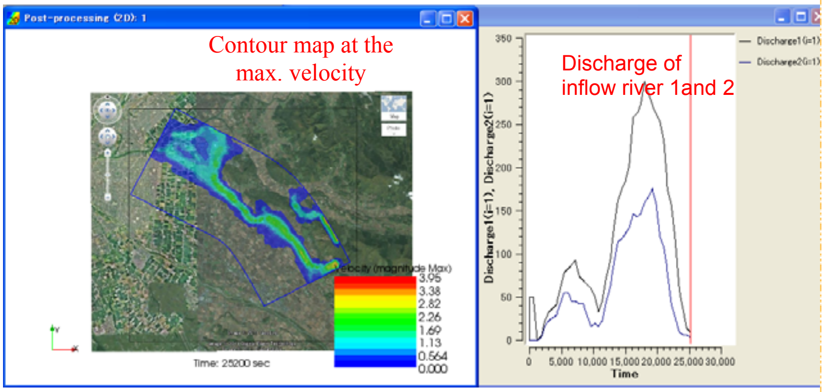
Discharge#(i=1)
The discharge of inflow river # that has been set at the inflow boundary conditions.

Discharge#(j=1)
The discharge of inflow river # that has been set at the boundary conditions for j=1.

Discharge#(j=nj)
The discharge of inflow river # that has been set at the boundary conditions for j=nj.

Unit: m^3/s

➢ It can be mapped with a contour map.



Chapter 3
Examples of Tsunami Runup Analysis Using SRTM data

◆ Objectives

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

◆ Outline

1. Retrieving and converting SRTM data

Download SRTM data available from **http://srtm.csi.cgiar.org/index.asp** for free, and convert it to DRM format (elevation data) adopted by iRIC.
You can covert and set the data range with Quantum GIS (QGIS), a free GIS software application. Other options for converting SRTM data to a format that is readable by iRIC are listed at Tips & Tools under http://i-ric.org/ja/downloads.

2. 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.

3. Setting the calculation conditions

Set a value for tsunami wave height assuming that a large-scale earthquake has occurred. Set various other conditions necessary for simulation.

4. Making a simulation

5. Visualizing the calculation results

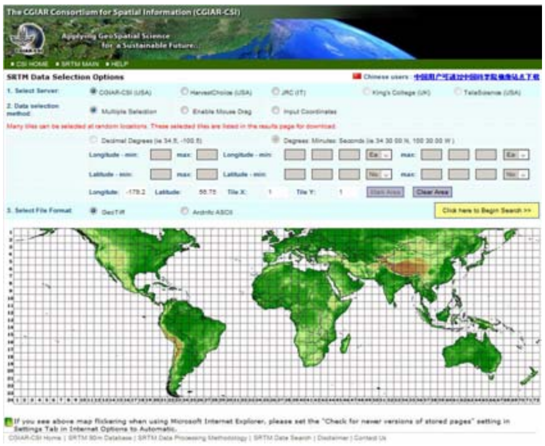
Here, we introduce how to display a water depth contour map and a flow velocity vector map.

1. Retrieving and converting SRTM data

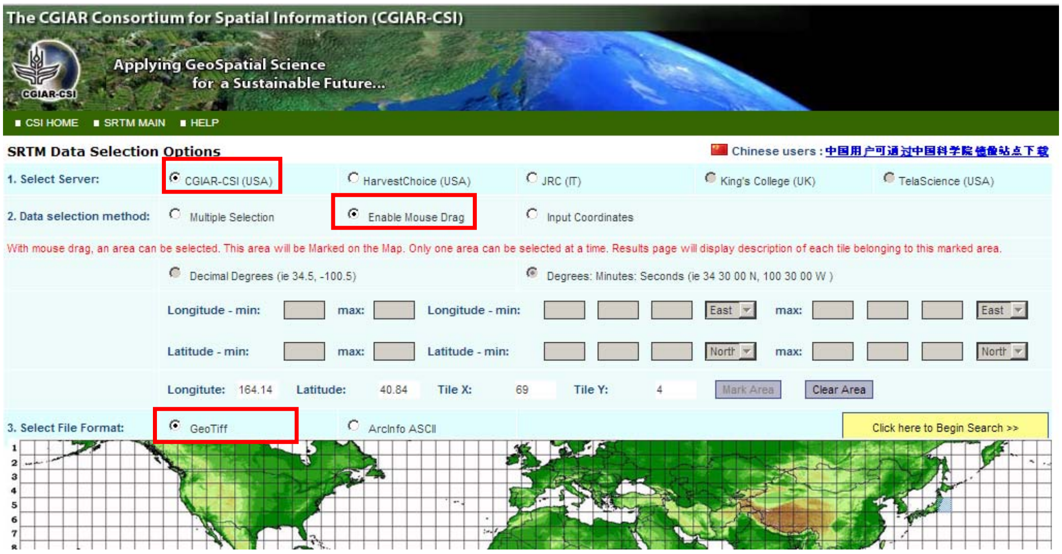
1. Retrieving SRTM data

① Data selection method

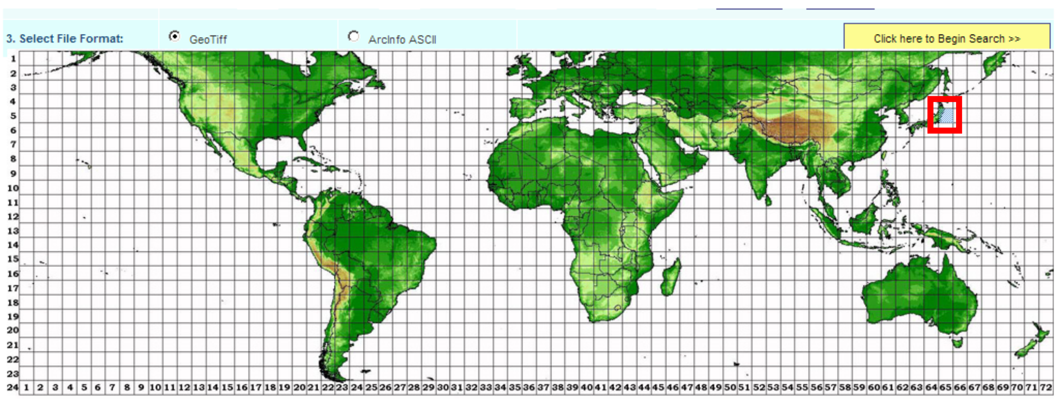
➢ Access the following website, where SRTM data are available.
(http://srtm.csi.cgiar.org/SELECTION/inputCoord.asp)



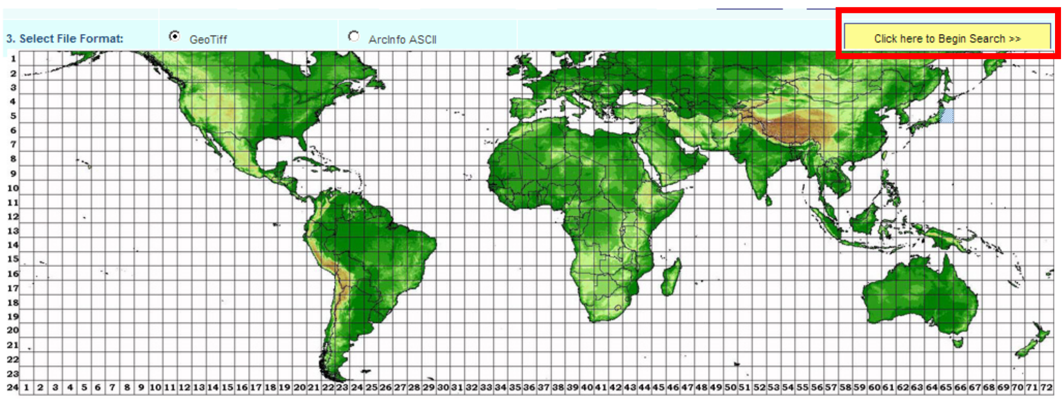
➢ Select [CGIAR-CGI] for [1. Select Server], select [Enable Mouse Drag] for [2. Data selection method], and select [GeoTiff] for [3. Select File Format]. You can also choose [Input Coordinates] for [2. Data selection method] and [ArcInfo ASCII] for [3. Select File Format].



➢ Using your mouse, click on the map cell that contains the area whose data you wish to download.

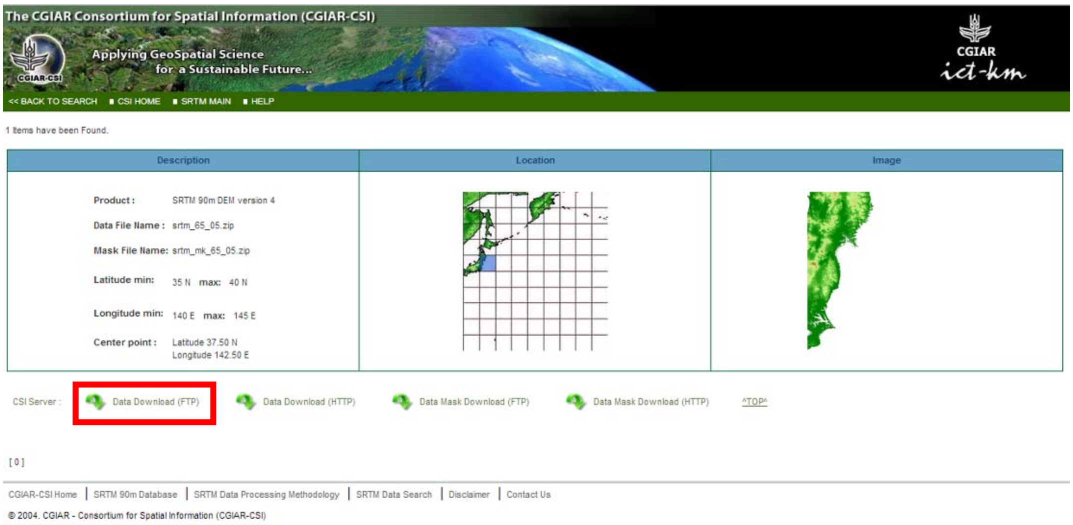


➢ After selecting the cell, click on [Click here to Begin Search] to go to the download page.



② Downloading SRTM data

➢ Select either [Data Download (FTP)] or [Data Download (HTTP)].



➢ The downloaded compressed data consist of the three files below.
This simulation example uses "srtm_65_05.tif", a geographic data.
We recommend that you read "*readme.txt*", which includes the terms and conditions for using SRTM.

srtm_*_*.zip
├ srtm_*_*.tif ：Geographic data (Geotiff)
├ srtm_*_*.tfw ：World file
(Position information of raster data (Latitude and longitude))
├ srtm_*_*.hdr ：
└ readme.txt ：Explanation of SRTM

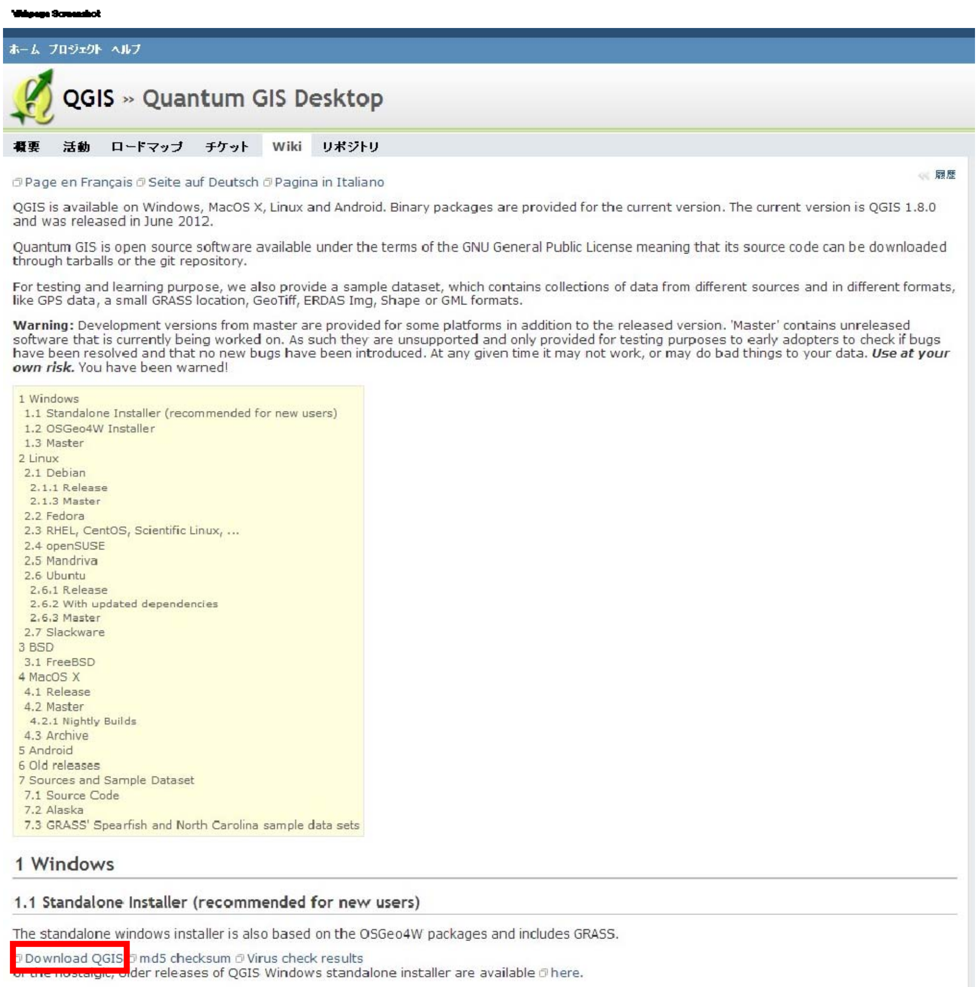
* ：number assigned to the selected area

2. Conversion of SRTM data by QGIS

① Installing QGIS

➢ Download the installer of QGIS (Quantum GIS) from the URL below and install it on your computer.

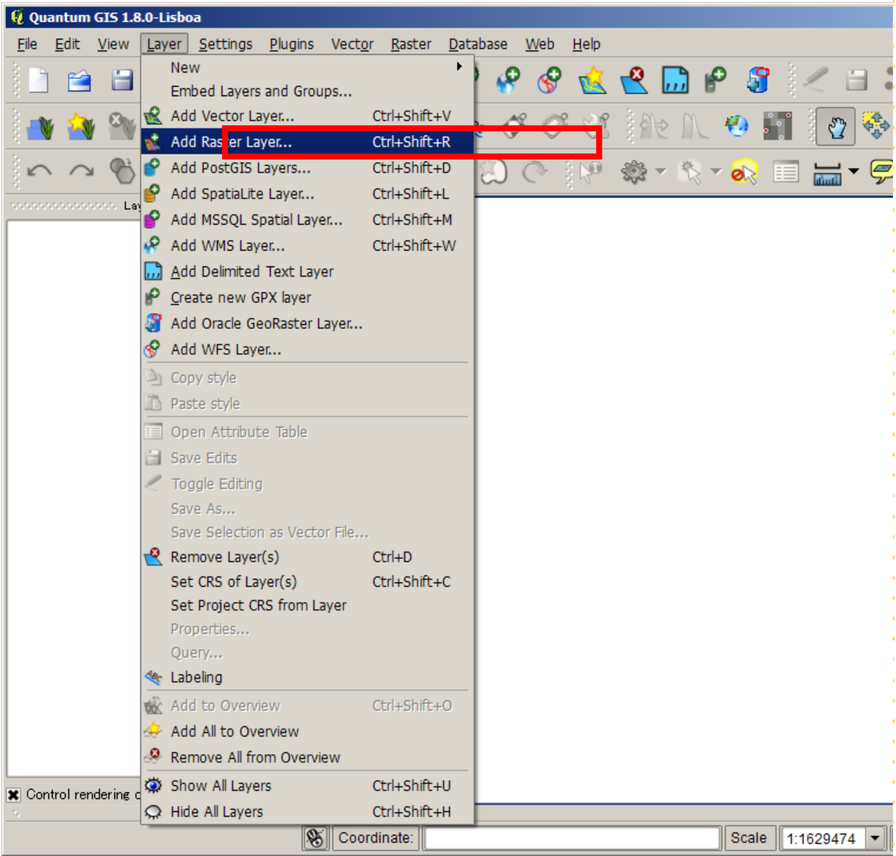
(http://hub.qgis.org/projects/quantum-gis/wiki/Download)

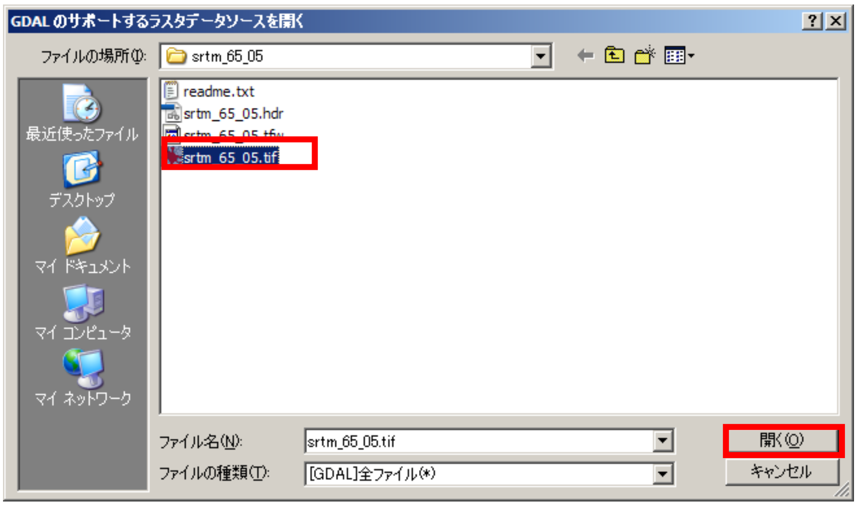


② Displaying SRTM data (QGIS)

➤ Launch QGIS and click on [Layer] - [Add Raster Layer] on the menu bar.

