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 3.0

Last Update: 2021.1.05
Release Date: 2015.3.17

Copyright 2013 iRIC Project All Rights Reserved.
Contents

CHAPTER 1 USING NAYS2D FLOOD ................................. 4
1. Nays2D Flood basic operating procedures ......................... 5
2. Launching Nays2D Flood .................................................. 6
3. The sample data ............................................................... 8
4. Simulation conditions of Nays2D Flood ............................... 9

CHAPTER 2 EXAMPLES OF FLOOD CALCULATION FOR AN
ACTUAL RIVER SECTION ................................................. 10
1. Creating the calculation grid .............................................. 11
   1 Importing cross-sectional river survey data ......................... 11
   2 Selecting an algorithm for creating a grid .......................... 17
   3 Creating a grid .............................................................. 18
2. Setting the calculation conditions ....................................... 20
   8 Open "Calculation Conditions" ......................................... 20
   9 Setting the inflow boundary conditions .............................. 21
  10 Setting the initial water surface profile ............................. 22
  11 Setting the time ............................................................ 23
  12 Other settings .............................................................. 24
  13 Setting inflow rivers ....................................................... 25
  14 Setting roughness condition ............................................ 30
  15 Setting obstacle cells ..................................................... 34
  16 Setting Areal fraction that buildings occupy ....................... 37
  17 Setting for the box culvert ............................................. 40
3. Making a simulation ......................................................... 45
4. Visualizing the calculation results ...................................... 46
   8 Open the "2D Post-Processing" window .............................. 46
   9 Visualizable quantities ................................................... 47
  10 Visualizing the max. water depth ..................................... 48
  11 Visualizing the max. flow velocity ................................... 50
  12 Visualizing the inflow discharge hydrograph ..................... 51

CHAPTER 3 EXAMPLES OF TSUNAMI RUNUP ANALYSIS
USING SRTM DATA ......................................................... 54
5. Creating the calculation grid ............................................. 55
   8 Importing geographic data .............................................. 55
   9 Selecting an algorithm for creating the grid ....................... 59
  10 Creating a grid ............................................................. 60
CHAPTER 4 EXAMPLES OF RAINFALL INDUCED FLOODING IN ACTUAL RIVER

1. Creating the calculation grid ................................................................. 93
   1 Importing geographic data .................................................................. 93
   2 Selecting an algorithm for creating the grid ......................................... 100
   3 Creating a grid ...................................................................................... 101

2. Setting the calculation conditions .......................................................... 103
   1 Open [Calculation Condition] ............................................................. 103
   2 Setting the inflow boundary conditions .............................................. 104
   3 Setting the initial water surface profile .............................................. 106
   4 Setting the time .................................................................................. 106
   5 Other settings ....................................................................................... 107
   6 Inflow settings .................................................................................... 108
   7 Setting the roughness .......................................................................... 112

3. Making a simulation ............................................................................ 114

4. Visualization of computational results ............................................. 115
   1 Open the 2D Post-processing window ............................................... 115
   2 Visualizing the water depth ................................................................ 116
   3 Visualizing flow vector ....................................................................... 118
   4 Visualizing the max. flow velocity ..................................................... 119
Chapter 1

Using Nays2D Flood

This manual explains the basic operation and startup procedures of Nays2D Flood, which is compiled with iRIC. The Nays2D 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/downloads
Software: iRIC version3.0 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 [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 3.0 [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/downloads

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

<table>
<thead>
<tr>
<th>Data name</th>
<th>Contents</th>
<th>version</th>
</tr>
</thead>
<tbody>
<tr>
<td>Ex01</td>
<td>Examples of flood calculation for an actual river section</td>
<td>Nays2D Flood 5.0</td>
</tr>
<tr>
<td>Ex02</td>
<td>Examples of tsunami runup analysis using SRTM data</td>
<td>Nays2D Flood 5.0</td>
</tr>
</tbody>
</table>
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 [\YSampleData\YN2F], 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.

**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.

![Confirmation]

Do you want to map geographic data to grid attributes now?

Yes  No

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

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]

![Boundary Condition dialog box with options for Inflow, Edit, and OK/Cancel buttons.]

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

![Discharge time series dialog box with options for Time, Discharge, Add, Remove, Import, Export, Clear, OK/Cancel buttons.]
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.
1) Find the area of each land use within each mesh. Land use: building A1, road A2 and other land use A3. 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:

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

where, \( \theta \) 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 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.

