Prepared for the Port and Airport Research Institute

CADMAS MESH MULTI GUI

User Manual

Auxiliary Services Connected

to the Input Data Creation Tool (CADMAS MESH MULTI) Upgrade

March 2014

Mizuho Information & Research Institute

Contents

1. Overview

This chapter provides an overview of CADMAS MESH MULTI.

1.1. Operation Sequence

Figure 1 provides the typical CADMAS MESH MULTI operation sequence.

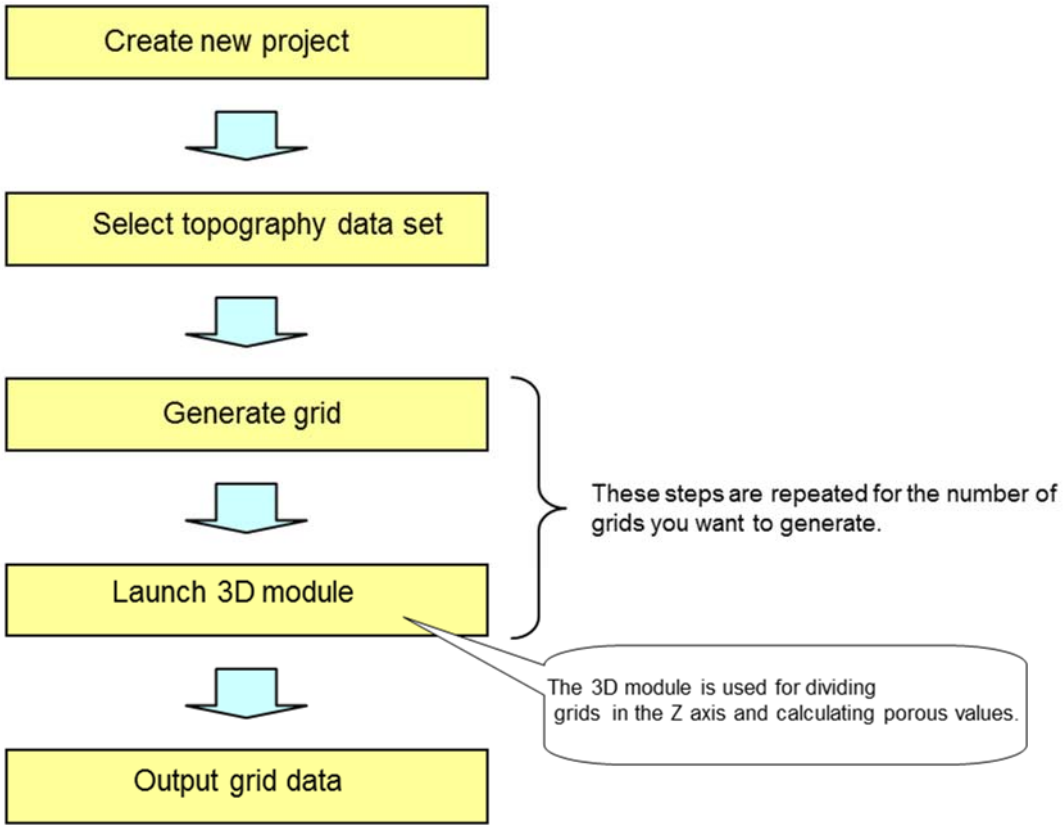

Figure 1 — Typical CADMAS MESH MULTI operation sequence

Window Structure

Figure 2 illustrates the CADMAS MESH MULTI window structure.

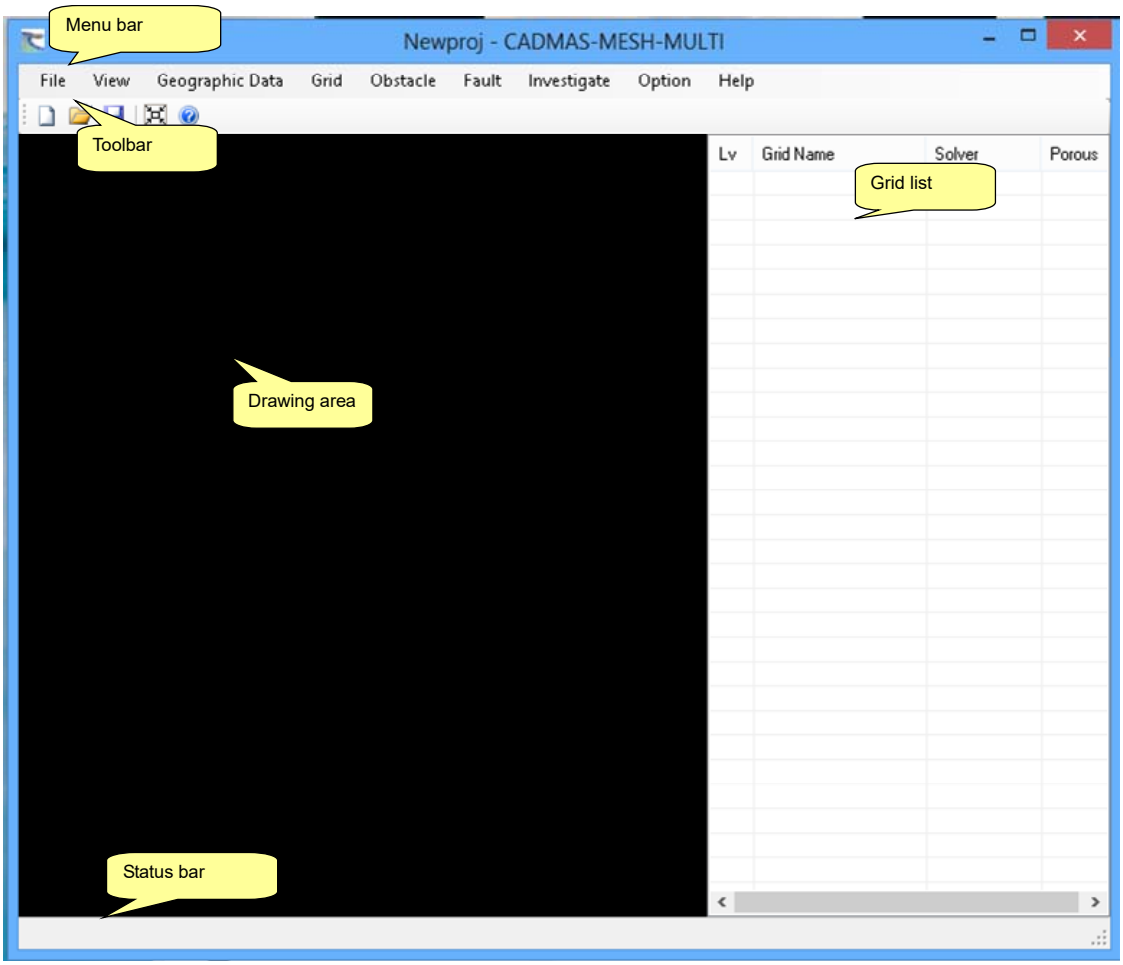

Figure 2 — CADMAS MESH MULTI window structure

Menu bar

Organizes functions by category.

Toolbar

Displays icons to access frequently used functions.

Drawing area

Displays topography data, generated grids, and other features.

Status bar

Displays the current status, as needed.

Grid list

Displays a list of the generated grids

- Lv

  This is the grid's hierarchal level. The Lv of the first generated grid is 1.

- Grid name

  Name specified when the grid was generated.

- Solver

  The type of solver specified when the grid was generated.

- P calculation

  Lists "ok" if the porous calculation has been run; "no data" if the calculation has not been run. Note that "ok" is displayed regardless of whether the porous calculation has been run for grids that have been passed to the 3D module.

1.1.1. Menu Structure

Table 1 lists the CADMAS MESH MULTI menu structure.

Table 1 — CADMAS MESH MULTI menu structure

Menu Option		Explanation	Page
File (F)	New Project (N)	Starts a new project	8
	Open Project (O)	Opens a project folder	8
	Save Project (S)	Saves a project to a folder	10
	Load ST Data (L)	Loads structures from an ST data set	10
	Discard ST Data (D)	Discards loaded ST data	11
	Load Fault Parameters (F)	Loads fault parameters	12
	Discard Fault Parameters (G)	Discards loaded fault parameters	12
	Import Obstacles (I)	Imports obstacle data	13
	Export Grid Data (E)	Exports grid data to a file	14
	Recent Projects (R)	Opens recent projects	
	Exit (X)	Exits CADMAS MESH MULTI	15
View (V)	Toolbar (T)	Toggles the toolbar display on / off	16
	Status Bar (S)	Toggles the status bar display on / off	16
	Default View (F)	Returns to the default view that fits all topography data, grids, and structure data on the screen	16
Topography Data (E)	Select Topography Data Set (S)	Selects a topography data set	17
	Add Topography Data (A)	Adds topography data to the currently selected topography data set	19
	Adjust Topography Data (E)	Adjusts the currently selected topography data set	20
Grids (G)	Add (A)	Adds a grid	21
	Edit (E)	Edits the currently selected grid	25
	Divide (I)	Divides the currently selected grid	25
	Delete (D)	Deletes the currently selected grid	26
	Delete Child Grids (R)	Deletes all child grids of the currently selected grid	26
	Calculate Porous (P)	Runs a porous calculation for the currently selected grid	28
	Calculate Water Surface (S)	Runs a calculation of water surface variations for the currently selected grid	28
	Launch 3D Module (L)	Launches the 3D module and passes the currently selected grid to the module	30

Table 2 — CADMAS MESH MULTI menu structure (continued)

Obstacles (O)	Specify Cells (C)	Selects a rectangular area on the grid and defines the grid cells in the selected area as an obstacle	31
	Specify Surface (F)	Specifies a line segment and defines the grid surface that crosses the selected line segment as an obstacle	33
	Move (M)	Selects a rectangular area and moves the obstacles contained in the area	34
	Delete (D)	Selects a rectangular area and deletes the obstacles contained in the area	34
	Delete All (A)	Deletes all obstacles	34
Faults (F)	View Parameters (V)	Displays loaded fault parameters	36
	Set Topology Variations (G)	Sets topology variations based on fault parameters	37
Inquiries (I)	ST Data (S)	Toggles the display of ST data on / off	38
	Overlapping Areas (V)	Toggles the display of areas where grids overlap on / off	38
	Obstacles (O)	Toggles the display of obstacles on / off	35
	Water Surface Variations (W)	Displays water surface variations	40
	All Z-Axis Obstacle Cells (A)	Toggles the display of all obstacle cells in the Z axis on / off	40
	Cells Under Porous Threshold (P)	Toggles the display of cells with a porous value under the threshold value on / off	43
Options (P)	Settings (S)	Opens the settings dialog	44
	Color Map Settings (C)	Opens the color map settings dialog	44
Help (H)	About (A)	Displays the current version of CADMAS MESH MULTI	45

1.1.2. Toolbar

Table 3 lists the CADMAS MESH MULTI toolbar structure.

Table 3 — CADMAS MESH MULTI toolbar structure

Button	Title	Description	Page
	New Project	Starts a new project	8
	Open Project	Opens a project folder	8
	Save Project	Saves a project to a folder	10
	Default View	Returns to the default view that fits all topography data, grids, and structure data on the screen	16
	About	Displays the current version of CADMAS MESH MULTI	45

1.2. Basic Operations

1.2.1. View Controls

Views in the drawing area are controlled by holding down the left or right mouse button and dragging. Table 4 gives the controls and their functions.

Table 4 — View controls

Function	Controls	Cursor
Move horizontally	Left-button drag	
Zoom in / zoom out	Right-button drag	

1.2.2. Selecting Areas, etc.

Area selections made when generating a grid, etc. are done with a left-button drag in the drawing area. While dragging, the selected area is displayed as a red square.

2. File (F)

2.1. New Project (N)

This menu option starts a new project. The steps are as follows.

Menu: File (F) → New Project (N)

Toolbar: 

The dialog shown in Figure 3 will open. Enter a name for the project and click the OK button. This action will create a folder with the project's name in the My Documents¥CADMAS-MESH¥Projects folder. Project files will be created in this folder from this point on, as grids are generated, etc.

Figure 3 — Example dialog for starting a new project

2.2. Open Project (O)

This menu option opens a previously saved project. The project opens in the state it was in when last saved. The steps are as follows.

Menu: File (F) → Open Project (O)

Toolbar: 

The dialog shown in Figure 4 will open. Select the project's folder and click the OK button. The project will open in the state it was in when last saved.

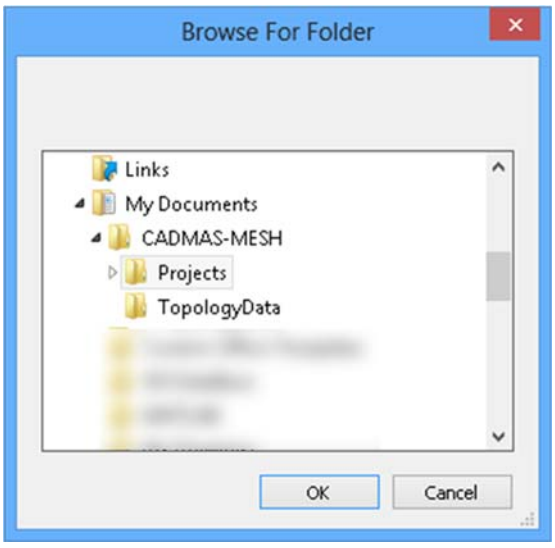

Figure 4 — Example dialog for selecting a project folder

2.3. Save Project (S)

This menu option saves the current project. The steps are as follows.

