■ Operations Guide

The following user guide explains how to operate the LS-RAPID system. Using this software, the user can simulate the initiation and motion of rapid landslides triggered by earthquakes (seismic loading) and rainfall (pore-water pressure).

■Operations Guide

The following user guide explains how to operate the LS-RAPID system. Using this software, the user can simulate the initiation and motion of rapid landslides triggered by earthquakes (seismic loading) and rainfall (pore-water pressure).

1. Introduction

1.1. About LS-RAPID

The LS-RAPID system is an application designed for Microsoft Windows based on the program, "Integrated model simulating the initiation and motion of rapid landslide initiation and motion" which produced by Dr. Kyoji Sassa (International Consortium on Landslides). The software uses a visual interface which enables the user to input topography parameters, sliding surface parameters, soil and water parameters, and landslide triggering factors (earthquakes and water pressure) to simulate the initiation and motion of landslides in 3D. It is designed to be easily operated by both experienced and first-time users.

1.2. System Requirements

The following are the minimum system requirements for running LS-RAPID.

Operating System	Windows 7 / 8 / 10
Main Processor	At least processor requirement for OS Multi-Core processor (Example: Core2Duo)
Main Memory	Recommend 2GB RAM or greater
Hard Disk Drive	1GB free hard disk space or greater
Optical Drive	CD-ROM (necessary for application setup)
Display Monitor	1024x768 pixels or greater
Mouse	Mouse equipped with a scroll wheel
Video Card	nVIDIA (GeForce) or ATI Technologies (RADEON) Note: LS-RAPID may not run on graphics cards other than the ones listed

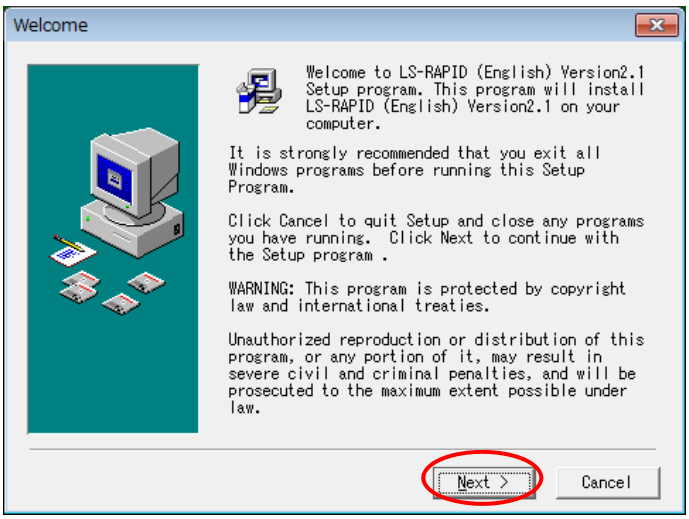
Note: Graphics cards other than the ones listed above may be incompatible (the display may become extremely slow). In that case:

Go to [Property] from the screen → [Setting] → [Details] → [Troubleshooting]

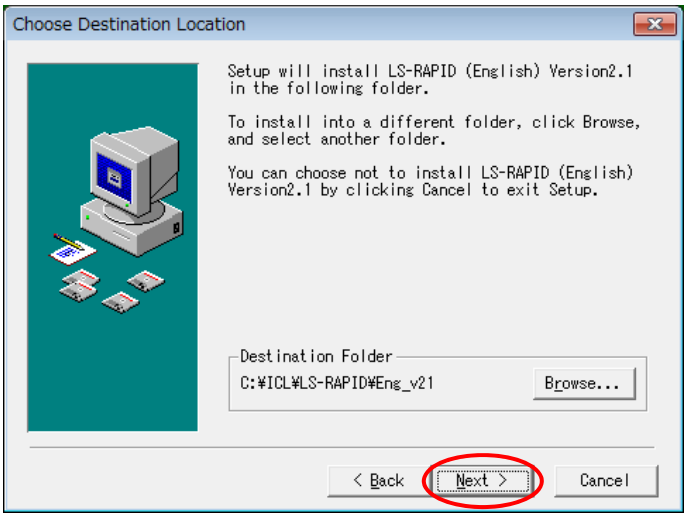
Choose [none] in Hardware accelerator.

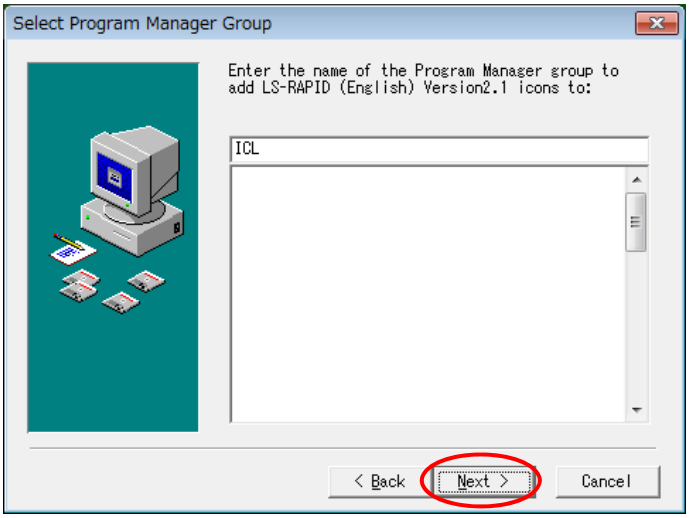
1.3. Installation Instruction

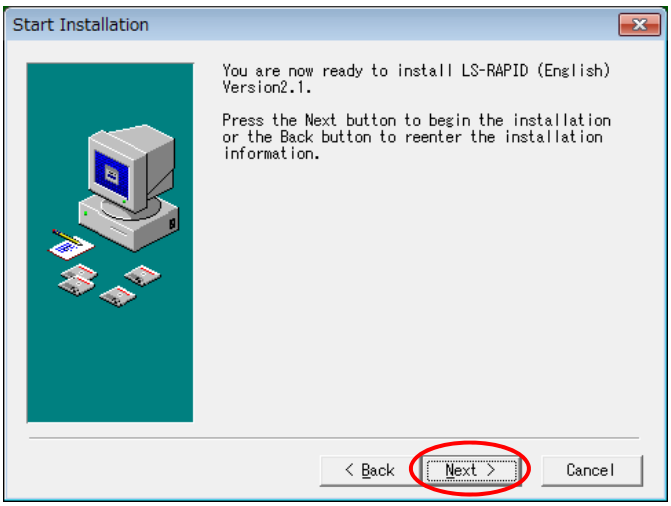
This section explains steps to install LS-RAPID. Once the CD of LS-RAPID is inserted into CD-ROM drive, the setup program will automatically launch. Click “Next” to start the installation.



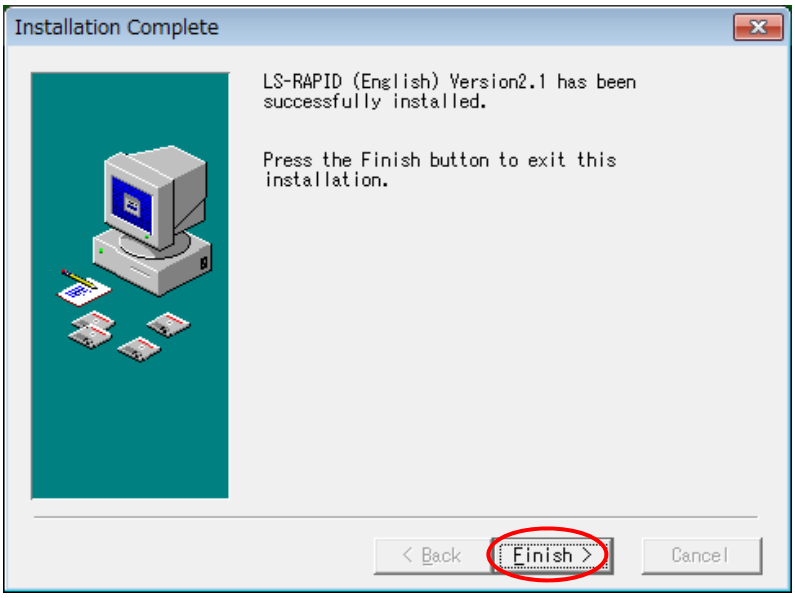
If do not wish to install the software to drive “C:”, then we can select the drive and folder where the software will be installed.





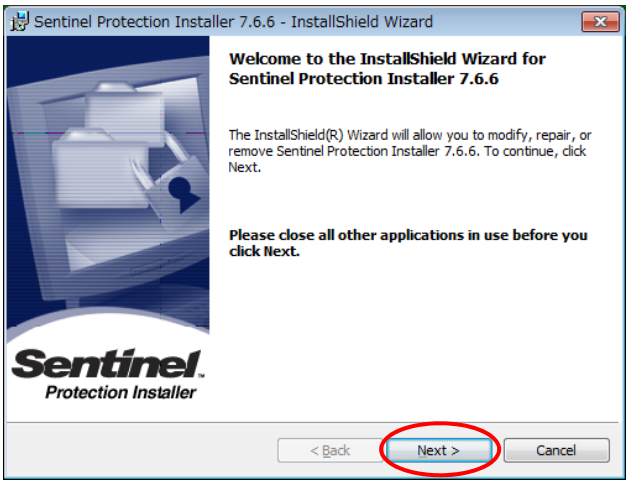


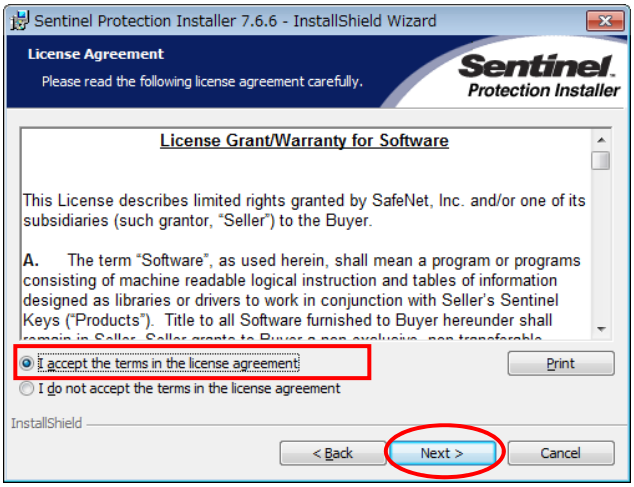
The following dialog box indicates that the installation has completed. Click “Finish” to complete the installation process.



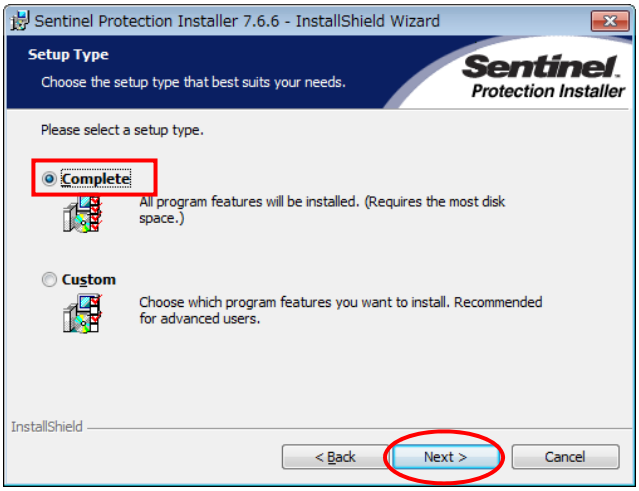
If the LS-RAPID software is installed into computer for the first time, then it is important to install also the Sentinel Protection Installer (as shown in the boxes below), as the driver for the USB Key Protector, to ensure the LS-RAPID software works properly.

(*The commands are unnecessary in a Trial Version*)

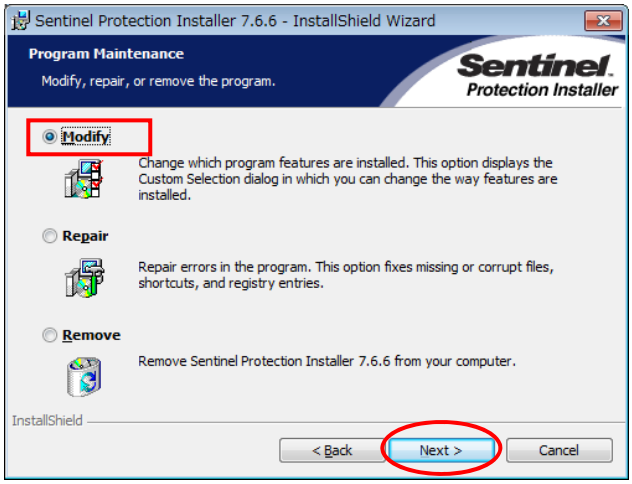


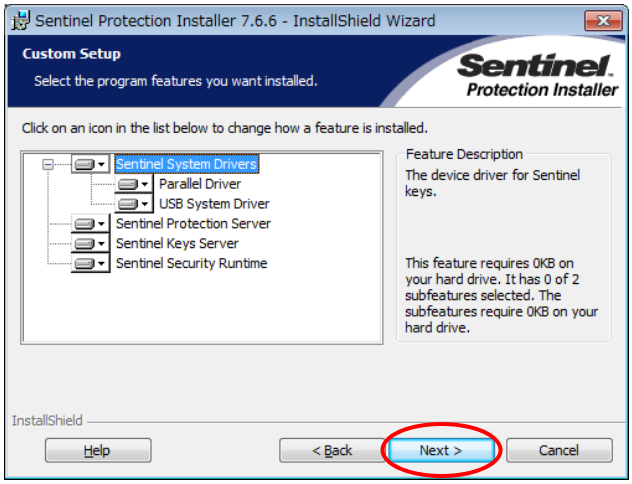


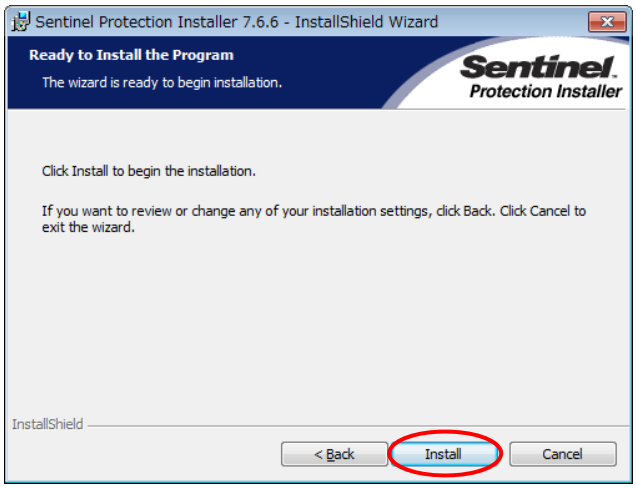
(New)

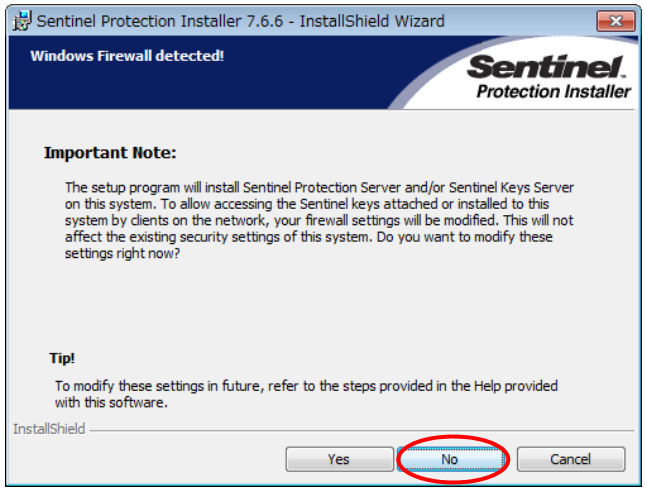


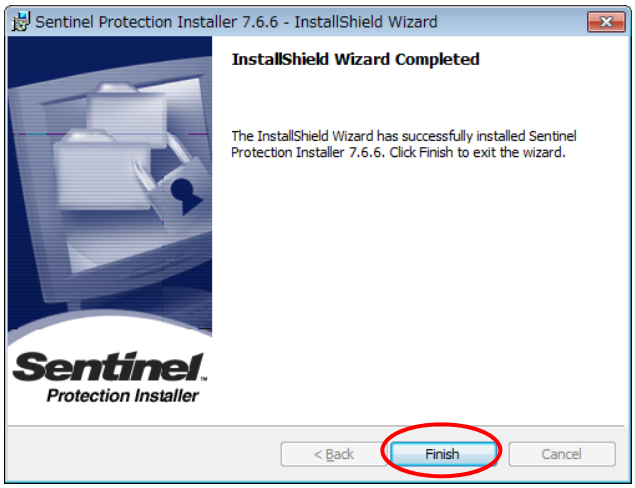
(Update)


Sentinel Protection Installer 7.6.6 - InstallShield Wizard
Program Maintenance
Modify, repair, or remove the program.
Sentinel Protection Installer
Modify
Change which program features are installed. This option displays the Custom Selection dialog in which you can change the way features are installed.
Repair
Repair errors in the program. This option fixes missing or corrupt files, shortcuts, and registry entries.
Remove
Remove Sentinel Protection Installer 7.6.6 from your computer.
InstallShield
< Back
Next >
Cancel


Sentinel Protection Installer 7.6.6 - InstallShield Wizard
Custom Setup
Select the program features you want installed.
Sentinel Protection Installer
Click on an icon in the list below to change how a feature is installed.
Sentinel System Drivers
Parallel Driver
USB System Driver
Sentinel Protection Server
Sentinel Keys Server
Sentinel Security Runtime
Feature Description
The device driver for Sentinel keys.
This feature requires 0KB on your hard drive. It has 0 of 2 subfeatures selected. The subfeatures require 0KB on your hard drive.
InstallShield
Help
< Back
Next >
Cancel


Sentinel Protection Installer 7.6.6 - InstallShield Wizard
Ready to Install the Program
The wizard is ready to begin installation.
Sentinel Protection Installer
Click Install to begin the installation.
If you want to review or change any of your installation settings, click Back. Click Cancel to exit the wizard.
InstallShield
< Back
Install
Cancel


Sentinel Protection Installer 7.6.6 - InstallShield Wizard
Windows Firewall detected!
Sentinel Protection Installer
Important Note:
The setup program will install Sentinel Protection Server and/or Sentinel Keys Server on this system. To allow accessing the Sentinel keys attached or installed to this system by clients on the network, your firewall settings will be modified. This will not affect the existing security settings of this system. Do you want to modify these settings right now?
Tip!
To modify these settings in future, refer to the steps provided in the Help provided with this software.
InstallShield
Yes
No
Cancel

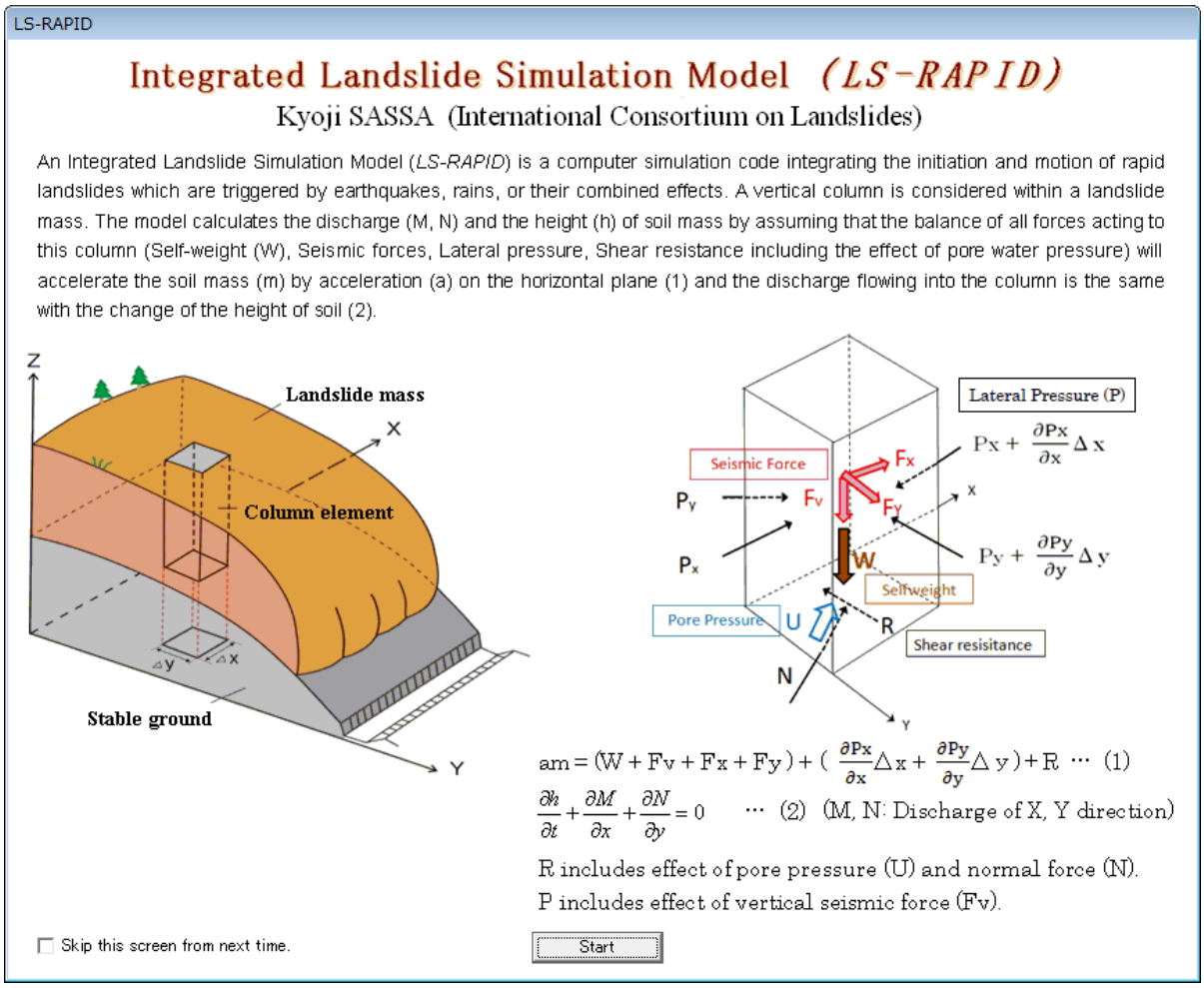

Sentinel Protection Installer 7.6.6 - InstallShield Wizard
InstallShield Wizard Completed
The InstallShield Wizard has successfully installed Sentinel Protection Installer 7.6.6. Click Finish to exit the wizard.
Sentinel
Protection Installer
< Back
Finish
Cancel

1.4. Starting the LS-RAPID System

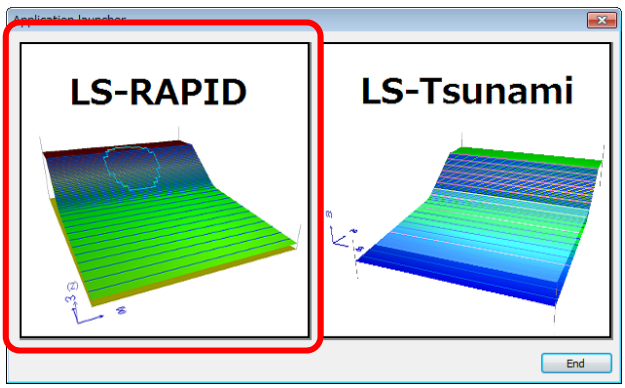
To start the software, follow the steps below:

Click [Start] →[Programs] → [ICL] → [LS-RAPID]

After the startup of LS-RAPID, the following window will appear.