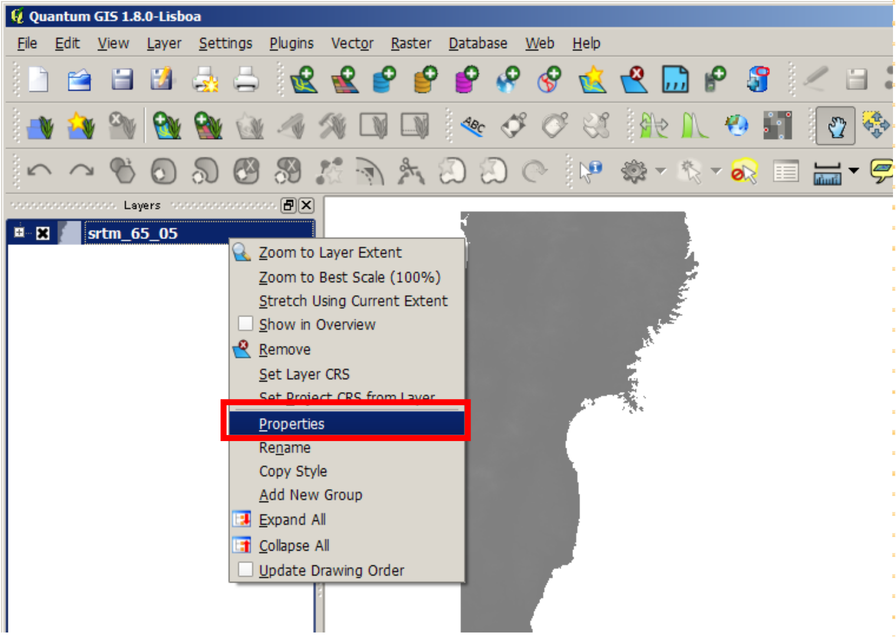
➤ Select downloaded [srtm_65_05.tif] data and click on [Open].



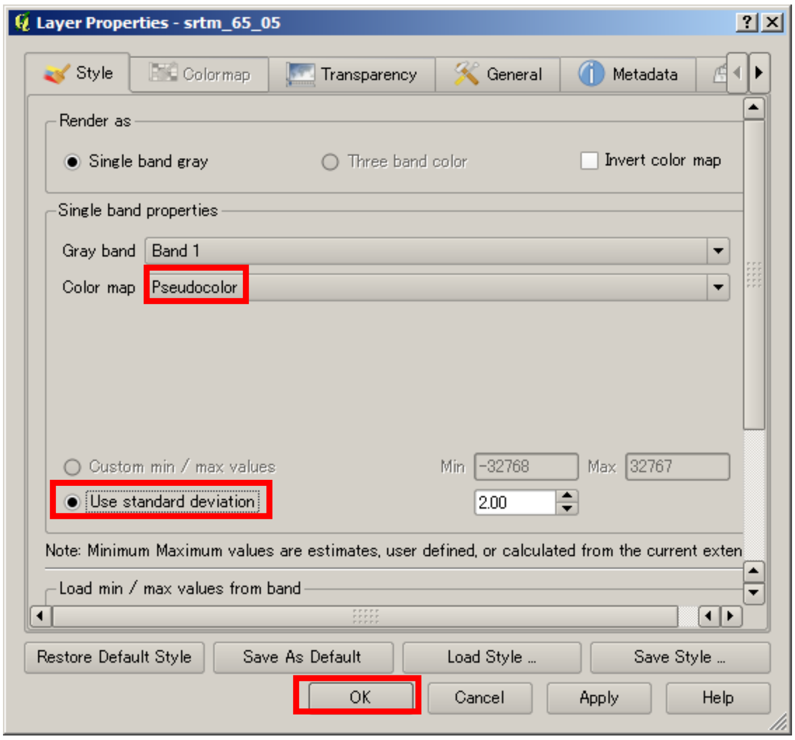


③ Changing the color (QGIS)

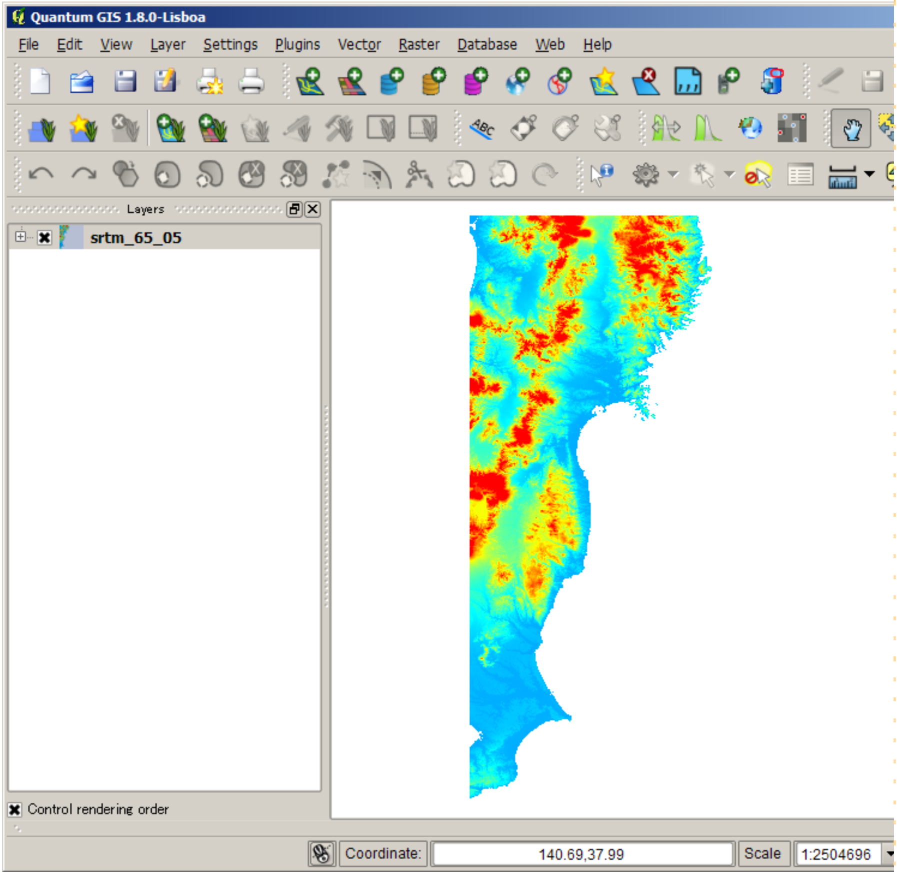
➢ Select [srtm_65_05] in the left window (layer) and right click to select [Properties].



➢ Under the [Style] tab, select [Pseudocolor] for [Color map]. Check [Use standard deviation] and click on [OK].



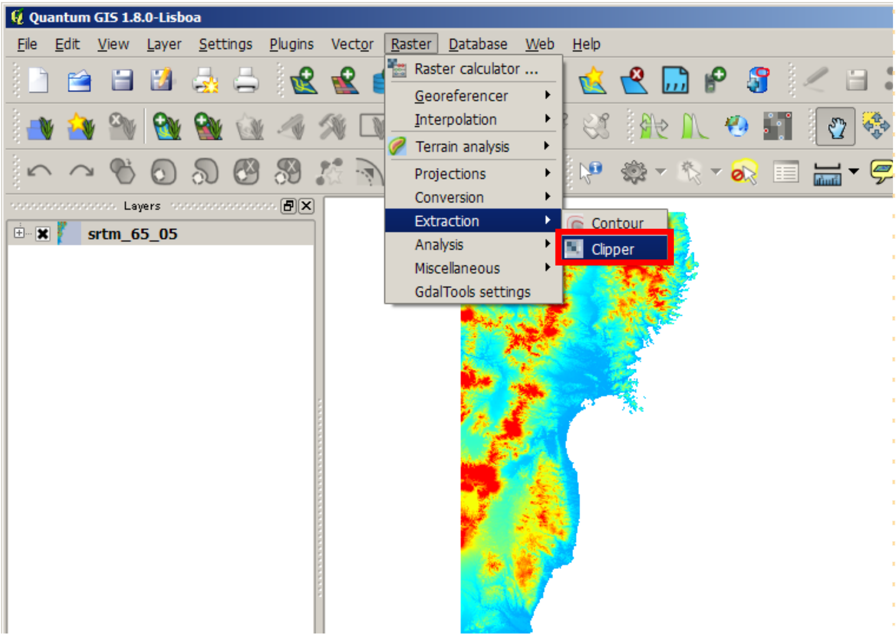
The SRTM data are now displayed in colors.



④ Selecting data you need (QGIS)

Because obtained SRTM data are very large and may cause instability of iRIC, specify the extent of data you need.

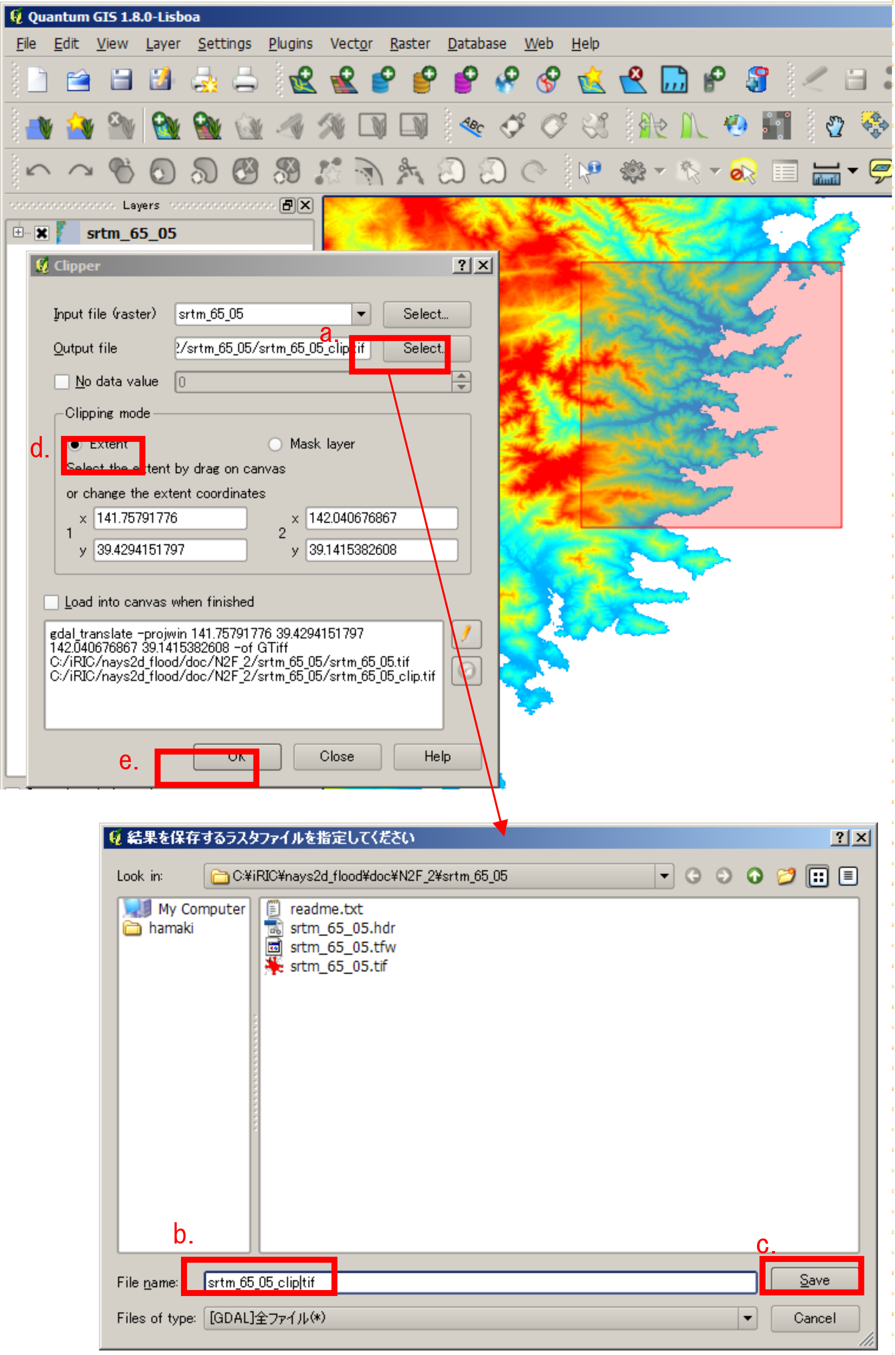
➢ On the menu bar, select [Raster] - [Extraction] - [Clipper].



➢ Open [¥SampleData¥N2F_2] from [Output file], enter [srtm_65_05_clip.tif] and click on [Save].

➢ Select [Extent] for [Clipping mode]. Specify the extent by dragging as shown in the figure below.

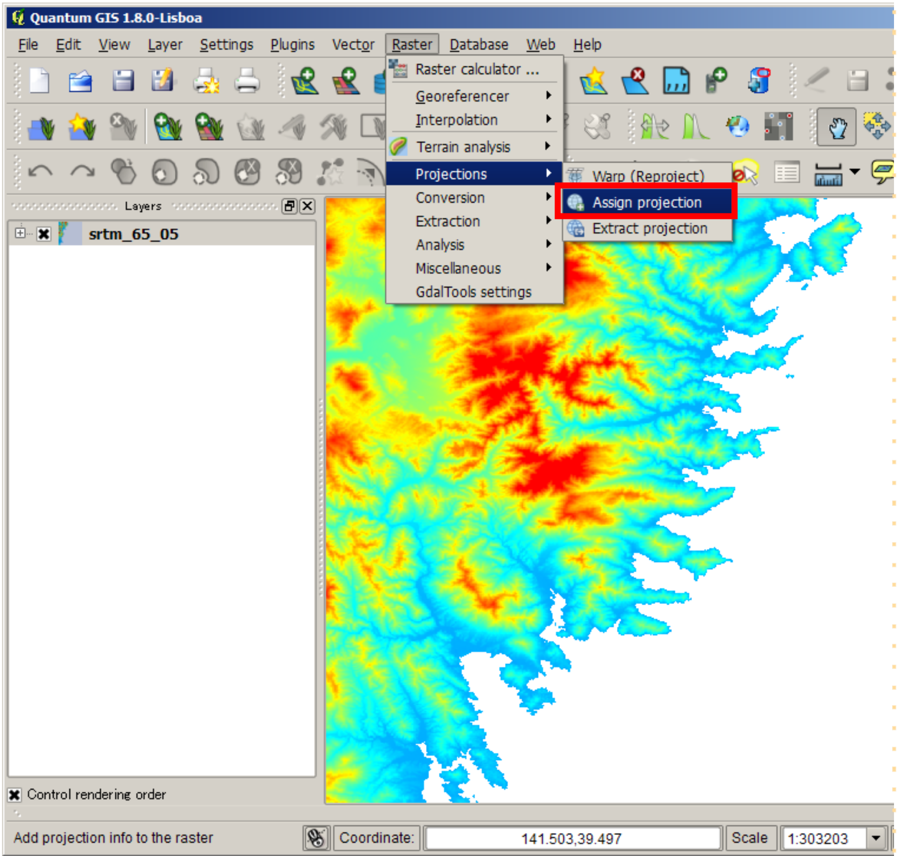
➢ Click on [OK].



⑤ Covering the coordinates system (QGIS)

The coordinates of retrieved SRTM data are the latitude and the longitude. Convert the data to the rectangular coordinate system for processing by iRIC. (In this case, the X system of plane rectangular coordinate system is used.)

➢ On the menu bar, select [Raster] - [Projections] - [Assign projection].

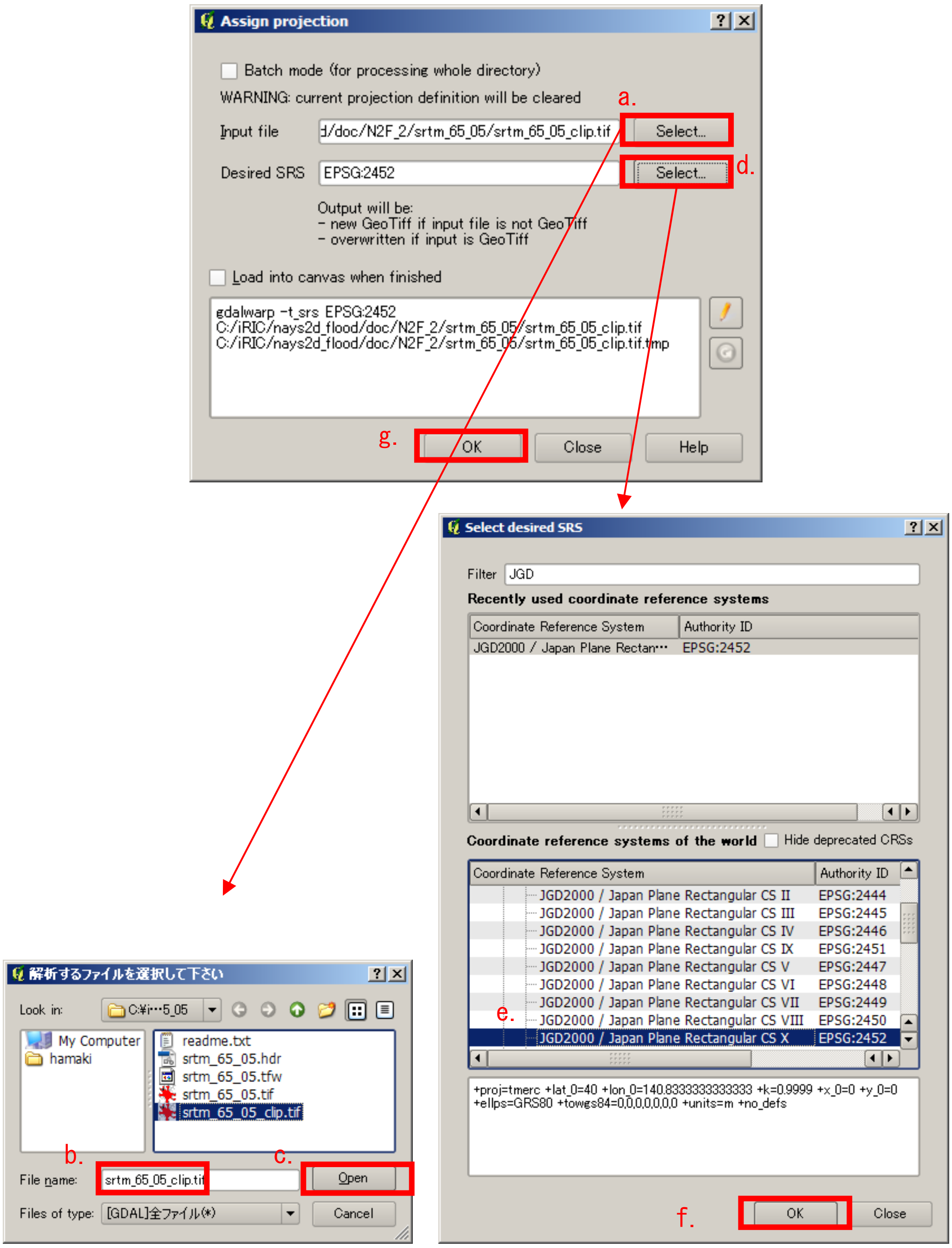


➤ For [Input file], select [srtm_65_05_clip.tif] and click on [Open].

➤ For [Desired SRS], select [JGD2000 / Japan Plane Rectangular CS X(EPSG:2452)] and click on [OK].