Menu: File (F) → Save Project (S)

Toolbar: 

After the project is saved, the dialog shown in Figure 5 will open.

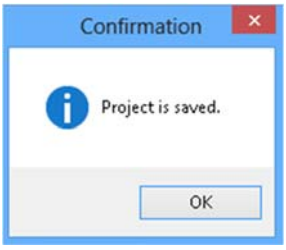

Figure 5 — Example dialog confirming the project has been saved

2.4. Load ST Data (L)

This menu option loads structure data from an ST data set. The steps are as follows.

Menu: File (F) → Load ST Data (L)

The dialog shown in Figure 6 will open. Select the ST data set you want to load and click the Open (O) button. This action will load the ST data set and display structures in gray. Figure 7 provides an example of a display of loaded structures.

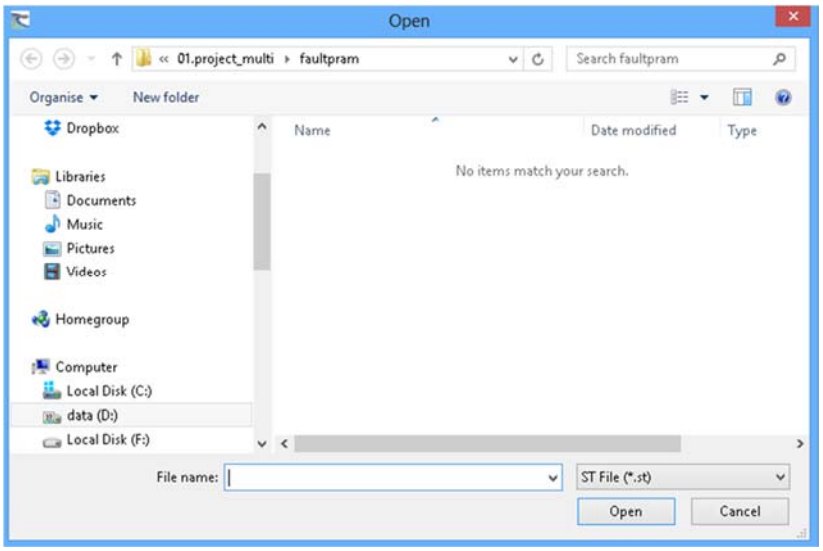

Figure 6 — Example dialog for selecting ST data

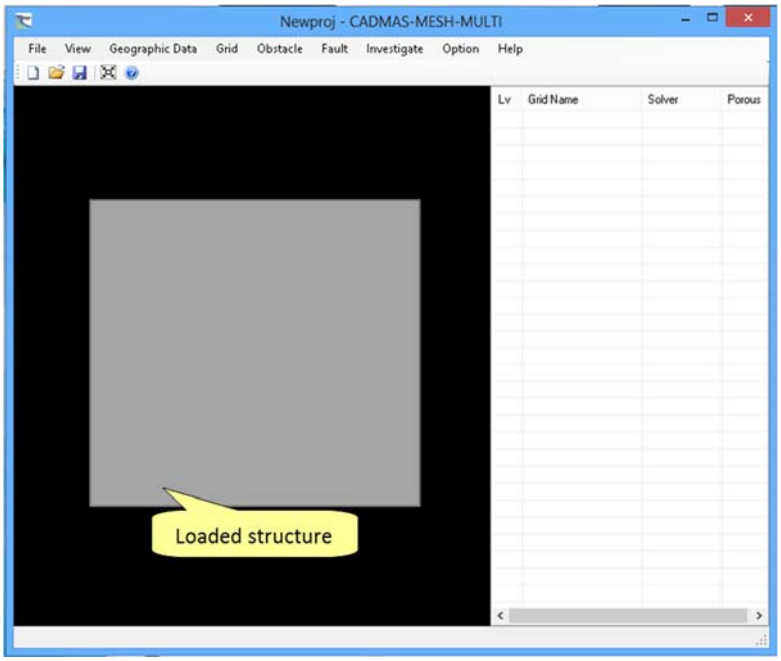

Figure 7 — Example display of a loaded structure

2.5. Discard ST Data (D)

This menu option discards loaded ST data. The steps are as follows.

Menu: File (F) → Discard ST Data (D)

The dialog shown in Figure 8 will open. Click the OK button.

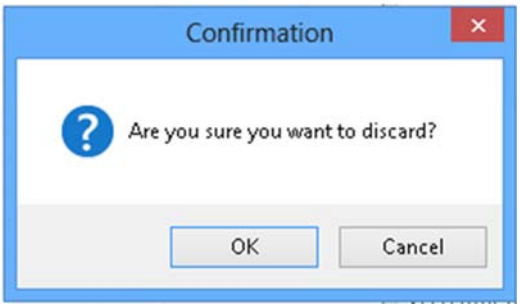

Figure 8 — Example confirmation dialog for discarding ST data

After discarding the ST data, the ST data will be cleared from the drawing area.

2.6. Load Fault Parameters (F)

This menu option loads a fault parameter file. The steps are as follows.

Menu: File (F) → Load Fault Parameters (F)

The dialog shown in Figure 9 will open. Select the fault parameter file you want to load and click the Open (O) button. This action will load the fault parameters.

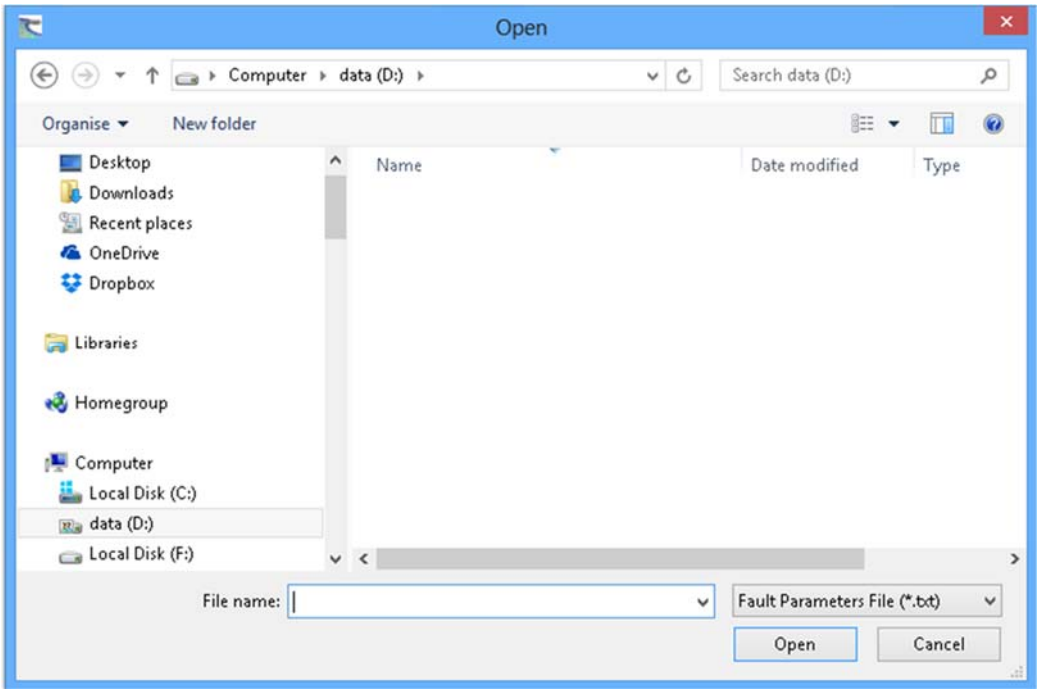

Figure 9 — Example dialog for selecting a fault parameter file

Confirm the loaded fault parameters with the following menu option.

Menu: Faults (F) → View Parameters (V)

2.7. Discard Fault Parameters (D)

This menu option discards loaded fault parameters. The steps are as follows.

Menu: File (F) → Discard Fault Parameters (G)

The dialog shown in Figure 10 will open. Click the OK button.

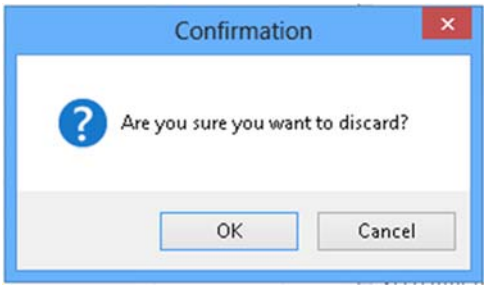

Figure 10 — Example confirmation dialog for discarding fault parameters

2.8. Import Obstacles (I)

This menu option imports obstacle data from an external file. The steps are as follows.

Menu: File (F) → Import Obstacles (I) → Cell Obstacles (C)
Menu: File (F) → Import Obstacles (I) → Surface Obstacles (F)

The type of obstacles you want to import determines which menu option to select.

The dialog shown in Figure 11 will open. Specify the file to import, select the value interpretation policy, and click the OK button.

The value interpretation policy determines how the program maps multiple obstacles to the same cell (or surface), which may occur because the imported obstacles are smaller than the grid width. Table 5 describes the mapping behavior for each policy option.

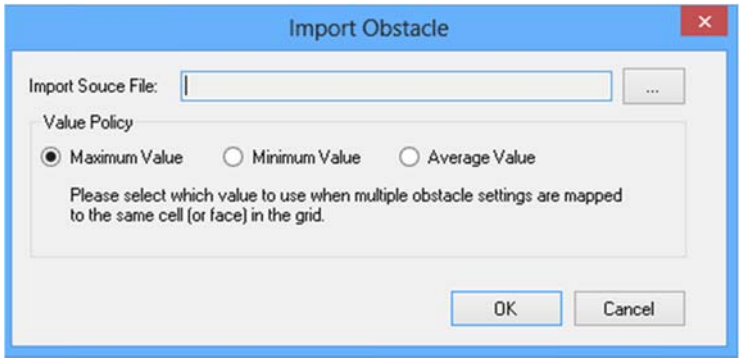

Figure 11 — Example dialog for importing obstacles

Table 5 — Value interpretation policy options

No.	Option	Mapping behavior
1	Largest	Of the obstacles mapped to the same cell (or surface), uses the obstacle with the largest height value
2	Smallest	Of the obstacles mapped to the same cell (or surface), uses the obstacle with the smallest height value
3	Average	Uses the average value of the heights of the obstacles mapped to the same cell (or surface)

2.9. Export Grid Data (E)

This menu option exports grid data to a file. The steps are as follows.

Menu: File (F)→ Export Grid Data (E)

The dialog shown in Figure 12 will open. Enter the project name and click the OK button. This action will export the grid data and open the dialog shown in Figure 13.

Grid data are exported to the project folder. Table 6 lists the grid data files that are exported. The number of exported files depends on the number of generated grids.

Table 6 — List of output files

File Name	Description
<Project name>_data.in	STOC area specification file
<Project name>_<grid name>.sgrid	STOC grid data
<Project name>_<grid name>.str	STOC topography data file
<Project name>_data.env	CADMAS-SURF/3D-MG area specification file
<Project name>_<grid name>.cgrid	CADMAS-SURF/3D-MG grid data file

If the number of data points in the porous calculation for the grid does not match the number of grid divisions because, for example, the porous calculation was not run after editing the grid in the 3D module, the dialog shown in Figure 14 will open and the export will be canceled.

If the porous calculation results are older than the last updated or saved grid data, or if the porous calculation has not been run, the dialog shown in Figure 15 will open. You can either click the OK button to export the grid data as is or click the Cancel button, run the porous calculation for the required grid, and then export the grid data.

If a file starting with "<Project name>_" already exists, the dialog shown in Figure 16 will open. Click the OK button to export the data by deleting and overwriting the existing files.