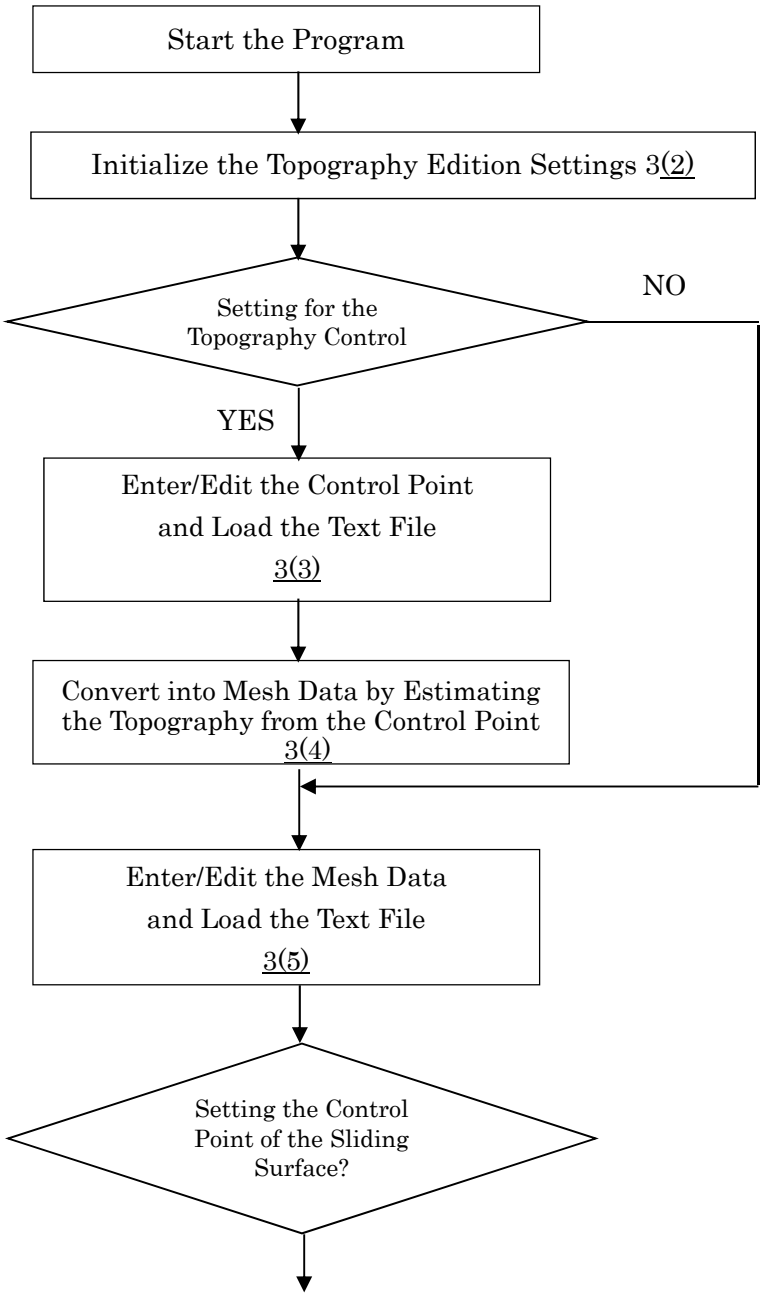


※The following initial screen will show both options of LS-RAPID and LS-Tsunami. Please choose the box of "LS-RAPID" in case of conducting the landslide simulation initially before doing tsunami simulation through "LS-Tsunami" (which is triggered by landslide).

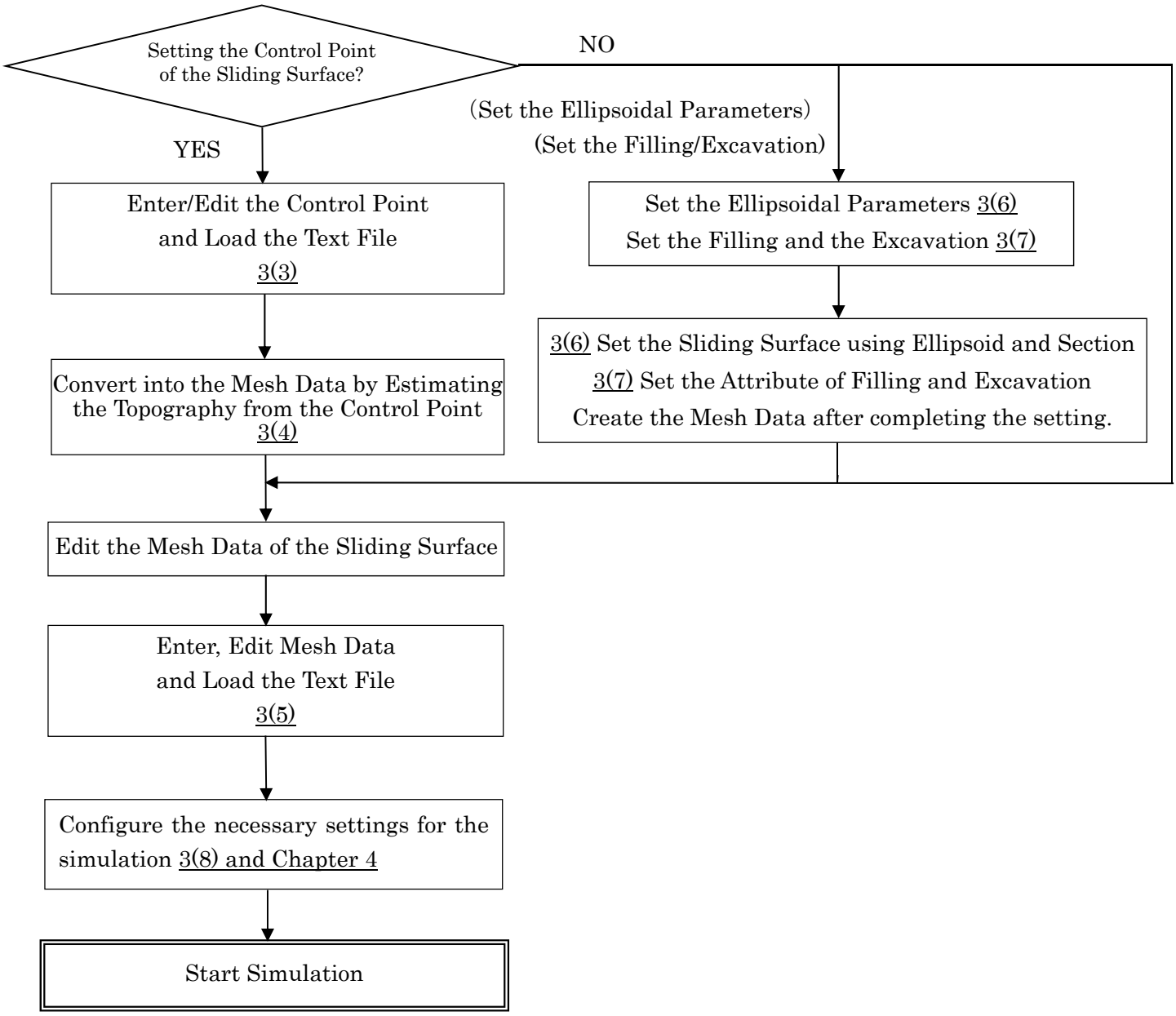


2. Simulation Flow Chart

The following chart showing the holistic flow for landslide simulation using the LS-RAPID: (See more details in Chapter 3)



Continued on to the next page....


Setting the Control Point of the Sliding Surface?
NO
YES
(Set the Ellipsoidal Parameters)
(Set the Filling/Excavation)
Enter/Edit the Control Point and Load the Text File 3(3)
Set the Ellipsoidal Parameters 3(6)
Set the Filling and the Excavation 3(7)
3(6) Set the Sliding Surface using Ellipsoid and Section
3(7) Set the Attribute of Filling and Excavation
Create the Mesh Data after completing the setting.
Convert into the Mesh Data by Estimating the Topography from the Control Point 3(4)
Edit the Mesh Data of the Sliding Surface
Enter, Edit Mesh Data and Load the Text File 3(5)
Configure the necessary settings for the simulation 3(8) and Chapter 4
Start Simulation

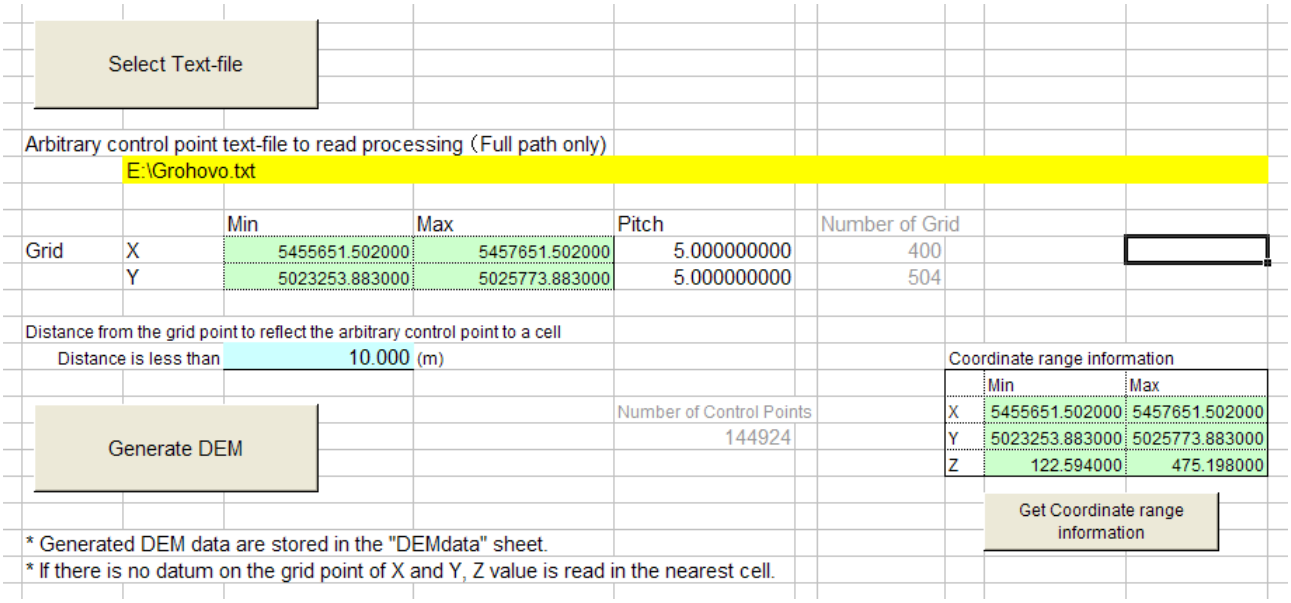
3. Topography Edition

3.1. Preparation of DEM Data

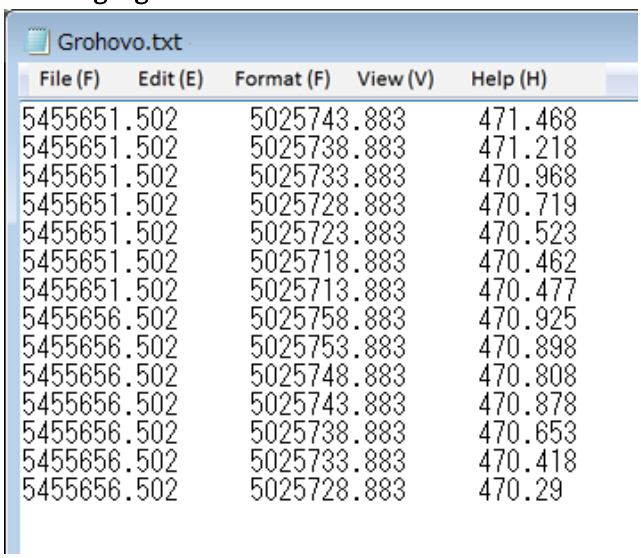
(*Skip this section if not necessary*)

Generally the DEMs are provided with a text file (X, Y, Z format) including Latitude, Longitude and Elevation. To input DEM data for LS-RAPID simulation, text file (X, Y, Z format) should be converted to elevation mesh data in advance. The accompanying software (DEMmake) is available to convert a text file to elevation mesh data.

Open DEMmake (Excel file) and the following screen appears.
(Macro feature should be enabled in advance.)



(1) Click [Select Text-file] to select the text file which contains XYZ coordinates data like the following figure.



** Three columns should be separated with blank or comma*

(2) Click [Coordinate range information] and automatically the range of coordinate appears.

In this case:

The range of X is 5455651.502 ～ 5457651.502

The range of Y is 5023253.883 ～ 5025773.883

(3) Input the data of coordinate range gained by the previous operation into Grid cells.

Pitch means the interval of X and Y coordinates. If we want to make Pitch bigger than original, input the interval necessarily, then the number of grids will be calculated automatically.

When we have any designated coordinate point of the upper left and lower right of the simulation area within this range, we can input them as minimum and maximum points.

Do not input any number into the grey displayed cells

In this case (full range):

X min = 5455651.502	X max = 5457651.502	Pitch = 10	(Number of grids is 200.)
Y min = 5023253.883	Y max = 5025773.883	Pitch = 10	(Number of grids is 252.)

(4) Input the distance (m) from the grid point to reflect the arbitrary control point to a cell.

This operation aims to set up the distance between the grid point and the arbitrary point to adopt Z value, when the distance between the nearest grid and the arbitrary point is less than the designated distance.

If the distance is set up as 10.00 m:

Case 1

In the case that the grid point is (5455635.185, 5025800.000, -44.0), the nearest point is (X, Y) = (5455651.502, 5025773.883). In this case, Z value of this point, -44.0 is not adopted for the nearest point, because the horizontal distance is 30.795 m, bigger than 10.00 m.

Case 2

In the case that the grid point is (5455785.251, 5024215.957,-55.0), the nearest point is (X, Y) = (5455781.502, 5024213.883). In this case, Z value of this point, -55.0 is adopted for the nearest point, because the horizontal distance is 4.284 m, less than 10.00 m.

(5) Click [Generate DEM] and the generated DEM data are stored in the DEMdata sheet.

*If there is no data near the grid point, the Z value at this point will not appear.
In this case, the distance should be set up with a bigger value.

Sample of the generated DEM data：

	5455682	5455692	5455702	5455712	5455722	5455732	5455742	5455752	5455762	5455772	5455782	5455792
5025774	469.246	469.629	469.824	471.807	474.222	474.136	471.56	468.444	464.228	459.381	453.41	447.352
5025764	469.473	468.963	469.031	472.206	473.58	472.88	471.362	465.723	463.587	456.212	453.271	444.377
5025754	469.316	469.295	469.659	469.956	471.115	473.565	469.984	466.535	462.814	458.077	452.93	447.75
5025744	469.408	469.647	469.731	469.955	470.29	471.422	469.823	464.267	462.493	455.345	452.994	444.51
5025734	469.26	468.948	469.15	469.346	469.414	469.948	470.2	467.027	462.883	458	453.188	448.007
5025724	469.154	468.812	468.912	469.135	468.909	470.027	469.827	465.694	463.441	456.022	453.553	445.588
5025714	468.828	468.041	468.192	468.147	467.2	467.003	467.819	468.549	465.348	460.466	455.445	450.238
5025704	468.742	467.864	467.894	467.57	466.642	465.808	466.234	467.622	465.859	458.629	456.041	448.101
5025694	468.358	467.309	466.714	466.707	465.938	465.446	465.141	467.438	467.261	463.185	457.78	452.605
5025684	468.238	466.714	466.401	466.3	465.986	465.252	465.13	468.439	467.477	461.038	458.355	450.601
5025674	467.068	466.643	465.916	465.504	465.638	465.315	465.892	468.437	468.168	464.531	460.026	455.121
5025664	466.445	465.956	465.664	465.124	465.124	464.775	465.645	469.089	468.378	462.434	460.448	453.507
5025654	464.57	464.342	464.116	463.839	462.349	461.247	463.388	466.272	467.335	464.319	461.488	458.056
5025644		463.602	463.49	462.901	461.837	460.936	462.451	465.981	466.361	463.126	461.881	456.637
5025634		462.578	462.433	462.665	461.096	460	460.042	461.539	463.863	462.925	461.015	458.063
5025624			462.201	462.156	460.915	460	460	461.283	462.319	460.795	460.133	455.938
5025614				462.68	462.333	460.651	460.199	460	460	459.899	458.289	456.22
5025604				462.836	462.869	461.221	460.683	460	459.922	458.968	458.163	455.12
5025594				462.503	462.54	462.498	460.712	459.29	458.405	457.686	457.113	455.716
5025584						461.396	460.42	458.25	457.831	456.83	456.549	454.572
5025574						461.771	459.015	457.395	456.234	455.395	455.052	454.771
5025564							459.121	456.203	455.643	455.012	454.958	454.255

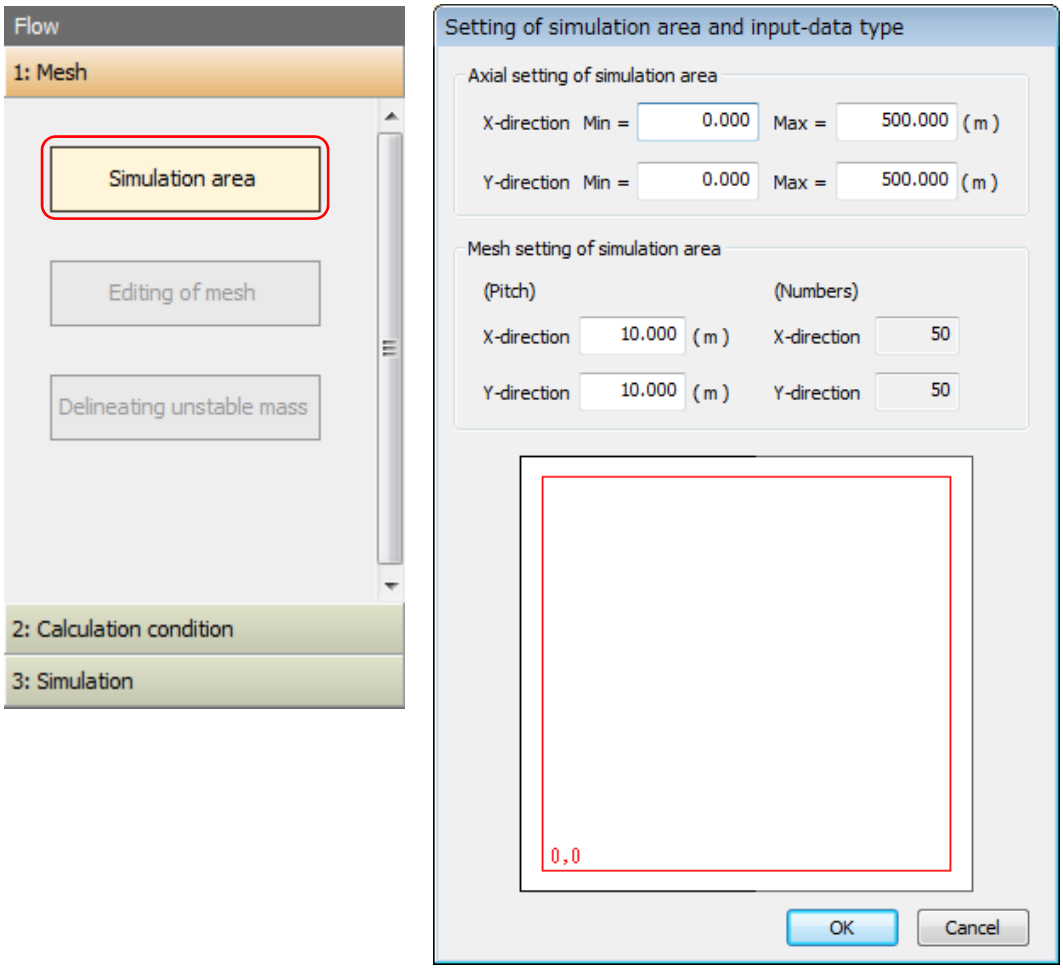
(6) Choose the desired area to do simulation using the LS-RAPID. Then count the number of grids on X and Y.