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.

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

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

<table>
<thead>
<tr>
<th>Descriptions in the Object Browser</th>
<th>Description of each quantity</th>
</tr>
</thead>
<tbody>
<tr>
<td>• Contour</td>
<td></td>
</tr>
<tr>
<td>Depth (Max)</td>
<td>The max. water depth by the time of visualization (m).</td>
</tr>
<tr>
<td>Depth</td>
<td>The water depth at the time of visualization (m).</td>
</tr>
<tr>
<td>Elevation</td>
<td>The ground height of calculation grid (m).</td>
</tr>
<tr>
<td>WaterSurfaceElevation</td>
<td>The water surface elevation at the time of visualization (m).</td>
</tr>
<tr>
<td>Velocity (magnitude Max)</td>
<td>The max. flow velocity by the time of visualization (m/s).</td>
</tr>
<tr>
<td>Velocity (magnitude)</td>
<td>Flow velocity at the time of visualization (m/s).</td>
</tr>
<tr>
<td>• Vector</td>
<td></td>
</tr>
<tr>
<td>Velocity</td>
<td>Vector of flow velocity (m/s) at the time of visualization.</td>
</tr>
<tr>
<td>• Streamline</td>
<td></td>
</tr>
<tr>
<td>Velocity</td>
<td>Displays a streamline.</td>
</tr>
<tr>
<td>• Particles</td>
<td></td>
</tr>
<tr>
<td>Velocity</td>
<td>Displays particles.</td>
</tr>
<tr>
<td>• Cell attributes</td>
<td></td>
</tr>
<tr>
<td>Normal cell (a normal cell)</td>
<td>Displays a normal cell.</td>
</tr>
<tr>
<td>Obstacle cell (a cell with an obstacle)</td>
<td>Displays an obstacle cell.</td>
</tr>
</tbody>
</table>
10 Visualizing the max. water depth

In the Object Browser, select [Nays2D Flood Grids] - [iRIC Zone] - [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
- **Remove ✓ from Fill lower area**
- **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 clear.
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
Remove ☑ from Fill lower area.

Color map: No change
Display Setting: Contour Figure
Division Number: 15
Transparent: No change
Region Setting: No change
Color Bar Setting: No change
Open [New Graph Window].
[Data Source Setting] screen is displayed.

Unit: $\text{m}^3/\text{s}$
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 3
Examples of Tsunami Runup Analysis Using SRTM data

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

3. Making a simulation

4. Visualizing the calculation results
   Here, we introduce how to display a waterdepth contour map and a flow velocity vector map.
5. Creating the calculation grid

8 Importing geographic data

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

[Select Coordinate System] screen is displayed. Search with JGD and select [EPSG:2452:JGD2000/Japan Plane Rectangular CS X].

The coordinate system of the target area can be understood from the hint on the coordinate system at the bottom of the screen.
[Select Region] screen is displayed. Drag the target part and click [Next].

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

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 the [Pre-processing Window] screen.

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

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.
Since the coordinate system is selected, it is set automatically.

If your PC specifications are insufficient, you may not be able to process very large arrays 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). 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 process large numbers of elevation data.

To speed up processing, remove geographic data in the areas that are not needed.

Such removal is not necessary when your PC has specifications high enough to maintain stable operation even when large numbers of data are processed.
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.
10 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].

- \( n_j \) (Number of divisions in the longitudinal direction): 100
- \( n_i \) (Number of divisions in the transverse direction): 130
- \( W \) (Grid width): 3000 m

Note: In this example, grid cell intervals \( d_i \), \( d_j \) 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.
6. 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.
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 the 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.
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
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.
Inflow 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 m³/s) is applied.

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

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

![Boundary Condition Window]

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

![Discharge time series Window]
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.
In the menu bar, click [Grid]-[Attributes Mapping]-[Execute].