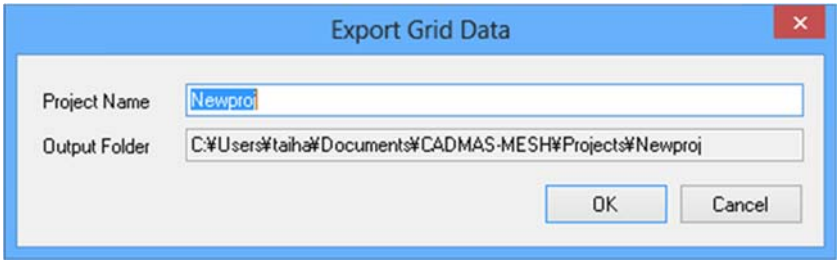

Figure 12 — Example dialog for exporting grid data

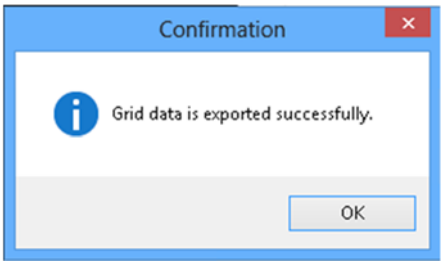

Figure 13 — Example dialog confirming the grid data has been exported

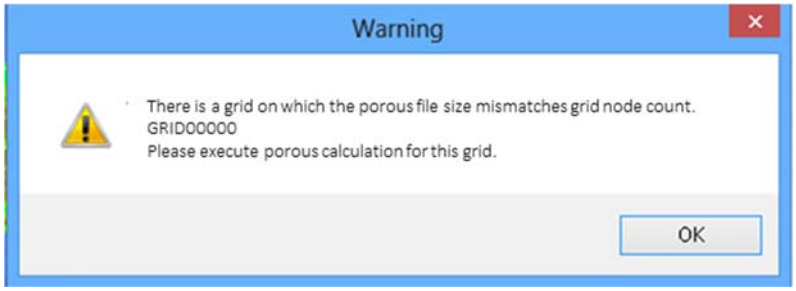

Figure 14 — Example 1 of a warning dialog while exporting grid data

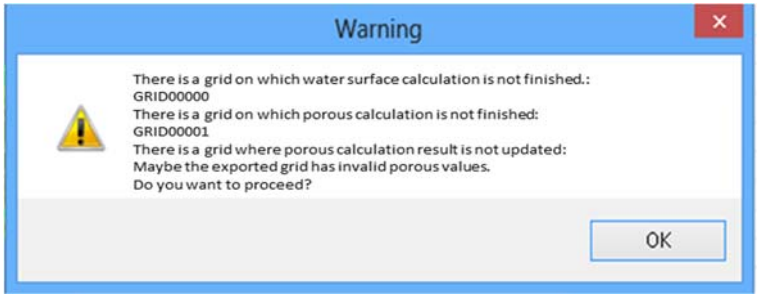

Figure 15 — Example 2 of a warning dialog while exporting grid data

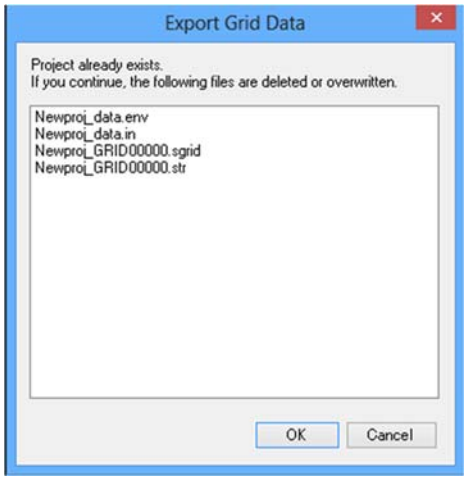

Figure 16 — Example dialog indicating that the project already exists

2.10. Exit (X)

This menu option exits CADMAS MESH MULTI. The steps are as follows.

Menu: File (F) → Exit (X)

3. View (V)

3.1. Toolbar (T)

This menu option toggles the toolbar display on and off. The steps are as follows.

Menu: View (V) → Toolbar (T)

A check appears beside this menu option when the toolbar is displayed.

3.2. Status (S)

This menu option toggles the status bar display on and off. The steps are as follows.

Menu: View (V) → Status (S)

A check appears beside this menu option when the status bar is displayed.

3.3. Default View (F)

This menu option returns to the default view that fits all topography data, grids, and structure data on the screen. The steps are as follows.

Menu: View (V) → Default View (F)

Toolbar: 

4. Topography Data (E)

4.1. Select Topography Data Set (S)

This menu option selects the topography data set used in grid porous calculations. This menu option is also used to add, edit, and delete topography data sets. The steps are as follows.

Menu: Topography Data (E) → Select Topography Data Set (S)

The dialog shown in Figure 17 will open. After selecting the topography data set you want to use and clicking the OK button, the dialog will close and the topography data set will load.

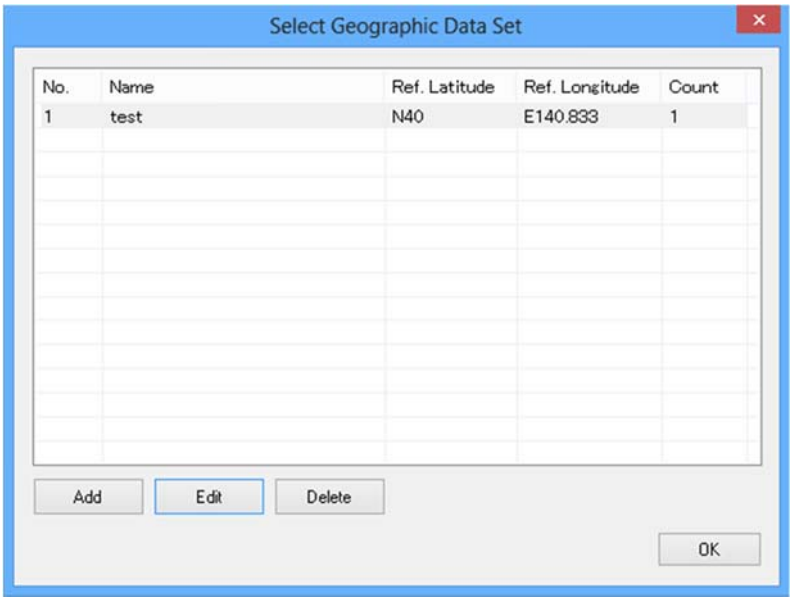

Figure 17 — Example dialog for selecting topography data sets

To add a new topography data set, click the Add (A) button on the dialog shown in Figure 17. The dialog shown in Figure 18 will open. After entering the data set's name and reference point, click the OK button. The dialog shown in Figure 19 will open. After setting conditions and clicking the OK button, the topography data will be added to the newly created topography data set.

See Chapter 11 for a description of the formats of the files that can be selected on the dialog shown in Figure 20.

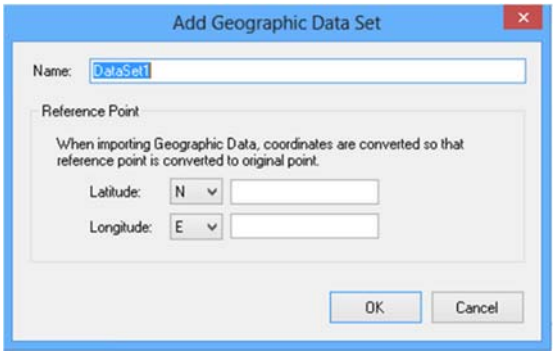

Figure 18 — Example dialog for adding topography data sets

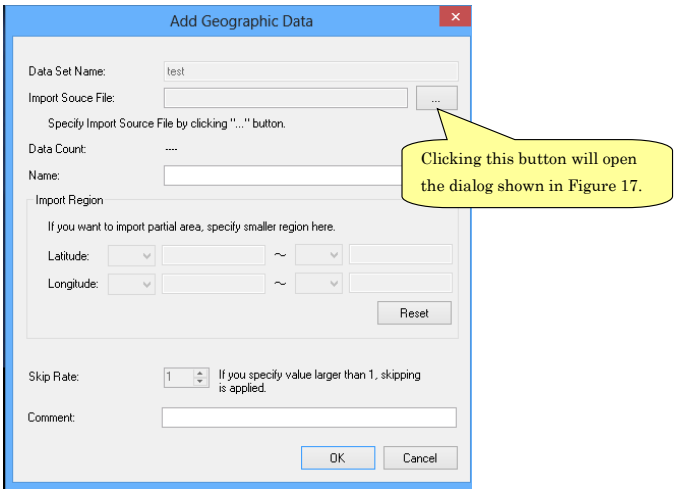

Figure 19 — Example dialog for adding topography data

Figure 20 — Example dialog for selecting the source file for importing topography data

To edit an existing topography data set, select the topography data set you want to edit on the dialog shown in Figure 17 and click the Edit (E) button. The dialog shown in Figure 21 will open. On this dialog, you can add, edit, and delete topography data. See Section 4.3 for a description of the topography data editing operations on this dialog.

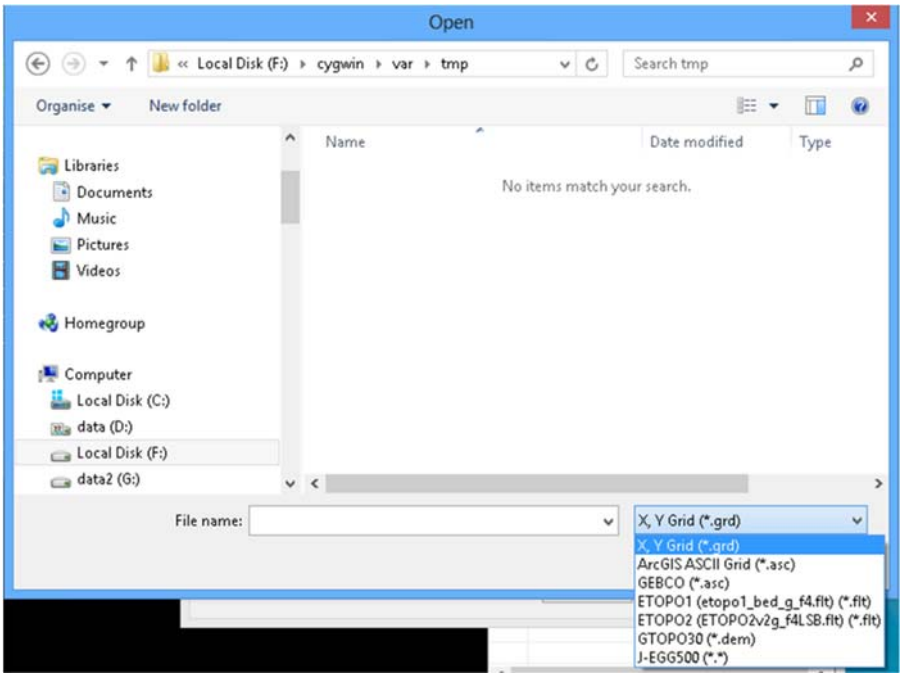

Figure 21 — Example dialog for editing topography data sets

To delete a topography data set, select the topography data set you want to delete on the dialog shown in Figure 17 and click the Delete (D) button.

4.2. Add Topography Data (A)

This menu option adds new topography data to the currently selected topography data set. The steps are as follows.

Menu: Topography Data (E) → Add Topography Data (A)

The dialog shown in Figure 19 will open. Adjust the settings as necessary and click the OK button.

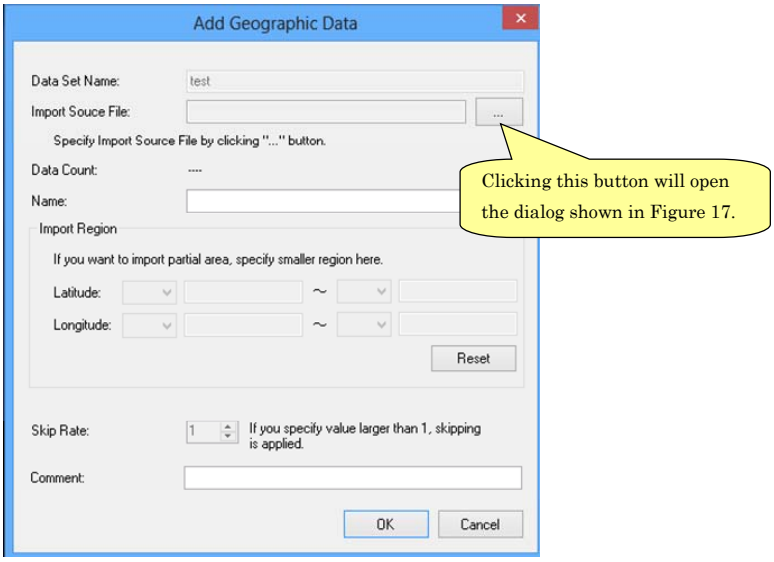

Figure 19 — Example dialog for adding topography data (figure shown previously)

4.3. Adjust Topography Data (E)

This menu option adjusts the currently selected topography data set. The steps are as follows.

Menu: Topography Data (E) → Adjust Topography Data (E)

The dialog shown in Figure 21 will open. Adjust the settings as necessary and click the OK button.

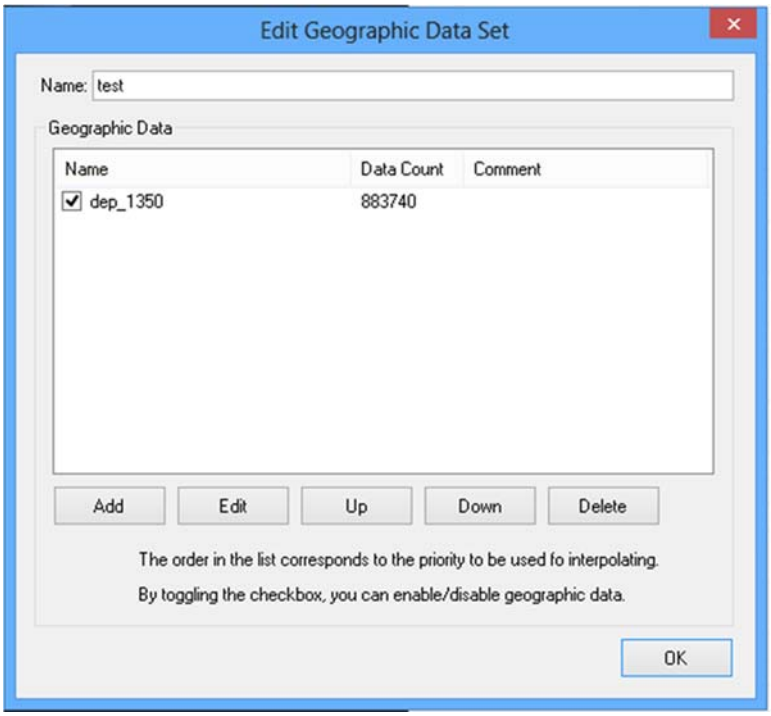

Figure 21 — Example dialog for editing topography data sets (figure shown previously)

To add topography data, click the Add (A) button on the dialog shown in Figure 21. The dialog hown in Figure 19 will open. On this dialog, set conditions and click the OK button.