In this case, the numbers of grids are 8 on X, and 18 on Y. The number of grids minus 1 indicates the number of mesh at the setting of Calculation Area. The selected elevation mesh data can be copied and paste to the Slope Surface Elevation sheet. If we set up the area before generating DEM, the numbers of grids are calculated automatically. (*See the next section*)

3.2. Initial Setting of Topography Edition

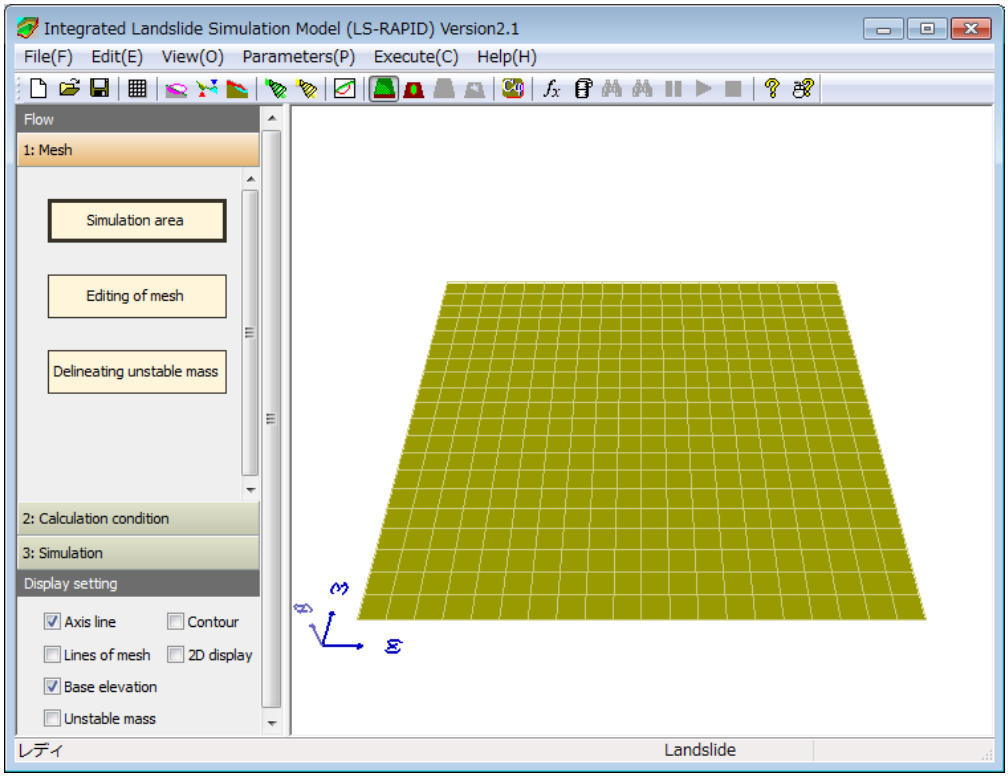
When the program has been started by creating a new document, there will be no topography data or settings appear for the simulation. This section explains the initial settings required to view topography.

Click “Flow” panel on the left side of the window → then [1: Mesh] option → [Simulation area], or just click the [Simulation Area Setting] icon from the toolbar. The dialog of “Setting of simulation area and input-data type” then appears as follow,



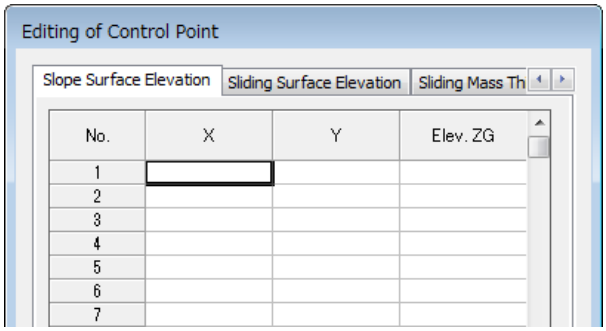
Item	Description
Axial setting of simulation Area	Enter the range for X-direction and Y-direction of the calculation area for the mesh. Range (-9999999.999～+9999999.999(m))
Mesh setting of simulation Area	Enter the number of mesh and pitch in both X-direction and Y-direction of the mesh in the calculation area. The number of mesh (2～9999) Pitch (0.001～999999.999(m))

After setting the area then click “OK”, the following window will appear.



3.3. Editing the Topography Control Point
(*Skip this section if not necessary*)

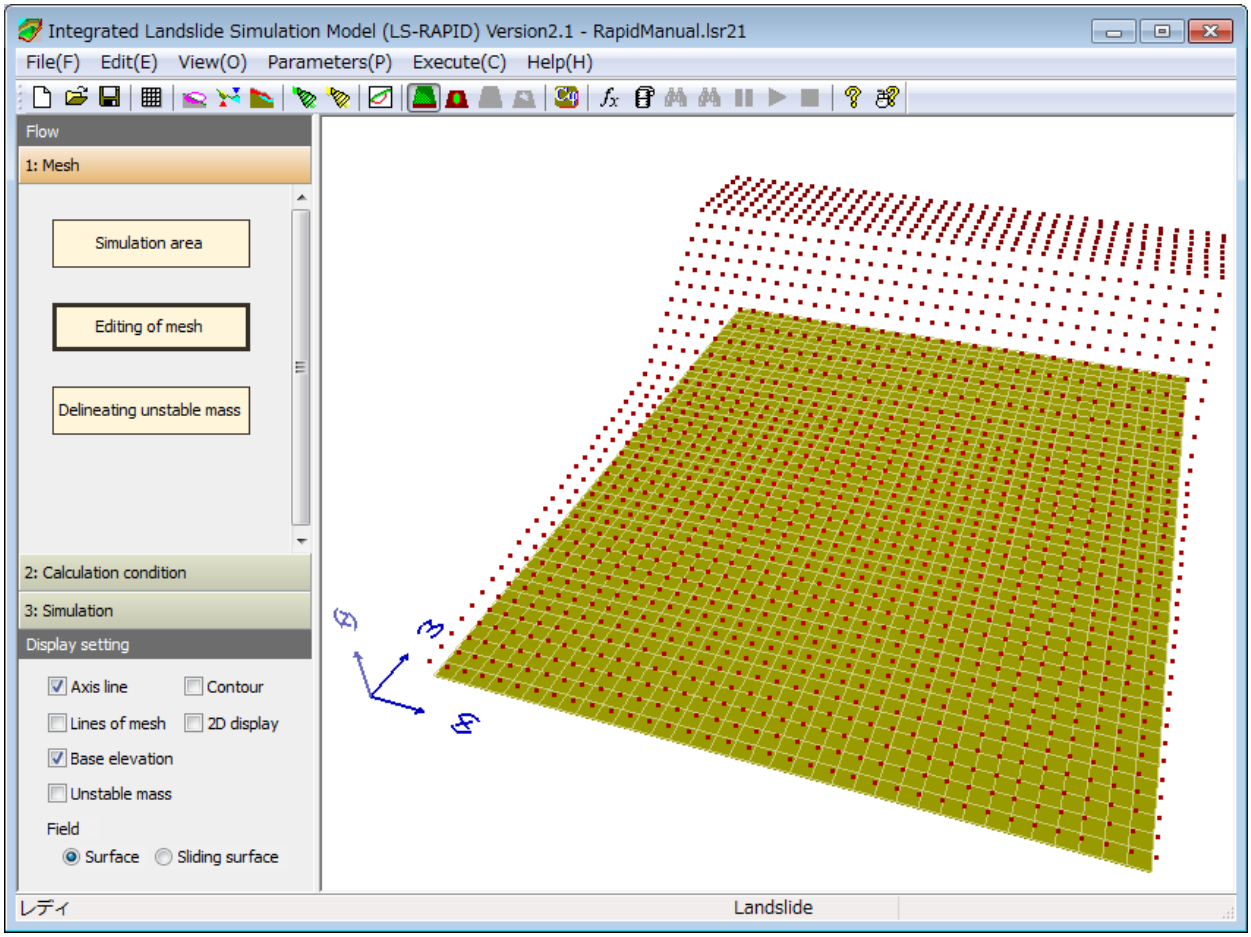
If we wish to edit the topographic control point manually, select the required tab provided in the "Editing of Control Point" option that opened from the [Edit] menu as follows,



If we already have a text file for the Control Point data, then from the [File] menu, select [Read text data file] → [Read (ground elevation・sliding surface elevation・sliding mass thickness) Control Point] to read the appropriate file.

Note: The data in the text file should be separated by a space, tab, or comma (CSV format) to load correctly. Other text file formats may fail to load properly.

After loading the text file and editing the data manually, the following window will appear.



3.4. Converting the Control Point Data into the Mesh Data by estimating the Topography (*Skip this section if not necessary*)

Once the Control Point data has been edited (as explained in Section 3.3), the control point data can then be transferred into the mesh data.

Go to [Menu] → [Edit] → [Transfer control point data to mesh data]

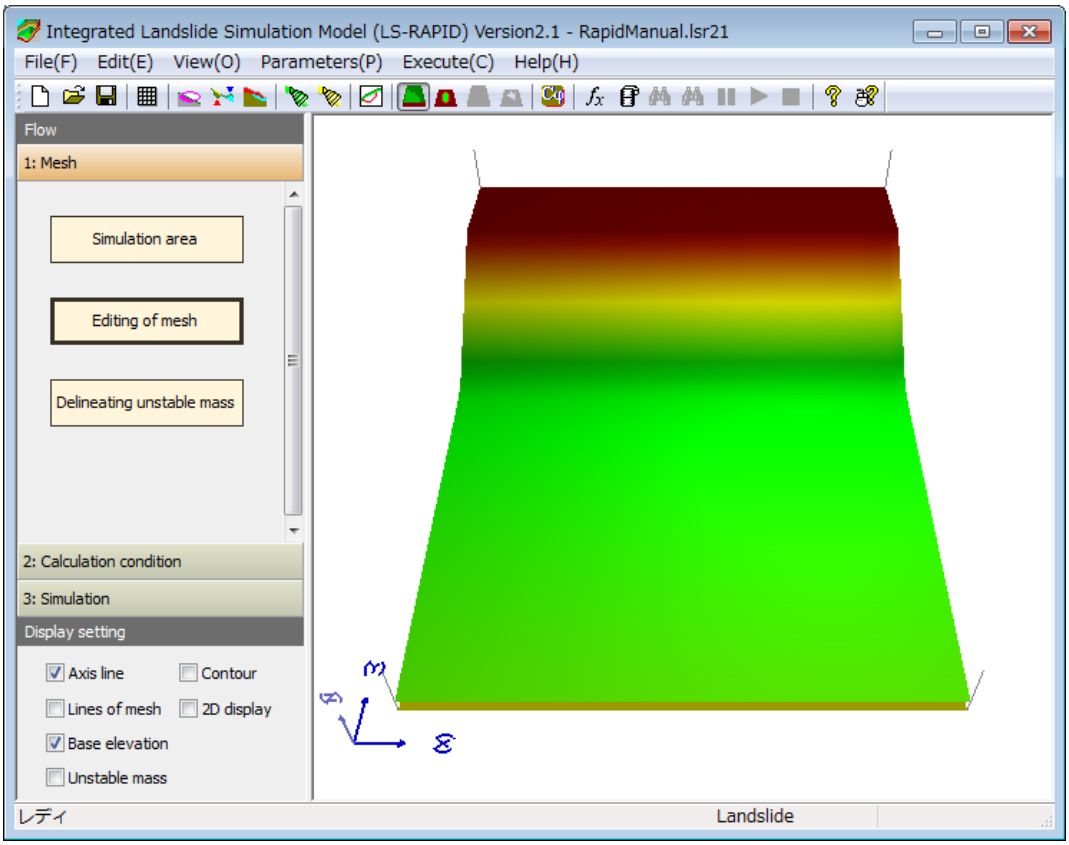
After seeing the message in dialog box, the image of mesh topography with the reflected Control Point data will appear, indicating the converted process is done successfully.

Note: After creating the Mesh data, we can still edit the Control Point. In this case, go back to the Edit the Control Point and then return to this section.

3.5. Editing the Topography Mesh Data

If we already have a text file for the Mesh data, from the [File] menu select the [Read text data file] option → then [Read (ground elevation・sliding surface elevation・sliding mass thickness) mesh data] and choose the appropriate file.

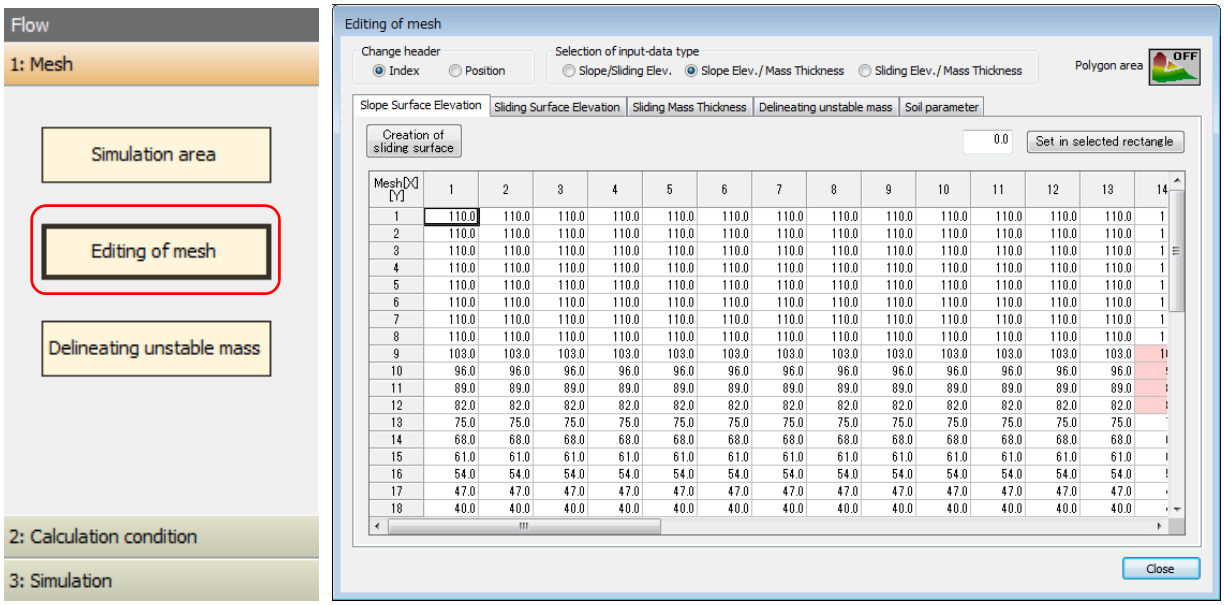
After loading the text file and editing the data manually, we will see the following window.



The calculation area can be edited with mesh data unit. If we wish to edit the values of Mesh data manually, select the required tab provided in the “Editing of Mesh” option that opened from the [Mesh] panel on the left side of the window as shown below.

Each tab, the “Slope Surface Elevation”, “Sliding Surface Elevation”, and “Sliding Mass Thickness”, displays the Mesh Data topography. We can edit the following options for each tab.

Note: If the drawing option from toolbar changed, the drawing may not be consistent with the tabs

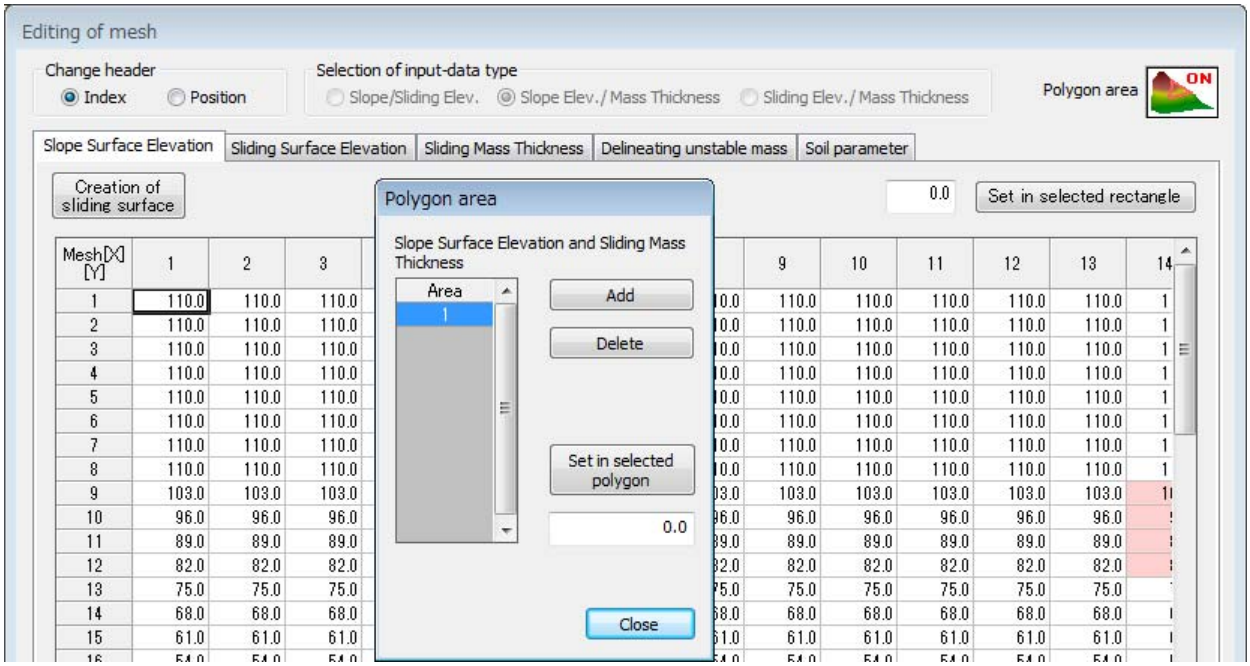


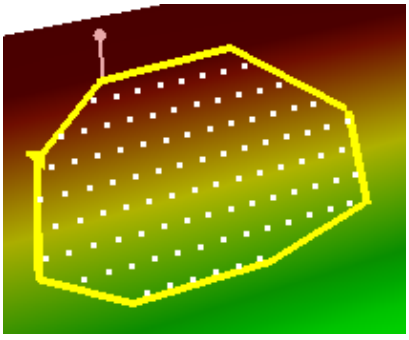
Item	Description
Header Notation	The header notation on the mesh item stated as mesh number or represent as the X-Y coordinate values
Selection of input-data type	Choose from the following list of data types to edit: ○ Slope Surface Elevation and Sliding Surface Elevation ○ Slope Surface Elevation and Sliding Mass Thickness ○ Sliding Surface Elevation and Sliding Mass Thickness (*Note: It is impossible to load and edit the text file if parameters other than those listed above are modified*)
Creation of sliding surface	By clicking this button, the sliding surface can be created by setting the Ellipsoidal body or by filling/excavation.
Polygon area editing	By clicking the button below, ON or OFF condition can be chosen.  When ON is selected, we can create the any-shape area by using left click while pressing the keys [Ctrl] + [Shift].
“Set in selected rectangle” button	First, input the value of elevation in the blank box beside the button of “Set in selected rectangle”. Then, choose the objective mesh area. Finally, click the button of “Set in selected rectangle”. As a result, the same values of elevation will be set for all the

mesh that selected (this can save the time if we want to input the same elevation values in the mesh).

If we use the “Polygon area editing” button clicked as ON mode, we can input the same value for any-shape area that have been created. In the case of OFF mode, it will be assigned the same value for selected cell blocks.