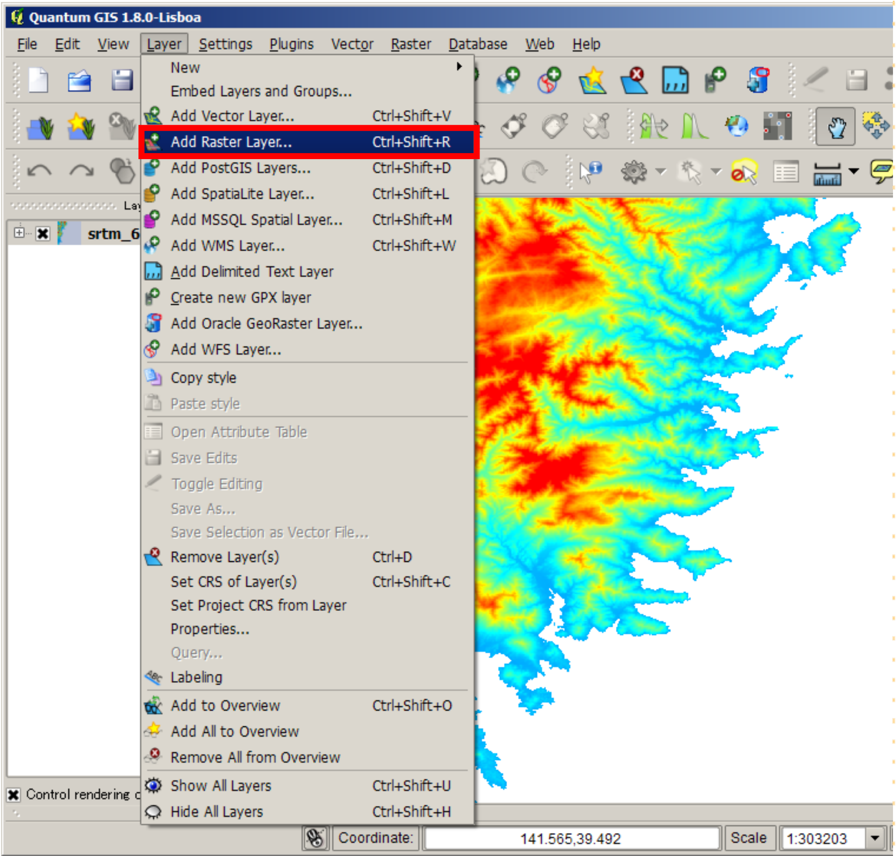
➤ Finally click on [OK] of the [Assign projection] window.

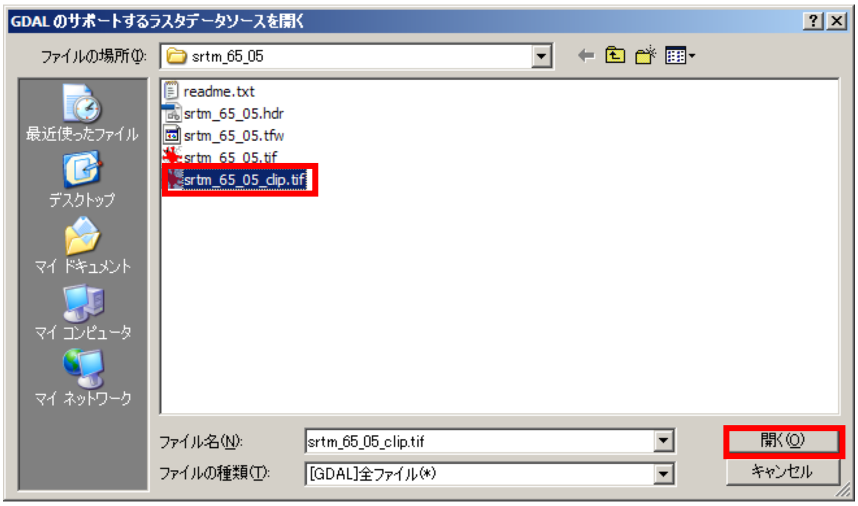
[srtm_65_05_clip.tif] which has been assigned for [Input file] is now overwritten.



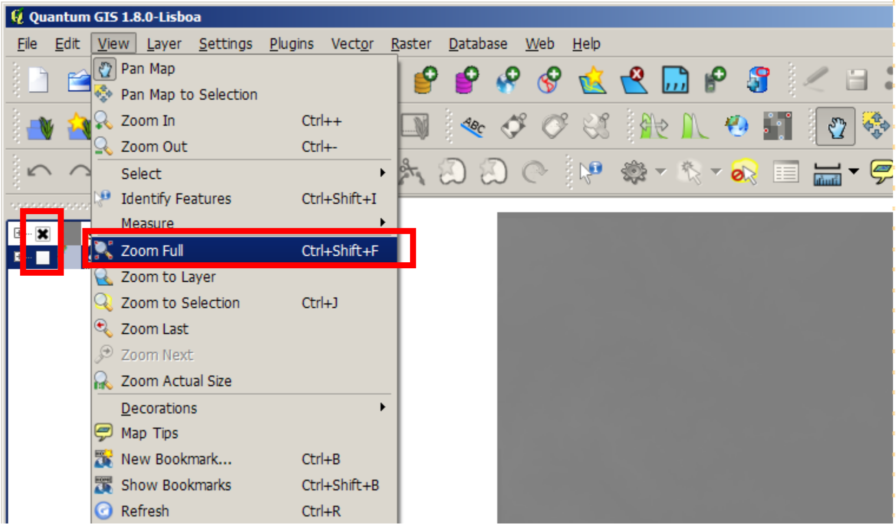
⑥ Converting SRTM data (QGIS)

- On the menu bar, select [Layer] - [Add Raster Layer].
- Select the coordinate-converted [srtm_65_05_clip.tif] and click on [Open].

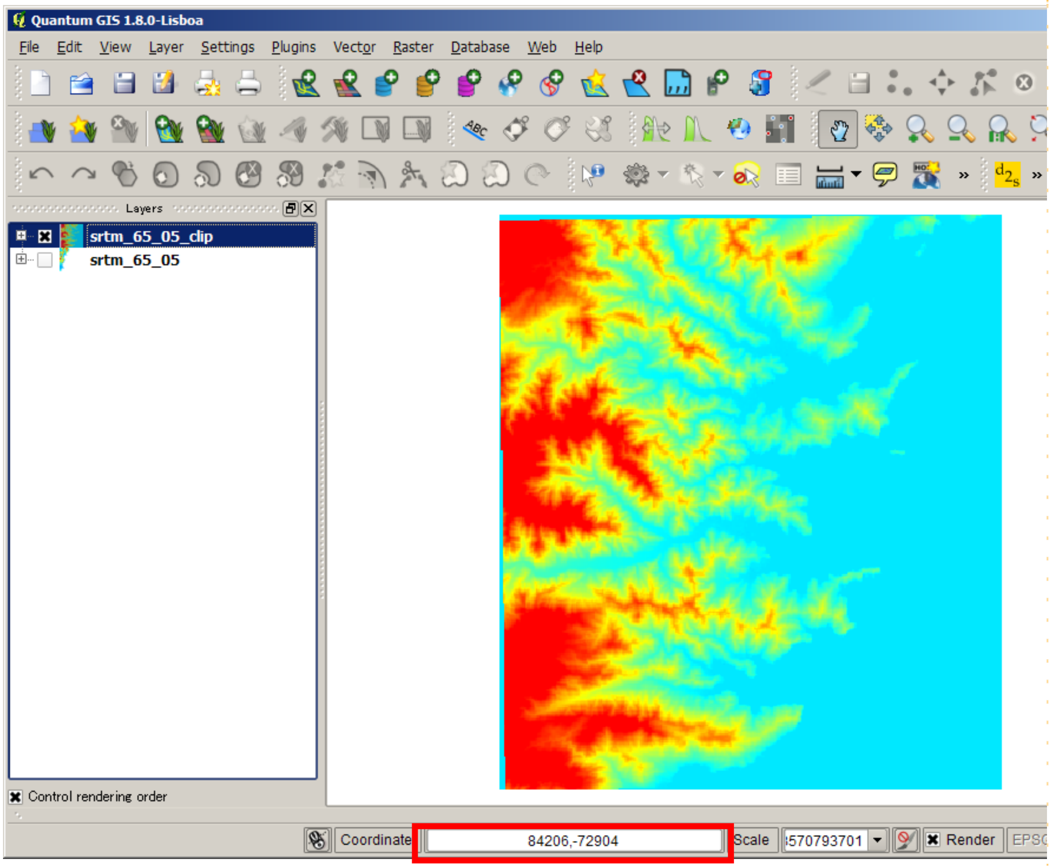




➢ While making only [srtm_65_05_clip] effective in the left window by clicking the left "X", select [View] - [Zoom Full] on the menu bar.



To display [srtm_65_05_clip] in colors, apply the same method as was used by "⑥ Changing the color", and check whether data of the specified extent have been extracted. Also, check whether the values on the coordinates are expressed not as latitude/longitude coordinates but as X/Y coordinates.

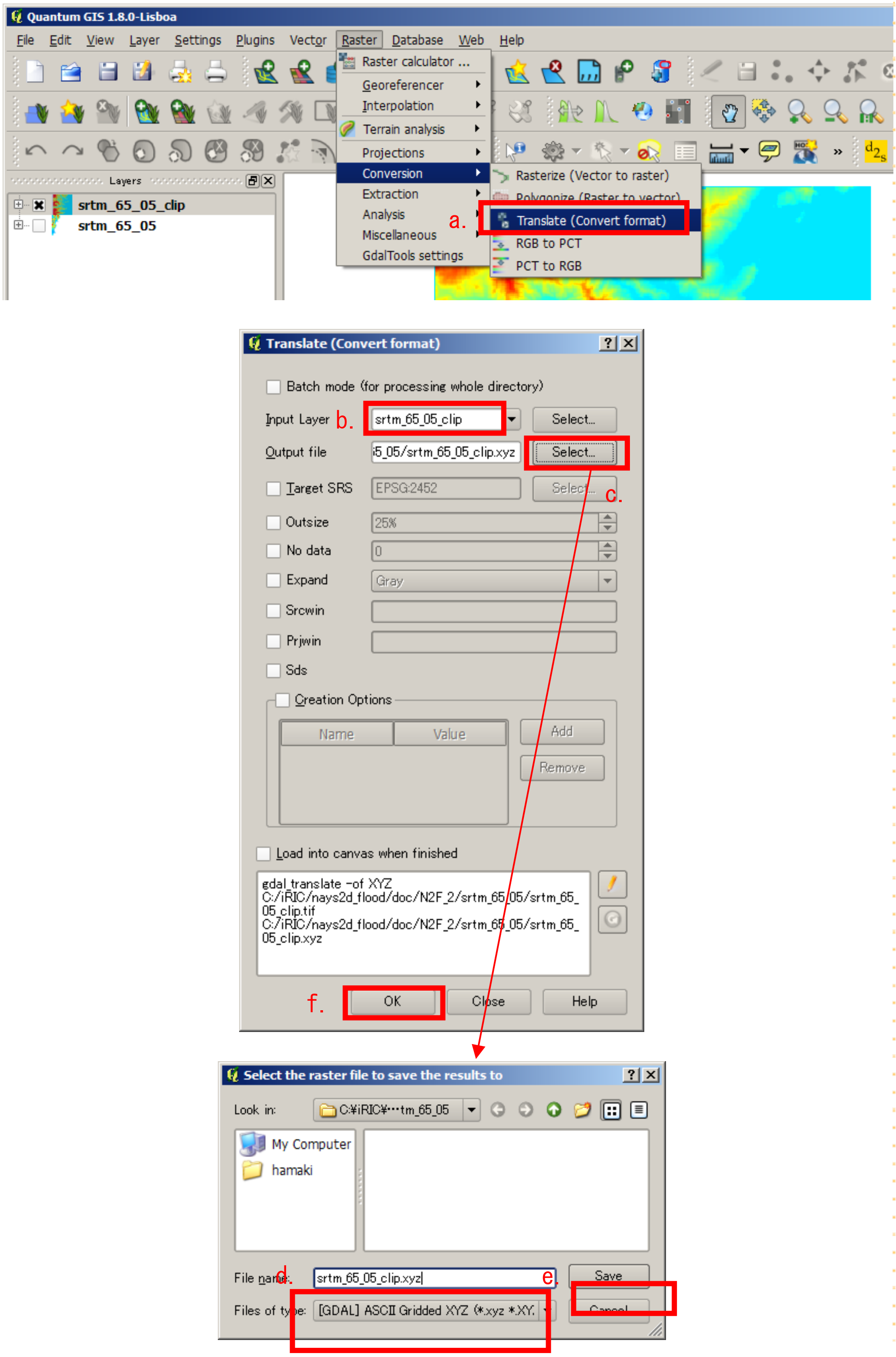


➢ On the menu bar, select [Raster] - [Conversion] - [Translate (Convert format)].

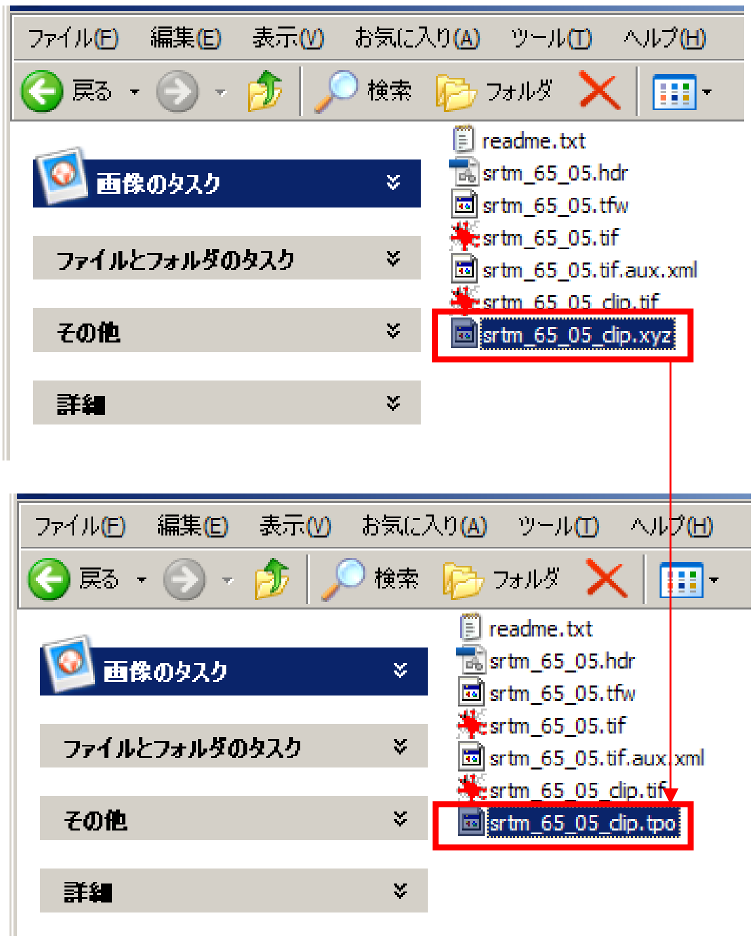
➢ Select [srtm_65_05_clip] for [Input layer].

➢ Input [srtm_65_05_clip.xyz] in the [Output file] box. Select [ASCII Gridded XYZ] for [Files of type] in the [Select the raster file to save the results to] window and click on [Save].

➢ Finally click on [OK] of the [Translate (Convert format)] window.

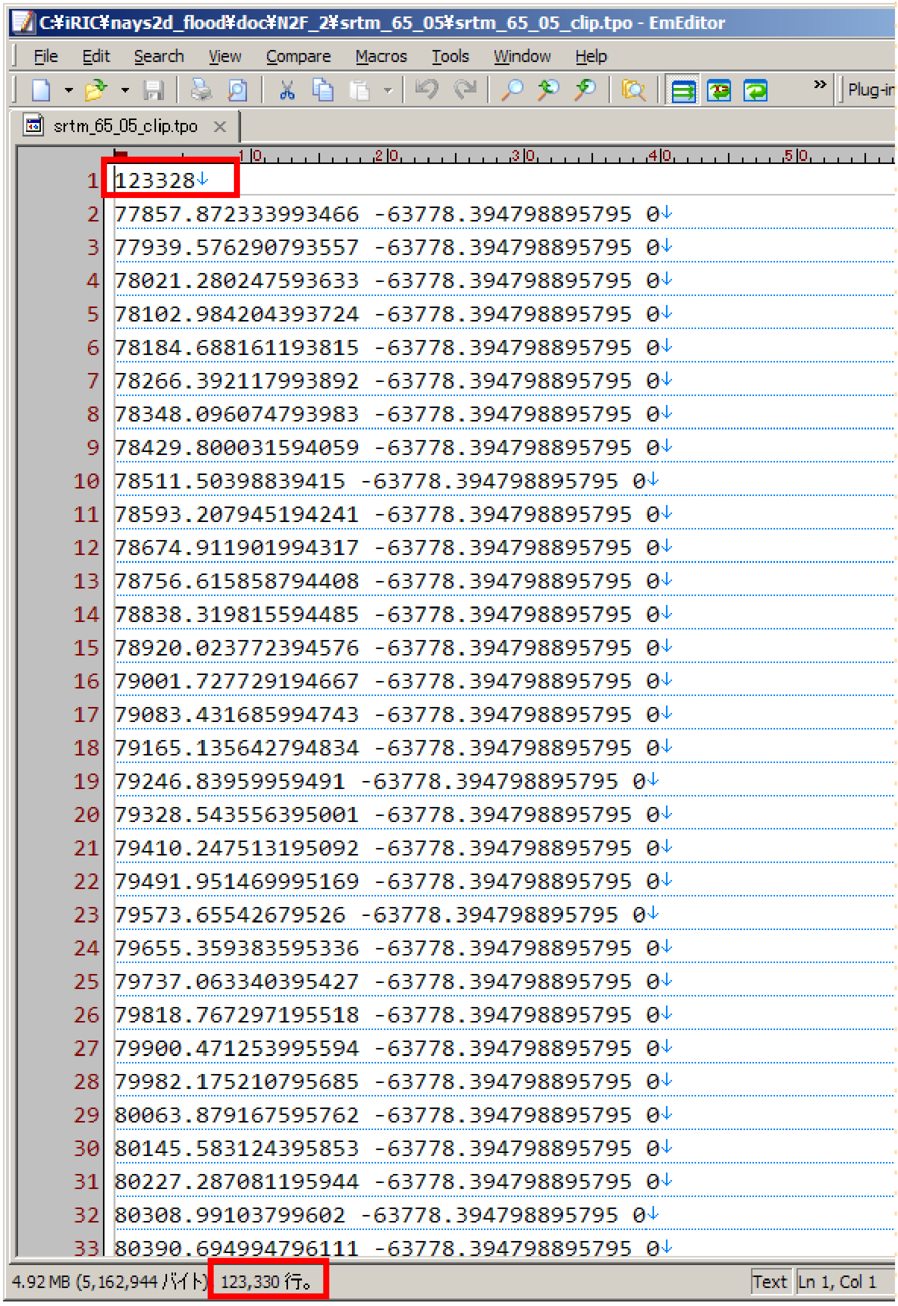


➢ Select the outputted [srtm_65_05_clip.xyz] file and change its file extension such that the file name is [srtm_65_05_clip.tpo].