To edit topography data, select the topography data you want to edit from the list on the dialog shown in Figure 21 and click the Edit (E) button. The dialog shown in Figure 19 will open. On this dialog, set conditions and click the OK button.

To change the priority order of topography data, select the topography data you want to move and click either the Up (U) or Down (D) button. This will move the topography data up or down the list.

To delete topography data, select the topography data you want to delete from the list on the dialog shown in Figure 21 and click the Delete button.

To enable or disable topography data, check the check box beside the appropriate topography data. Checked topography data are enabled.

5. Grids (G)

5.1. Add (A)

This menu option adds a grid. There are two ways to add grids: one when generating the first grid and one when generating a child grid for an existing grid.

Generating the first grid:

The steps are as follows for generating a grid.

Menu: Grid (G) → Add (A)

When area selections are enabled, select the area in which to generate the grid with a left drag in the main window. Releasing the mouse's left button will open the dialog shown in Figure 22. On this dialog, set parameters and click the OK button.

If you check the Run Porous Calculation When Generating Grid check box and click the OK button, the program will automatically run a porous calculation after generating the grid. Note that the ST data is not be reflected in the porous calculation. If the generated grid includes ST data in the grid's area, run the porous calculation in the 3D module.

If an area without topography data is included in the specified grid area and the OK button is clicked, the warning dialog shown in Figure 23 will open. If a Zmin value is specified that is larger than the minimum Z value of the topography data in the specified area, the warning dialog shown in Figure 24 will open.

When the grid is completely generated, the grid will appear on the main window as shown in Figure 25.

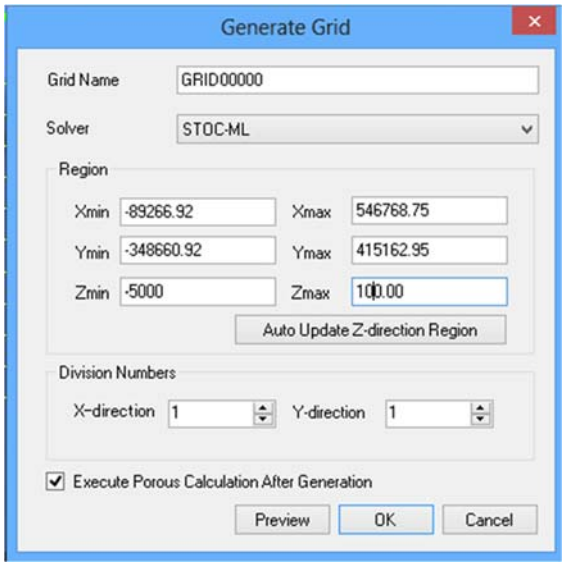

Figure 22 — Example dialog for generating a grid

Figure 23 — Example warning dialog indicating insufficient topography data for porous calculations

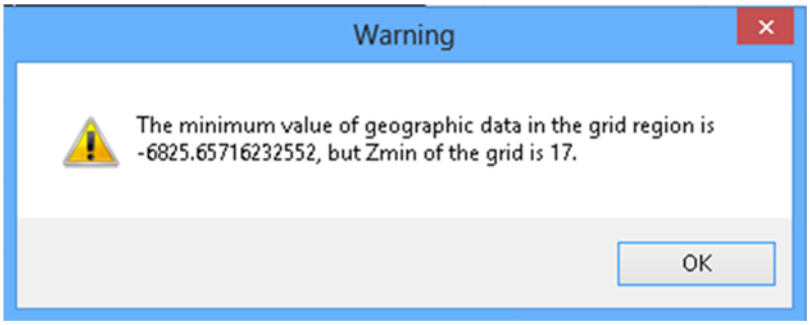

Figure 24 — Example warning dialog about the minimum Z-axis value when generating a grid

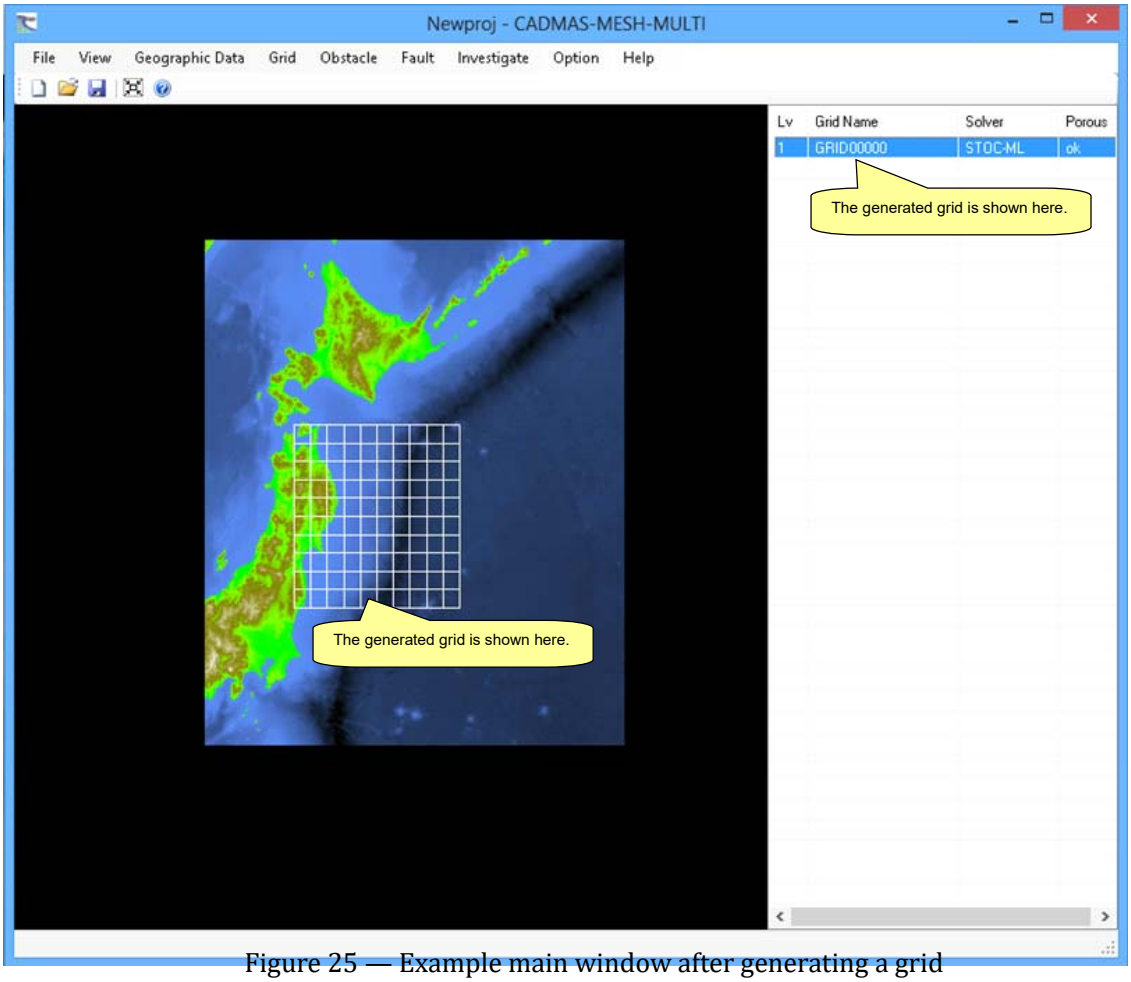

Figure 25 — Example main window after generating a grid

Generating a child grid for an existing grid:

Select the grid you want to be the parent grid from the list of generated grids and carry out the following steps.

Menu: Grid (G) → Add (A)

When area selections are enabled, select the area in which to generate the grid with a left drag in the main window. Releasing the mouse's left button will open the dialog shown in Figure 26. On this dialog, set parameters and click the OK button. Zmin can only be set when the parent grid is not divided in the Z axis. Note that Zmin cannot be set smaller than the parent grid's Zmin.

When the grid is completely generated, the grid will appear on the main window as shown in Figure 27.

You can make any number of nested child grids by repeating the child grid generation process.

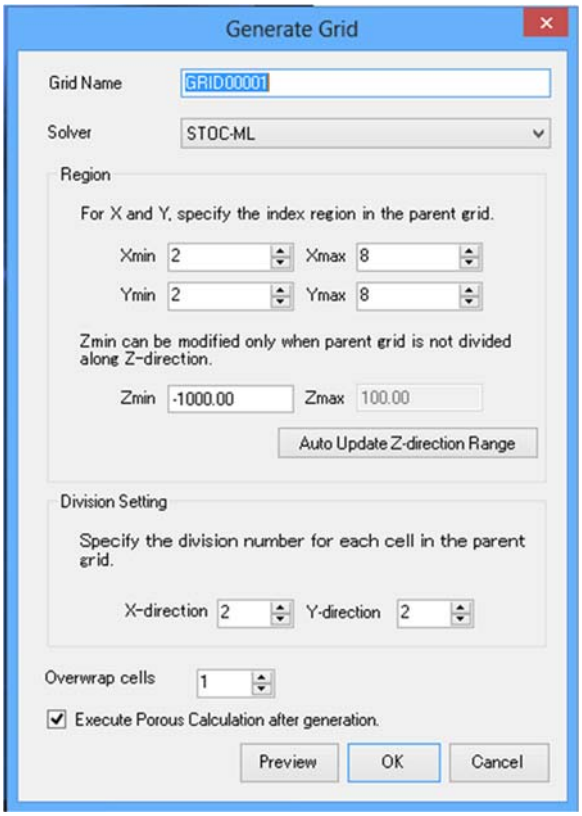

Figure 26 — Example dialog for generating a child grid

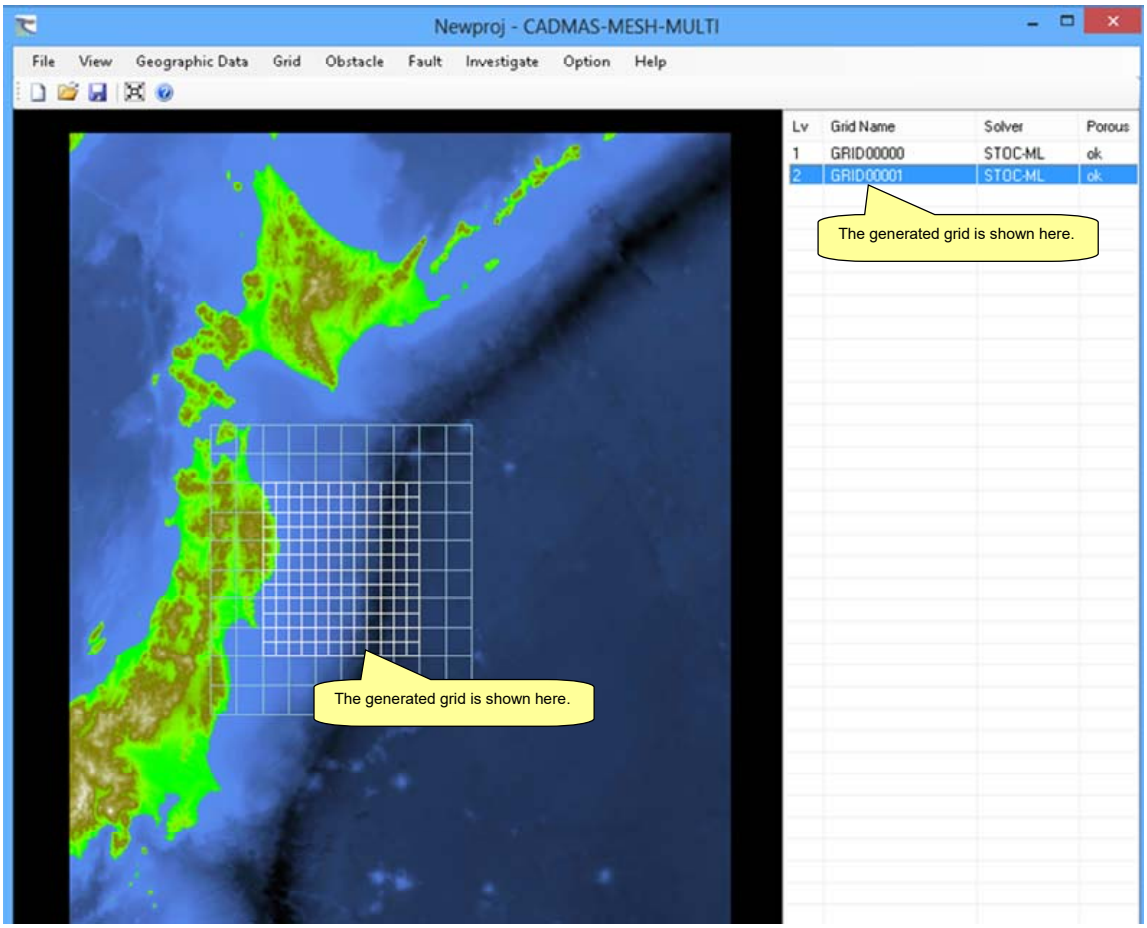

Figure 27 — Example main window after generating a child grid

5.2. Edit (E)

This menu option edits the currently selected grid. The steps are as follows.