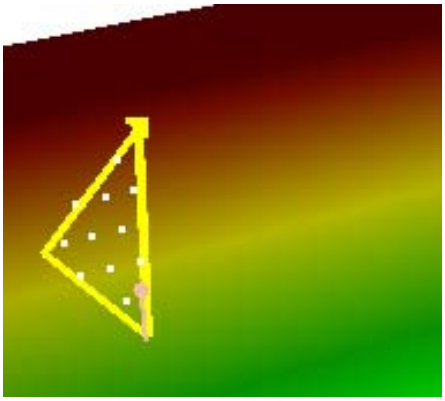
If we click the “Polygon area editing” button to choose any-shape area, the figure below can be seen as an example. If we right click on the windows of 3D-View, the pop-up menu will appear and we can delete the last one point or delete all area points.





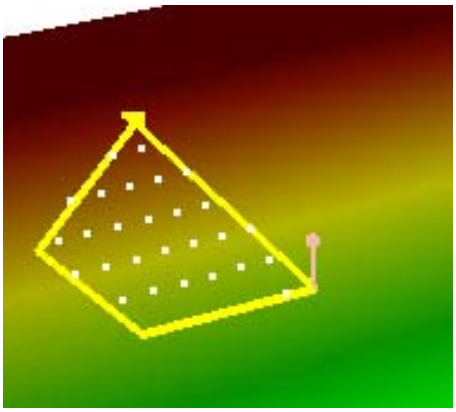
Method to edit the selected polygon

Example to edit the initial Polygon (triangle) area



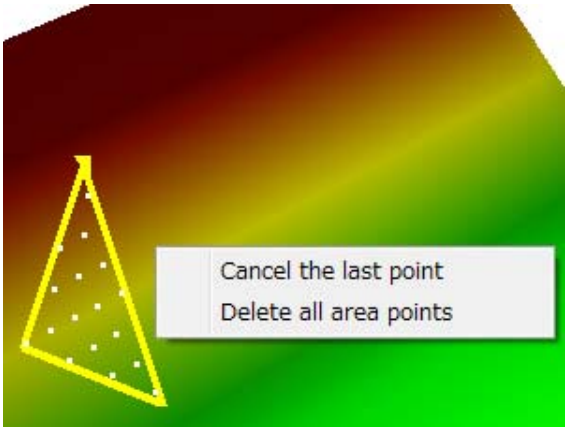
(Figure 1)

[Ctrl] + [Shift] + left click the windows of 3D-View
One area point is added to the initial triangle.



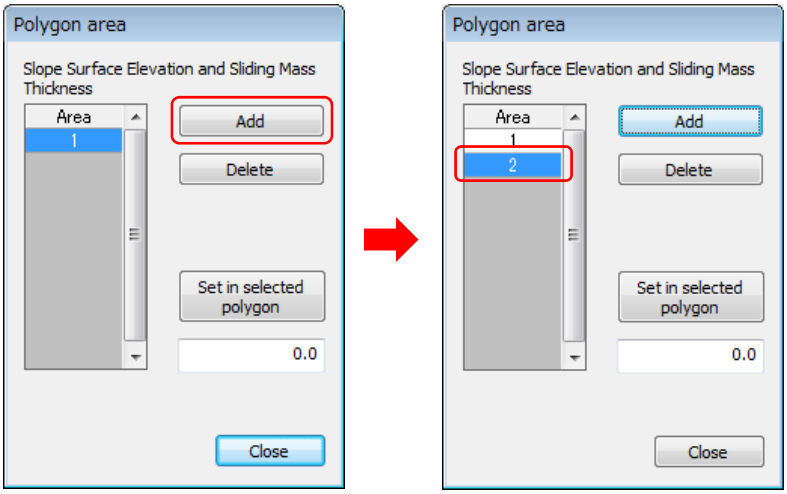
(Figure 2)

Right click the windows of 3D-View
The pop-up menu will appear and the last editing point can be cancelled or all area points can be deleted.

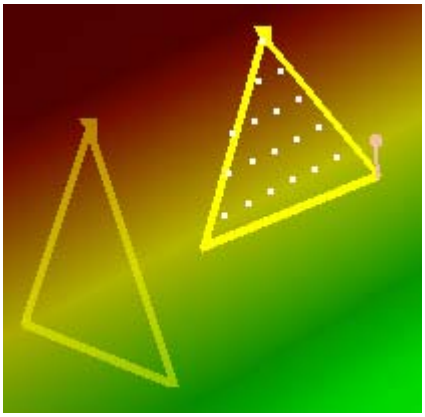


(Figure 3)

Multiple polygonal areas can be set using the polygon edit mode as follows,



Several numbers of new polygon area can be added by clicking the button “Add”
If we edit with this state, a new polygonal area can be carried out.



3.6. Creating the Possible Sliding Surface using the Ellipsoidal Parameter (*Skip this section if not necessary*)

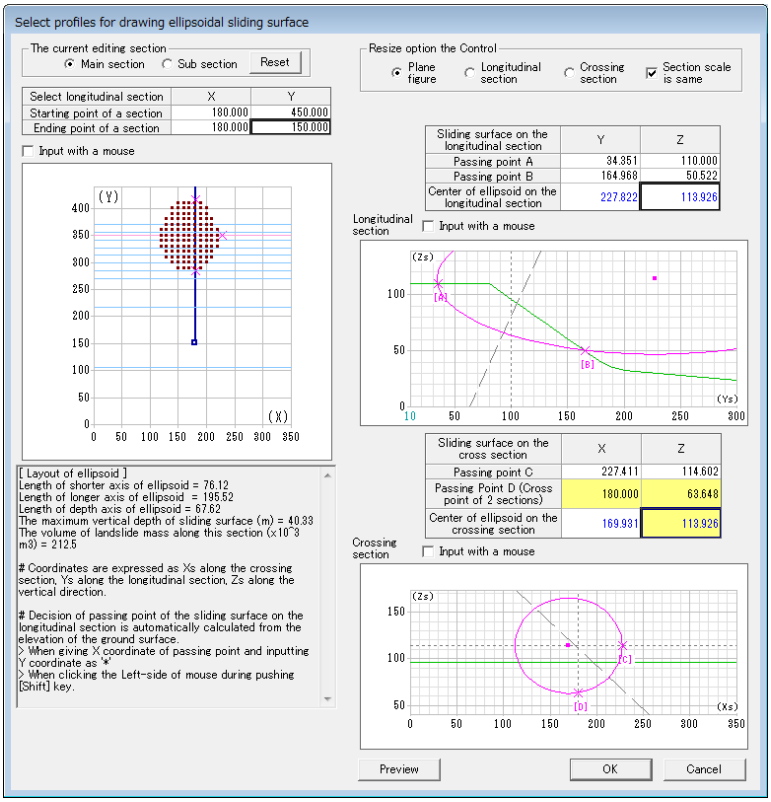
If the Mesh data of the Slope Surface is already created, we can build the Sliding Surface using the ellipsoidal parameter.

Go to the [Edit] menu and select [Ellipsoid Sliding Surface setting].

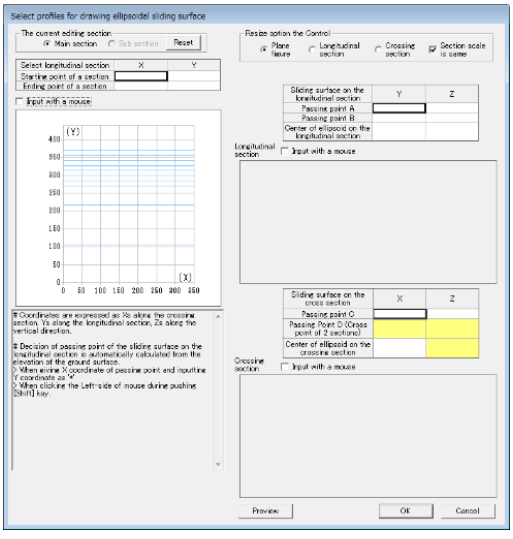
Or select [Ellipsoid Sliding Surface setting] from the toolbar.

Or by opening the "Editing of mesh" dialog, click [Creation of sliding surface] and choose [Create sliding surface with section profile of Ellipsoid].

This will display the dialog box for building the ellipsoidal sliding surface from the selected section.

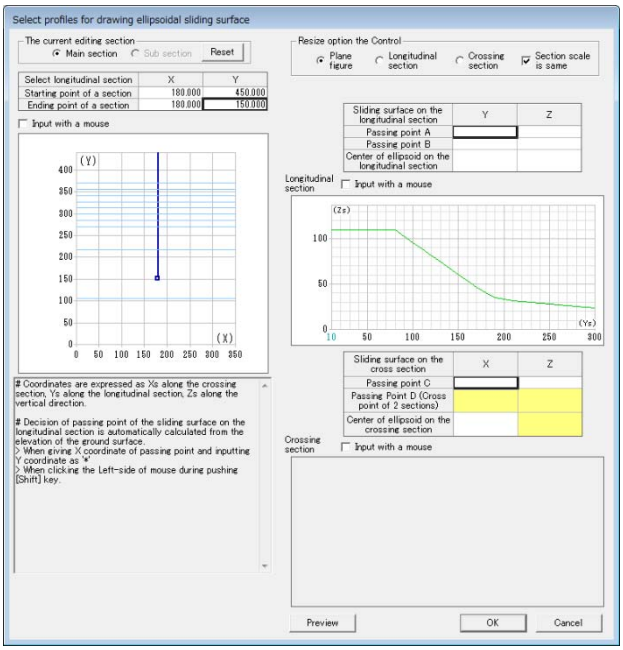


The following explains the steps to create the ellipsoidal sliding surface using this dialog box.
(The Initial View)



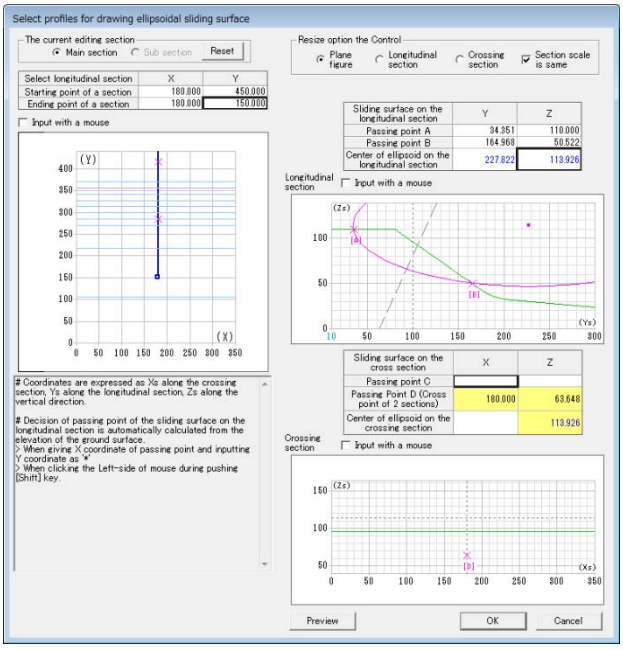
(1) Setting the Longitudinal Section of the landslide

We can create an ellipsoidal sliding surface by setting the longitudinal section of the landslide on the left side of this dialog box. Once we enter the starting and ending points, the corresponding longitudinal section will appear on the right side of the dialog box (we can also input the starting and the ending points by using a computer mouse).



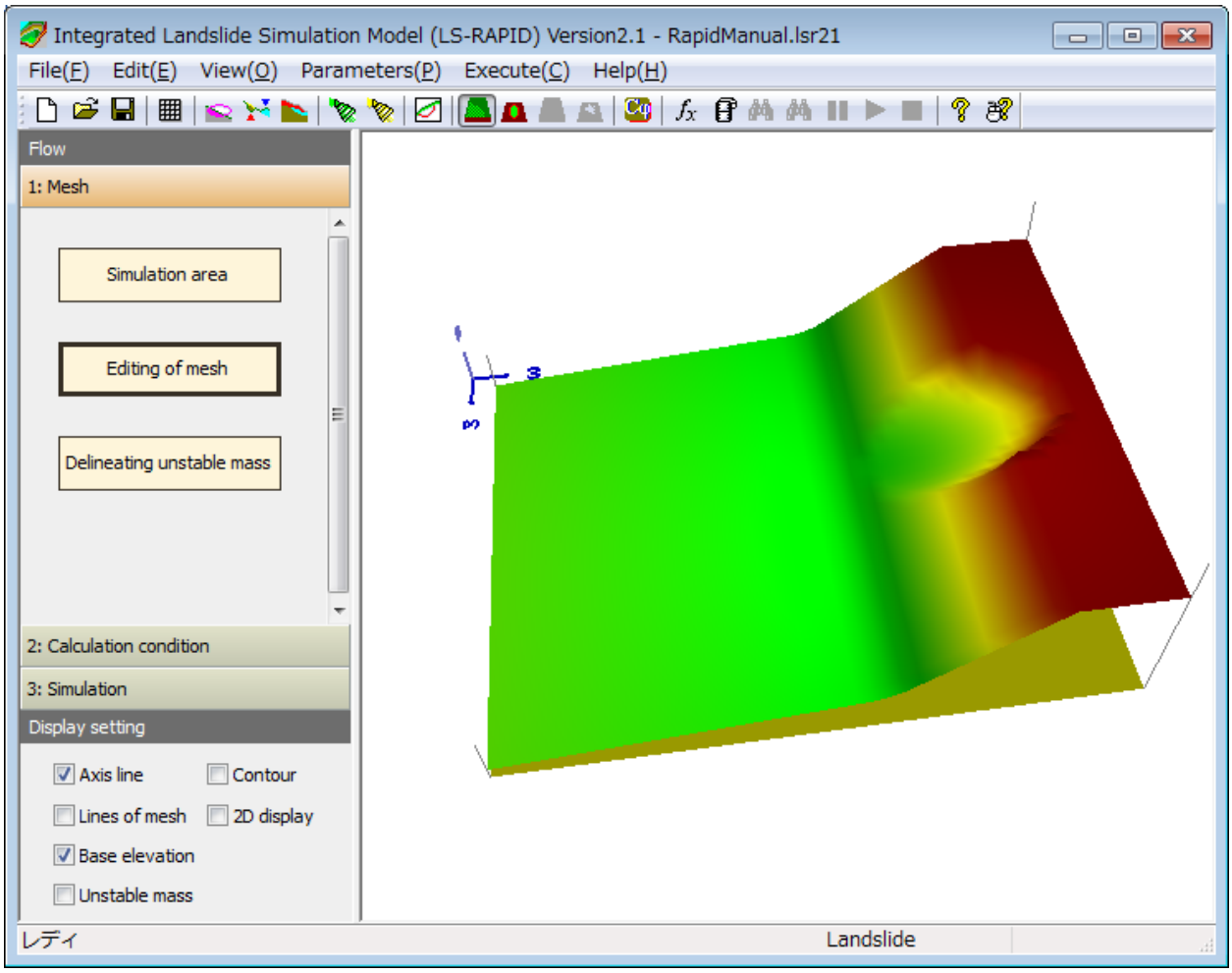
(2) Setting the Sliding Passing Points and the Center Point of Ellipsoid on the Longitudinal Section

After setting the longitudinal section, we can set the passing points of the sliding surface. By setting the passing points, the crossing section which passes through the center of the passing points will appear. The left part of the screen will display the position information. When we enter the center point of the ellipsoid, the ellipsoid will be displayed on the longitudinal section. Crossing points between the cross section and the ellipsoid will appear. (Note: we may not see the ellipsoid even after entering its center point. In that case, adjust the location of the center point and the passing points. We can input the passing points and the center point using a computer mouse.)



(3) Setting the Sliding Passing Points and the Center Point of Ellipsoid on the Crossing Section
Similar to the longitudinal section, we can also set the passing point and the center point of the ellipsoid on the crossing section. We can set the crossing point between the sliding surface, the slope, and the X-coordinate of the center point of the ellipsoid. The cross point of the two sections and the elevation of the center of ellipsoid are already determined on the crossing section.

(4) If the subsection is necessary to make, we can create the ellipsoidal subsection by following steps (1) to (3). Configure the ellipsoid by adjusting its location through steps (1) to (4). Click “OK”, then the ellipsoidal sliding surface will be built.



Item	Description
The current editing section	We can create the sliding surface using “Main section” or “Sub section”. If there is only one sliding surface, select “Main section.”
Select longitudinal section	Set the landslide longitudinal section to create an ellipsoidal sliding surface. After entering the starting point and the ending point of the longitudinal section, the corresponding image will appear. We can enter the data using a computer mouse.
Resize option	○ Plane Figure ○ Longitudinal Section ○ Crossing Section Set an option to change the size of view windows when the size of the dialogue changed. In addition, when the “Section scale is same” is checked, cross-section will be displayed in the same aspect ratio.
Sliding Surface (longitudinal section) Passing points A, B, Center of Ellipsoid Sliding Surface (cross section) Passing points C, D, Center of Ellipsoid	Set the passing points A and B which the sliding ellipsoid passes through and the center of the ellipsoid on the longitudinal section. After setting the passing points, the cross section which passes through the center of passing points will be displayed. We can enter this data using a computer mouse. Once the ellipsoid is created, the matched coordinates will appear on the crossing section. Similar to the longitudinal section, for the crossing section, set the passing points C and D which the ellipsoid passes through, and the center point of the ellipsoid (point D may be determined depending on the longitudinal section). After entering the X-coordinate of the sliding surface passing point, input the Y-coordinate as “*” or by left-clicking while holding down the [Shift] key; the elevation of the slope surface corresponding X-coordinate will be automatically calculated.

After creating the ellipsoid on the longitudinal and crossing sections, the appropriate ellipsoid will appear. We will see the region for the ellipsoid on the longitudinal section screen and the 3D view screen. The characteristics of the ellipsoid will appear on the dialog box

Note: Coordinates are expressed as X's along the crossing section, Y's along the longitudinal section, and Z's along the vertical direction. The X and Y coordinates on the longitudinal and the crossing sections represent the distances from the starting point of the sections and the Z coordinate represents the elevation.

3.7. Filling and Excavating the Current Topography to Estimate Pre-failure Topography (*Skip this section if not necessary*)

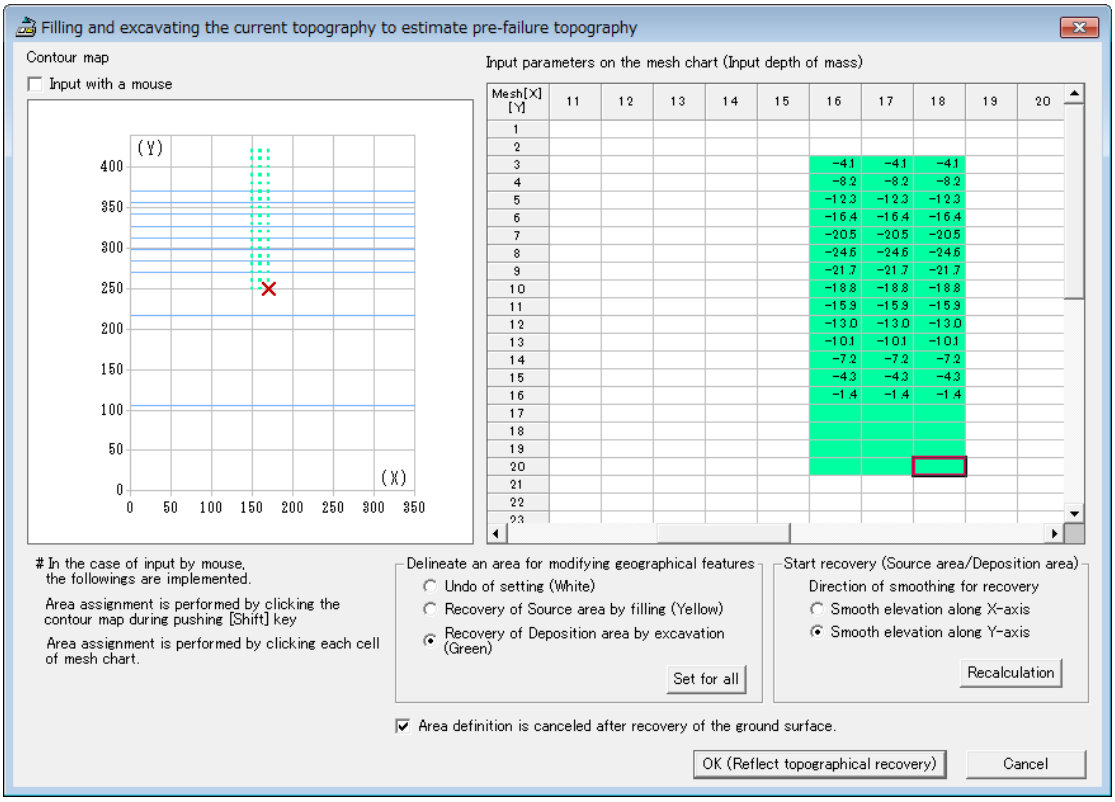
Once the mesh data of the slope surface is provided, we can create the sliding surface and the original topography before triggering land sliding by filling and excavating the current slope surface. To set the filling and excavation, follow the steps below.

Go to the [Edit] menu and select [Recover (Source area/Deposition area)].