➢ Open [srtm_65_05_clip.tpo] with a text editor to count the number of lines of data.

➢ At the top of the data, add the number of the data lines and save the data.

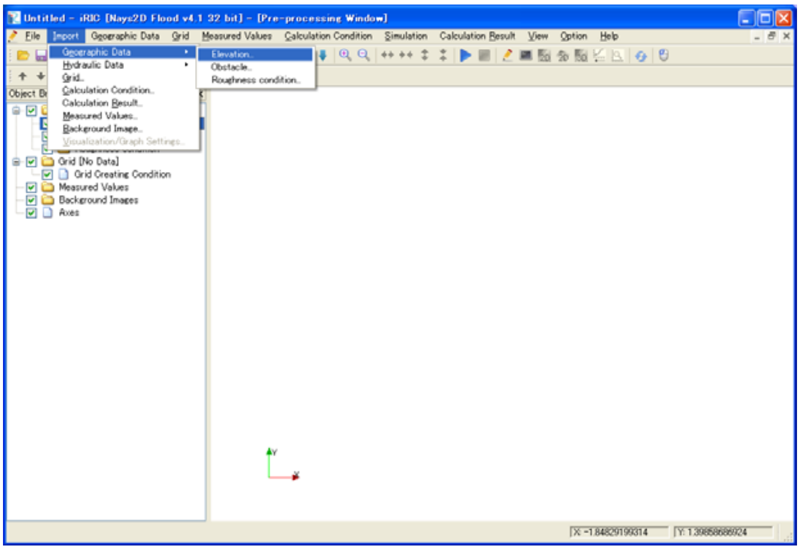


2. Creating the calculation grid

1. Importing geographic data

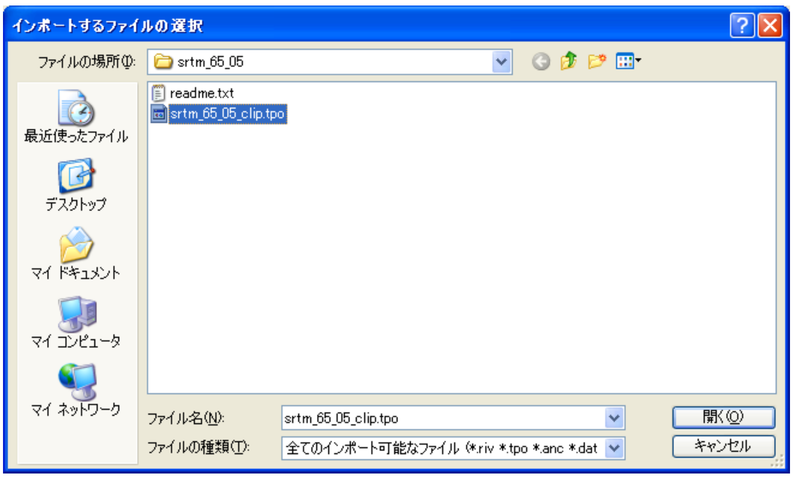
① Importing geographic data

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



➢ Select [srtm_65_05_clip.tpo], which was prepared in the previous section, and click on [Open].

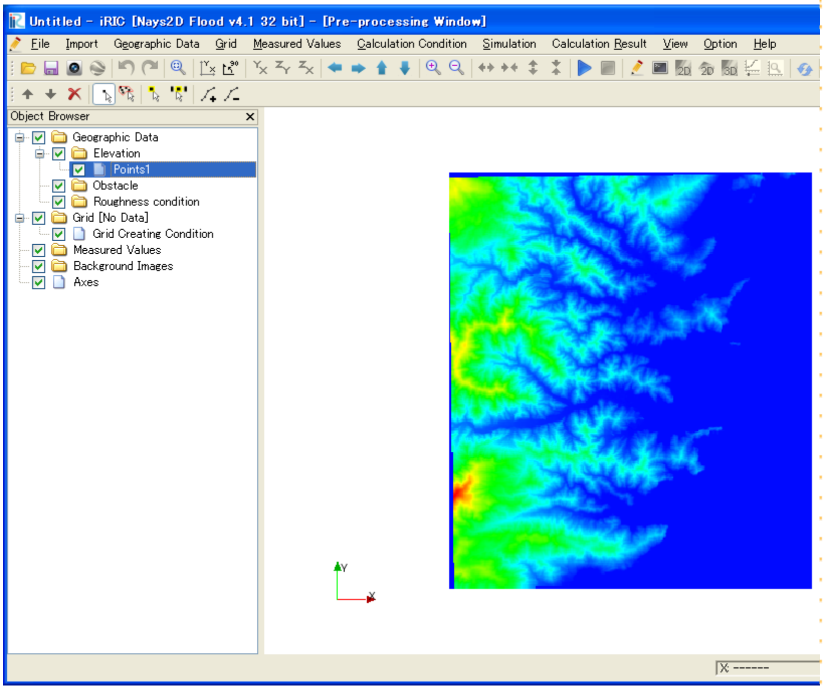
➢ Or, open the [¥SampleData¥N2F_2] folder to select [srtm_65_05_clip.tpo] and click on [Open] to import the data.



➢ Select [1] for the value of [Filter] in the [Filtering Setting] window.

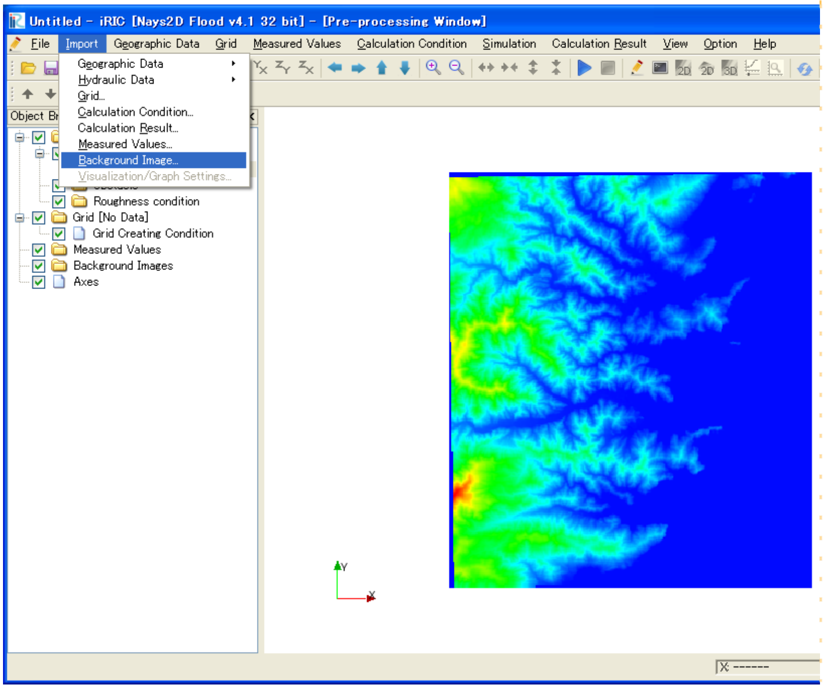
In the event that data processing is slow due to the large volume of data, set a larger value for [Filter] to reduce the number of data.

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



② Importing a background image

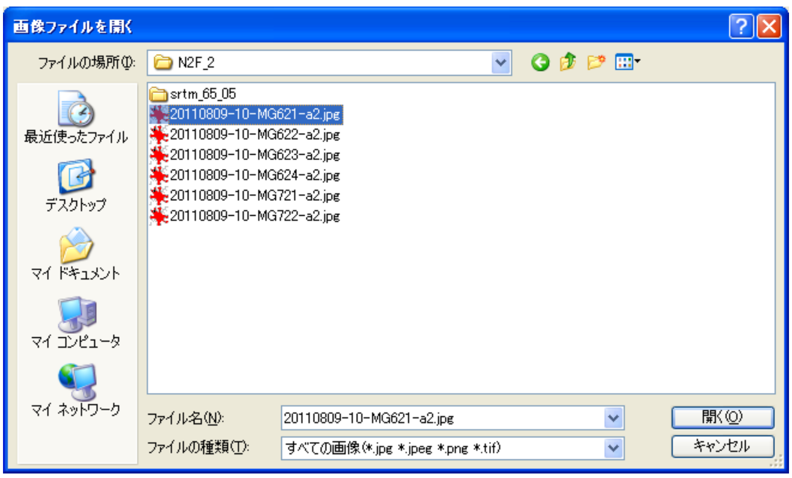
➢ On the menu bar, select [Import] - [Background Image].



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 [20110809-10-MG621-a2.jpg] in the [¥SampleData¥N2F_2] folder.



Orthoimage

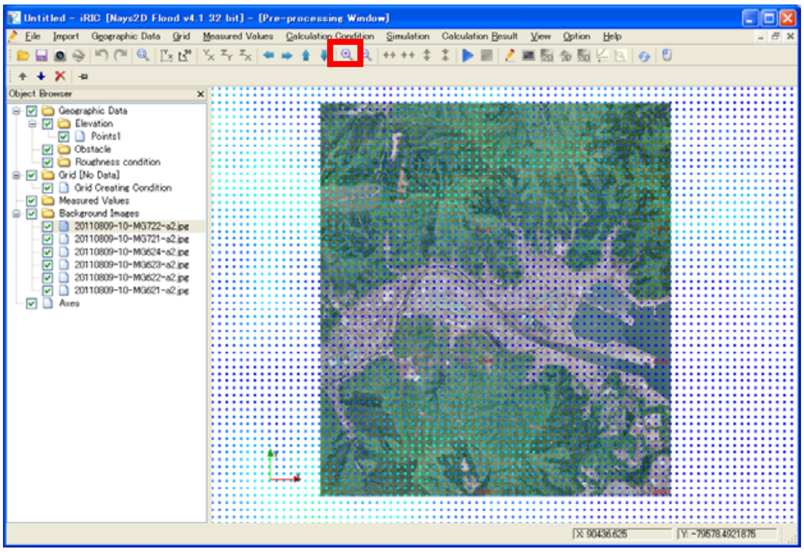
An orthoimage is an aerial photograph that has been geometrically corrected to eliminate all distortion. In addition, correct positioning information is compiled with the image data. Use of orthoimage enables geographic data to be overwrapped with various types of geospatial information data. Compared with an aerial photograph, it has a wider range of use.

If you store the world file (such as .jpw) in the same folder as the data file of an orthoimage, the positioning information will be automatically corrected.

In this tutorial, the world file and the orthoimage have been downloaded from the URL below.
http://saigai.gsi.go.jp/h23taiheiyo-zort/index.html

➢ By consecutively opening the other jpg fies in the [¥SampleData¥N2F_2] folder, the positions of background images are automatically corrected.

➢ Enlarge the display image and check the background images.



Tips:
When using images obtained from Google Earth or the like, y need to manually correct the positioning information of such images.
With this sample, because ther is a world file that includes positioning information of the orthoimage, you can automatically position the background images.

In the event that the PC specifications are insufficient, you may not be able to process large numbers of elevation data.
To speed up processing, remove geographic data in the areas that are not needed.

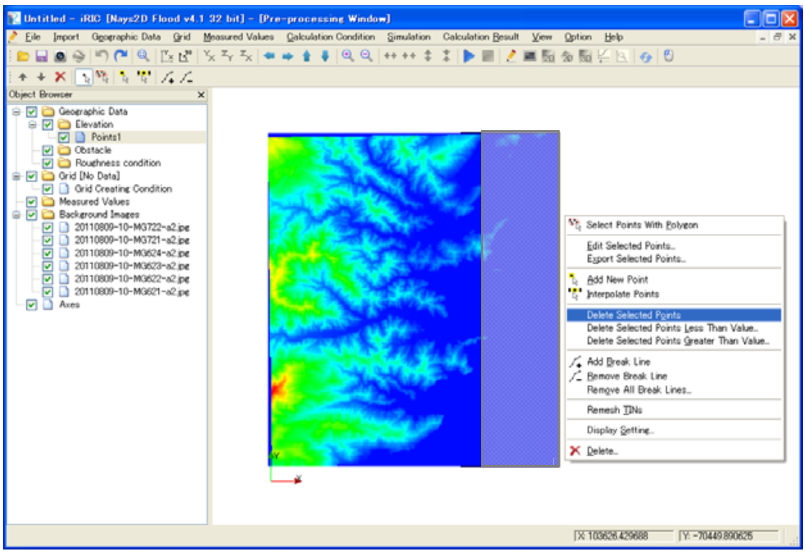
➢ Click on [Fit] to display the entire data.

➢ Select [Object Browser] - [Geographic data 1] and keep the data ([Geographic data 1]) selected.

➢ Select the area you wish to delete by dragging and right clicking. (The gray area in the figure below. An area slightly larger than the background image must remain).

➢ To delete the unnecessary areas by clicking [Delete selected points].

Repeat the above procedure. The area slightly larger than the background image will be left.

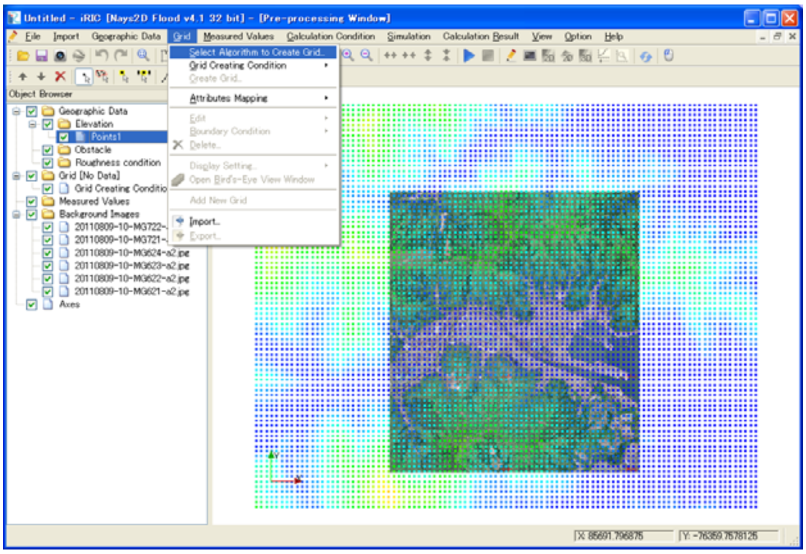


Tips:
In the event that the PC specifications are insufficient, you may not be able to proces large numbers of elevation dat
To speed up processing, remove geographic data in the areas that are not needed.
Such removal is not necessa when your PC has specificatio high enough to maintain stable operation even when large numbers of data are processe

2. Selecting an algorithm for creating the grid

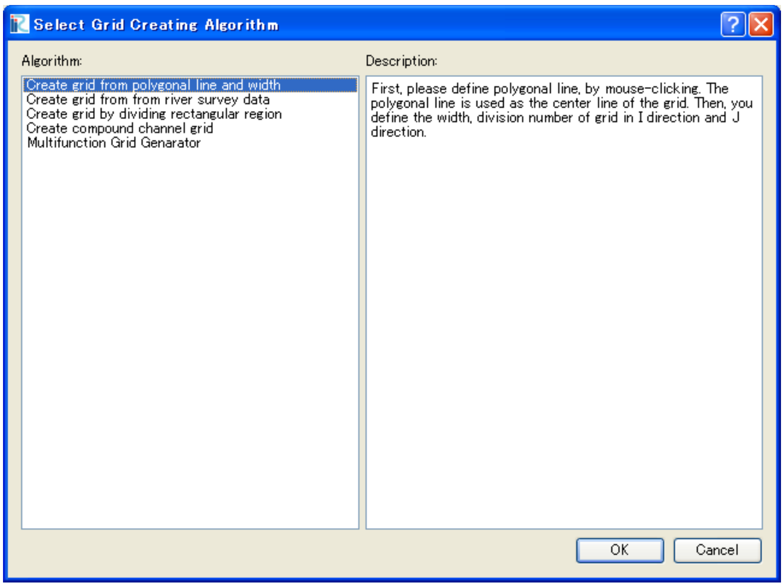
➢ On the menu bar, select [Grid] - [Select algorithm to create grid].

The [Select Algorithm to Create Grid] window will be displayed.



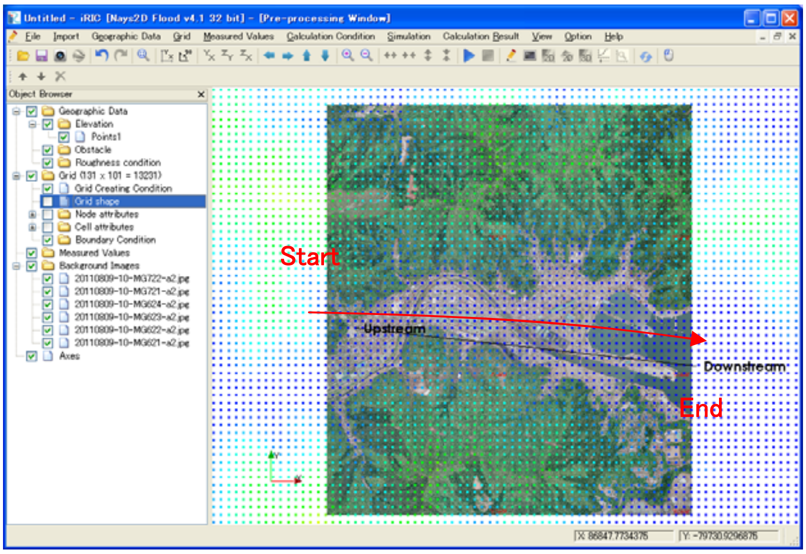
➢ Select [Create grid from polygonal line and width] from the list under the [Select Grid Creating Algorithm] window, and click on [OK].

Primary, Nays2D Flood generates a grid from polygonal lines and widths.



3. Creating a grid

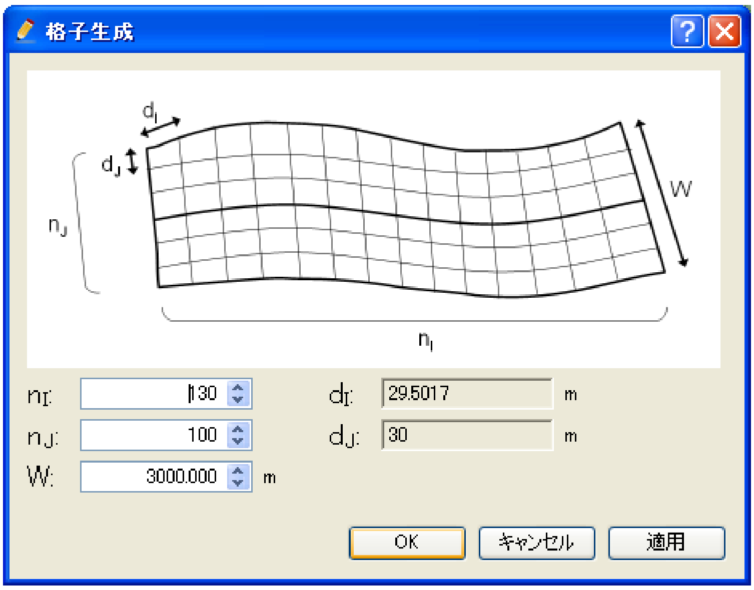
➢ Right click on several points through which the grid centerline passes and press the Enter key on your keyboard to draw a centerline.