Menu: Grid (G) → Edit (E)

The dialog shown in either Figure 22 or Figure 26 will open depending on whether the currently selected grid is the highest level grid or a child grid. On this dialog, change the settings and click the OK button. Note that if the edited grid has been divided in the Z axis with the 3D module or if a child grid exists in the edited grid, the dialog shown in Figure 28 will open. Click the OK button to proceed.

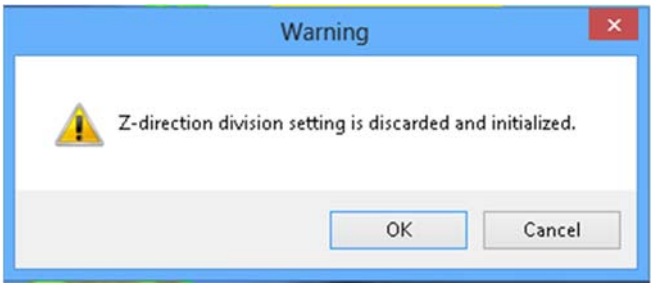

Figure 28 — Example dialog to confirm the initialization of divisions or deleting the child grids when editing a grid

5.3. Divide (I)

This menu option generates a grid that divides the currently selected grid. The original grid is deleted. The steps are as follows.

Menu: Grid (G) → Divide (I)

The dialog shown in Figure 29 will open. Specify the number of divisions in the X-axis direction and the Y-axis direction and click the OK button. The new grid inherits the selected solver, number of overlapping cells, and other properties from the original grid. The dialog shown in Figure 30 will open if another child grid exists in the parent grid of the currently selected grid. And the dialog shown in Figure 31 will open if the currently selected grid is too small to be divided.

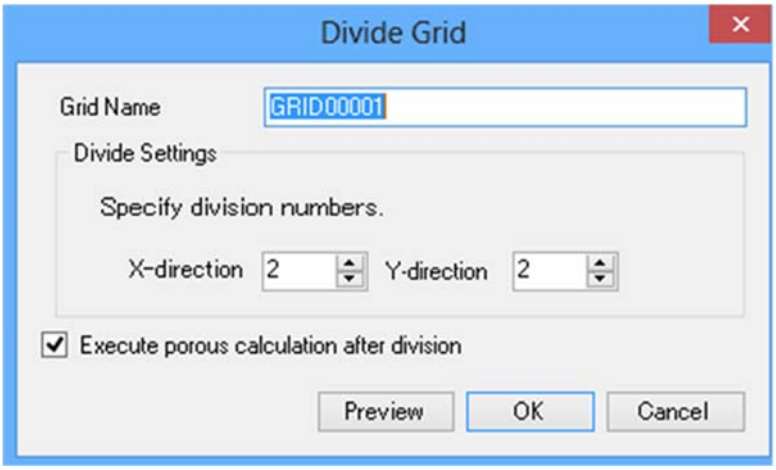

Figure 29 — Example dialog for dividing a grid

Figure 30 — Example dialog indicating the presence of another child grid besides the grid being divided

Figure 31 — Example dialog indicating that the selected grid cannot be divided

5.4. Delete (D)

This menu option deletes the currently selected grid. The steps are as follows.

Menu: Grid (G) → Delete (D)

The dialog shown in Figure 32 will open. Click the OK button to confirm the deletion.

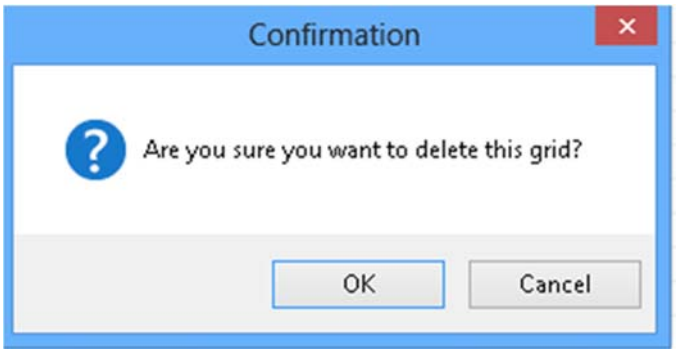

Figure 32 — Example dialog to confirm the deletion of a grid

5.5. Delete Child Grids (R)

This menu option deletes all child grids of the currently selected grid. The steps are as follows.

Menu: Grid (G) → Delete Child Grids (R)

The dialog shown in Figure 33 will open. Click the OK button to confirm the deletion. Note that the dialog shown in Figure 34 will open if there are no child grids in the currently selected grid.

Figure 33 — Example dialog to confirm the deletion of child grids

Figure 34 — Example dialog indicating that there are no child grids

5.6. Calculate Porous (P)

This menu option runs a porous calculation for the currently selected grid. The steps are as follows.

Menu: Grid (G) → Calculate Porous (P)

The porous calculation will run even if the selected grid's area contains areas without topography data. In this case, the warning dialog shown in Figure 35 will open. Additionally, the porous calculation will run even if the minimum Z-axis value of the selected grid is larger than the topography data's minimum Z value, but the warning dialog shown in Figure 36 will open.

Note that the ST data is not be reflected in the porous calculation. If the grid includes ST data in the grid's area, run the porous calculation in the 3D module.

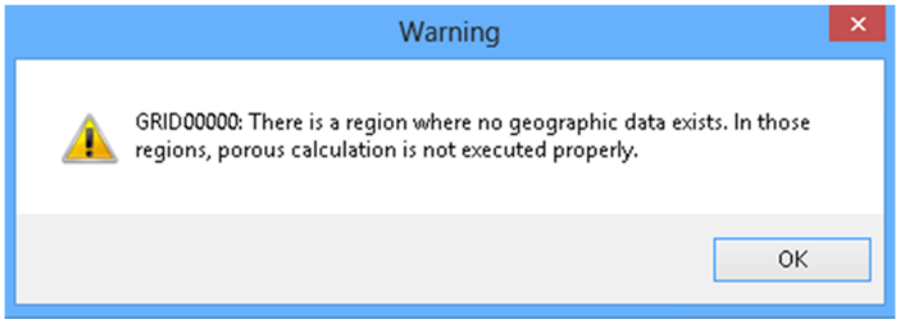

Figure 35 — Example warning dialog indicating insufficient topography data for porous calculations

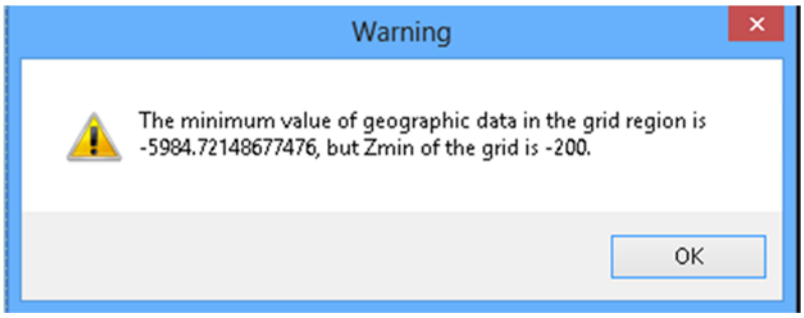

Figure 36 — Example warning dialog about the minimum Z-axis value

5.7. Calculate Water Surface (S)

This menu option runs a calculation of water surface variations for the currently selected grid based on fault parameters. The steps are as follows.

Menu: Grid (G) → Calculate Water Surface (S)

When the calculation finishes, the dialog shown in Figure 37 will open.

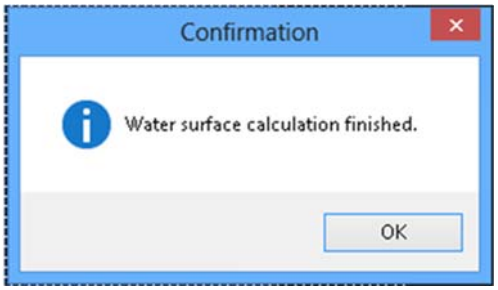

Figure 37 — Example dialog indicating the water surface calculation is complete

Note that a warning message will appear if fault parameters have not been loaded. In this case, load fault parameters with the following menu selection.

Menu: File (F) → Load Fault Parameters (F)

5.8. Launch 3D Module (L)

This menu option launches the 3D module and passes the currently selected grid to the module. The steps are as follows.

Menu: Grid (G) → Launch 3D Module (L)

The 3D module window as shown in Figure 38 will open and porous calculations and other operations will run. The 3D module closes on completion of the processing.

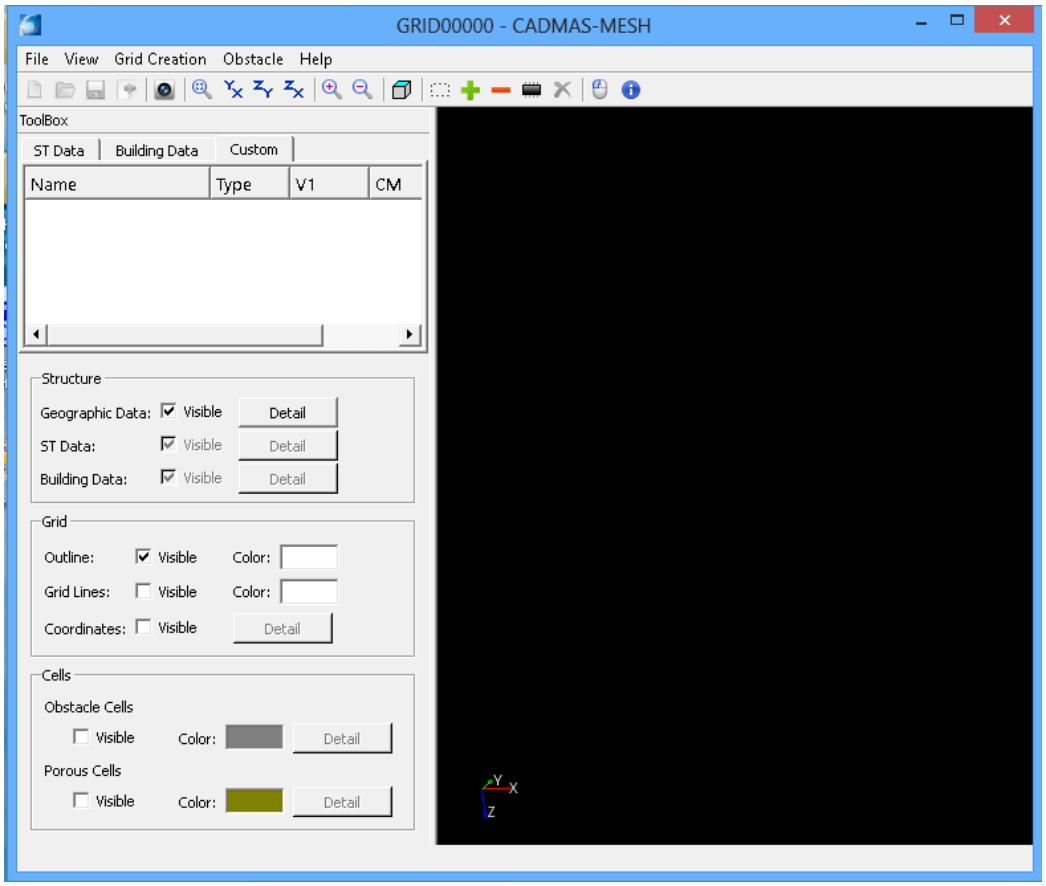

Figure 38 — Example 3D module window

6. Obstacles (O)

6.1. Specify Cells (C)

6.1.1. Rectangular Area (R)

This menu option selects a rectangular area on a grid and defines the grid cells contained in the selection as an obstacle. The steps are as follows.

Menu: Obstacles (O) → Specify Cells (C) → Rectangular Area (R)

After making the menu selection, hold down the left mouse button and drag in the drawing area to select the area you want to define as an obstacle. When you release the left mouse button, the dialog shown in Figure 39 will open. Confirm the coordinates of the selected area, set the height of the obstacle, and click the OK button.

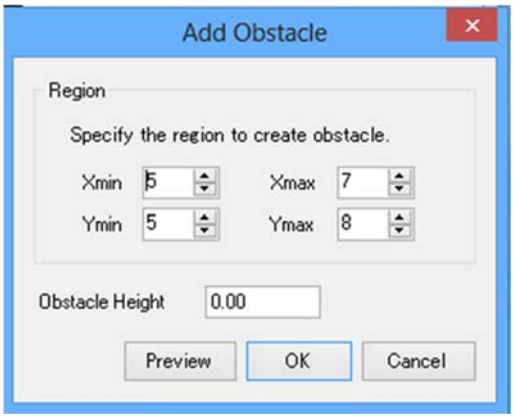

Figure 39 — Example dialog for defining an obstacle (by specifying cells)

6.1.2. Line Area (L)

This menu option draws a line segment on a grid and defines cells on the line segment as obstacles. The steps are as follows.

Menu: Obstacles (O) → Specify Cells (C) → Line Area (L)

After making the menu selection, hold down the left mouse button and drag in the drawing area to select with a line segment the area you want to define as an obstacle. When you release the left mouse button, the dialog shown in Figure 40 will open. Confirm the coordinates of the selected area, set the height of the obstacle, and click the OK button.