Or click the icon [Recovery (Source area/Deposition area)] from the toolbar.

Or by opening the “Editing of mesh” dialog, click [Creation of sliding surface] and choose [Create sliding surface with recovery setting].

We will see the following dialog box.



Item	Description

Delineate an area for modifying geographical features

- Undo of setting [White]
- Recover of Source area by filling [Yellow]
- Recovery of Deposition area by excavation [Green]

Set for all button

The filling and excavation attributes can be specified for each mesh data of the current geographical feature. If we click the button [Set for all] after selecting the mesh, the area will be updated by the selected attribute.

Start recovery (Source area/Deposition area)

Direction of smoothing for recovery

- Smooth elevation along X-axis
- Smooth elevation along Y-axis

Recalculation button

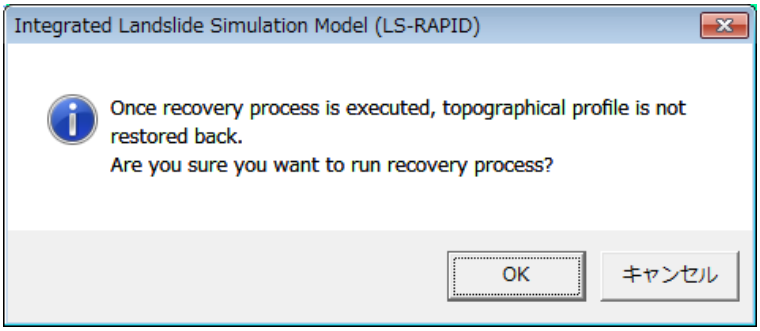
After deciding on the attribute, if we click the [Recalculation] button, the value of fluctuations from the current elevation of the geographical features will be calculated and appear on the mesh chart. (Note: if we change the direction of smoothing for recovery, the re-calculation may start automatically.)

We can input the depth of mass arbitrarily. The input data will be valid until we re-calculate.

Note: If we just change the attribute of the area, the re-calculation will not start.

Note: If we select the checkbox, "Input with a mouse," and click the contour map, the cursor on the mesh chart will move. If we click the map while holding down the "Shift" key, the delineated area (created through undo of setting, recovery of source area by filling or recovery of deposition area by excavation) will be updated. We can also update the delineated area by clicking on a cell in the mesh chart.

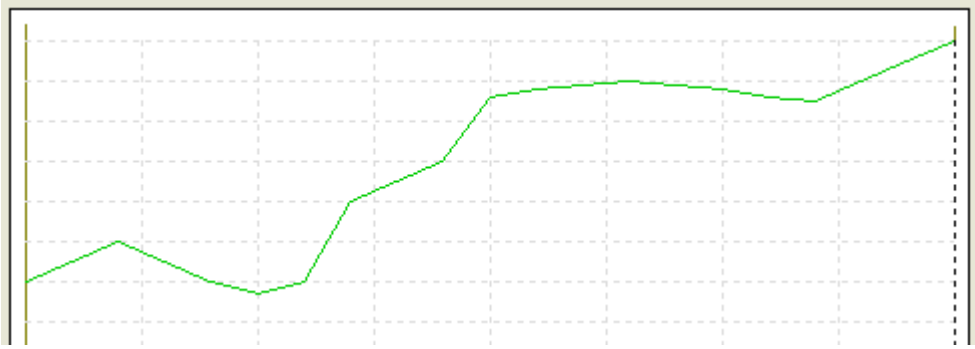
After finishing with all the settings, click [OK (Reflect topographical recovery)]. We will see the following final confirmation message.



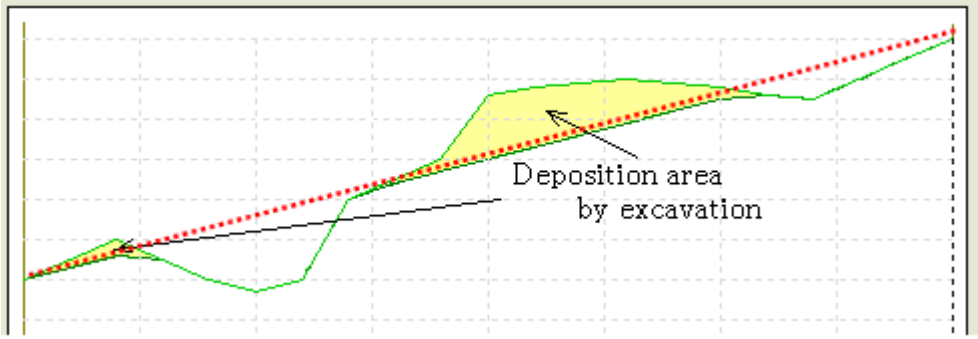
By clicking [OK] button, the recovery process will be executed based on the current settings, while clicking [Cancel] button will take us back to the previous settings dialog box.
After running the recovery process, the slope surface of the current settings will become the sliding surface. The source area created by filling will become a new slope surface. If we only do the process of recovery of deposition area by excavation, the elevation of sliding surface and the slope surface will be the same. In this case, the range of the original slope surface will be deleted.

About the Recovery Calculation

The following shows the calculation of filling and excavation for an example elevation distribution.

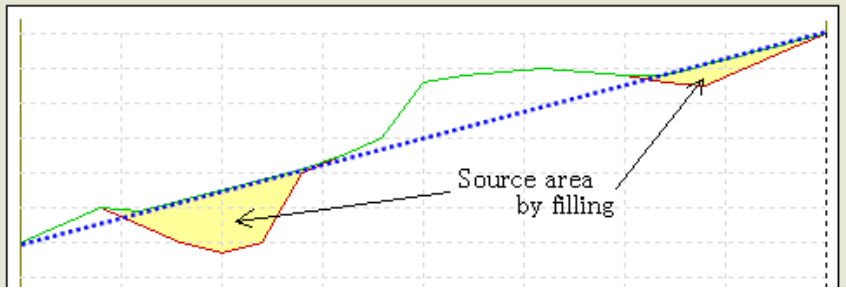


1 Calculation of the "Excavation"



If we select the entire section as the deposition area by excavation, the red dotted line will show the standard elevation line for excavation. If a mesh elevation value exceeds the standard line, the deposition area will be excavated up to the elevation of the standard line. Nothing will be done for the area below the standard line.

2 Calculation of the “filling”



If we select the entire section as the source area by filling, the blue dotted line will indicate the standard elevation line for filling. If a mesh elevation values falls below the standard line, the source area will be filled up to the elevation of the standard line. Nothing will be done for the area above the standard line. The geographical feature before filling will be a new sliding surface. The geographical feature after filling will be a new slope surface.

3.8. Delineating Unstable Mass (Source area and Moving area)

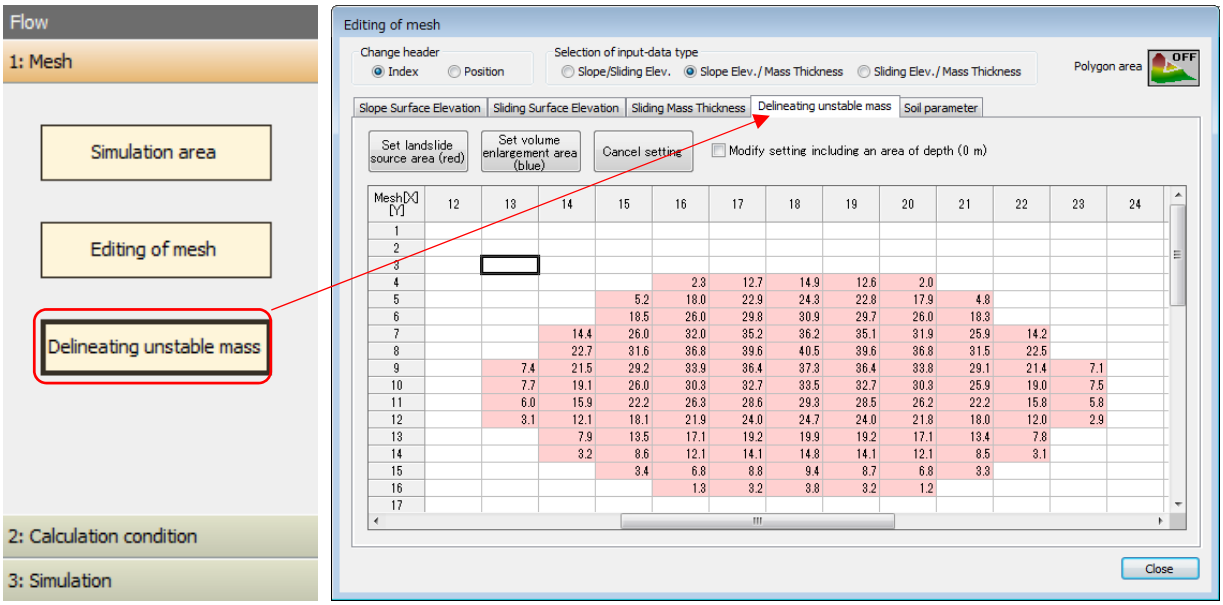
Once the geographical features of both the slope surface and the sliding surface are created, we will see the distribution of the unstable mass height. In the distribution, we can specify the delineation of elements such as “landslide source area” and “volume enlargement area.”

To set the delineation of unstable mass, we can follow the steps below.

Go to the [Edit] menu and select the [Delineation of unstable mass] option

Or click the [Delineation of unstable mass] icon from the toolbar.

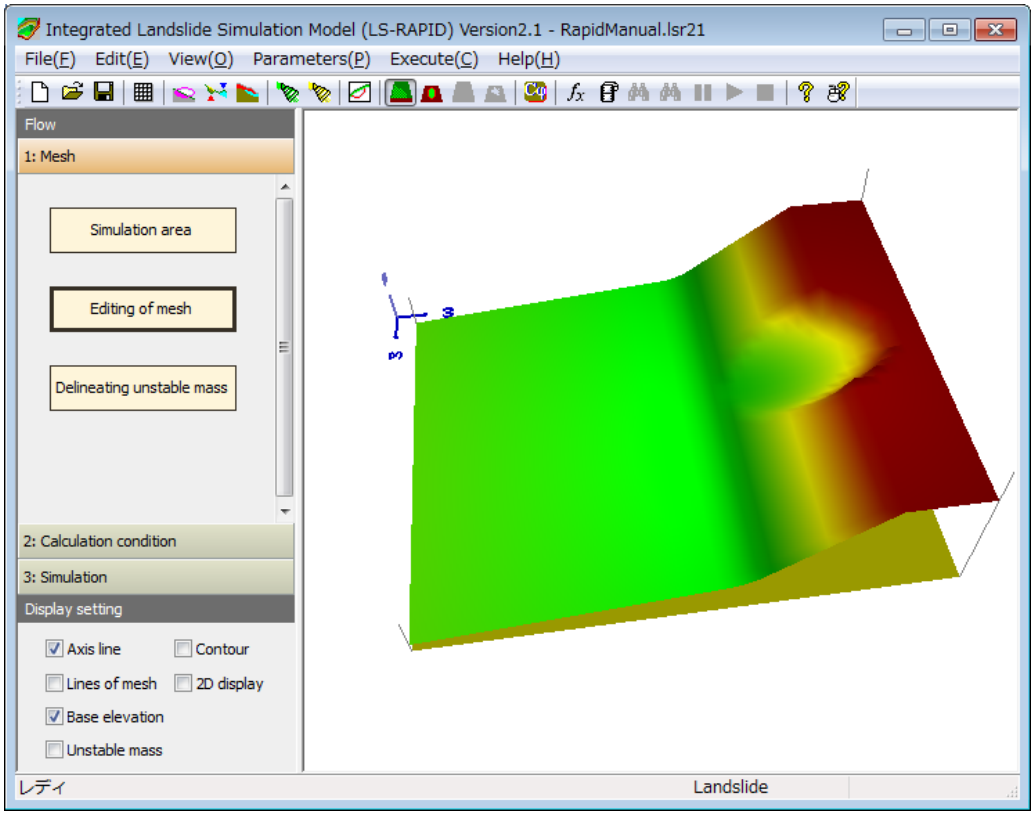
When we click the tab “Delineating unstable mass” at the “1: Mesh” of the “Flow” panel on the left side of the window, the following dialog panel “Editing of mesh” will be displayed.



Item	Description
Set landslide source area (red) Set volume enlargement area (blue)	Clicking this button will set the selected range on the height distribution mesh of the delineation of unstable mass to either “landslide source area” or “volume enlargement area”.
Cancel setting	Clicking this button will cancel the selected range on the height distribution mesh of the delineation of unstable mass.
Polygon area editing	By clicking the button below, either ON or OFF can be activated.  When ON is chosen, we can create the any-shape area by using left click and pushing the keys of [Ctrl]+[Shift].
Modify setting including an area of depth (0 m)	If we select this checkbox, we can set an area which does not contribute to the depth of the unstable mass (cannot specify this area by default).

Note: This area setting is effective for performing “Initiation + Motion + Expansion Simulation”

After finishing with the settings, click [Set completed] to close the dialog box. When the process is completed, we can see the mass distribution in the elevation edit area as follows,



4. Parameters and Conditions for Simulation

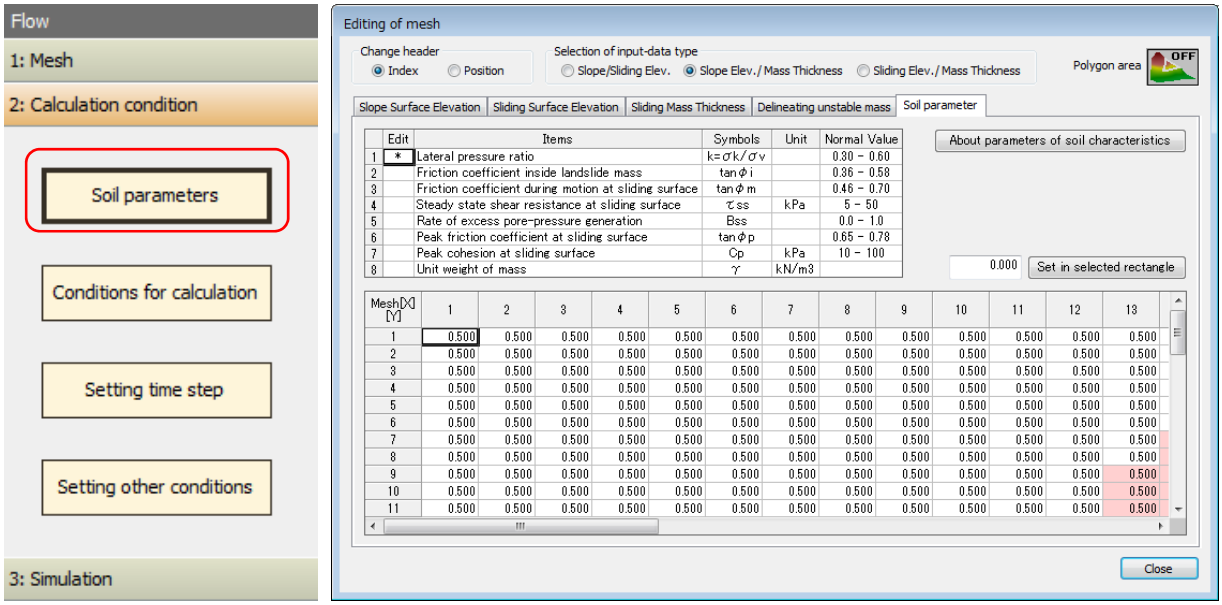
After creating the mesh data representing the geographical features of the slope surface and the sliding surface, we need to configure the necessary settings for simulation.

4.1. Setting the Soil Parameters

Soil parameters are set for every mesh by following steps,

Go to the [Parameters] menu, select [Setting the soil parameters], and choose [Input for each mesh]. Or click the [Soil Parameter] icon from the toolbar.

When we click the “Soil Parameters” option at the “2: Calculation condition” of the “Flow” panel on the left side of the window, the following dialog box of “Editing of mesh” will be displayed.



The “*” in the Edit column indicates the current parameter of being edited. We do not need to enter the data for the gray colored items.
If we press the keys [Ctrl]+[Shift] and left click on the 3D-View window, the corresponding mesh cell in red color will be viewed in the window for mesh editing.

Item	Description
Editing items for parameters of soil characteristics	Set the parameters for different grid points.
Polygon area editing	By clicking the button below, either ON or OFF can be activated.  When ON is chosen, we can create the any-shape area by using left click while pressing the keys [Ctrl]+[Shift].
"Set in selected rectangle" button	First, we input the value of elevation in the blank box beside the button of "Set in selected rectangle". Then, choose the objective mesh area. Finally, click the button of "Set in selected rectangle". As a result, the same values of elevation will be set for all the meshes we selected (this can save the time if we want to input the same elevation values in the meshes). If we have activated the "Polygon area editing" button in ON mode, we can input the same value for any-shape area that already created. In the case of OFF mode, it will be assigned the same value for selected cell blocks.
About parameters of soil characteristics button	Open the description dialog box about the parameters of the soil characteristics.

The following table provides seven parameters that can be set for different grid points.

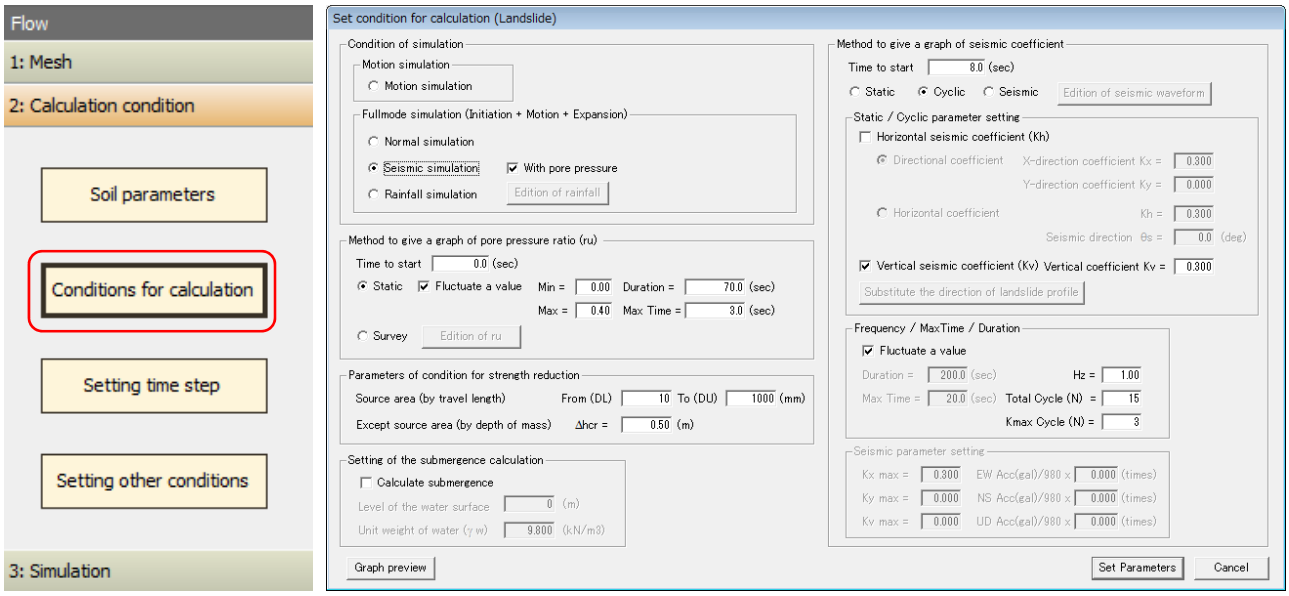
Item	Symbol (Unit)	
Lateral pressure ratio	k	○
Friction coefficient inside landslide mass	$tan\,\phi_i$	○
Friction coefficient during motion at sliding surface	$tan\,\phi_m$	○
Steady state shear resistance at sliding surface	τ_{ss} (kPa)	○
Rate of excess pore-pressure generation	B_{ss}	○
Peak friction coefficient at sliding surface	$tan\,\phi_p$	△
Peak cohesion at sliding surface	c_p (kPa)	△
Unit weight of mass	γ (kN/m^3)	○

Symbol "○" represents the required items, while "△" represents the required items for considering the strength reduction for the "Initiation + Motion + Expansion Simulation."

4.2. Conditions for Calculation

In the LS-RAPID system, we can set various patterns for landslides simulation.

Go to the [Parameters] menu and select [Condition for calculation].
Or by clicking the "Conditions for calculation" option at the "2: Calculation condition" of the "Flow" panel on the left side of the window, the following dialog box of "Set condition for calculation (Landslide)" will be displayed.