Setting the gird 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.

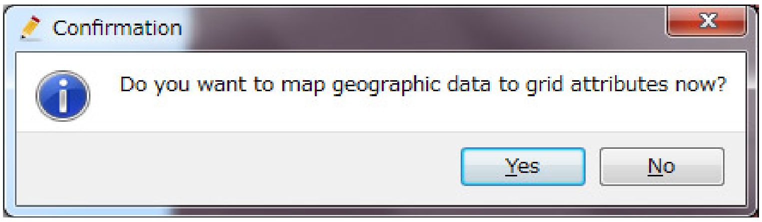
➢ In the [Grid Creation] window, make the following settings and click on [OK].



- [nj] (Number of divisions in the longitudinal direction): 100
- [ni] (Number of divisions in the transverse direction): 130
- [W] (Grid width): 3000m

Note: In this example, grid cell interv di, dj are set at around 30 m.

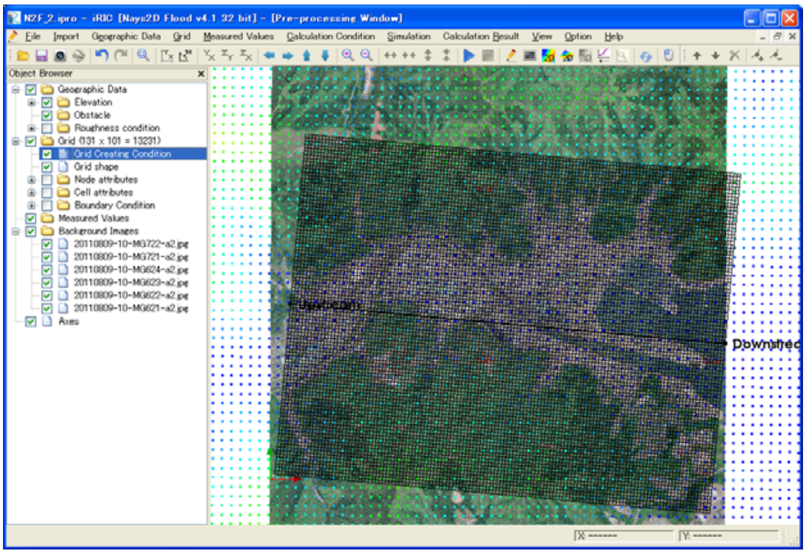
➢ Click on [Yes] in the [Confirmation] window.



Mapping geographic data

Elevation data are applied to the grid.

A grid will be created.



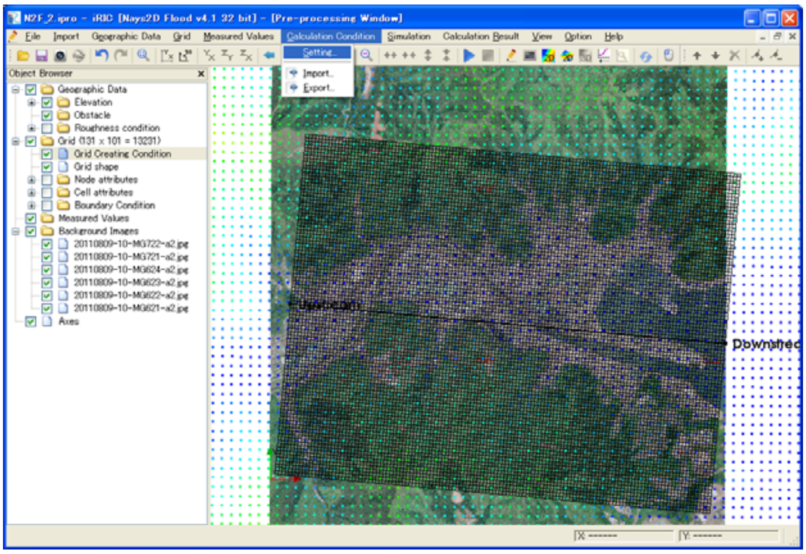
Adjusting the calculation grid

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

3. Setting the calculation conditions

1. Open [Calculation Condition]

➢ On the menu bar, select [Calculation Condition] - [Setting].
The [Calculation Condition] window will open.

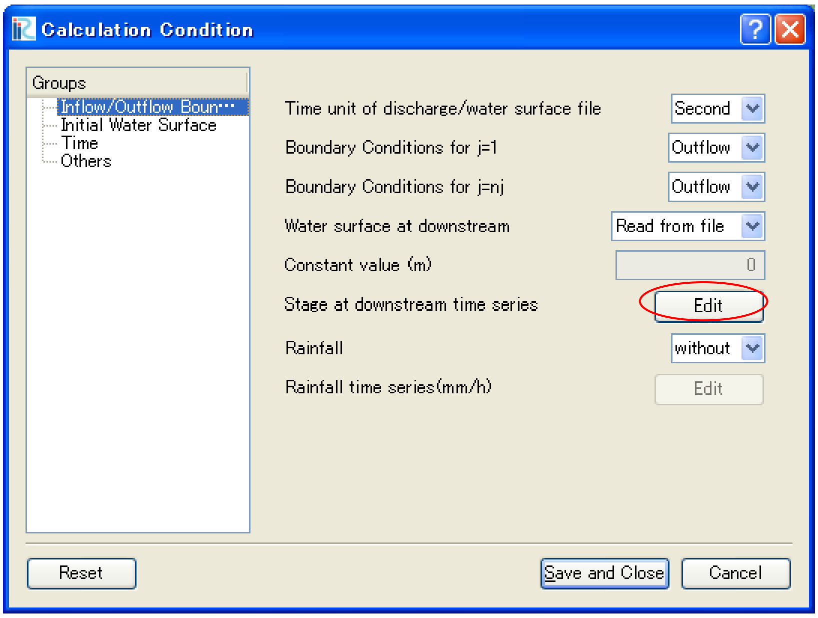


2. Setting the inflow boundary conditions

➢ 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.

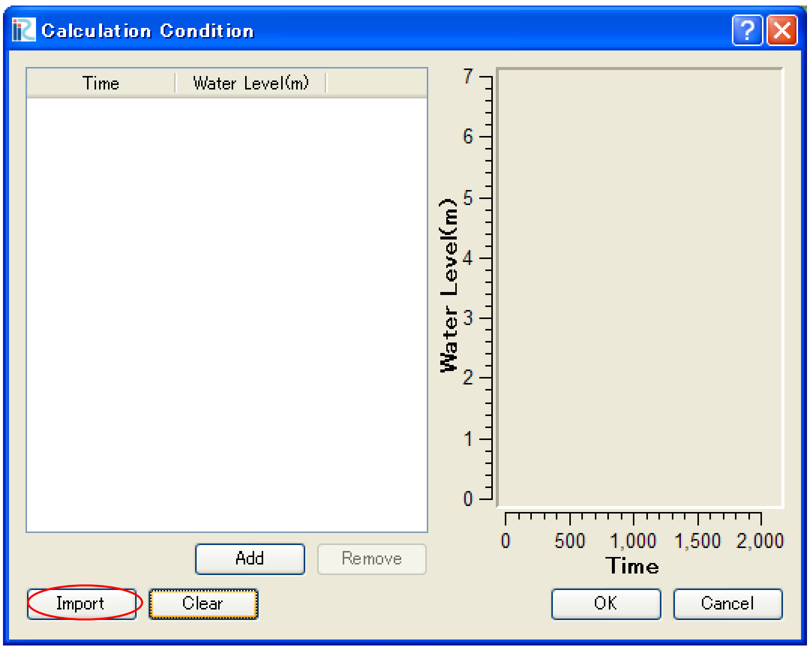


- [Time unit of discharge/water surface file]: Second
- [Boundary Conditions for j=1]: Inflow
- [Boundary Conditions for j=nj]: Outflow
- [Water surface at downstream]: read from the file
- Rainfall: without

Note 1: Inflow boundary conditions are set at "6. Inflow Settings".
Note 2: When "Water surface elevation at downstream end"is affected by the sea surface level or flood water level of downstream, select [Constant value] or [Read from file].

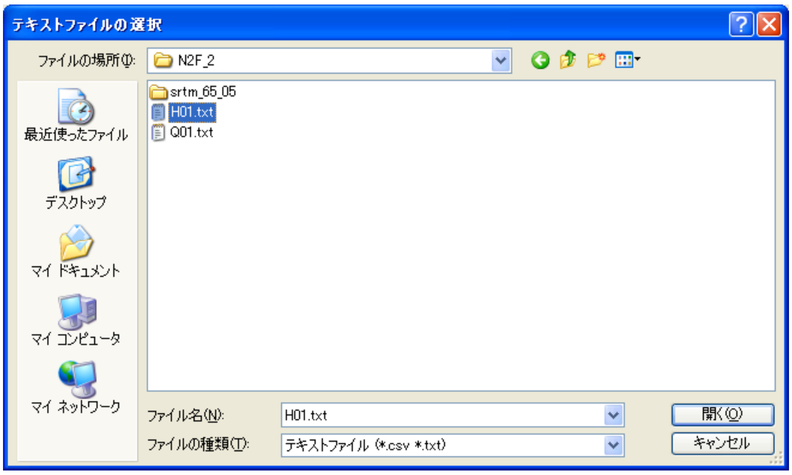
➢ Click on [Import] on the [Calculation condition] window.

The [Select text file] window will be displayed.



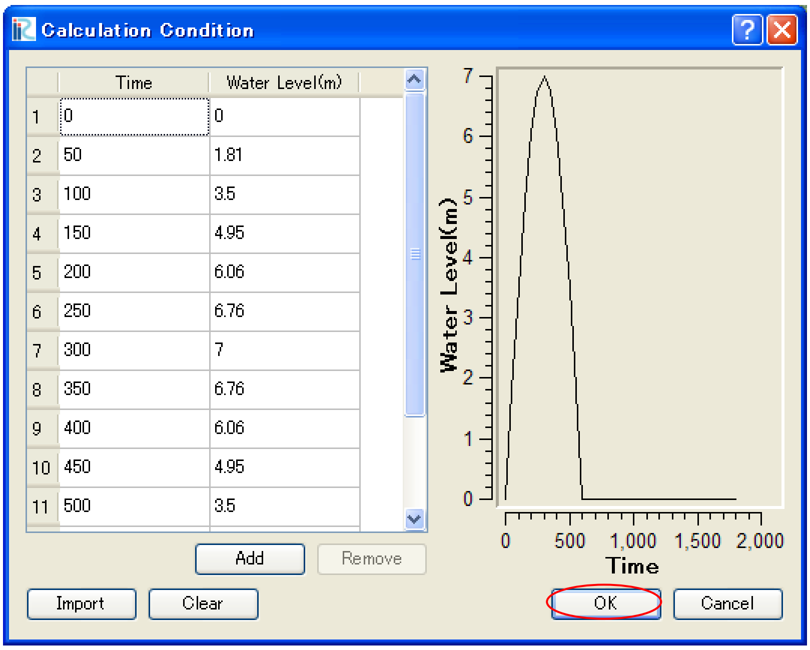
➢ Select [¥SampleData¥N2F_2] - [H01.txt] and click on [Open].

Time series of tsunami wave height data is displayed.



➢ Click on [OK].

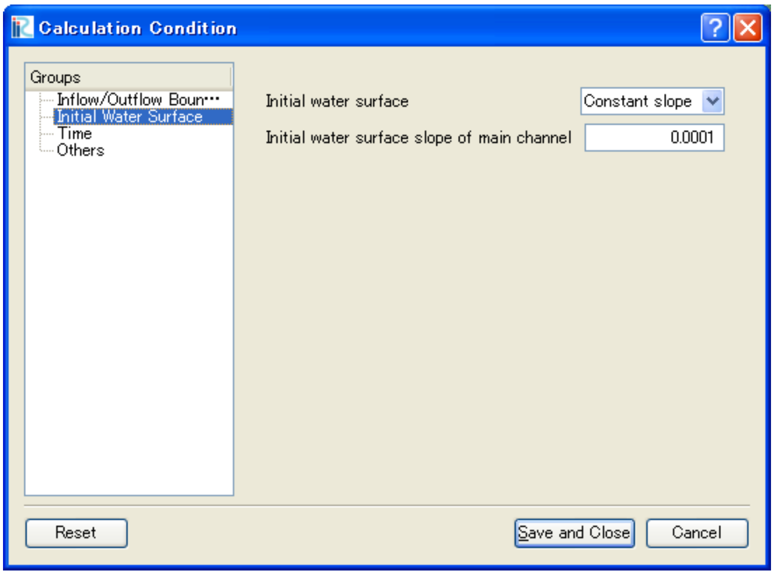
Here, we assume a tsunami whose wave height is 7 m and half-wavelength is 600 sec.



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:

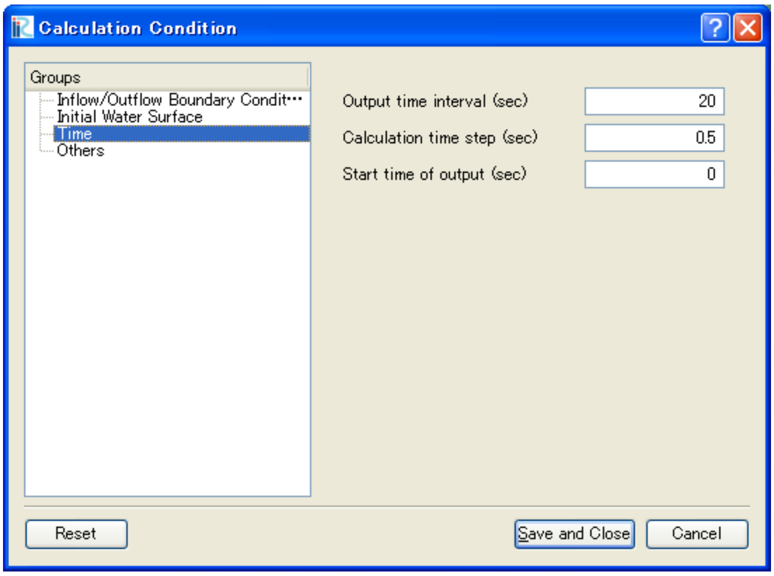


● [Initial water surface]: constant slope
●[Initial water surface slope of main channel]: 0.0001

Note: In order to give a nearly horizontal sea level in downstream area, a very small value is given to [initial water surface slope of main channel].
The first water level data set for [Stage at downstream time series] (0 m in this sample) is regarded as the water level at the downstream end, and the initial water level is set to achieve nearly horizontal sea water level.
At locations where the ground elevation is set higher than the water surface elevation, the water surface elevation will not be set.

4. Setting the time

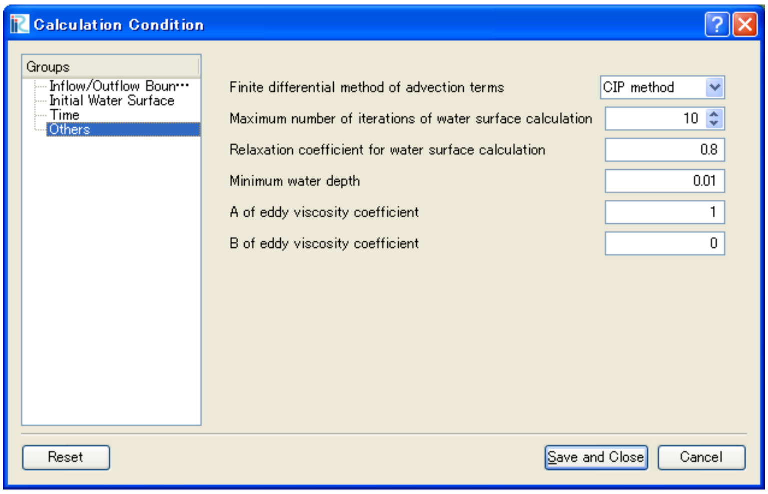
➢ Click on [Time] from the [Group] list to make the following settings:



- [Output time interval (sec)]: 20
- [Calculation time step (sec)]: 0.5
- [Start time of output (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]: 001
- ● [A of eddy viscosity coefficient]: 1
- ●[B of eddy viscosity coefficient]: 0

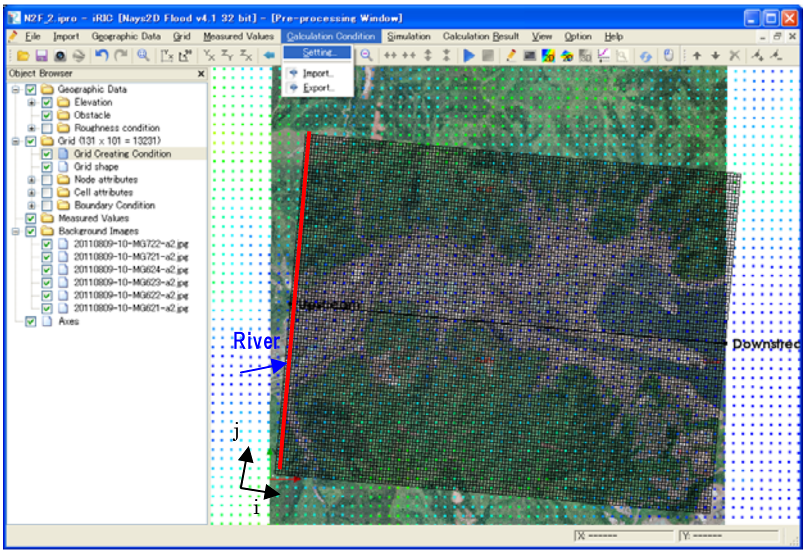
After making the settings above, click on [Save and Close] to close the window.

6. Inflow settings

① The number of inflows

Inflow river settings (or bank failure location settings) are made at the upstream end (red line).