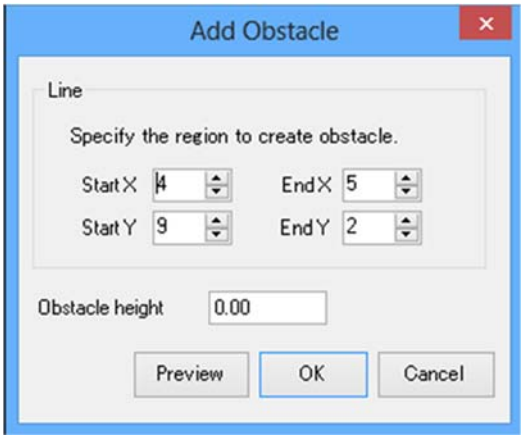

Figure 40 — Example dialog for defining an obstacle (by specifying cells)

6.2. Specify Surface (F)

This menu option specifies a line segment joining two grid points and defines the grid surface that crosses the selected line segment as an obstacle. The steps are as follows.

Menu: Obstacles (O) → Specify Surface (F)

After making the menu selection, hold down the left mouse button and drag in the drawing area to select the line segment. When you release the left mouse button, the dialog shown in Figure 41 will open. Confirm the coordinates of the selected area, set the height of the obstacle, and click the OK button.

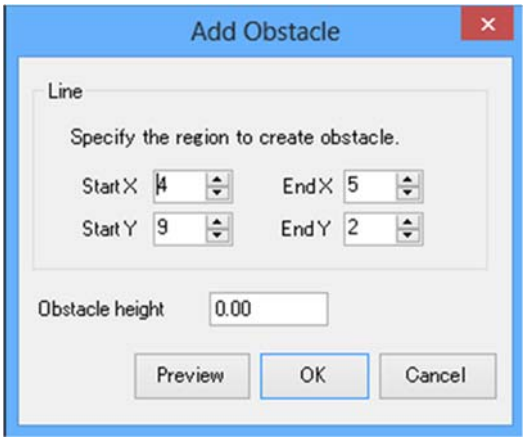

Figure 41 — Example dialog for defining an obstacle (by specifying a surface)

6.3. Move (M)

This menu option moves obstacles. The steps are as follows.

Menu: Obstacles (O) → Move (M)

After making the menu selection, hold down the left mouse button and drag in the drawing area to select a rectangular area containing the obstacle or obstacles you want to move. When you release the left mouse button, the dialog shown in Figure 42 will open. Confirm the coordinates of the selected area containing the obstacles to be moved, set how far to move the obstacles, and click the OK button.

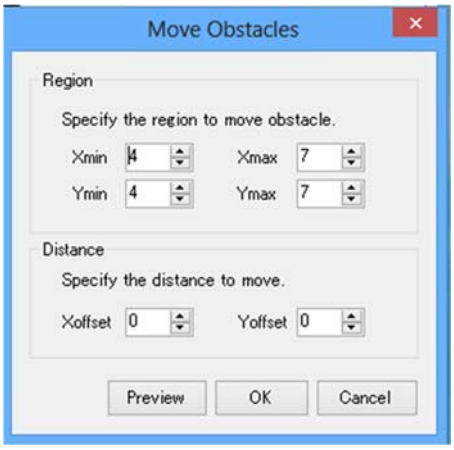

Figure 42 — Example dialog for moving obstacles

6.4. Delete (D)

This menu option deletes obstacles. The steps are as follows.

Menu: Obstacles (O) → Delete (D)

After making the menu selection, hold down the left mouse button and drag in the drawing area to select a rectangular area containing the obstacle or obstacles you want to delete. When you release the left mouse button, the dialog shown in Figure 43 will open. Confirm the coordinates of the selected area containing the obstacles to be deleted and click the OK button to confirm the deletion.

Figure 43 — Example dialog for deleting obstacles

6.5. Delete All (A)

This menu option deletes all obstacles. The steps are as follows.

Menu: Obstacles (O) → Delete All (A)

The dialog shown in Figure 44 will open. Click the OK button to confirm the deletion.

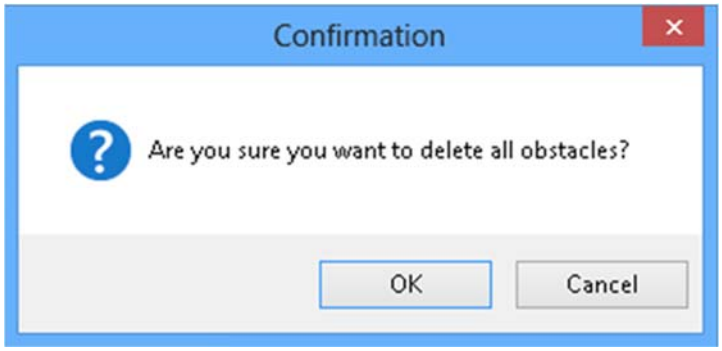

Figure 44 — Example dialog to confirm the deletion of all obstacles

7. Faults (F)

7.1. View Parameters (V)

This menu option displays loaded fault parameters. The steps are as follows.

Menu: Faults (F) → View Parameters (V)

The dialog shown in Figure 45 will open. Click the Previous and Next buttons to move through the loaded parameter sets in order.

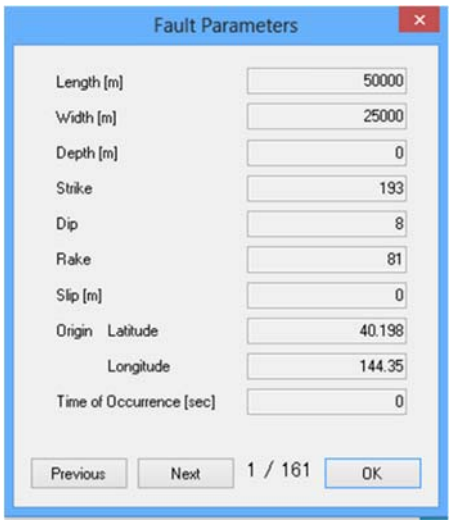

Figure 45 — Example dialog for viewing fault parameters

Note that a warning message will appear if fault parameters have not been loaded. In this case, load fault parameters with the following menu selection.

Menu: File (F) → Load Fault Parameters (F)

7.2. Set Topology Variations (G)

This menu option sets topology variations based on the loaded fault parameters. The steps are as follows.

Menu: Faults (F) → Set Topology Variations (G)

The dialog shown in Figure 46 will open. Change the setting with the radio buttons and click the OK button.

Note that when this setting is changed, porous calculations must be run again for each grid.

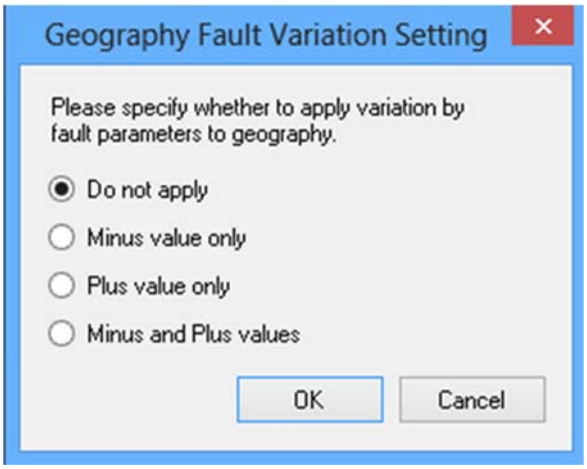

Figure 46 — Example dialog for setting variations based on topology fault parameters

8. Inquiries (I)

8.1. ST Data (S)

This menu option toggles the display of ST data on and off. The steps are as follows.

Menu: Inquiries (I) → ST Data (S)

By default, ST data are displayed.

8.2. Overlapping Areas (V)

This menu option toggles the display of areas where grids overlap. The steps are as follows.

Menu: Inquiries (I) → Overlapping Areas (V)

By default, overlapping areas are hidden. Figure 47 provides an example of the main window with overlapping areas hidden and displayed.

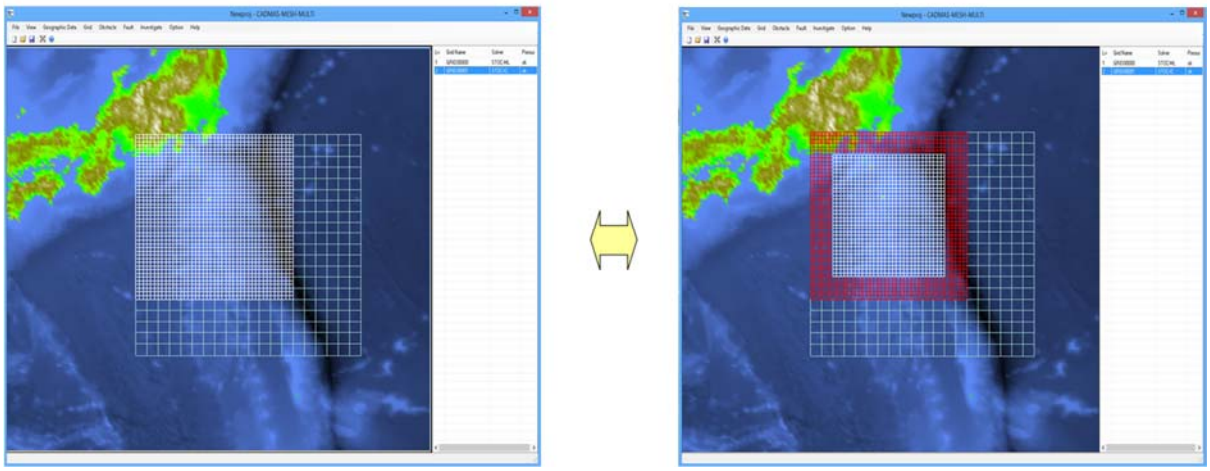

Figure 47 — Example of the main window with overlapping areas hidden and displayed

8.3. Obstacles (O)

This menu option toggles the display of obstacles set with the obstacle definition function. The steps are as follows.

Menu: Inquiries (I) → Obstacles (O)

By default, obstacles are displayed. Obstacles defined by specifying cells are displayed as colored rectangles and obstacles defined by specifying surfaces are displayed as broken colored lines. Figure 48 provides an example of the main window with obstacles hidden and displayed.

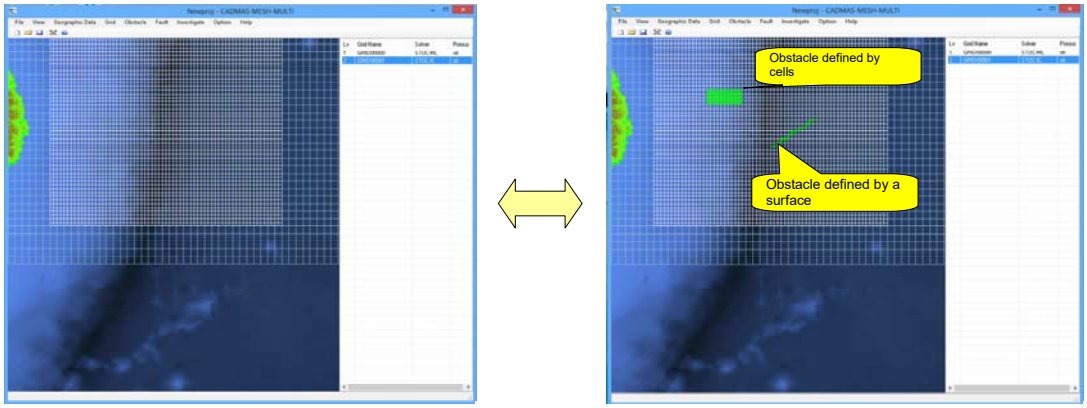

Figure 48 — Example of the main window with obstacles hidden and displayed

8.4. Water Surface Variations (S)

This menu option displays water surface variations. The steps are as follows.

Menu: Inquiries (I) → Water Surface Variations (S)

Water surface variations are displayed as shown in Figure 49.