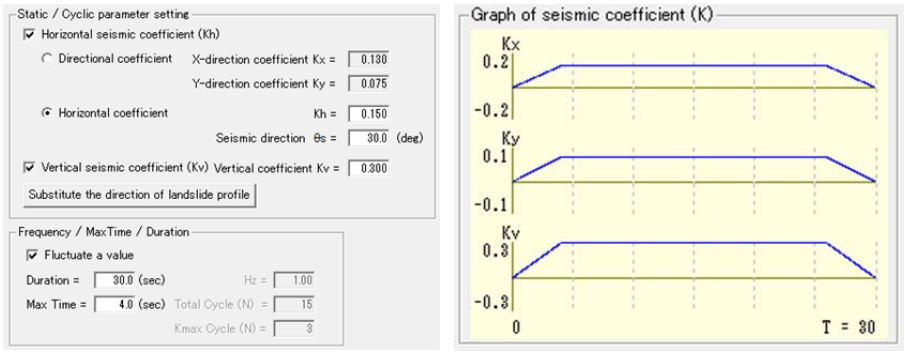
Item	Description
Condition of the simulation	Select the simulation type that we would like to perform. (Motion simulation) ○ Motion simulation (Full mode simulation: Initiation + Motion + Expansion) ○ Normal simulation ○ Seismic simulation ○ Rainfall simulation ☐ With pore pressure If we choose the "With pore pressure" box, pore water pressure will be considered as landslide trigger (Only when selecting Seismic simulation. Time of heavy rainfall is required).

◆Seismic simulation

Item	Description
Seismic loading (K)	Set the seismic loading force as an inducement of the landslide. We can set a constant seismic load or a fluctuating seismic load over time. We can choose one of the following seismic operation methods. ○ Static ○ Cyclic ○ Seismic In the Static method, the seismic coefficient varies linearly. In the Cyclic method, the seismic coefficient varies as a sine wave while the value of the amplitude changes linearly. In the Seismic method, the seismic coefficient varies based on the actual seismic wave form data.
Edition of seismic waveform button	If the seismic method is selected, we need actual seismic wave data. By clicking this button, the dialog box can be seen for editing the waveform data.
Substitute the direction of landslide profile button	By clicking this button, if we created the sliding surface (in the main section) from an ellipsoid, the direction of the land sliding longitudinal section will be set as the seismic direction.

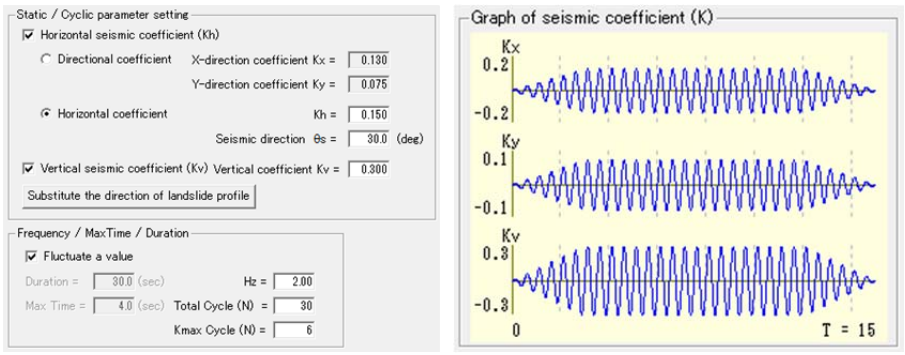
<Example of settings>

(1) Static Method



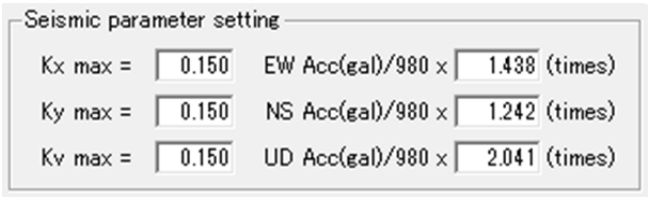
The above Static setting will cause the seismic coefficient to go up from K=0 at the beginning of the simulation to Kmax in four seconds and it will remain fixed for thirty seconds.

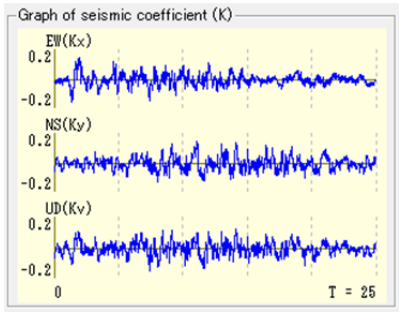
(2) Cyclic Method



The above Cyclic setting will cause the amplitude of the seismic coefficient to go up from K=0 at the beginning of the simulation to Kmax in three seconds (=6/2Hz) and it will remain fixed for fifteen seconds (=30/2Hz).

(3) Seismic Method





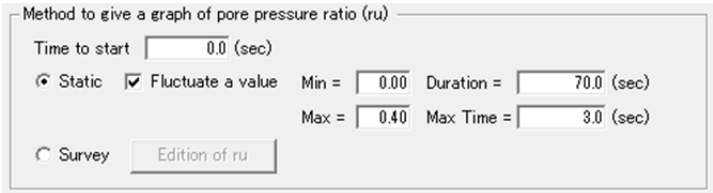
The above Seismic setting will cause the seismic coefficient to become Kmax=0.15 based on the edited seismic waveform. We can set the value of "Kmax" for the seismic coefficient in the X-direction, Y-direction, and the Vertical direction (*Note: see more details about the editing of the seismic waveform in the next section*)

In addition, when conducting a Seismic simulation, if we choose "With Pore Pressure", pore pressure can also be considered to be a factor to trigger landslide.

Item	Description
Water pressure (pore pressure ratio: r_u) ($r_u = u/\sigma$ … 0.0 ～1.0)	Setting the pore pressure as a factor to trigger landslide. Pore-water pressure can be set as a constant value or variable value changing with time. The start time can also be set. The pore pressure can be set as: ○ Static ○ Survey In case of Static, the pore pressure will be constant. In case of Survey, the monitored data of pore pressure can be used.
Input pore pressure: r_u	If we select Survey as an operating method, it is necessary to input actual data of pore pressure. After clicking this button, we can edit r_u in the editing dialog of pore pressure data.

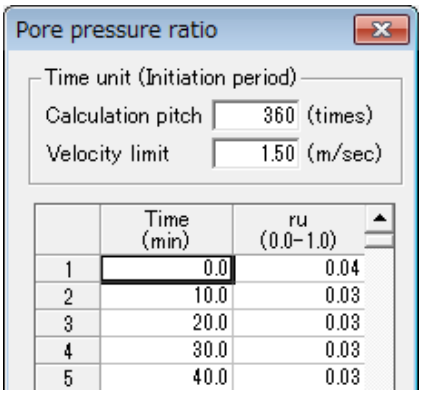
<Example of settings>

(1) Static Method



The above setting means that the pore pressure will go up from r_u = 0.0 at the beginning of the simulation to r_u = 0.4 in three seconds and remain fixed for seventy seconds.

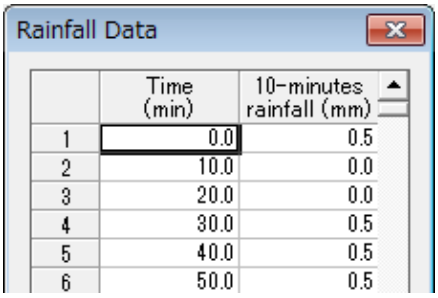
(2) Survey Method



Item	Description
Calculation Time Unit (Initiation period)	
Calculation pitch Velocity limit	This button is to set the landslide velocity faster than the planned velocity for each calculation step, when it is not yet reach the velocity limit.
Monitored value of r_u	Input the pore pressure ratio (r_u)

◆ About the simulation in heavy rainfall

Item	Description
Input rainfall data	Input the rainfall data as the factor to trigger landslide. Input unit is mm/10 min. By clicking this button, we can edit rainfall data.



Setting of pore pressure ratio is necessary for the simulation of rain-induced landslides.
To estimate the pore pressure ratio from the rainfall record, we can refer to "3.Estimation from the rainfall record to the pore pressure ratio of the Theoretical Background of LS-RAPID".

◆Parameters of condition for strength reduction

Item	Description
Parameters of condition for strength reduction Source area (by travel length)	In the (Initiation + Motion + Expansion) simulation mode, the simulation will reduce the friction coefficient and the cohesion from their peak values to the normal motion time values within the source area in the determined distribution of the unstable mass (reducing from "$tan\,\phi_p$" to "$tan\,\phi_m$" and from "c_p" to "c_m").

	Note: Essentially, when the travel length becomes DL (mm), the reduction will be started. Once the travel length reaches DU (mm), the reduction will be completed and then normal motion simulation will start.
Parameters of condition for strength reduction Except source area (by depth of mass)	In the (Initiation + Motion + Expansion) simulation mode, the simulation will reduce the friction coefficient and the cohesion from their peak values to the normal motion time values within the source area in the determined distribution of the unstable mass (reducing from "$tan\,\phi_p$" to "$tan\,\phi_m$" and from "c_p" to "c_m"). *Note: Essentially, when the depth of mass reaches* $\triangle h_{cr}$ (m), *the reduction will be completed and the normal motion simulation will start.*

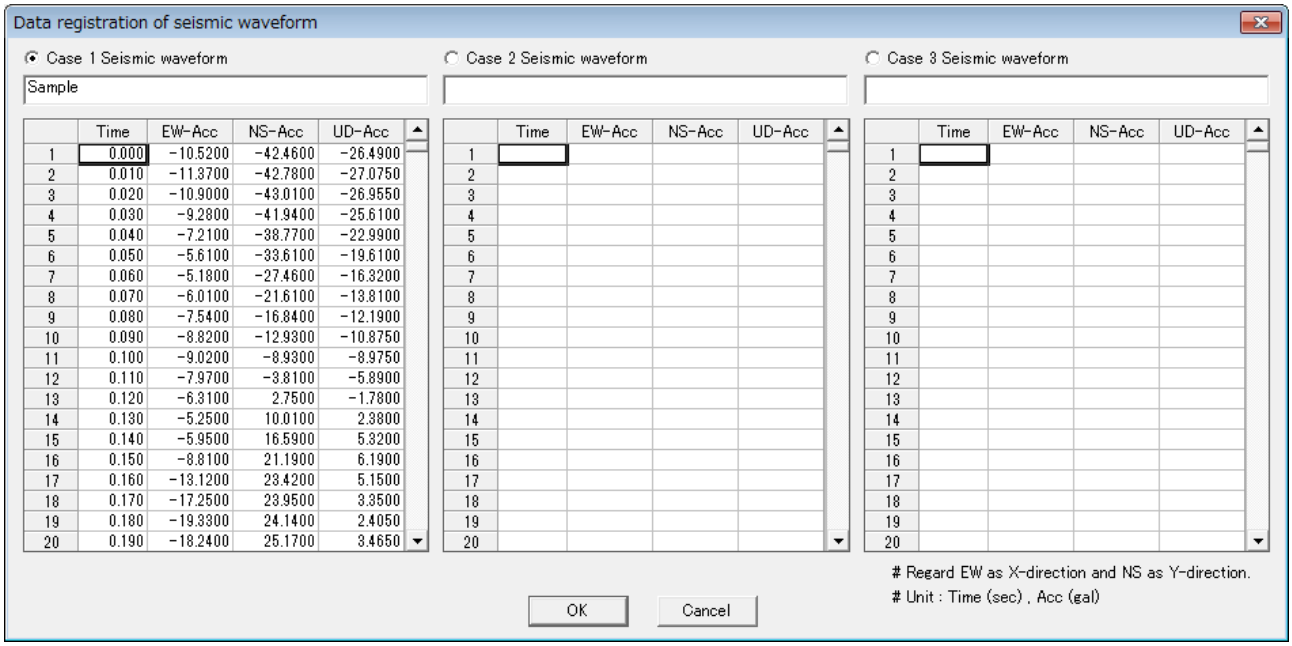
◆ Setting of the submergence calculation

Item	Description
Calculate submergence	When choose this button, the soil mass under the water will be considered as buoyancy. (*example：the simulation of submarine landslide*)
Level of the water surface	Set the depth of the interface between the submerged and non-submerged soil mass.
Unit weight of water	Set the unit volume weight of water

4.3. Registration of Seismic Waveform

In the landslide simulation, we can set the seismic waveform based on the model seismic waveform as well as actual observation data by the initiation simulation of inducement of seismic loading. The following section describes steps for setting the seismic waveform based on actual observed data.

To start editing, we need to select the Seismic method in the previous section, "Condition of calculation", and click [Edition of seismic waveform]. Or go to the [Parameters] menu and select [Registration of seismic waveform]. The following window will be displayed.



	Time	EW-Acc	NS-Acc	UD-Acc
1	0.000	-10.5200	-42.4600	-26.4900
2	0.010	-11.3700	-42.7800	-27.0750
3	0.020	-10.9000	-43.0100	-26.9550
4	0.030	-9.2800	-41.9400	-25.6100
5	0.040	-7.2100	-38.7700	-22.9900
6	0.050	-5.6100	-33.6100	-19.6100
7	0.060	-5.1800	-27.4600	-16.3200
8	0.070	-6.0100	-21.6100	-13.8100
9	0.080	-7.5400	-16.8400	-12.1900
10	0.090	-8.8200	-12.9300	-10.8750
11	0.100	-9.0200	-8.9300	-8.9750
12	0.110	-7.9700	-3.8100	-5.8900
13	0.120	-6.3100	2.7500	-1.7800
14	0.130	-5.2500	10.0100	2.3800
15	0.140	-5.9500	16.5900	5.3200
16	0.150	-8.8100	21.1900	6.1900
17	0.160	-13.1200	23.4200	5.1500
18	0.170	-17.2500	23.9500	3.3500
19	0.180	-19.3300	24.1400	2.4050
20	0.190	-18.2400	25.1700	3.4650

Here, we can register three observational cases. In the actual simulation, we can select one of these cases and start the calculation. For each case, enter a name for the data-set in the top box and enter the values of the observation times and the acceleration K (unit: Gal) in the appropriate columns.

<Things to Note before Entering the Data>

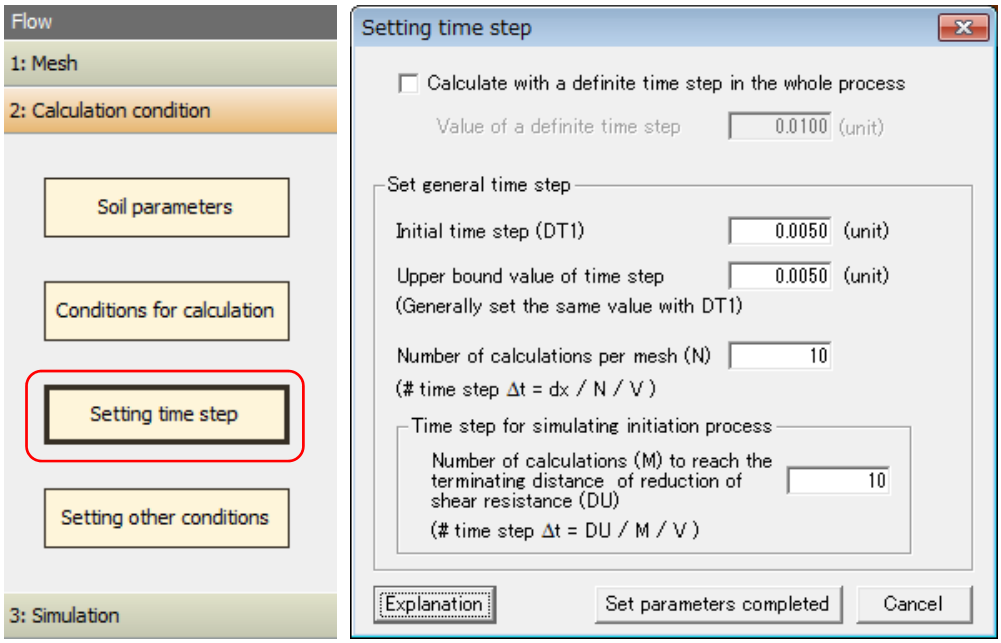
1. Enter the observation time in chronological order.
2. When entering the seismic coefficient, the EW means the X-direction and NS means the Y-direction.

4.4. Setting Time Step

In the landslides simulation, we can adjust the motion process degree using a specified time step.

Go to the [Parameters] menu and select [Set time step].

Or by clicking the "Setting time step" option at the "2: Calculation condition" of the "Flow" panel on the left side of the window, the following dialog box of "Setting time step" will be displayed.



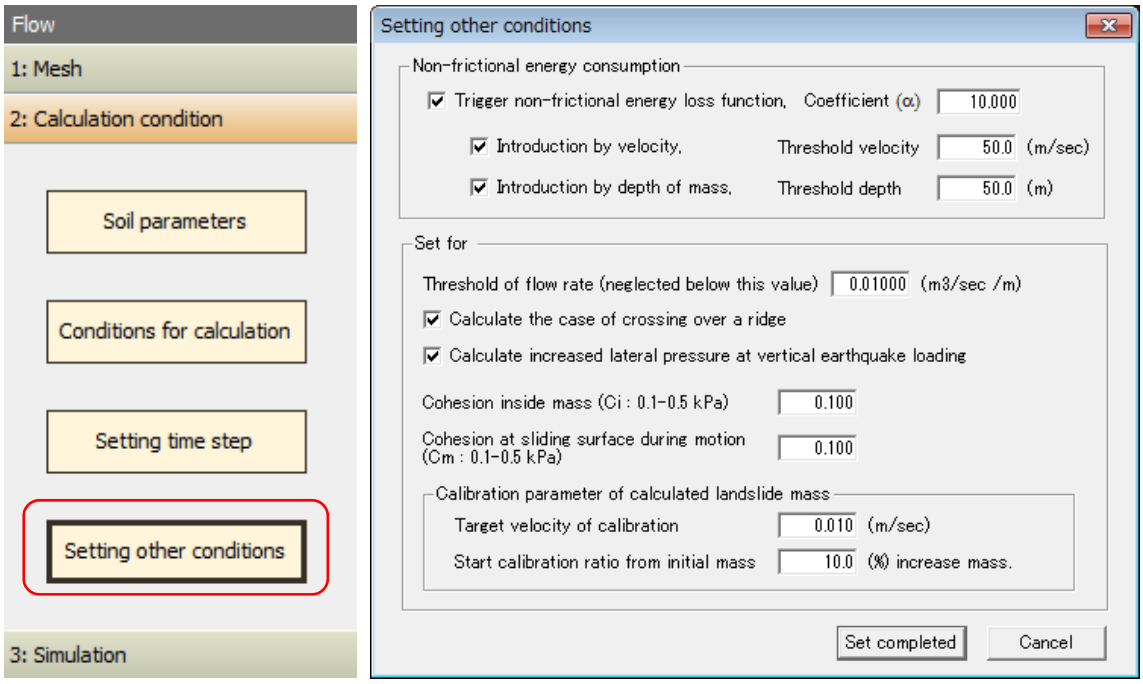
Item	Description
Calculate with a definite time step in the whole process	By selecting this checkbox, the simulation time step will remain constant as the simulation proceeds.
Initial time step (DT1)	Set the time step for the first step of the simulation.
Upper bound value of time step	If the next time step calculated from the average maximal velocity v=$\sqrt{u_{max}{}^2 + v_{max}{}^2}$ is more than the entered value, set the next time step to the value entered here.
Number of calculations per mesh (N)	Set the number of calculations by mesh N where the next time step is defined by dx(=mesh pitch)/N/v.
Number of calculations (M) to reach the terminating distance of reduction of shear resistance (DU)	Set the number of calculations M to define the time step by DU/M/v in the parameters of the initial simulation. After reaching the terminating distance of reduction of shear resistance, the simulation proceeds with the time step of the motion simulation.

4.5. Setting Other Conditions

This section explains the system's less-frequently used simulation settings.

Go to the [Parameters] menu and select [Set other conditions].

Or by clicking "Setting other conditions" option at the "2: Calculation condition" of the "Flow" panel on the left side of the window, the following dialog box of "Setting other conditions" will be displayed.



◆Non-frictional energy consumption

Item	Description
Trigger non-frictional energy loss function	By checking this box, the non-frictional energy consumption (=α*1/2*m*v²) will be considered in the simulation.
Coefficient α	This is the coefficient (α) for the simulation when triggering the non-frictional energy loss function.
Introduction by velocity Introduction by depth of mass	By checking this box, the non-frictional energy consumption (=α*1/2*m*v²) will be considered, corresponding to the mesh which reaches the specified velocity (depth of mass).

◆Other settings

Item	Description
Threshold of flow rate (neglected below this value)	The threshold of flow rate is the minimum amount the soil is moving in the calculated mesh within the unit time. The unit is m^3/sec. In the case of landslide, the "mm" unit landslide is thinner than water; therefore, the landslide cannot usually flow. Thus, if the flow rate goes below threshold, the mesh does not accept the flow. The value depends on the size of the landslides, but ~0.01 m^3/sec can be used as the first approximation for usual landslides
Calculate the case of crossing over a ridge	When the landslide mass travels down a curved torrent, most of it travels along the torrent. However, there is a case in which the upper part of the landslide mass, if it is higher than the ridge, goes over the ridge. If we would like to include this effect in the simulation, select the checkbox or deselect it if not necessary (there is no harm in leaving the checkbox selected whether there is a case of crossing over the ridge or not).
Calculate increased lateral pressure at vertical earthquake loading	As we can see in "Explanation of Section" 1.1., three seismic stress factors are considered in the simulation. When the vertical stress is increased, the lateral stress should be increased accordingly to the lateral pressure coefficient (the ratio between the vertical pressure and the lateral pressure). If we concerned about this effect, select this checkbox, if not then deselect the checkbox (the checkbox should be selected under normal conditions).
Cohesion inside mass (Ci・kPa)	Set the cohesion inside mass for all the geographical features.
Cohesion at sliding surface during motion (Cm・kPa)	Set the cohesion at sliding surface during motion for all the geographical features.
Calibration parameter of calculated landslide mass - Target velocity of calibration - Start calibration from initial mass	In general, the total volume of soil mass does not change during simulation. However, the volume is possible to increase when calculation reach unsteady situation. In this case, we can adjust the velocity by increasing x % of the velocity before simulation.