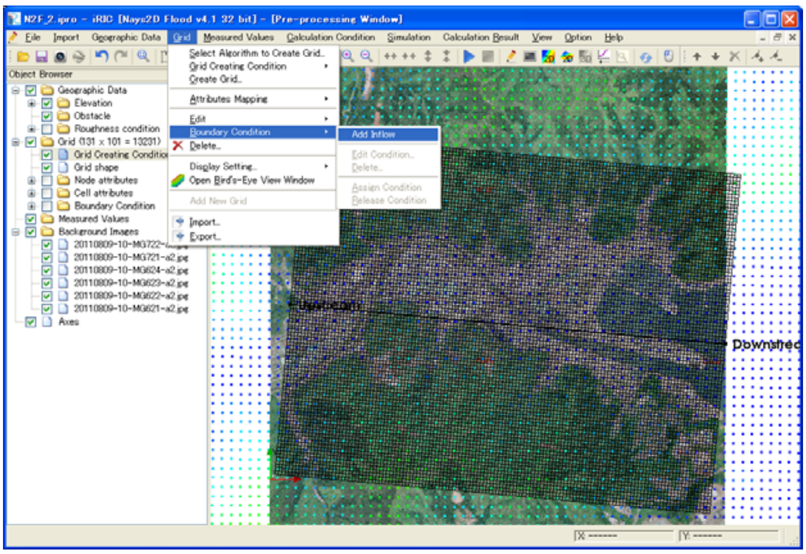
Note: This sample does not incorporate river discharge. However, Nays2D Flood requires at least one inflow location to be set on the upstream end.



Note: In this basin, an inflow to the mainstream occurs on the red line, as is shown by the blue arrow. Because this sample focuses on tsunami runup, a dummy inflow discharge datum, (Discharge 0 m^3/s) is applied.

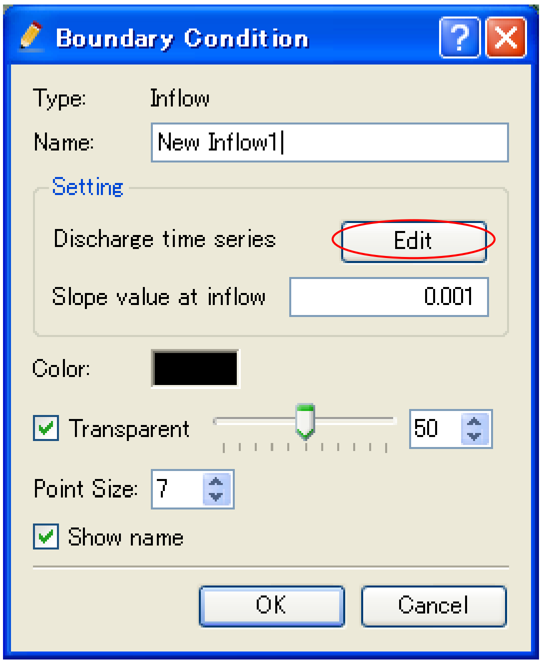
② Adding inflow boundary conditions

➢ On the menu bar, select [Grid] - [Boundary Condition] - [Add Inflow].



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

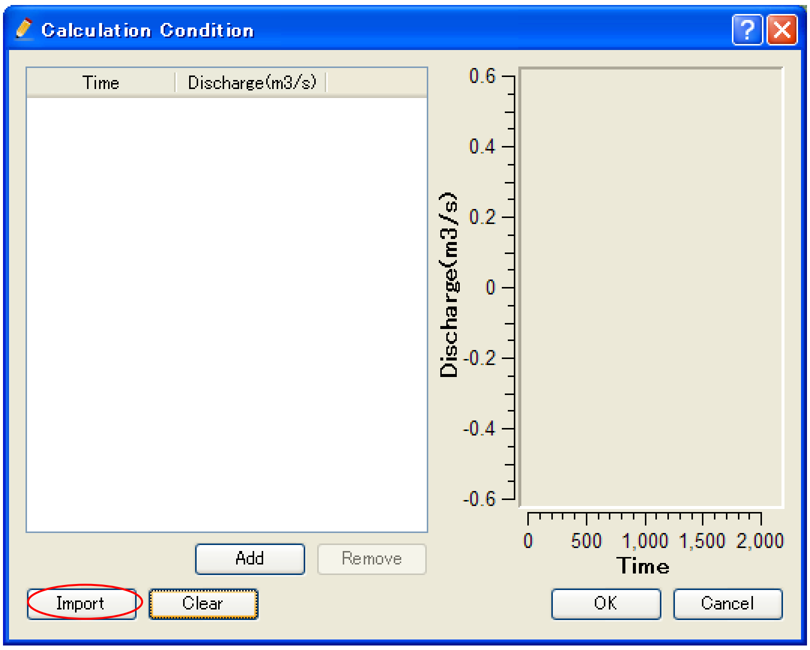
The [Calculation Condition] window will be displayed.



- [Name]: Arbitrary name of inflow
- [Slope value at inflow]: 0.001

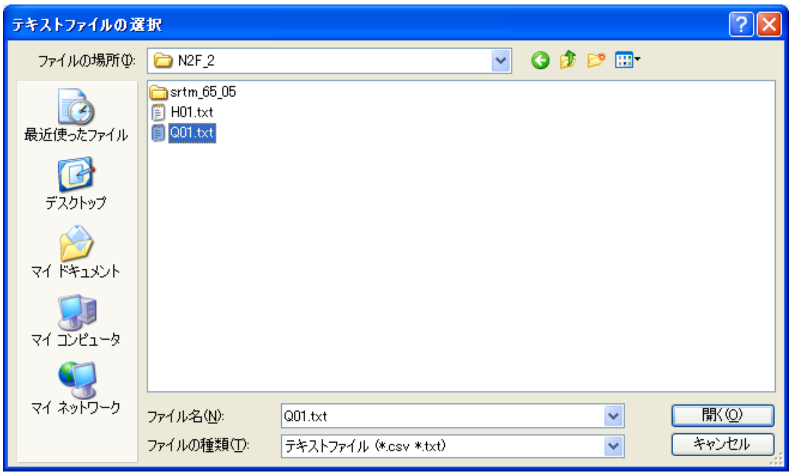
➢ Click on [Import] on the [Calculation Condition] window.

The [Select text file] window will be displayed.

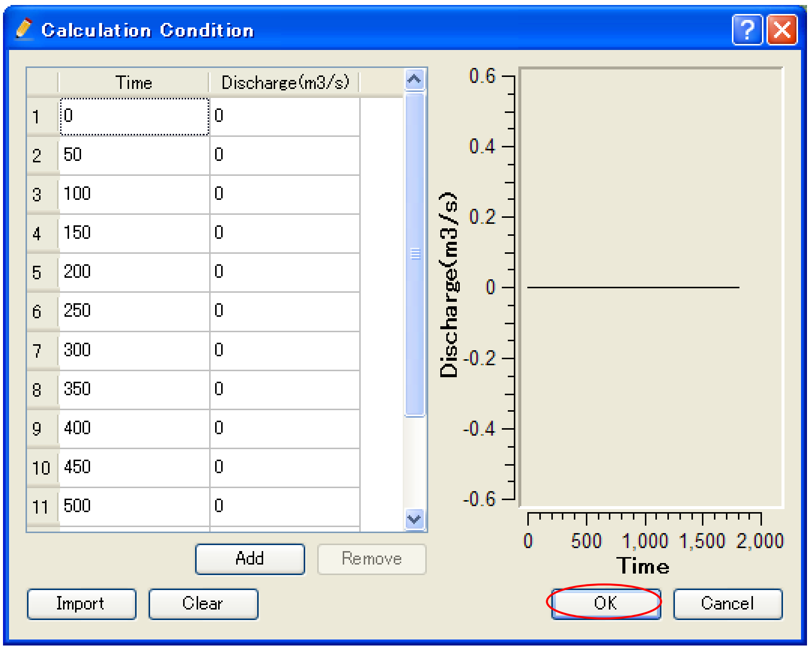


➢ Open [¥SampleData¥N2F], select [Q01.txt] and click on [Open].

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



➢ Click on [OK].



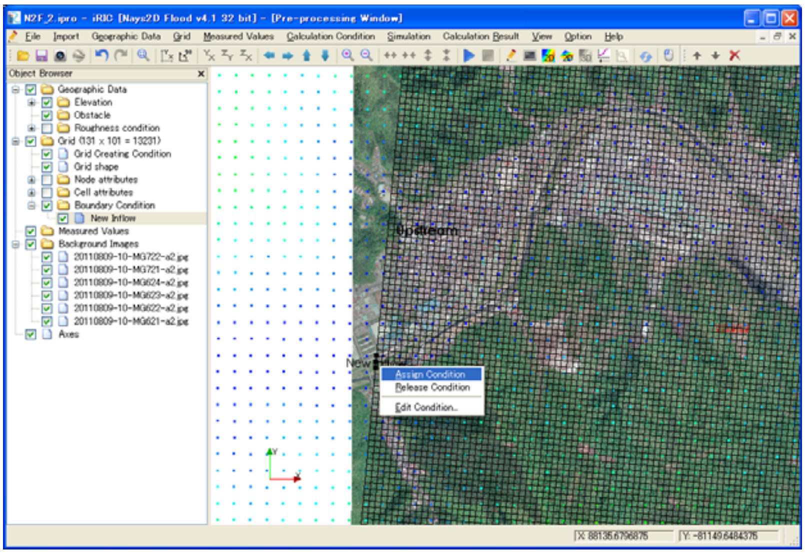
The time steps of the tsunami wave height must be kept at the same value, including at the inflow location.

③ Setting locations under the inflow boundary conditions

- In the Object Browser, select [Grid] - [Boundary Condition] - [New Inflow1].
- Select the grid node to be set as the inflow location by dragging and right-clicking.
- Click on [Add].

The grid node indexes of inflow location of the sample data is as below:

Inflow	Grid node indexes (i, j)	Discharge data	Note
Inflow 1	(1, 28)～(1, 30)	Q01.txt	



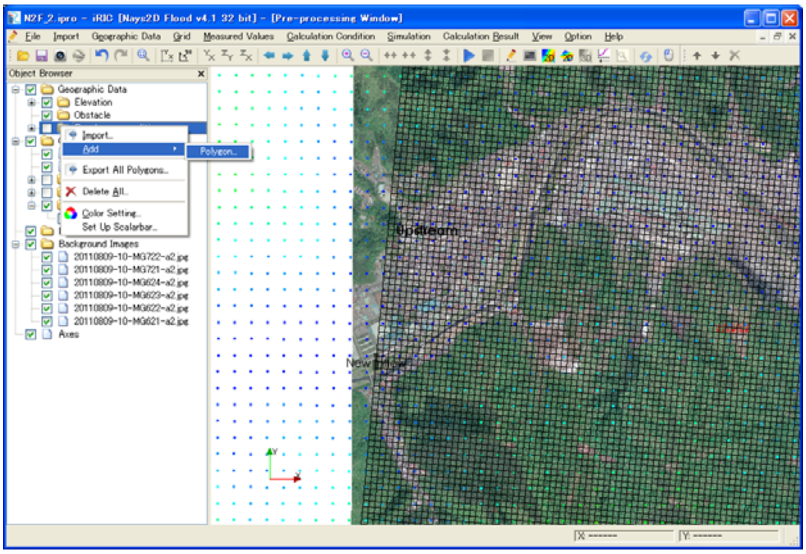
Inflow locations cannot be set at a grid node other than i=1, j=1 or j=nj.

When you set other gird nodes than above as inflow locations, an error occurs and simulation stops.

In such a case, select the grid node and right click to select [Delete].

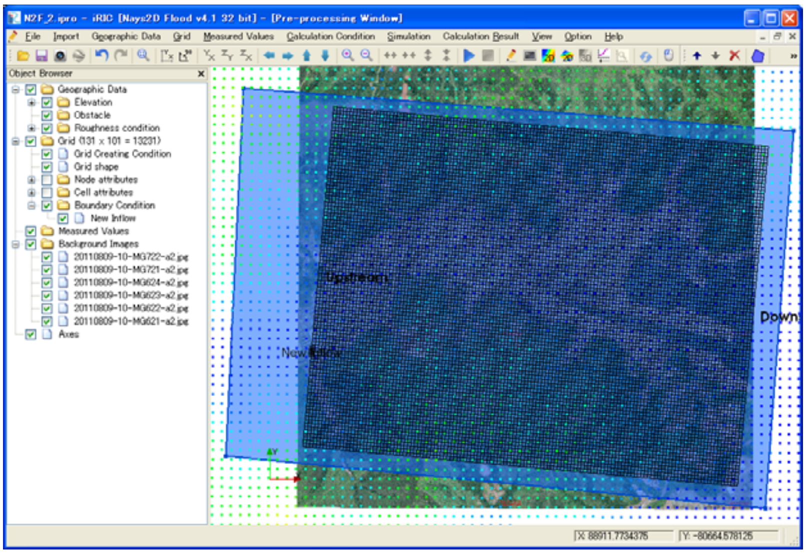
7. Setting the roughness

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



➢ Click on several points to set several "vertexes" such that the vertexes enclose the grid area, and press the Enter key on your keyboard to complete the operation.

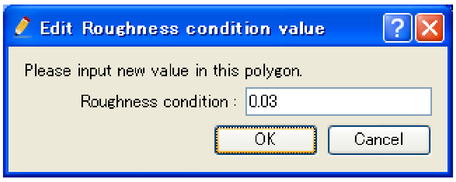
➢ Then, input a value of Manning's roughness coefficient for [Roughness condition] in the [Edit roughness condition value] window.



Setting Polygon

A polygon is set by enclosing the area by the line linking vertexes 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. Then, you make polygons that cover details. In this way, you can set roughness coefficients, where necessary.



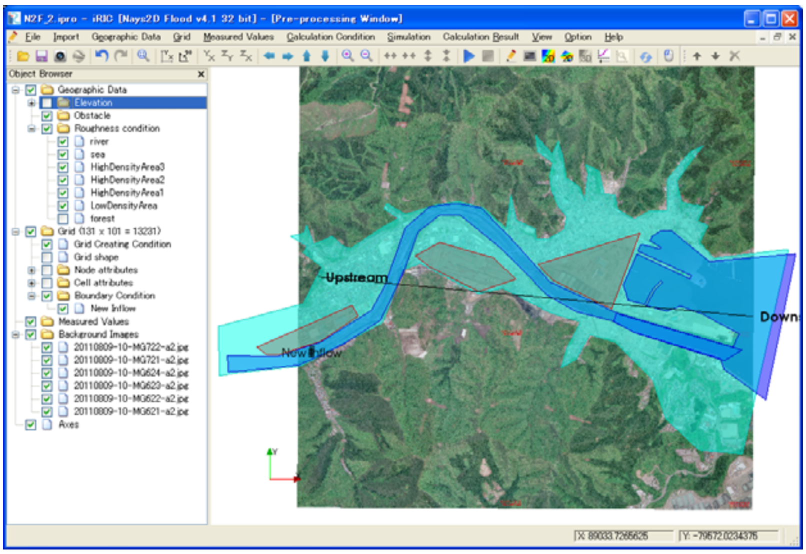
➢ Referring to the background image, repeat the procedure to add polygons as shown the figure below.

➢ Name each polygon: In the Object Browser, select [Geographic Data] - [Roughness] and right click to open the [Edit Name] box. Input a simple name that expresses the roughness of the area specified by the polygon.

In the sample data, roughness coefficients are set as below:

Name of polygon	Description	Roughness coefficient	Note
forest	Forested mountains	0.030	
LowDensityArea	Low building density area	0.040	Residential area, etc.
HighDensityArea1	High building density area	0.080	Industrial area, etc.
HighDensityArea2	Ditto	0.080	
HighDensityArea3	Ditto	0.080	
Sea	Sea area	0.025	
River	River	0.025	

Note: Manning's roughness coefficient is comprehensively set by considering simulation models, land use in the basin and past records. Refer to page 33 of *Flood Simulation Manual (Draft)* and page 89 of *The Collection of Hydraulic Formulae* (Japan Society of Civil Engineers (FY 1999).

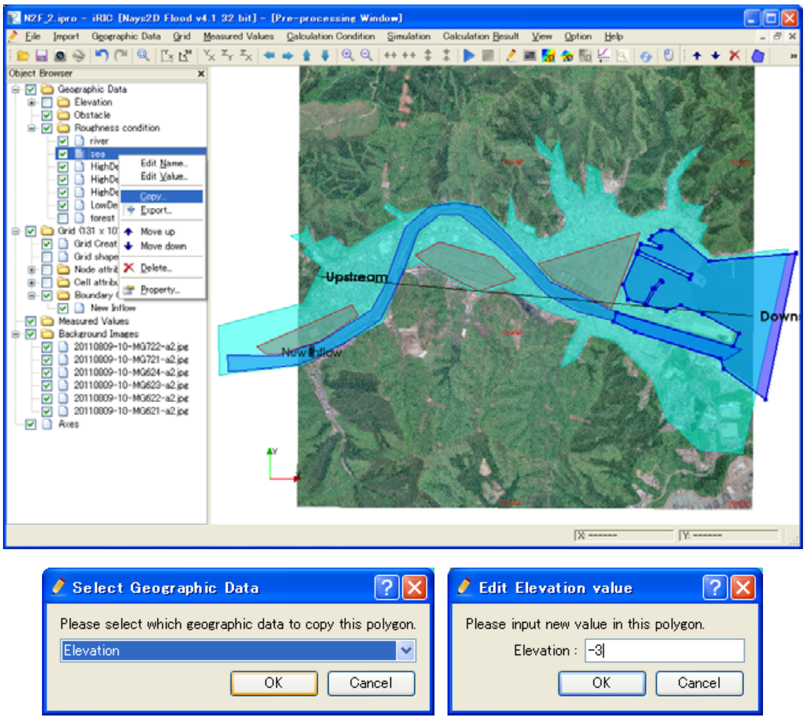


8. Correcting elevation of sea area

➢ In the Object Browser, select [Geographic Data] - [Roughness] - [sea]. Then, select [Copy] from right click.

➢ Select [Elevation] on the [Select Geographic Data] window, and click on [OK].

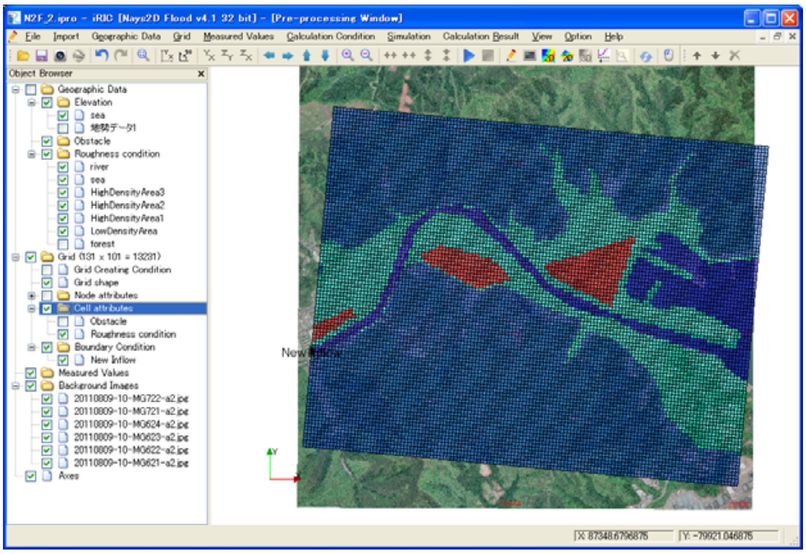
➢ Input [-3] on the [Edit Elevation value] window, and click on [OK].