Inflow rivers are set on the sides.
The grid node indexes of the inflow location of the sample data is given below:

<table>
<thead>
<tr>
<th>Inflow</th>
<th>Grid node indexes (i, j)</th>
<th>Discharge data</th>
<th>Note</th>
</tr>
</thead>
<tbody>
<tr>
<td>Inflow 1</td>
<td>(1, 26)~(1, 29)</td>
<td>Q01.txt</td>
<td></td>
</tr>
</tbody>
</table>

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

When you set grid nodes other than those above as inflow locations, an error occurs and the 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 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. Then, you make polygons that cover smaller regions with additional detail, if desired. In this way, you can set roughness coefficients, where necessary.
Referring to the background image, repeat the procedure to add polygons as shown in 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:

<table>
<thead>
<tr>
<th>Name of polygon</th>
<th>Description</th>
<th>Roughness coefficient</th>
<th>Note</th>
</tr>
</thead>
<tbody>
<tr>
<td>Forest</td>
<td>Forested mountains</td>
<td>0.030</td>
<td></td>
</tr>
<tr>
<td>LowDensityArea</td>
<td>Low building density area</td>
<td>0.040</td>
<td>Residential area, etc.</td>
</tr>
<tr>
<td>HighDensityArea1</td>
<td>High building density area</td>
<td>0.080</td>
<td>Industrial area, etc.</td>
</tr>
<tr>
<td>HighDensityArea2</td>
<td>Ditto</td>
<td>0.080</td>
<td></td>
</tr>
<tr>
<td>HighDensityArea3</td>
<td>Ditto</td>
<td>0.080</td>
<td></td>
</tr>
<tr>
<td>Sea</td>
<td>Sea area</td>
<td>0.025</td>
<td></td>
</tr>
<tr>
<td>River</td>
<td>River</td>
<td>0.025</td>
<td></td>
</tr>
</tbody>
</table>

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.
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. 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

<table>
<thead>
<tr>
<th>Names on the Object Browser</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>● Contour</td>
<td></td>
</tr>
<tr>
<td>Depth(Max)</td>
<td>The max. depth achieved by the end time of visualization (m)</td>
</tr>
<tr>
<td>Depth</td>
<td>Water depth at the time of visualization (m)</td>
</tr>
<tr>
<td>Elevation</td>
<td>Altitude of the grid for calculation (m)</td>
</tr>
<tr>
<td>WaterSurfaceElevation</td>
<td>Water level at the time of visualization (m)</td>
</tr>
<tr>
<td>Velocity (magnitude Max)</td>
<td>The max. flow velocity achieved by the end time of visualization (m/s)</td>
</tr>
<tr>
<td>Velocity (magnitude)</td>
<td>Flow velocity at the time of visualization (m/s)</td>
</tr>
<tr>
<td>● Vector</td>
<td></td>
</tr>
<tr>
<td>Velocity</td>
<td>Vector of flow velocity (m/s) at the time of visualization</td>
</tr>
<tr>
<td>● Streamline</td>
<td></td>
</tr>
<tr>
<td>Velocity</td>
<td>Displays the streamline</td>
</tr>
<tr>
<td>● Particles</td>
<td></td>
</tr>
<tr>
<td>Velocity</td>
<td>Displays particles</td>
</tr>
<tr>
<td>● Cell attributes</td>
<td></td>
</tr>
<tr>
<td>Obstacle cell (normal cell)</td>
<td>Displays normal cells</td>
</tr>
<tr>
<td>Obstacle cell (obstacle cell)</td>
<td>Displays obstacle cells</td>
</tr>
</tbody>
</table>
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] - [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: 7
- Min: 0.01
- Remove ☑ from [Fill lower area]

Color map setting: 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.
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 indices.
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: m³/s
The grid node indices from an arbitrary location that were identified before are input to the [I] and [J] boxes 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
- **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.

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 Time steps
5. Important information

◆ Tsunami runup

In this tutorial, tsunami waves are generated by inputting only the water level 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 at the downstream end of the simulation.

For details of updates, refer to http://i-ric.org/ja/downloads.
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)]

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

Google Map can also be used instead of Ortho-images but it requires registration with Google.
In the Object Browser, right-click [Geographic Data]-[Elevation]. Click [Import 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.
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].

- \( n_j \) (Number of divisions in the longitudinal direction): 120
- \( n_i \) (Number of divisions in the transverse direction): 100
- \( W \) (Grid width): 10000 m

Note: In this example, grid cell intervals \( d_i \), \( d_j \) 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]: Second
- [Boundary Conditions for \( j=1 \)]: Inflow
- [Boundary Conditions for \( j=n_j \)]: 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] [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 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
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.

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 [YYSampleData\YYN2F_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]
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.
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]

- **Color map setting:** 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.
3 Visualizing flow vector

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

On the menu bar, drag time hand to t=0
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) Writing
Eriko KAKEGAWA (Hokkaido Univ., Japan) Writing

Supported by

River center of Hokkaido, Japan.