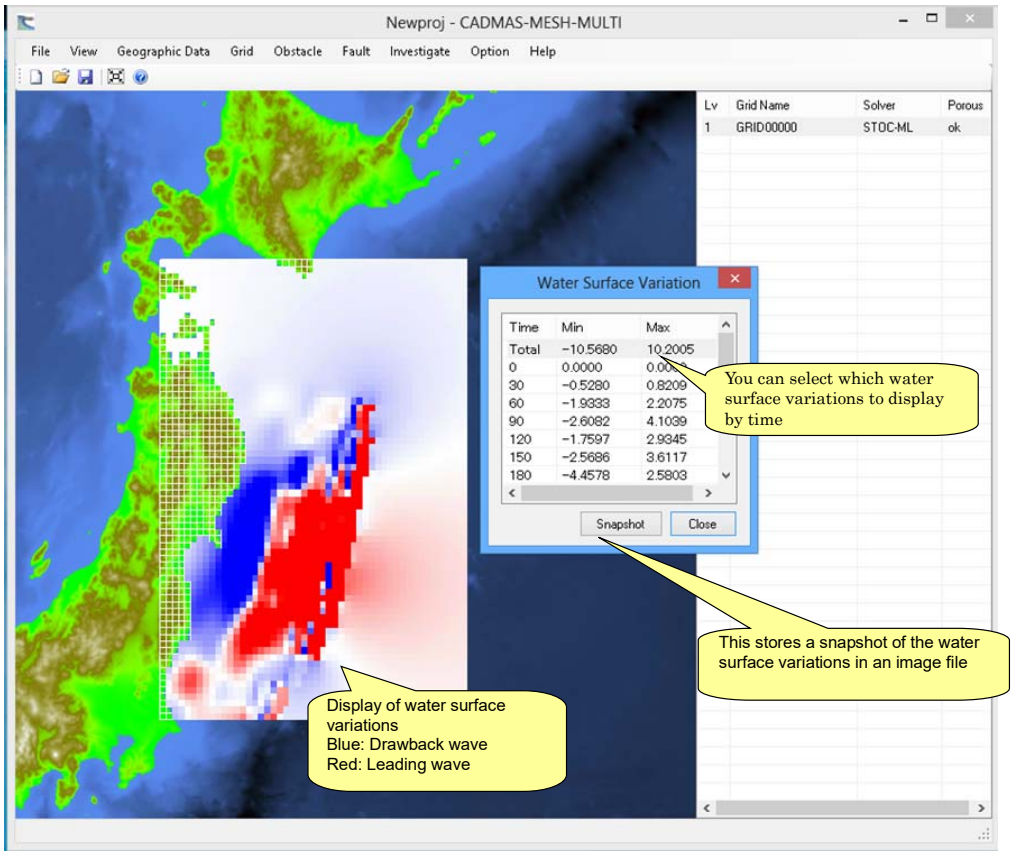

Figure 49 — Example of a display of water surface variations

Note that a warning message will appear if fault parameters have not been loaded. In this case, load fault parameters with the following menu selection.

Menu: File (F) → Load Fault Parameters (F)

8.5. All Z-Axis Obstacle Cells (A)

This menu option toggles the display of all cells defined as obstacles in the Z axis on and off. The steps are as follows.

Menu: Inquiries (I) → All Z-Axis Obstacle Cells (A)

By default, all obstacle cells in the Z axis are hidden. When displayed, obstacle cells are represented by dark gray rectangles. Figure 50 provides an example.

Note that the dialog shown in Figure 51 will open if the number of data points resulting from the porous calculation for the selected grid does not match the number of grid divisions, which could happen if the porous calculation is not run after editing the grid in the 3D module. In this case, no Z-axis obstacle cells will be displayed for the grid.

Furthermore, the dialog shown in Figure 52 will open if the porous calculation data for the selected grid are older than the most recently updated or stored grid data. In this case, all Z-axis obstacle cells will be displayed based on the old porous calculation data.

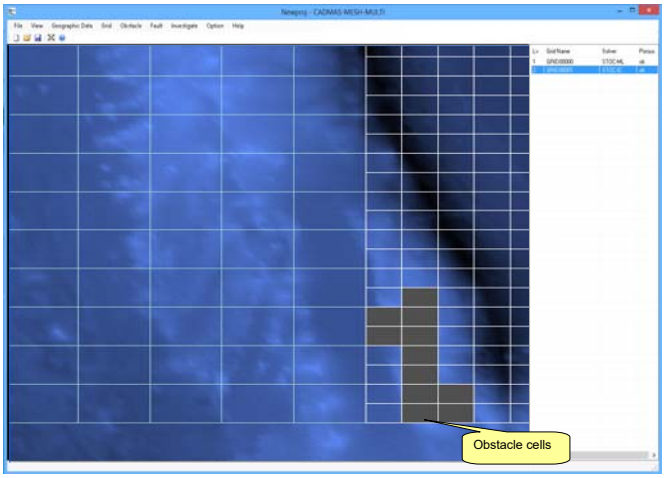

Figure 50 — Example of the main window with all Z-axis obstacle cells displayed

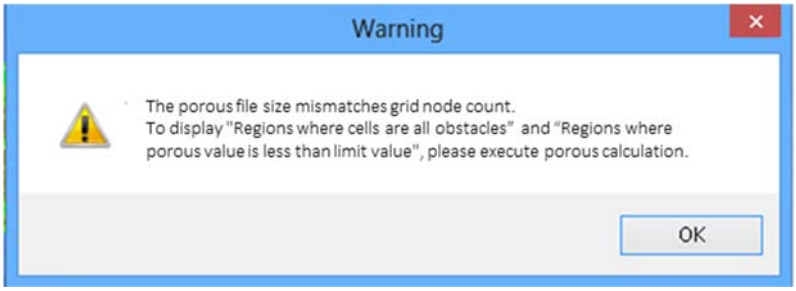

Figure 51 — Example 1 of a dialog warning about the display of all Z-axis obstacle cells and the display of cells under the porous threshold

Figure 52 — Example 2 of a dialog warning about the display of all Z-axis obstacle cells and the display of cells under the porous threshold

8.6. Cells Under Porous Threshold (P)

This menu option toggles the display of cells with a porous value under the threshold value on and off. The steps are as follows.

Menu: Inquiries (I) → Cells Under Porous Threshold (P)

By default, cells under the porous threshold value are hidden. When displayed, the cells are represented by red rectangles. Figure 53 provides an example.

Note that, as in Section 8.4, the dialog shown in Figure 51 will open if the number of data points resulting from the porous calculation for the selected grid does not match the number of grid divisions, and the dialog shown in Figure 52 will open if the porous calculation data for the selected grid are older than the most recently updated or stored grid data.

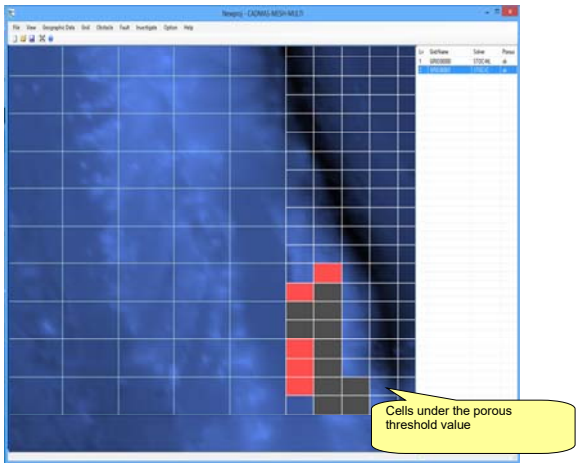

Figure 53 — Example of the main window with cells under the porous threshold value displayed

9. Options (P)

9.1. Settings (S)

This menu option opens the settings dialog. The steps are as follows.

Menu: Options (P) → Settings (S)

The dialog shown in Figure 54 where various settings are made will open.

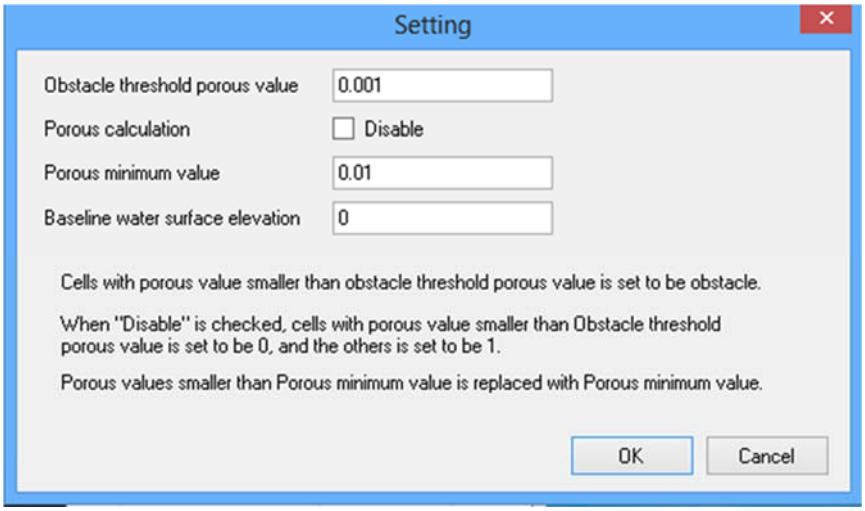

Figure 54 — Example of the settings dialog

9.2. Color Map Settings (C)

This menu option opens the color map settings dialog. The steps are as follows.

Menu: Options (P) → Color Map Settings (C)

The dialog shown in Figure 55 where various settings are made will open.

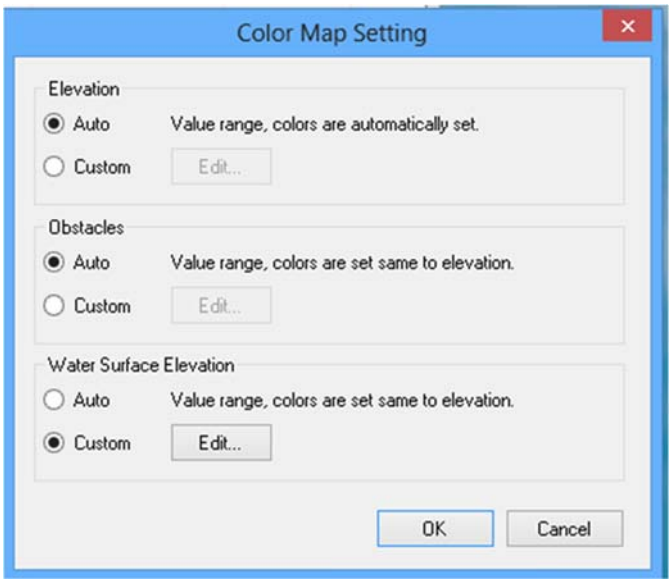

Figure 55 — Example of the color map settings dialog

Selecting Custom on the dialog shown in Figure 55 and clicking the Edit button will open the dialog shown in Figure 56. On this new dialog, you can make detailed color map settings.

Figure 56 — Example dialog for making detailed color map settings

10. Help (H)

10.1. About (A)

This menu option displays the current version of CADMAS MESH MULTI. The steps are as follows.

Menu: Help (H) → About (A)

Toolbar: 

The dialog shown in Figure 57 will open, where you can check the version information.

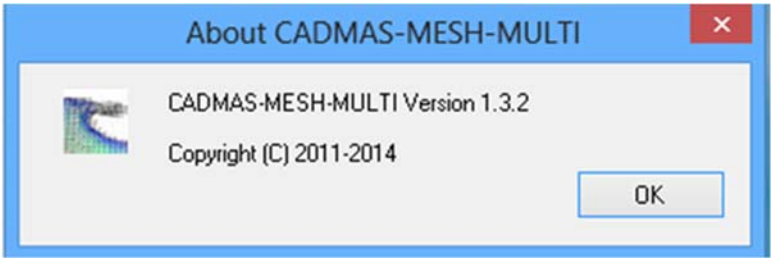

Figure 57 — Example of the version information dialog

11. File Formats for Topography Data

CADMAS MESH MULTI can import topography data in the formats listed below.

- GEBCO
- GTOPO30
- ETOPO2
- J-EGG500
- X, Y (original format)

This chapter describes each of these file formats in detail.

11.1. GEBCO

GEBCO provides bathymetry data sets covering all areas of the earth. The GEBCO Website can be accessed at the following URL (see Figure 58).

http://www.gebco.net/

Figure 58 — Screenshot of the GEBCO Website

11.2. GTOPO30

GTOPO30 provides global digital elevation model (DEM) data with a horizontal grid spacing of 30 arc seconds (approximately 1 kilometer). The GTOPO30 Website can be accessed at the following URL.

http://www1.gsi.go.jp/geowww/globalmap-gsi/gtopo30/gtopo30.html

The Website displays a map divided into areas (see Figure 59). Clicking on an area of interest on the map moves to a window (see Figure 60) where you can download the data set for that area. Download the data set by clicking on the data name link (e.g., E100N40).