➢ Remove the check mark from the [Geographic Data] box in the Object Browser.

➢ In the Object Browser, select [Grid] - [Cell attributes] - [Roughness] and make a check mark in the [Roughness] box.

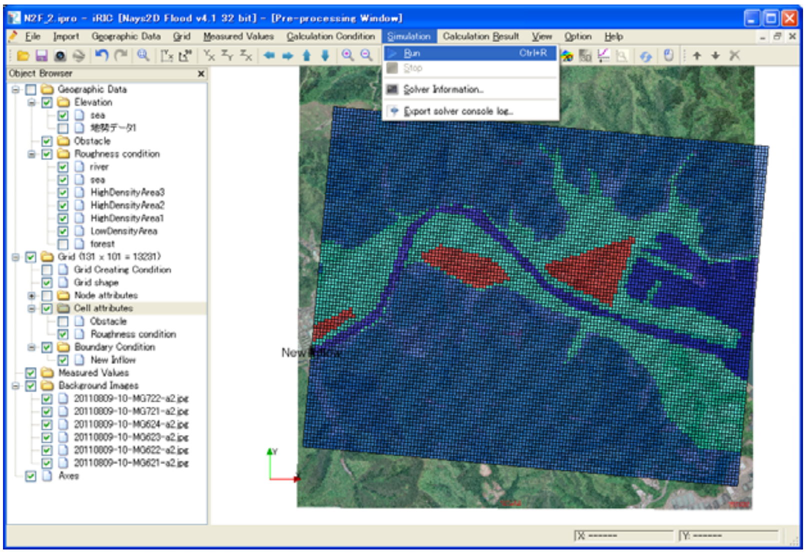
➢ In the Object Browser, select [Grid] - [Grid shape] and make a check mark in the [Grid shape] box.



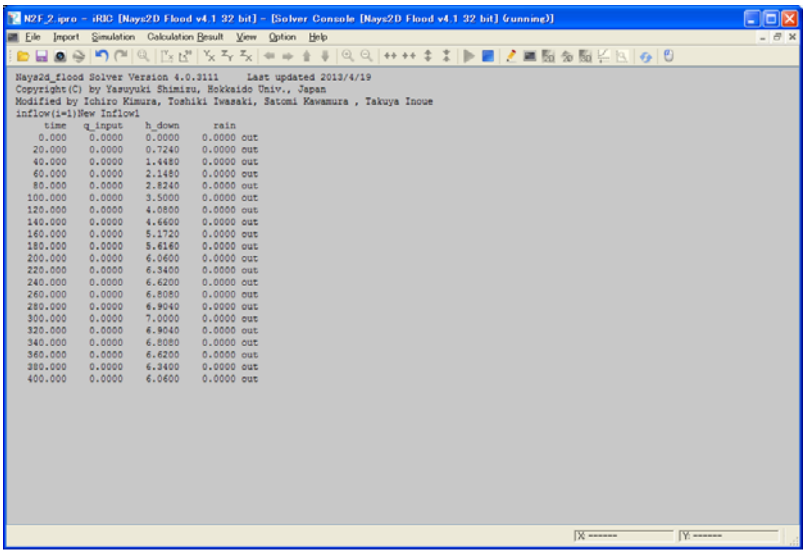
Note: Check whether the roughness coefficients are properly displayed on the grid.

4. Making a simulation

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



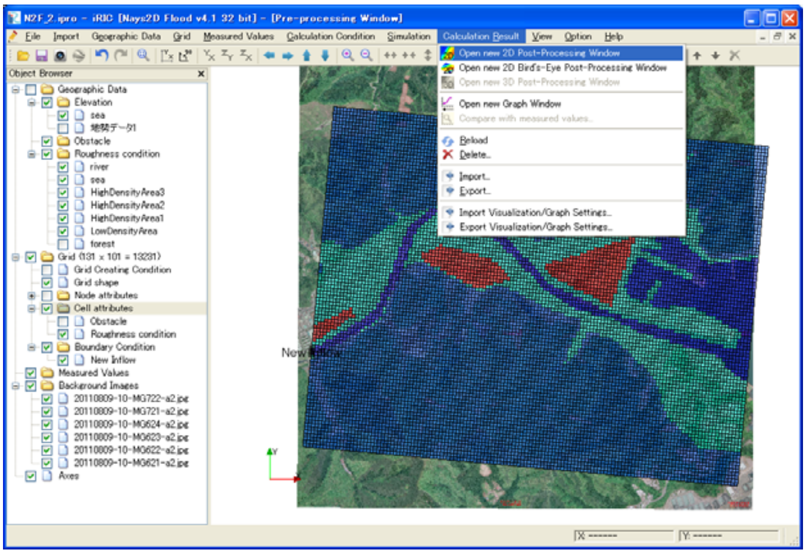
The [Solver Console [Nays2D Flood v4.1 32bit] (running)] window will open to start the simulation.



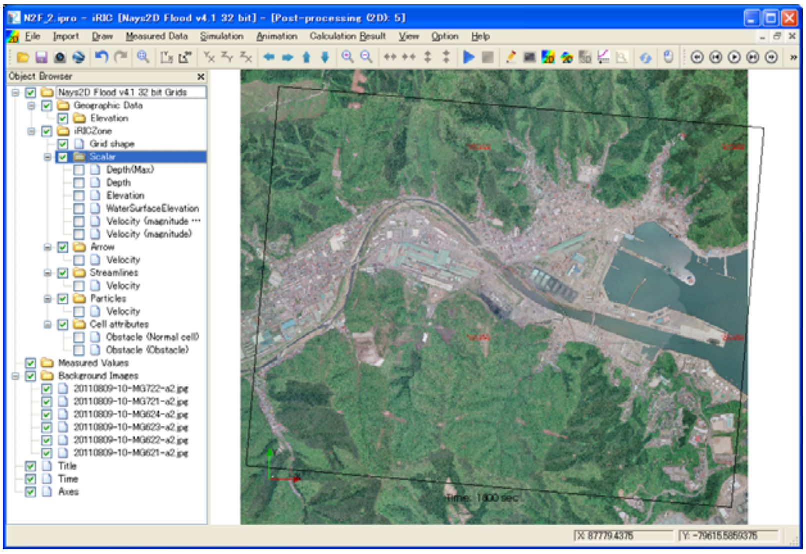
5. Visualization of computational results

1. Open the 2D Post-processing window

➢ On the menu bar, select [calculation results] - [Open New 2D Post-Processing Window].



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



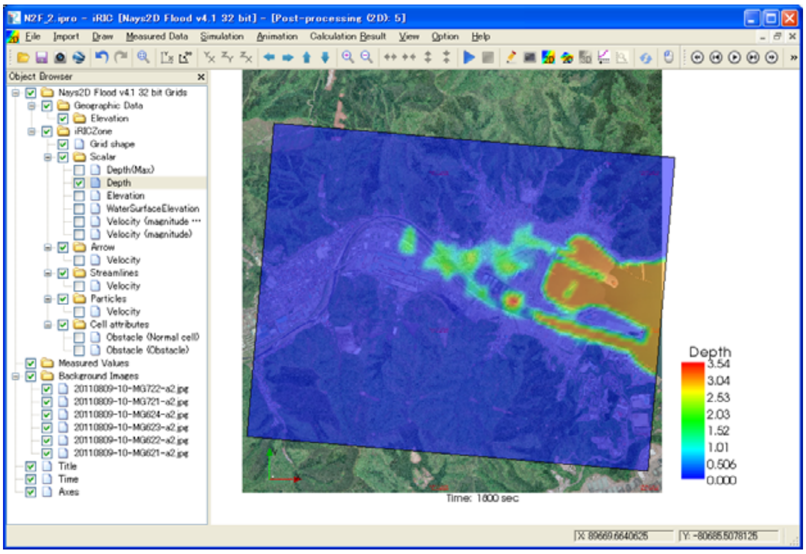
2. Quantities that can be visualized

Names on the Object Browser	Description
● Contour	
Depth(Max)	The max. depth achieved by the end time of visualization (m)
Depth	Water depth at the time of visualization (m)
Elevation	Altitude of the grid for calculation (m)
WaterSurfaceElevation	Water level at the time of visualization (m)
Velocity (magnitude Max)	The max. flow velocity achieved by the end 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 the streamline
● Particles	
Velocity	Displays particles
● Cell attributes	
Obstacle cell (normal cell)	Displays normal cells
Obstacle cell (obstacle cell)	Displays obstacle cells

3. Visualizing the water depth

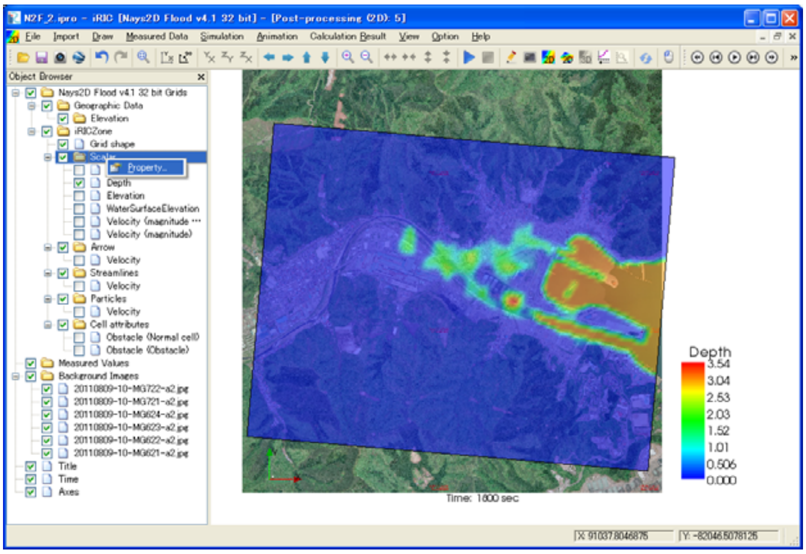
➢ In the Object Browser, select [Nays2D Flood Grids] - [iRICZone] - [Scalar] - [Depth] by making a check mark in each box.

A contour map of water depth will open.

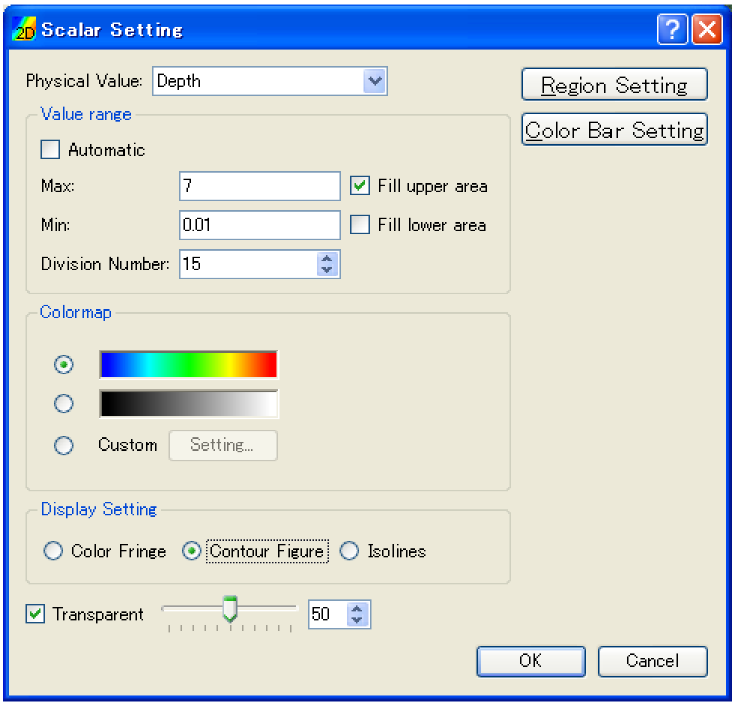


➢ In the Object Browser, select [Nays2D Flood Grids] - [iRICZone] - [Contour]. Right click on [Scalar] to select [Property].

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: 7
  Min: 0.01
  Remove ☑ from [Fill lower area]

- Colormap setting:
- [Color map]:
Do not change any settings.

- Display Setting
  [Contour Figure]

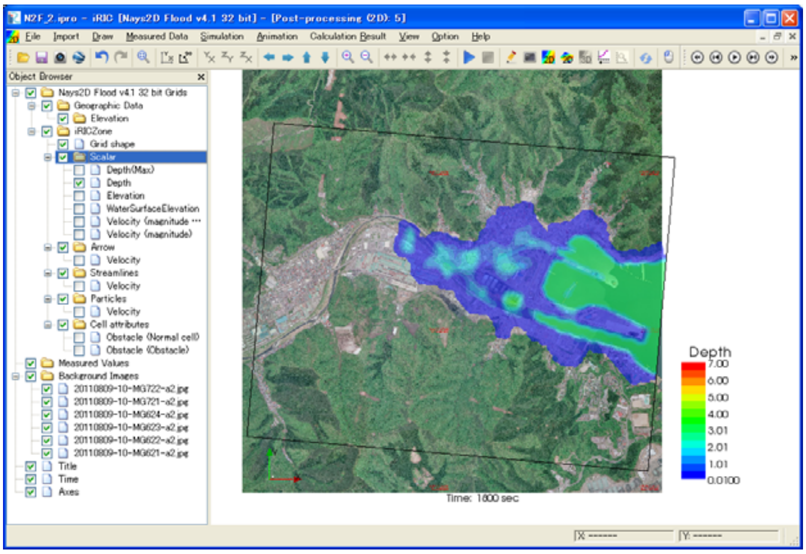
- Division Number: 15

Semi-transparent:
Do not change any settings.

[Region Setting]:
Do not change any settings.

- [Color Bar Setting]:
Do not change any settings.

The contour map is easier to see now.

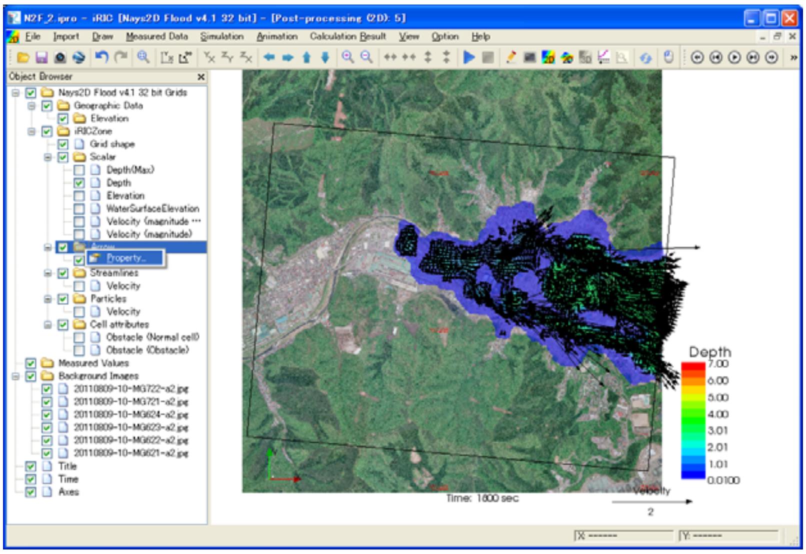


4. Visualizing flow vector

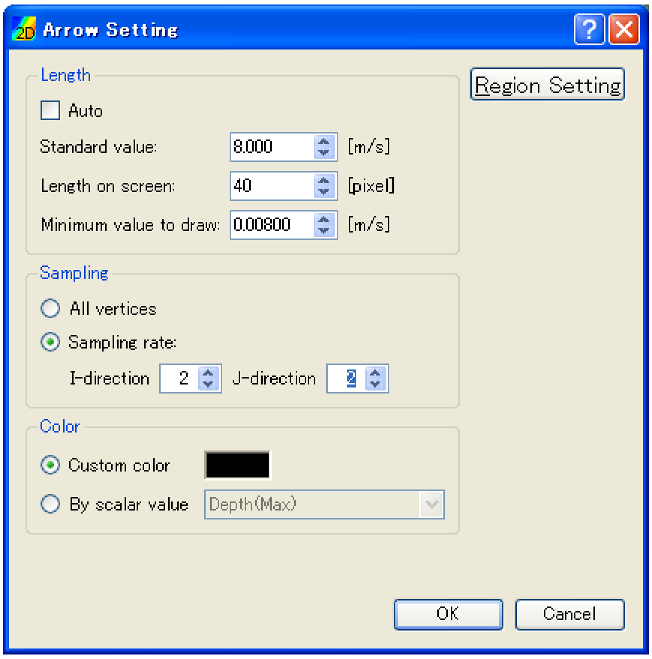
➢ In the Object Browser, select [Nays2D Flood Grids] - [iRICZone] - [Arrow] - [Velocity].

➢ 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

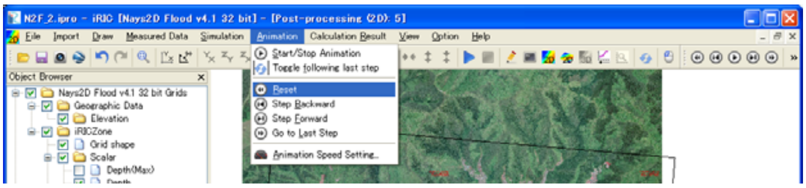
● Color:
Do not change any settings.

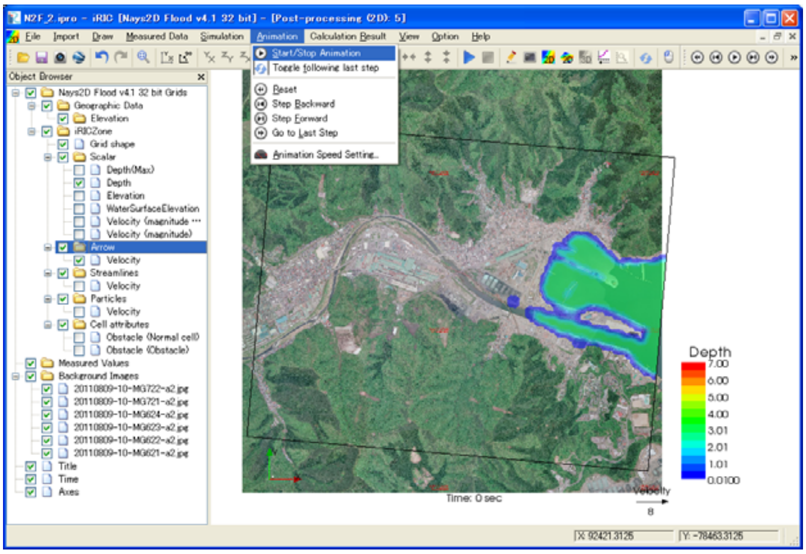
5. Visualizing the max. flow velocity

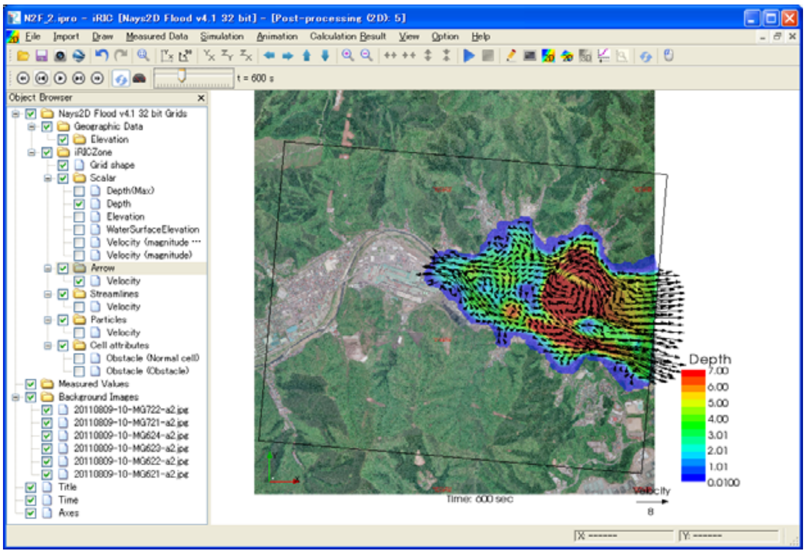
➢ On the menu bar, select [Animation] - [Reset].