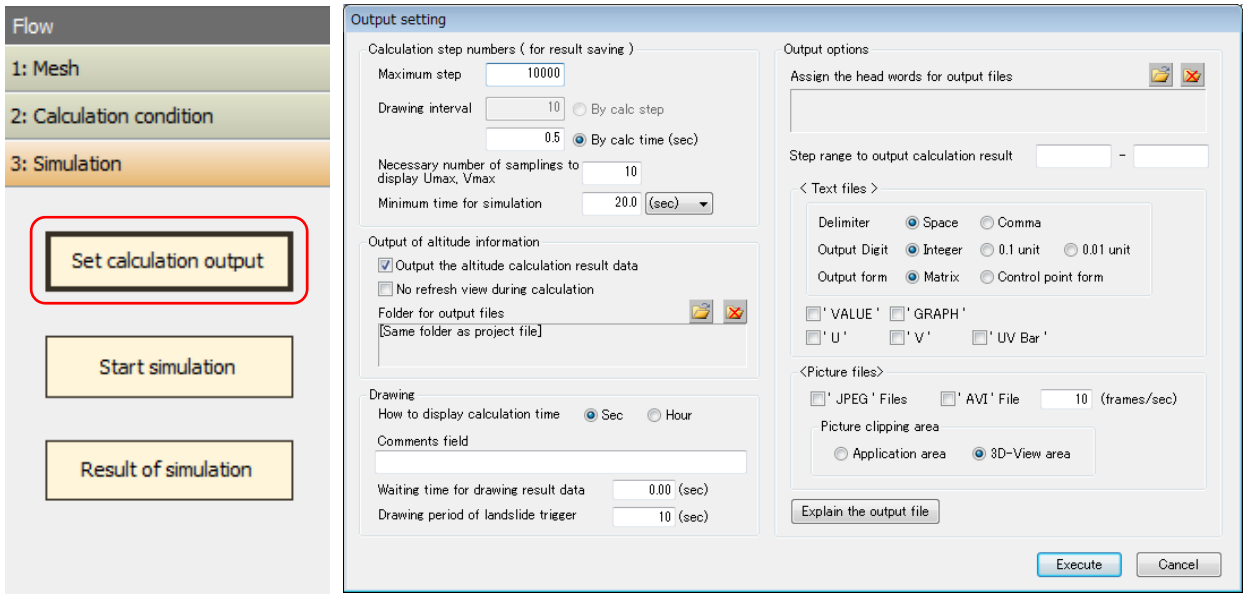
Note: The ranges listed beside "Ci" and "Cm" in the dialog box represents typical values.

5. Run Simulation

This section describes the output settings for the landslide simulation.

5.1. Output Settings for the Landslide Simulation

When we click the “Set calculation output” option at the “3: Simulation” of the “Flow” panel on the left side of the window, the following dialog box of the “Output setting” will be displayed.



◆Calculation step numbers (for result drawing and saving)

Item	Description
Calculation step numbers (for result drawing and saving)	Set the number of the calculation steps
Maximum step	Set the maximum step
Drawing interval o By calc. step o By calc. time (sec)	Set the time interval for updating the geographical features during the simulation. The intermediate simulation results will be saved in the output files at the end of each time interval.
Necessary number of samplings to display, Umax, Vmax	The number of samples of the velocity per mesh in order to display the X and Y components of the maximal velocity during the simulation (the displayed velocity values for each mesh are the average over the samples.
Minimum time for simulation	Essentially, the simulation will be finished when all the solid features are stopped. However, if we set the minimum time, the simulation will run until the end of the entered time, even if all the solids have stopped.
How to display calculation time	We can choose calculation time in second or in hour.

Item	Description
Comments field	Enter any comments that would like to be displayed during simulation.
Drawing period of landslide trigger	When conduct a simulation under earthquake or heavy rainfall, the graphs can be displayed together to compare them. The length of time axis can be set.

◆ Output the altitude information

Item	Description
Outputs the altitude calculation result data	If this column checked, the resulted calculation of elevation data after simulation is an output that will be stored into a same folder as the project file located. In the output options, we can set the display updating interval as "passed time". ※ the output file can be the examples as below: (when set the output interval as 1.0 second) G0000120.dat: the ground surface altitude after 12.0 seconds M0000120.dat: the altitude difference between the 12th second and 11th second. (= "G0000120.dat" - "G0000110.dat")
No refresh view during 3D calculation	If we choose this option, the 3D view will not be refreshed in calculation. The display time will be somehow shortened.
Waiting time for drawing altitude data	If we want to view the results with conducting the simulation, we can use this button to read the outputted altitude data. The waiting time can be set if we want to view the results earlier.

◆ Output options

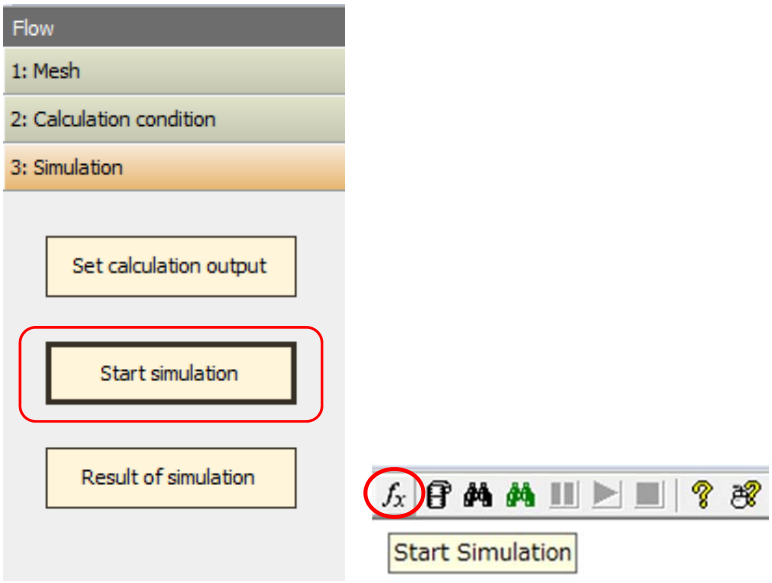
(*Options here is different with the above "Output the altitude information"*)

Item	Descriptions
Assign the head words for output files	We can set the head words for output files and folders
Step range to output calculation result	We can output the data in a range (As default, all data from the beginning will be outputted)
Text files	Here, the following files can be outputted: VALUE … Depth of unstable soil mass for each mesh GRAPH … Using graph to display the depth of unstable soil mass for each mesh U, V, UV Bar …Velocity of soil mass for each mesh (U: X direction、V: Y direction、UV Bar: mean value)

Delimiter (when output the text)	○ Space ○ Comma Setting the text between the values
Output Digit (when output the depth of soil mass)	○ integer ○ 0.1 unit ○ 0.01 unit Choose the unit for the depth of soil mass
Output Form (when output the depth of soil mass)	○ Matrix ○ Control Point Form When give an output for the soil depth, we can display them as matrix or control point form (output the depth for each mesh ID and soil mass depth in a row)
Picture clipping area (when output the images)	○ Application area ○ 3D-View area We can select the application area or choose the interested area from 3D display and then create them as JPEG image and AVI animation files. (In case the JPEG image cannot be displayed normally, we can choose 3D-View area)

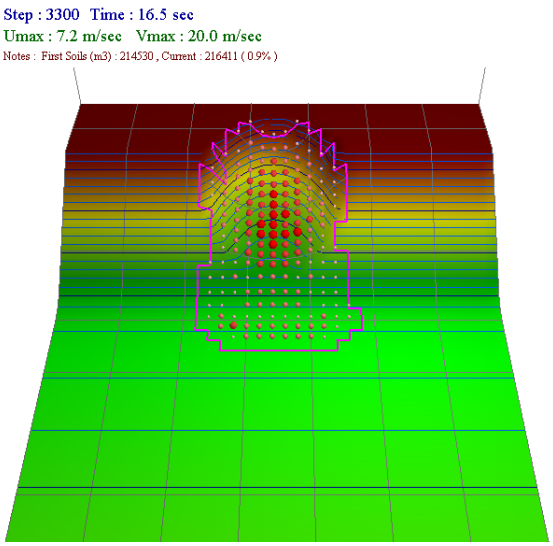
5.2. Start Simulation

Click from “Flow” panel → then [3: Simulation] option → and choose [Start simulation].
Or by clicking the [Execute] menu and select [Start Simulation],
Or click the [Start Simulation] icon “f_x” directly from the toolbar to begin the simulation.

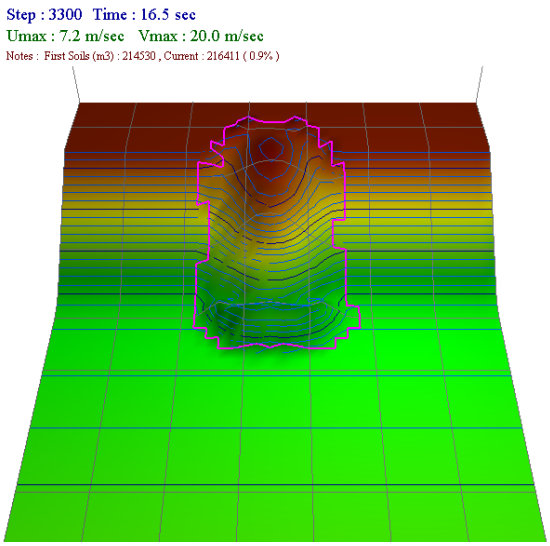


The following figures are the captured screens during simulation as examples

(Left: an illustration of the ellipsoid)



(Right: an illustration of the mass thickness expanded two times)



5.3. Commands during the Simulation

We can run the following commands during the simulation.

Calculation monitoring	Opens a dialog box showing the mass thickness and the values of different components of the velocity
Given triggering factors	Opens a dialog box showing the pore pressure and seismic loading and rainfall
Pause	Pauses the simulation temporarily
Restart	Restarts the simulation which has been paused temporarily
Stop	Forces the simulation to stop

Note: Both the "Calculation monitoring" and "Given triggering factors" commands can also be called from the [View] menu by selecting [Display of calculation values].

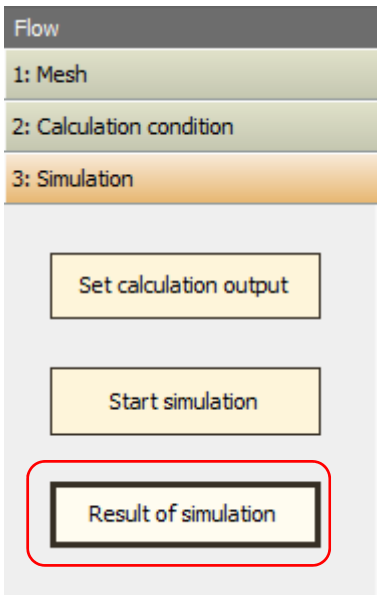
5.4. Display the simulation results

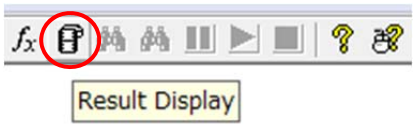
If we have selected the "Output of altitude information" in the setting of simulation output (described in the above Section 5-1), we can read and display the outputted altitude data when we run the simulation.

Click from the "Flow" panel on the left side of the window → then choose [3: Simulation] → and select the [Result of simulation],

Or by clicking the "Execute" from Menu and choose "Display the simulation results".

Otherwise, we can directly select the icon "Result Display" to display the simulation results.





6. Other Commands

This section explains other commands which have not been mention in the previous sections.

(1) The [File] menu

Menu Item	Description
New	Creates a new data file for the landslide simulation system
Open	Opens a data file
Save	Saves the data file
Save as	Saves the document as a different file. Choose the new file name and location for the document
Read text data file	The text data files (Control Point, Mesh Data) can be read. The tool transferring Control Point to Mesh Data is also prepared here
Write to a Text Data	Export the entered numerical values（Control Point、Mesh Data、Parameters of Soil Characteristics）to a text data file
Setting background image file	Setting the image files which can be pasted to the background
Save image by JPEG	Saves the image currently displayed in the 3D view as a JPEG file
Close of File	Close the editing file
Exit of Application	Exit

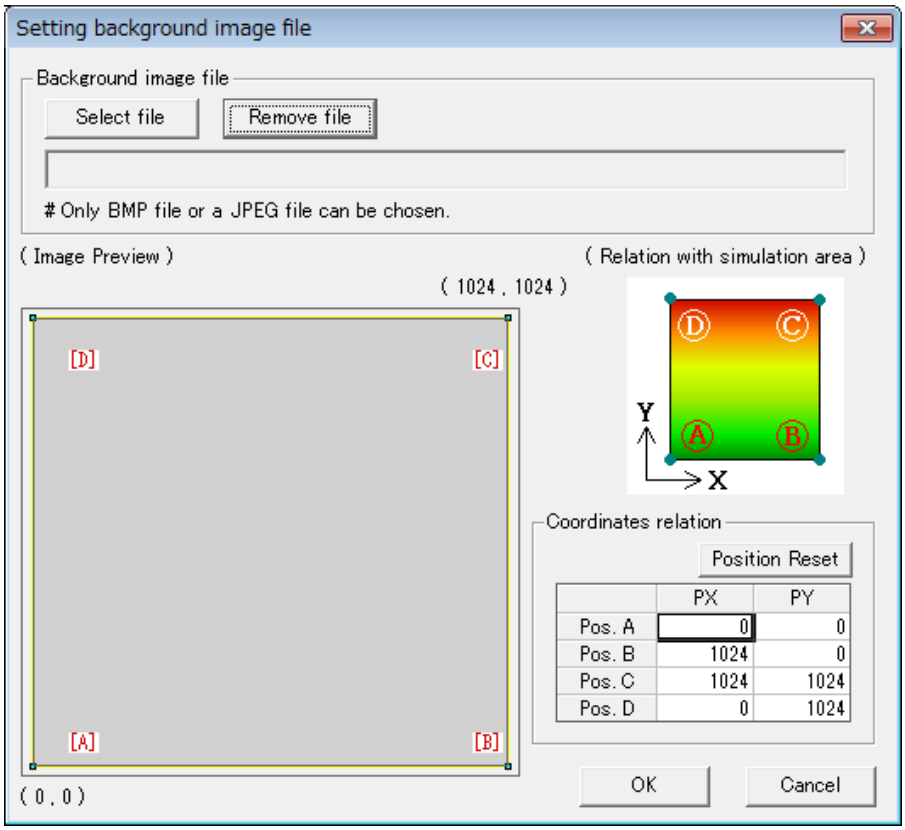
Note: There are two options under the “Write to a Text Data” item in [File] menu, namely, “All Data” and “Specific Data”. If we choose the “All Data” option, all values will be exported to a text file. We need to specify the location of the exported file the same way as we did for the simulation output file under the “assign the head words for output files” in the “output setting” dialog box (Section 4-6). If we choose the “Specific Data” option, we have to select the value to be exported from the list that will show up, and then select the location of the exported file and save it.

About the “Setting background image file”

After we choose this menu, we can paste the selected image to the background.

Opening the dialogue of Setting Background Image File, we may choose the objective file by clicking the [Select file] button. From the image preview window, we can select the interested area. When we want to remove it, just click the [Remove file] button. In addition, only BMP and JPEG files can be selected.

After the files are chosen, we may set the position where the selected image located in the simulation area. As defaults, the image’s triangular positions are exactly same with the simulation area’s triangular positions.

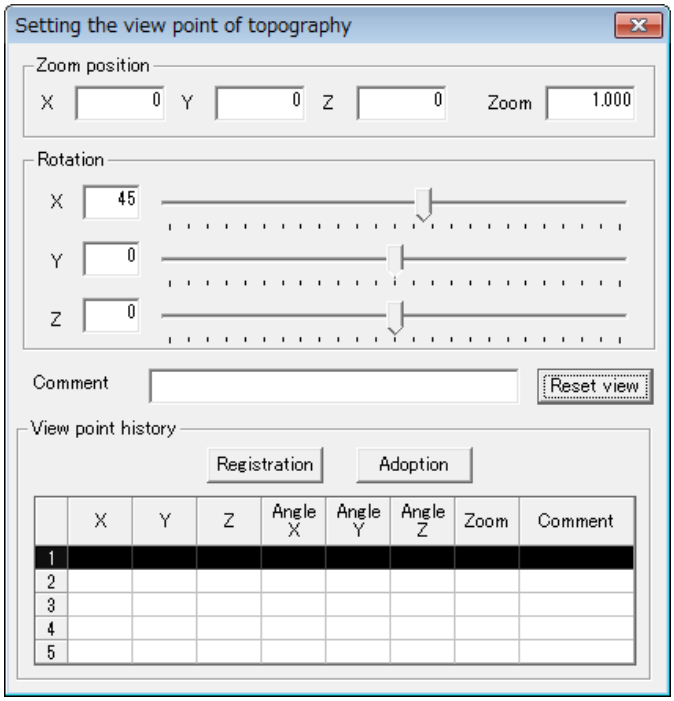


(2) The [View] menu

Menu Item	Description
Setting the View Point of Topography	Update the viewpoint by setting the angle of rotation for each axis. We can update the viewpoint of the screen by using a mouse
Reset geographical viewpoint	Reset the topography viewpoint to the default setting
Reset geographical feature	Reset the topography data that has been previously calculated
Setting of Topography View	Setting of topography view
Setting of Topography Color	Setting of topography color and contour
Current Figure	Select the topography that we would like to see from the following list ○ Slope surface・Sliding surface (Default) ○ Thickness of Sliding mass ○ The Last View of Calculation ○ Sliding Mass Thickness distribution
View Cross or Longitudinal Section	View the cross section at the selected location.
Output arbitrary section profile	Set the output section arbitrarily. We can do output for the section coordinates.
Display of calculation values	Display the value of each mesh data during the simulation. In “landslide inducement parameter”, the values indicate the pore pressure and the seismic loading for the initial simulation.

”Setting the view point of topography”

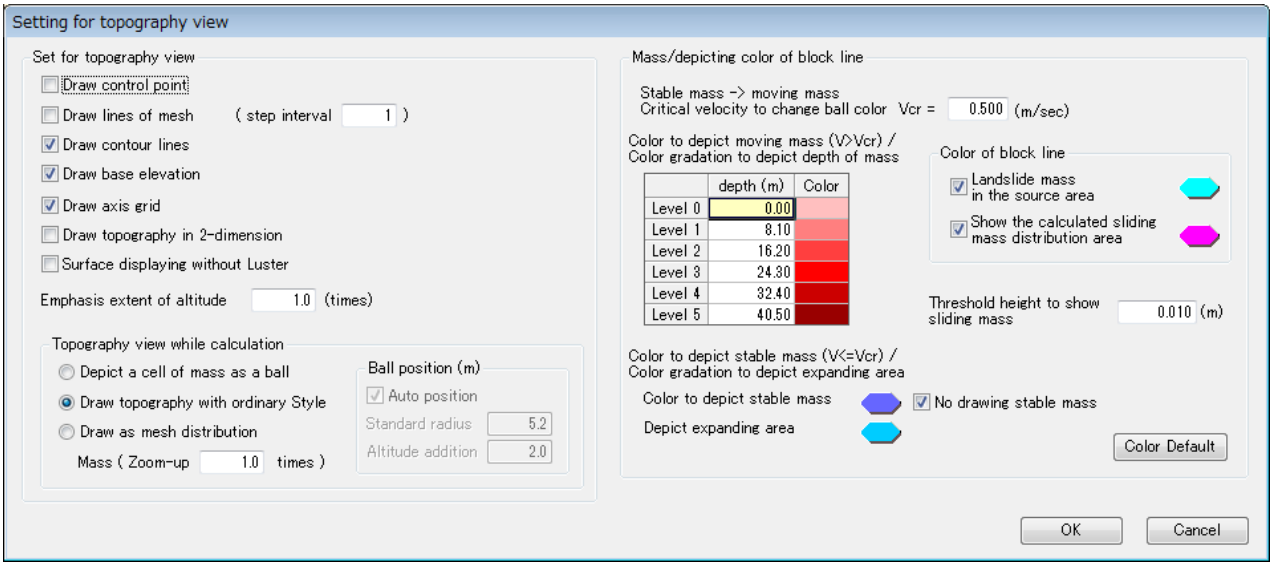
From this menu, we can use the following dialogue to set the parameters which are needed to set the viewpoints of topography.