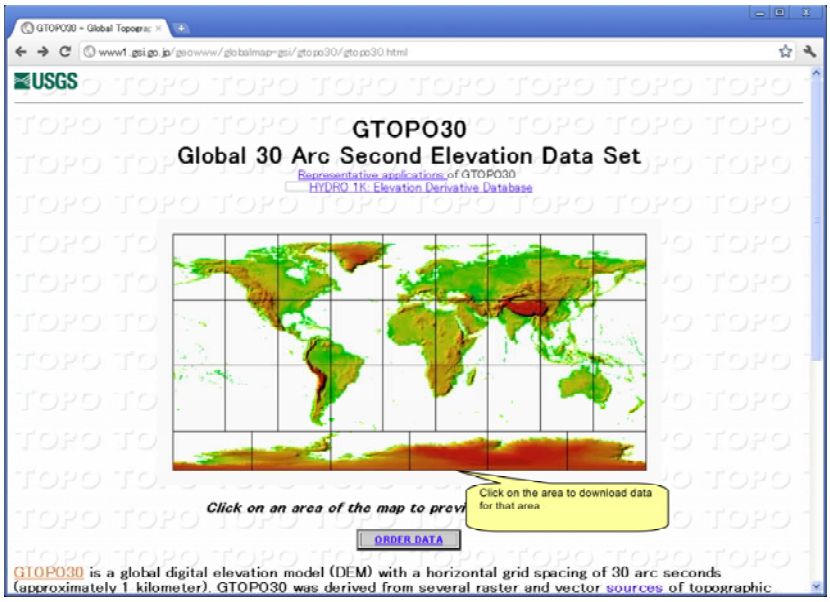

Figure 59 — Screenshot of the GTOPO30 Website

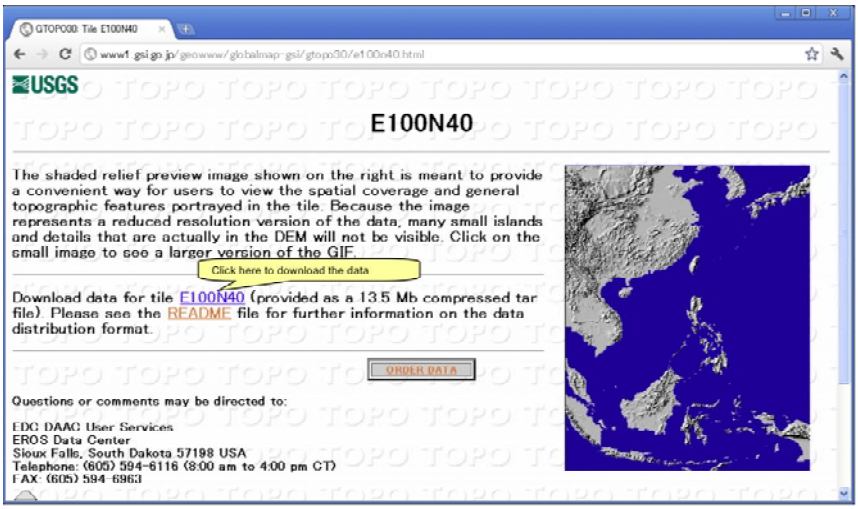

Figure 60 — Screenshot of GTOPO30 data download page

11.3. ETOPO2

ETOPO2 provides global digital elevation model (DEM) data with a horizontal grid spacing of two minutes. The ETOPO2 Website can be accessed at the following URL.

http://www.ngdc.noaa.gov/mgg/global/etopo2.html

You can download raw binary data from the Website and import it into CADMAS MESH MULTI. Figure 61 provides a screenshot of the Website and Figure 62, a screenshot of the download page.

Figure 61 — Screenshot of the ETOPO2 Website

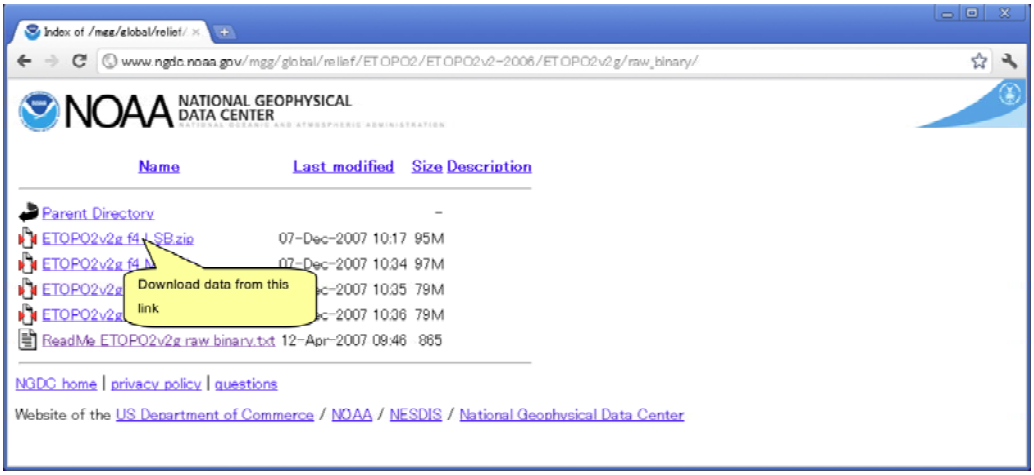

Figure 62 — Screenshot of the ETOPO2 download page

11.4. J-EGG500

J-EGG500 provides bathymetry data sets for Japan's coastal waters on a 500-meter grid. The J-EGG500 Website can be accessed at the following URL. Figures 63, 64, and 65 provide screenshots of the Website, the data range selection page, and the download page.

http://www.jodc.go.jp/data_set/jodc/jegg_intro_j.html

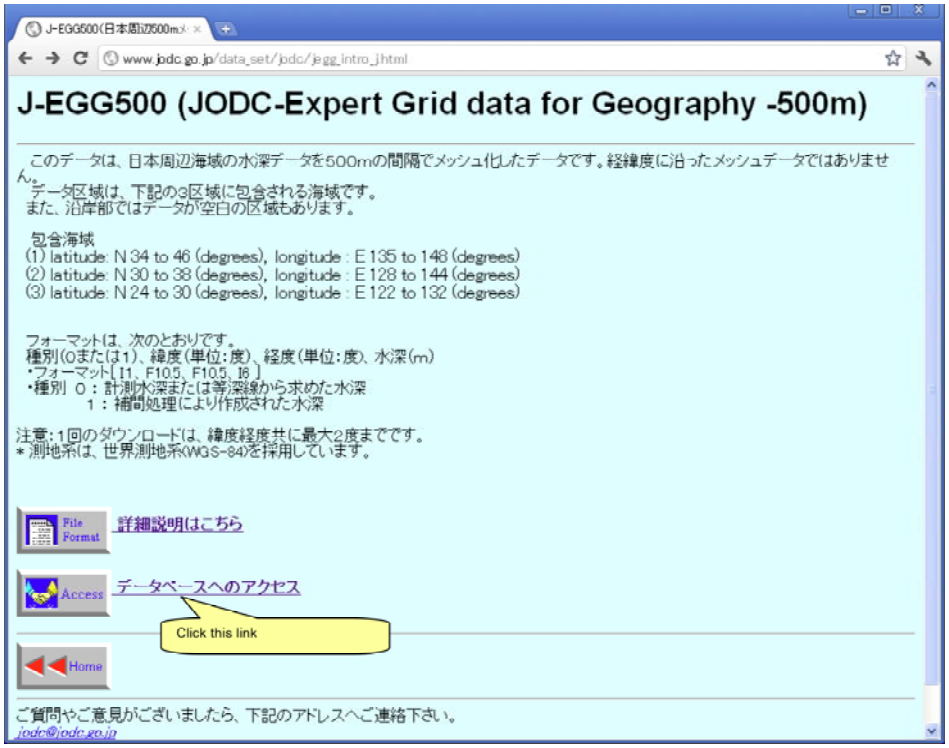

Figure 63 — Screenshot of the J-EGG500 Website

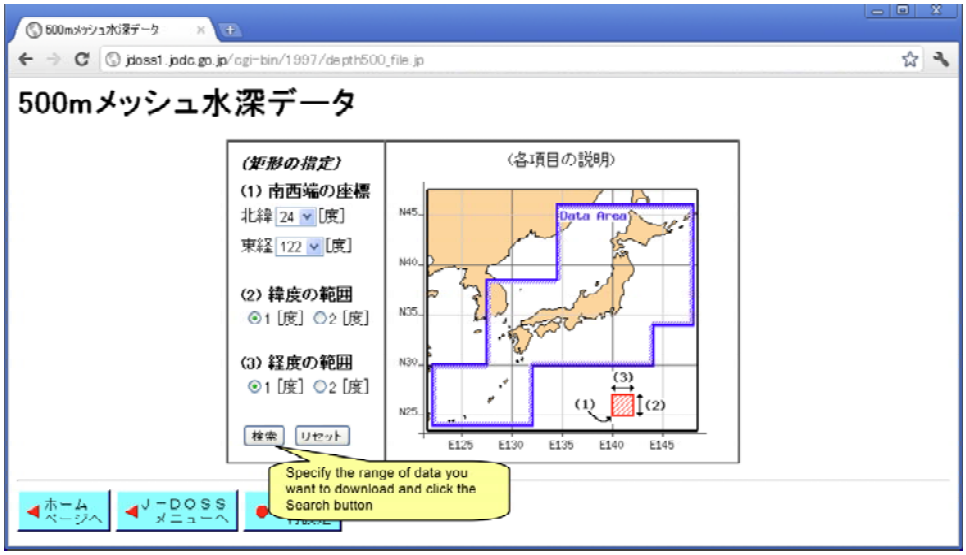

Figure 64 — Screenshot of the J-EGG500 data range selection page

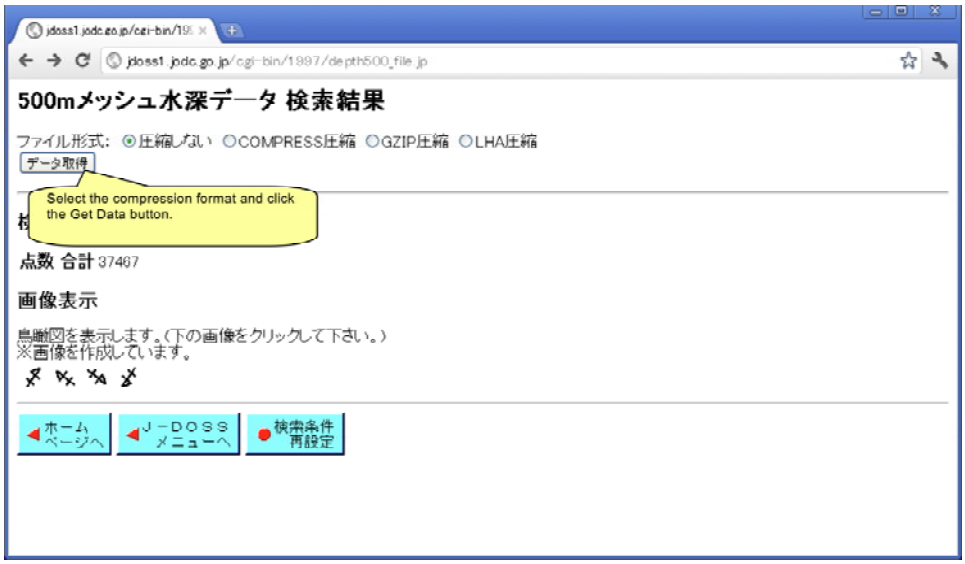

Figure 65 — Screenshot of the J-EGG500 data download page

11.5. X, Y (original format)

The X, Y data format is a format for users to create and import topography data with CADMAS MESH MULTI. Use the file extension *.grd. Table 7 gives an example of X, Y data.

Table 7 — Example of a file in X, Y format

4, 6 — Number of X-axis and Y-axis data points
0, 0, -32 — X coordinate, Y coordinate, height (0 is the water surface, a positive value indicates land)
10,0, -44
20,0,-47
30,0,-51
0,10,-41
(remainder omitted)

11.6. ArcGIS

The ArcGIS data format is a format for importing topography data from various GIS software. Use the file extension *.asc. Table 8 gives an example of a file in ArcGIS format.

Table 8 — Example of a file in ArcGIS format

```
ncols            1549
nrows            1850
xllcorner        3095.3929916298
yllcorner        59145.637722044
cellsize         2
NODATA_value     -9999
-7.8 8.5 9.5 9.5 9.6 9.4 9.4 9.5 9.4 9.4 9.4 9.4 9.2 3.3 -1111 -1111 10.1 10.3 10.3 10.4 10.2
10.2 [illegible]8 9.7 9.6 9.5 9.6 9.4 9.2 6.8 -1111 -1111 -1111 -9999 -9999
9.3 9.3 9.4 -1111 -1111 -1111 -1111 -1111 -1111 -1111 -1111 -1111 -1111 -1111 -1111 7.5
6.8 8.4 8.4 8.4 7.1 10.9 -1111 -1111 10.6 10.1 12.3 12.2 10 9.5 -1111 10.6 10.3 10.6 10.3 6.8
9.2 3.8 -1111 -1111 -1111 -1111 6.5 6.7 6.1 -1111 -1111 -1111 -1111 -1111 -1111 -1111
-1111
(remainder omitted)
```

Number of X-axis and Y-axis data points

X, Y coordinates at the lower left corner of the area

Grid width

Value indicating a point with no data

Height data

The height data is ordered, starting from the top left point of the area and moving to the right.

12. File Format for Fault Parameters

CADMAS MESH MULTI loads fault parameters in the format distributed by the International Institute of Seismology and Earthquake Engineering (IISEE) with incident times appended (in seconds).

Table 9 gives an example of a fault parameters file in the IISEE format. Table 10 gives an example of a fault parameters file with earthquake incident times (in seconds) appended.

Note that CADMAS MESH MULTI can load a fault parameters file without incident times, as shown in Table 9. In this case, the incident time parameters are all set to zero.

Table 9 — Example of a fault parameters file

Length	width	depth	strike	dip	rake	slip	lat	long	
50.0	50.0	0.0	193.0	14.0	81.0	14.73	39.738	144.331	# E01
50.0	50.0	0.0	193.0	14.0	81.0	10.07	39.3	144.20	# E02
50.0	50.0	0.0	193.0	14.0	81.0	6.60	38.862	144.069	# E03
50.0	50.0	0.0	193.0	14.0	81.0	28.77	38.424	143.939	# E04

Table 10 — Example of an edited fault parameters file

length	width	depth	strike	dip	rake	slip	lat	long	sec	
50.0	50.0	0.0	193.0	14.0	81.0	14.73	39.738	144.331	94	# E01
50.0	50.0	0.0	193.0	14.0	81.0	10.07	39.3	144.20	70	# E02
50.0	50.0	0.0	193.0	14.0	81.0	6.60	38.862	144.069	50	# E03
50.0	50.0	0.0	193.0	14.0	81.0	28.77	38.424	143.939	32	# E04