➢ On the menu bar, select [Animation] - [Start/Stop Animation].

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







6. Visualizing time series data of an arbitrary location

➢ On the menu bar, select [View] - [Pre-processor].

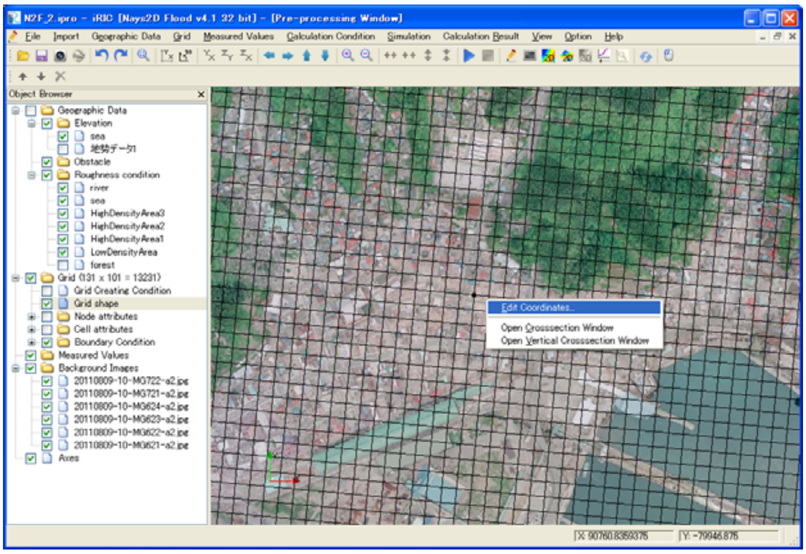
➢ Zoom in on an arbitrary location. (Refer to [Mouse Hints])

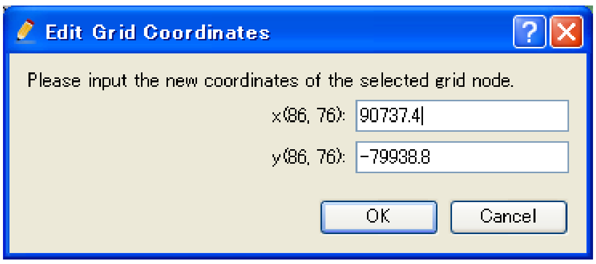
➢ In the Object Browser, select [Grid] - [Grid shape] and make a check mark in the [Grid shape] box.

➢ Select one grid cell of arbitrary location by dragging it, and right click.

➢ Click on [Edit Coordinates].

The [Edit Grid Coordinates] will open. Find the grid node indexes.

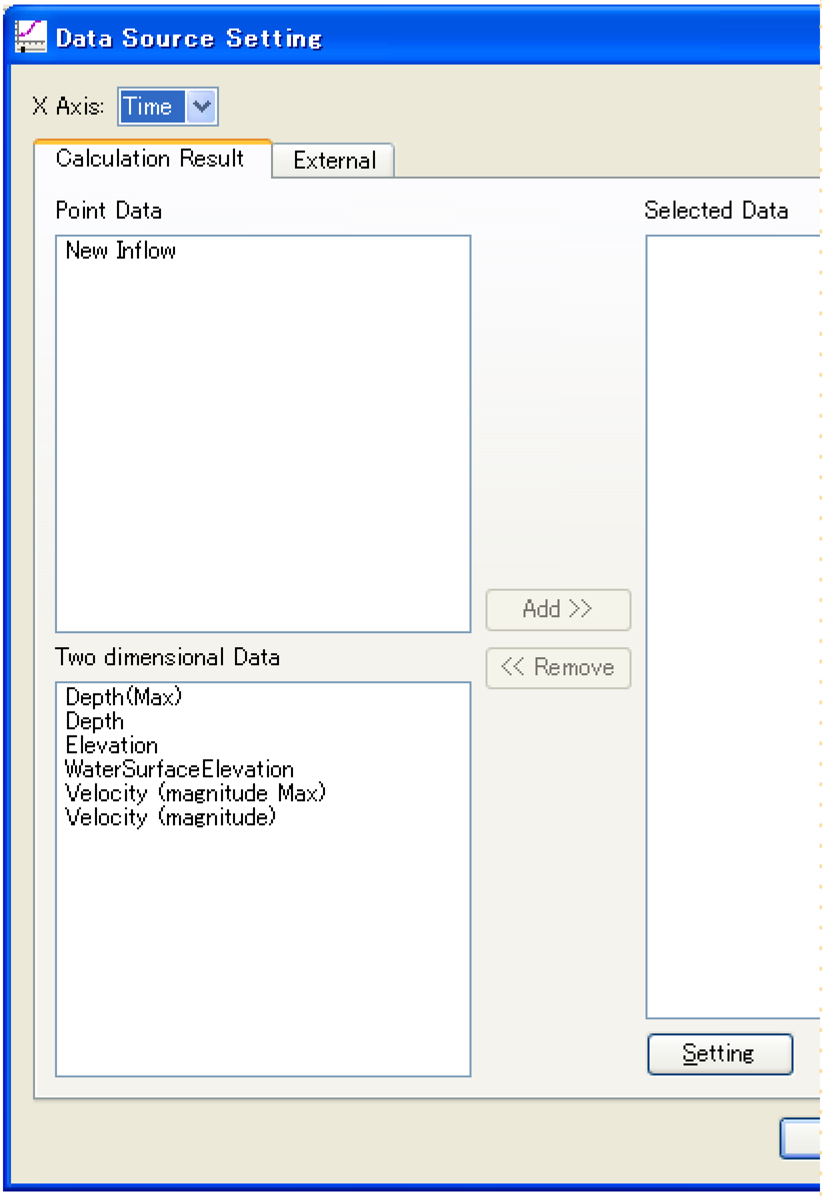




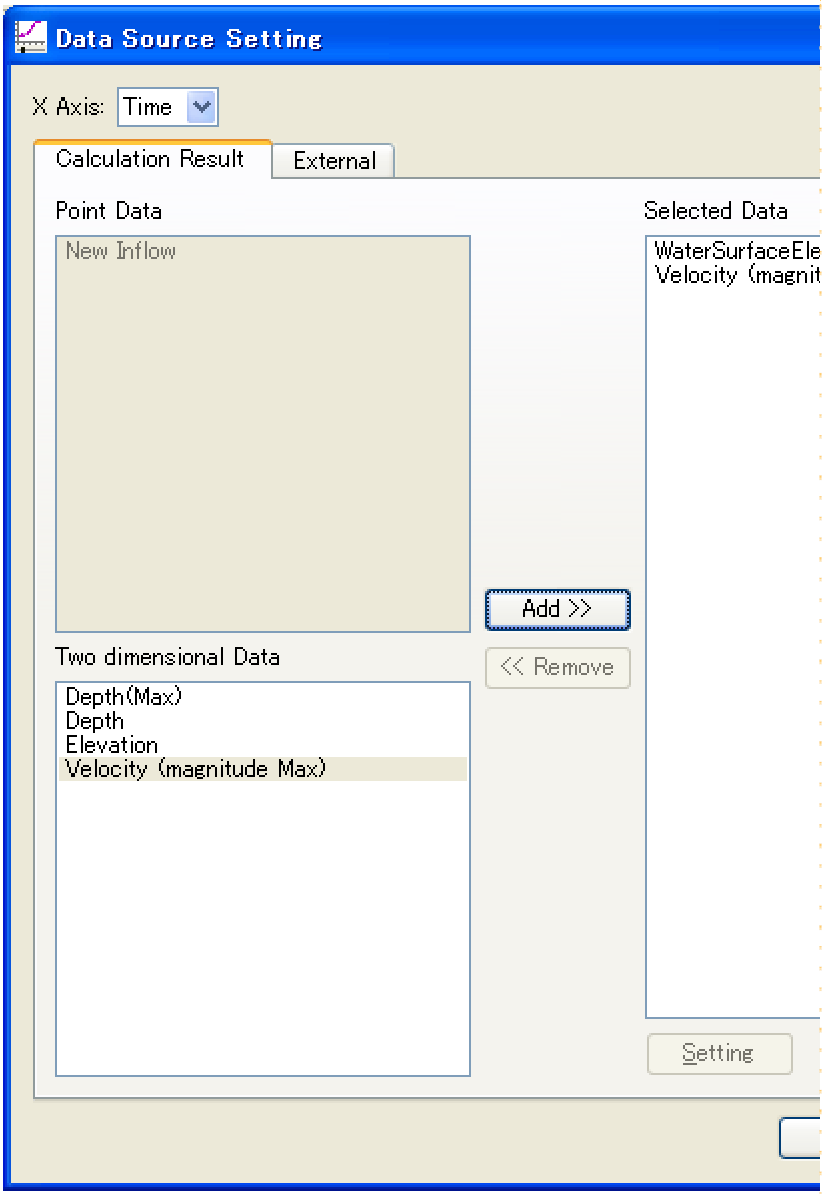
➢ On the menu bar, select [Calculation Results] - [Open New Graph Window].

➢ In the [Calculation Result] tab, select [WaterSurfaceElevation] and [Velocity (magnitude)] from [Two dimensional Data] to add [Selected Data].

➢ Click on [OK].



Note: Unit: m^3/s

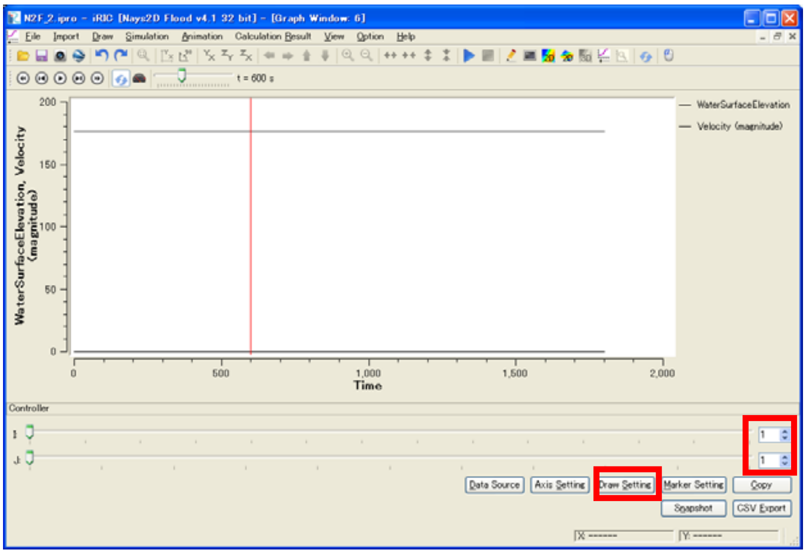

Data Source Setting
X Axis:
Time
Calculation Result
External
Point Data
New Inflow
Selected Data
WaterSurfaceEle
Velocity (magnit
Add >>
<< Remove
Two dimensional Data
Depth(Max)
Depth
Elevation
Velocity (magnitude Max)
Setting

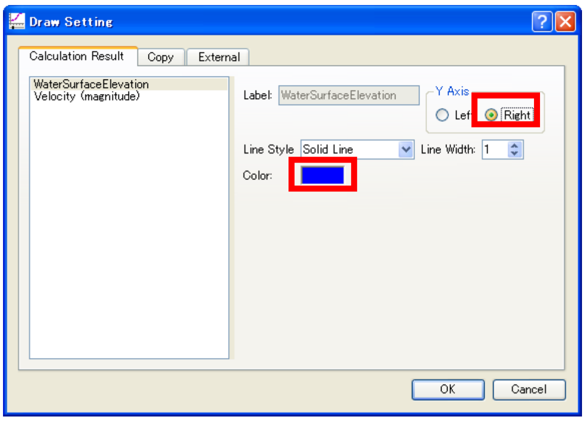
➢ The grid node indexes of arbitrary location that have been identified before are input to the [I] box and the [J] box of [Controller].

➢ Click on [Draw Setting] to set the [Draw Setting] window as below and click on [OK].

➢ Click on [OK].

Time series graph of the arbitrary location will be displayed.





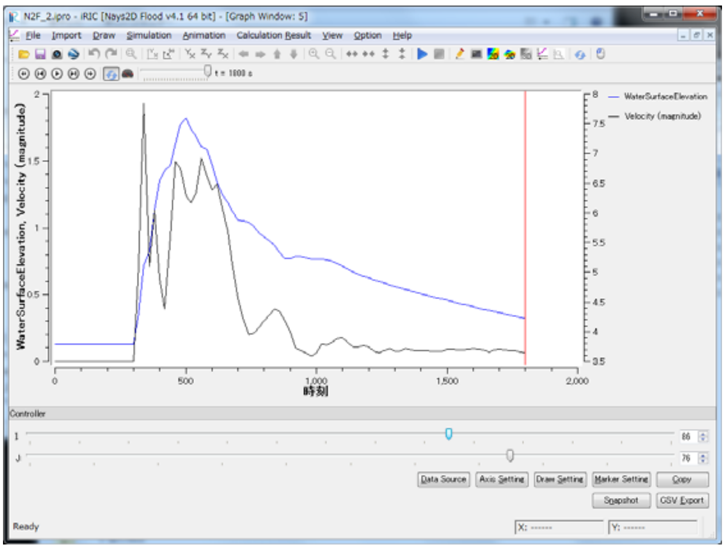
[Draw Setting]

● Y-axis: Right

● [Y Axis]:: Do not change any settings.

● [Line Style]: Do not change any settings.

● Color: Blue

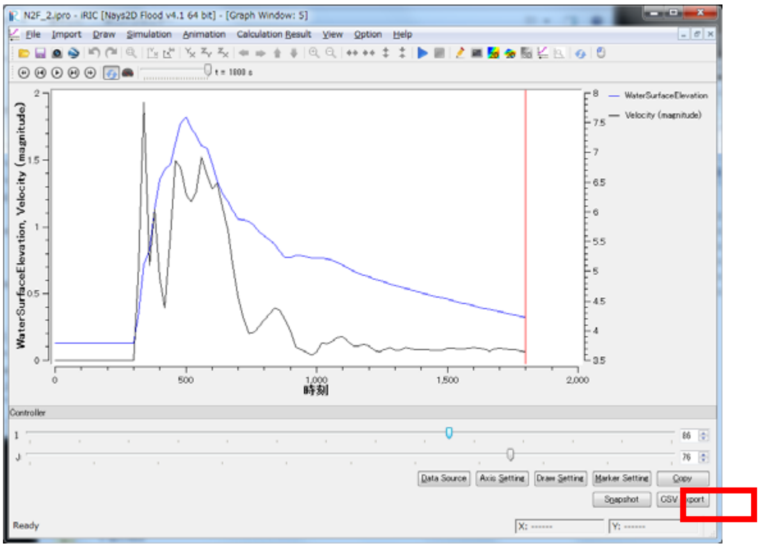


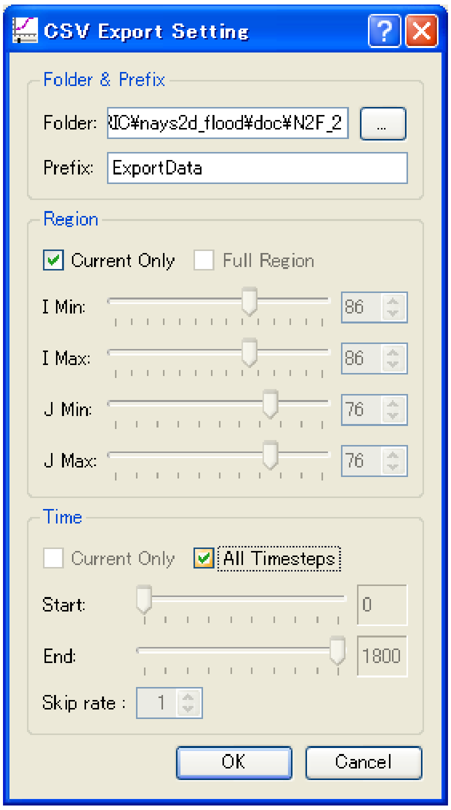
➢ Click on [CSV Export].

➢ Set the [CSV Export] window as below, and click on [OK].

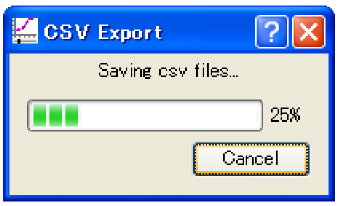
➢ Click on [OK].

The time series data of an arbitrary location will be exported.





[Draw Setting]
● [Folder]:
Any
(The folder can be the same as that for storing the sample data)
● [Prefix]:
Do not change any settings.
● [Region]:
Do not change any settings.
● [Time]:
[All Timesteps]



6. Important information

◆ Tsunami runup

In this tutorial, tsunami waves are generated by inputting only the water level of the boundary at the downstream end. For an accurate simulation, a wide-area analysis of tsunami wave propagation from the wave origin should be made to input boundary conditions of water surface elevation and flow velocity. The objective of the example introduced by this tutorial is to assess the inundation area by a simple method. In future, we will improve this simulation software so that the results of tsunami wave propagation analysis (water surface elevation and flow velocity) can be input as data to the boundary condition of the downstream end of the simulation.
For details of updates, refer to http://i-ric.org/ja/downloads.

Additional comments

$$\nu_t = A\frac{\kappa}{6}u_* h + B \tag{1}$$

where, A and B are correction coefficients. Their default values are A=1 and B=0. If you wish to modify eddy viscosity coefficients, modify values of A and B.

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

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)	
	Eriko KAKEGAWA (Hokkaido Univ., Japan)	
Supported by	River center of Hokkaido, Japan.	