In this dialogue, we may set parameters of Zoom Position (X, Y, Z), Zoom Ratio, Rotation Angle (X, Y, Z axis・degree unit). If we want to save the desired view points or to fix some view points, we may register the maximum 5 view points by click the button of Registration. When we want to use the registered view point, just select the row and then click the button [Adoption].

About the “Setting of Topography View” Menu

Once we select this menu, the following dialog box will appear to set the topography view as necessary.



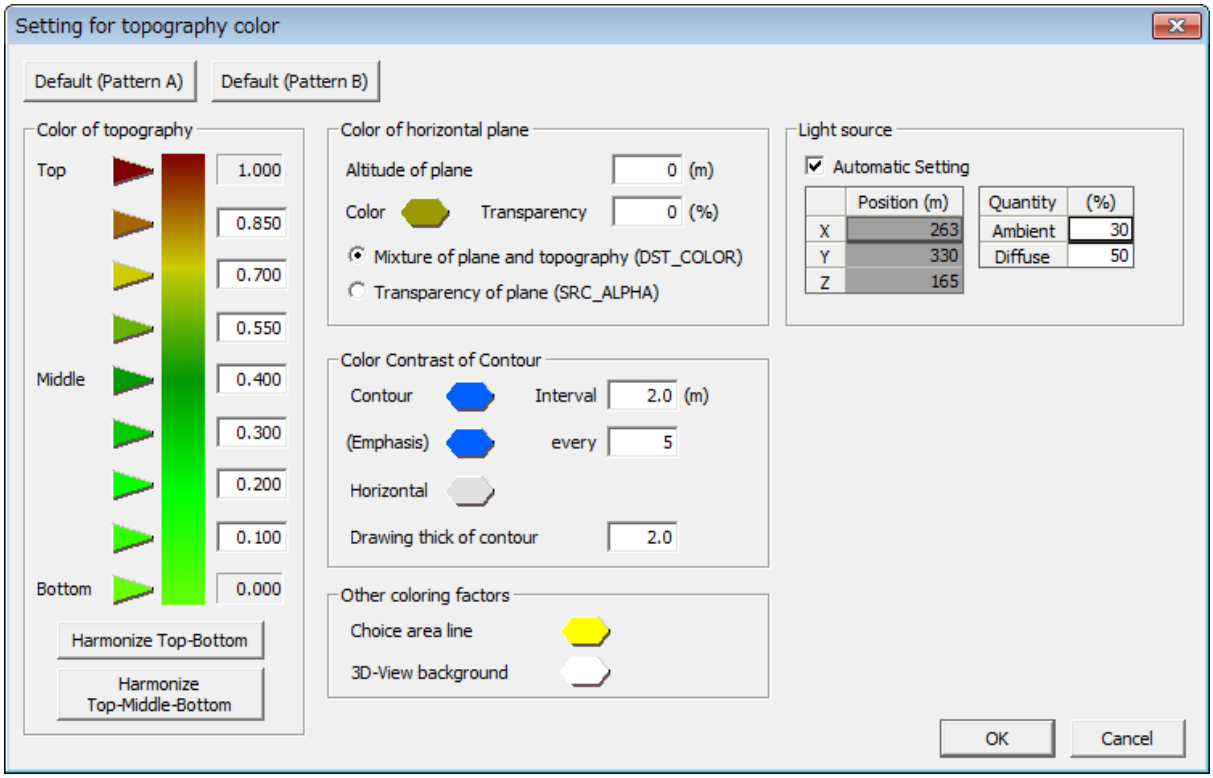
Item	Description
Draw control point	The software will display the control point if any.
Draw lines of mesh	The software will display the grid lines. We can specify the step interval.
Draw contour lines	The software will display the contour lines. We can also specify the interval.
Draw base elevation	Draw the base of elevation.
Draw axis-grid	Draw an axis of coordinates.
Draw topography in 2-dimension	Draw the topography in two-dimensions.
Surface displaying without Luster	Check this item if the topography does not show up in the 3D view (this setting is usually unnecessary).
Emphasis extent of altitude	If we want to emphasize the altitude compared to the real ones, we may type the emphasis extent value here.
Topography view while calculation	Select one from the list below: o Depict a cell of mass as a ball o Draw topography with ordinary style o Draw as a mesh distribution
Mass zoom up	When we select either “Draw topography with ordinary style” or “Draw as mesh distribution”, we can display the topography by customizing the zoom level of the mass.
Ball position	If we choose [Depict a cell of mass as ball], we can set the standard radius and altitude addition. Generally, they are set automatically.

Item	Description
Mass/depicting color of block line	
Critical velocity to change ball color V_{cr}	Sets the critical velocity (V_{cr}) at which the stable mass changes to a moving mass.
Threshold height to show sliding mass	Displays a mass depth greater than the specified value as "sliding mass".
Color gradation to depict depth of mass Color gradation to depict expanding area	Sets the color gradient to depict the depth of the mass when describing the moving mass and the stable mass by an ellipsoid or a mesh.
Color of block line	Displays the block line for showing the simulated sliding mass distribution area and for the landslide mass in the source area of the motion simulation.
Color Default	Return to the initial default values

※ If we want to depict the color of mass, just double click the "Color" palette or hexagon and change the color.

About the "Setting for Topography Color" Menu

After select this menu, the following dialog box will appear. We can set the topography view as necessary.



Item	Description
Default Color (Pattern A, B)	Set the color using the default patterns. Pattern A is mainly used for sub-aerial landslides, and Pattern B is mainly used for sub-marine landslides.
Color of Topography	We can set the color according the elevation of topography by clicking the triangle on the left side of color palette. On the right side, input the elevation ratio (0.0-1.0) according the color that we have specified.
Harmonize Color (Top-Bottom / Top-Middle-Bottom)	To adjust the color between the ranges. "Top Bottom" is to harmonize the color between Top and Bottom color. "Top Middle Bottom" is to harmonize the color between Top and Middle, and Bottom color.
Color of Horizontal Plane	Set the color/level/permeability for some topographic surface. If we wish to run the submarine landslide simulation, we can define the water surface here.
Color Contrast of Contour	When we want to depict contours, we can set the interval, color, thickness, horizontal option of the contour lines here.
Other coloring factors	
Choice area line	Set the color for the objective area we have selected by lines.
3D-view background	Setting the 3D-view background

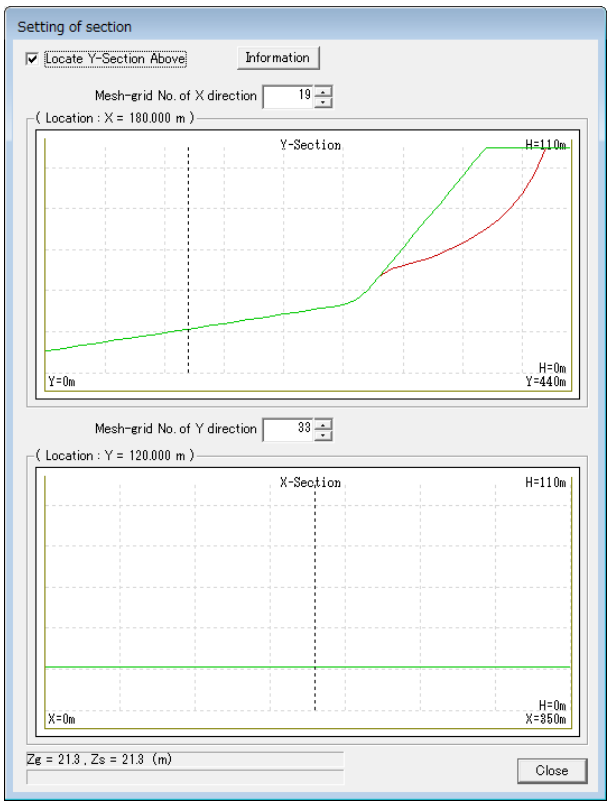
Item	Description
Light source	
Position	When displaying the altitude in 3D view, we may select the position of light source, even though we can use default values.
Quantity	This option is to set the parameters of light quantity. The Ambient or Diffuse light can be selected. Ambient light is used to set the light for entire ground altitude. Diffuse light is used to set the light from light source, which is similar to sport light.

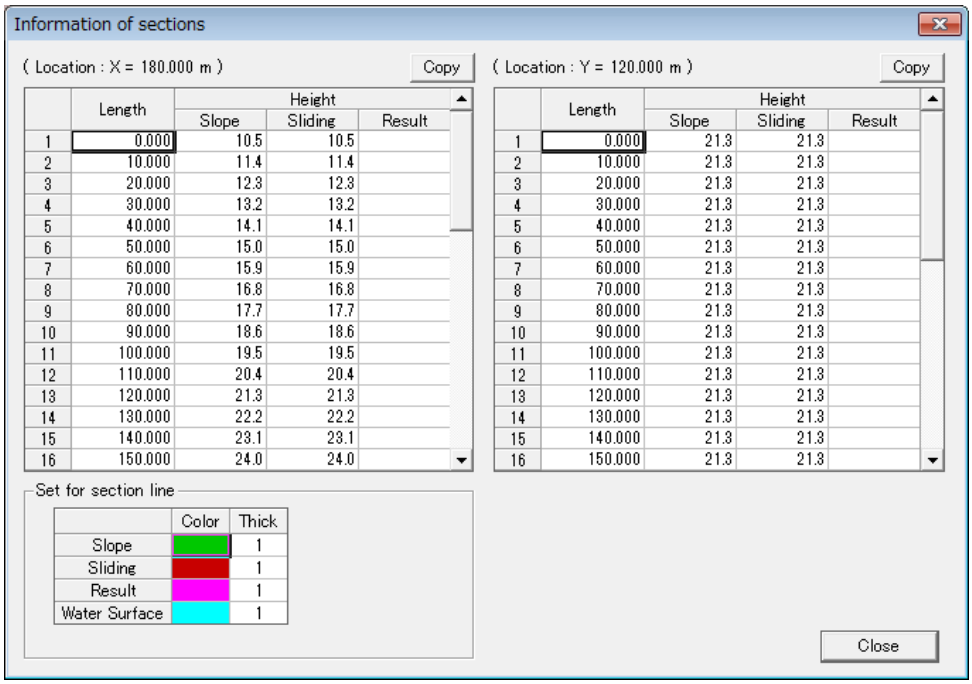
About the "View Cross or Longitudinal Section" Menu

In this menu, the cross sections parallel to the X and Y axis can be viewed as shown in the dialogue box below. In the default colors, the green means slope surface; the red means sliding surface; the pink means the section of the slope surface after simulation.

If we click "information", the dialogue of "information of sections" will appear and we can define the color and thick of the cross sections.

If we select the "Locate Y-Section Above" checkbox, the order of the section will be displayed reversely.





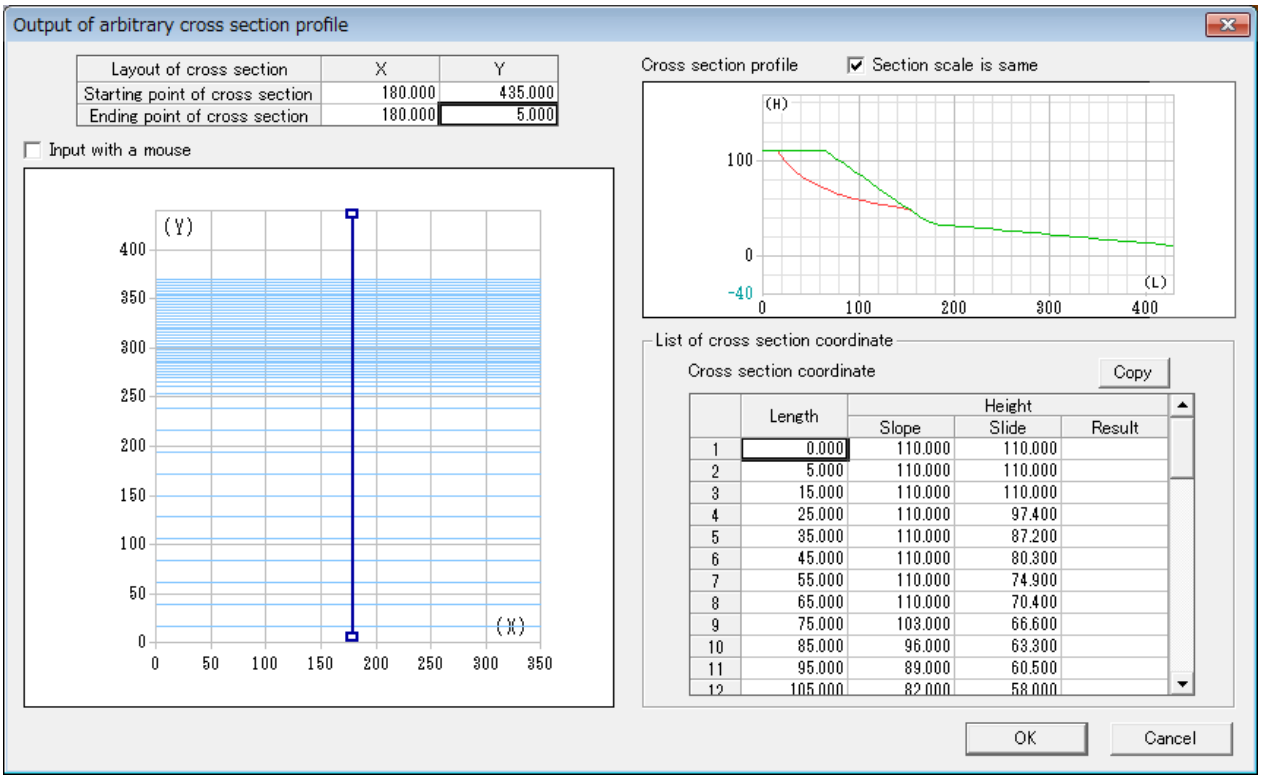
	Length	Height		
		Slope	Sliding	Result
1	0.000	10.5	10.5	
2	10.000	11.4	11.4	
3	20.000	12.3	12.3	
4	30.000	13.2	13.2	
5	40.000	14.1	14.1	
6	50.000	15.0	15.0	
7	60.000	15.9	15.9	
8	70.000	16.8	16.8	
9	80.000	17.7	17.7	
10	90.000	18.6	18.6	
11	100.000	19.5	19.5	
12	110.000	20.4	20.4	
13	120.000	21.3	21.3	
14	130.000	22.2	22.2	
15	140.000	23.1	23.1	
16	150.000	24.0	24.0	

	Length	Height		
		Slope	Sliding	Result
1	0.000	21.3	21.3	
2	10.000	21.3	21.3	
3	20.000	21.3	21.3	
4	30.000	21.3	21.3	
5	40.000	21.3	21.3	
6	50.000	21.3	21.3	
7	60.000	21.3	21.3	
8	70.000	21.3	21.3	
9	80.000	21.3	21.3	
10	90.000	21.3	21.3	
11	100.000	21.3	21.3	
12	110.000	21.3	21.3	
13	120.000	21.3	21.3	
14	130.000	21.3	21.3	
15	140.000	21.3	21.3	
16	150.000	21.3	21.3	

	Color	Thick
Slope		1
Sliding		1
Result		1
Water Surface		1

About the "Output of arbitrary section profile" Menu

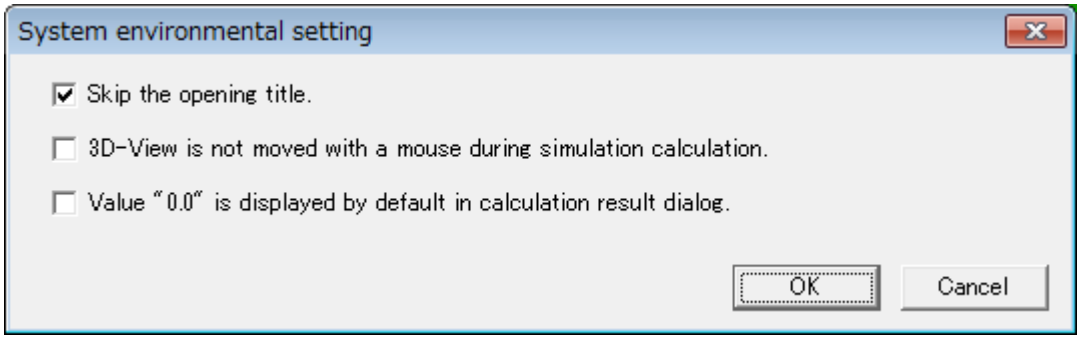
By selecting this item, the following window is shown where we can set the section layout as necessary.



Item	Description
Layout of cross section	Enter the starting and ending points of the cross section that we would like to display. We can enter the data by using a computer mouse.
Cross section profile	Displays the cross section profile based on the specified values of the section coordinates.
List of cross section coordinate	Displays the coordinate values of the cross sections of slope surface and sliding surface corresponding to the above cross section profile.

(3) The [Parameters] menu (System environmental setting)

This menu is used for the basic settings of the LS-RAPID model.



Item	Description
Skip the Opening Title	As default, the introduction window of LS-RAPID will appear when it is started.
3D-view is not moved with a mouse during simulation calculation	If we select this option, nothing will happen when we click or drag the mouse during the simulation. This function can be used in case the topography suddenly starts to move.
Value "0.0" is displayed by default in calculation result dialog	In the dialogue of the calculation result, we can choose whether the value of 0.0 will not be displayed. Generally, it will not be displayed when we started the LS-RAPID. When we checked this button, value "0.0" will be displayed in the calculation result.

(4) The [Help] menu

Menu Item	Description
Help	Displays the help for the software
Explanations of LS-RAPID	Describes the following concepts in the LS-RAPID simulator.
Control guide	Explains how to move the screen using a computer mouse
Check update	We can check whether the "LS-RAPID" is the latest version or not by connecting through the internet.
Version information	Displays the version of the LS-RAPID software

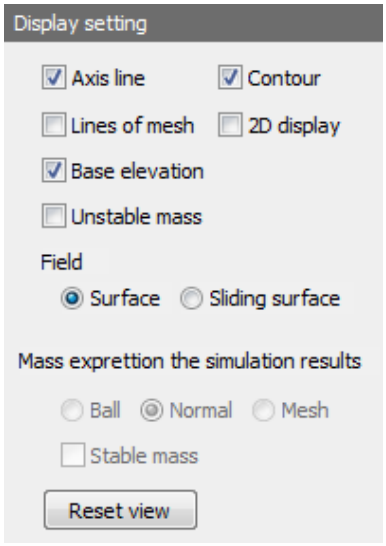
(5) Toolbar Commands

The LS-RAPID system uses the following toolbar commands
(The icons are described from left to right).

Icon	Description
New	Create a new text file
Open	Open a data file
Save	Save the current data file
Simulation area setting	The necessary initial settings to display the topography
Ellipsoid sliding surface setting	Edit the sliding surface and the section from the ellipsoid
Filling and Excavating	Filling and excavating the current topography
Delineating Unstable Mass	Delineating source area and volume enlargement area
Topography View	Setting of topography view
Topography Color	Setting of topography color and contour
View Cross Section	Display a section of the topography cut parallel to the X-axis and the Y-axis.
Geometry Style	Choose the topography to be displayed (the slope surface or the sliding surface)
Thick Area	Choose the topography to be displayed (The distribution of the depth of the unstable mass)
Calc. Result	Choose the topography to be displayed (The screen of the final calculation results)
Calc. Thick	Choose the topography to be displayed (The distribution of the mass after the calculation is done)
Soil Parameter	Set parameters of soils and pore pressure
Start Simulation	Start the simulation
Result Display	Display the calculation result
Calculation monitoring	Open the dialog box showing the mass thickness and the values of different components of the velocity
Given triggering factors	Open the dialog box showing the pore pressure and seismic loading
Pause	Pause the simulation temporarily
Restart	Restart the simulation which has been paused temporarily
Stop	Force the simulation to stop
Help	Display the help for the software
Guidance of Graph View	Explain how to move the screen with a computer mouse

(6) Command for Display Pane

This command is about the display setting of 3D view in the screen.



Item	Description
Axis line (Coordinate axis)	Display the axis coordinate
Contour (Contour line)	Display the lines of contour
Lines of mesh (Grid line)	Display of the grid lines. We can specify the grid line spacing through the "Display setting" screen
2D display	Displayed as the 2D form
Base elevation (Horizontal plane)	Display of the horizontal plane
Unstable mass (Display of unstable soil mass)	When this option is selected, the unstable soil mass are represented by spheres according to the thickness in the case of the “Surface” / “Sliding surface” drawing mode.
Field (Landform)	We can select whether to draw the ground surface by selecting the item “Surface” or slip surface through the “Sliding surface” drawing mode. It is linked with the tab on "Mesh edit" display.
Mass expression the simulation results (Soil mass displayed from calculation result)	The same command as the "Topography view while calculation" on the "Setting of Topography View" display.
Ball (Sphere)	Soil-mass is drawn with spheres.
Normal	Landform is drawn in normal.
Mesh (Sliding surface)	Soil mass is drawn in the distribution of each mesh.
Stable mass (Display of stable soil mass)	The stable soil mass is drawn.