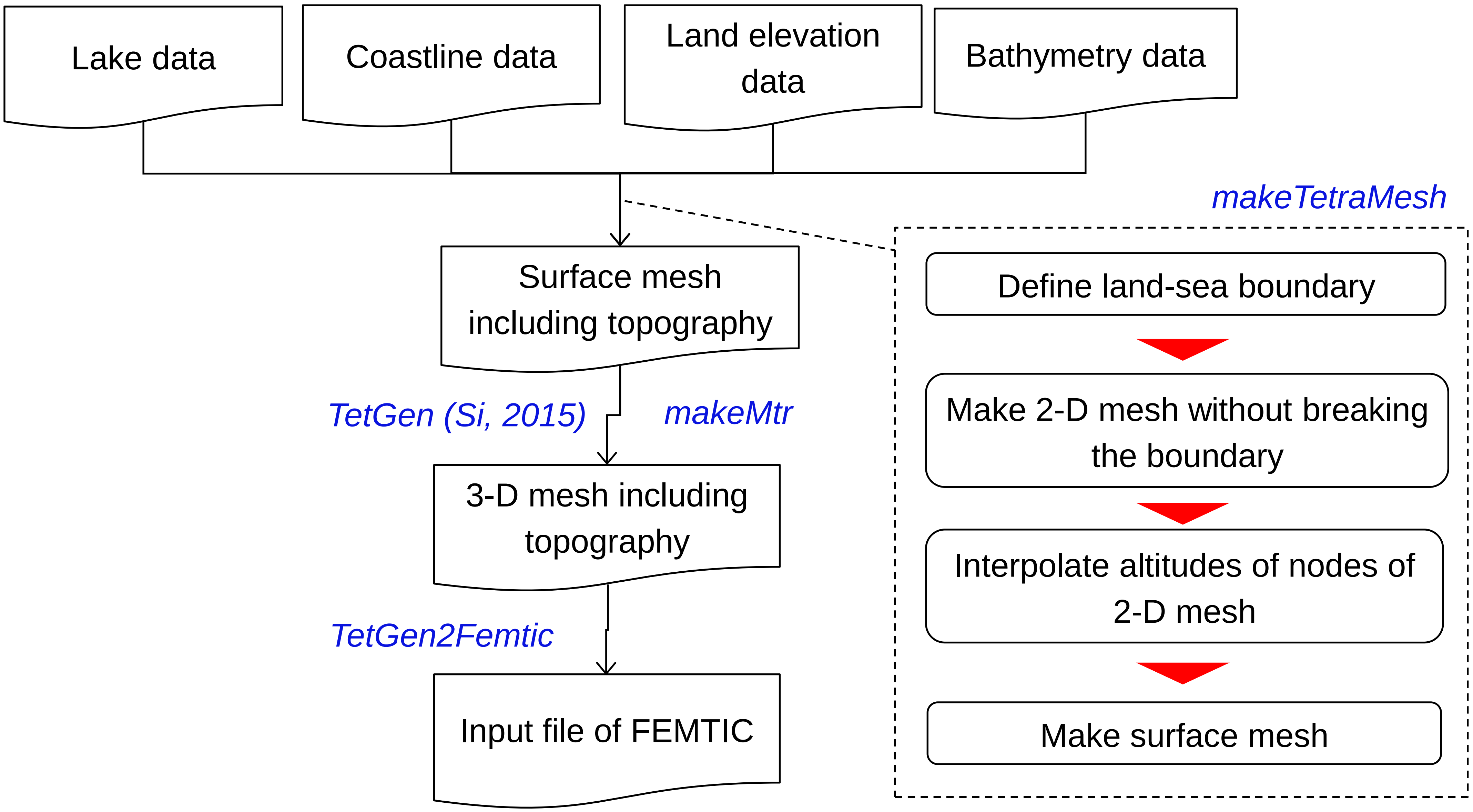
How to make 3-D mesh including topography



Procedures of makeTetraMesh

makeTetraMesh

Execution commands

	makeTetraMesh	Execution commands
Step 1	Define land-sea/land-lake boundaries	makeTetraMesh -stp 1
Step 2	Make 2-D mesh without breaking the boundaries	makeTetraMesh -stp 2
Step 3	Interpolate altitudes of nodes of 2-D mesh	makeTetraMesh -stp 3
Step 4	Make surface mesh	makeTetraMesh -stp 4

You need to execute the above commands in the directory where all input files exist.

Step 1: Define land-sea/land-lake boundaries

Input files of Step 1

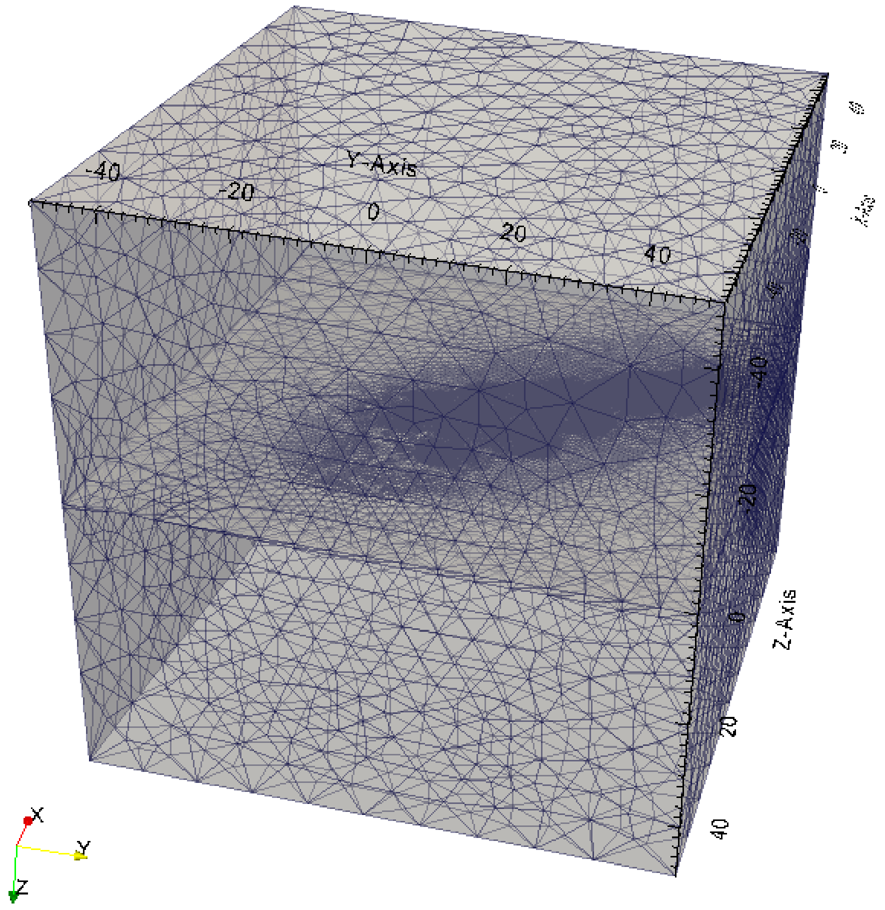
File name	Contents
analysis_domain.dat	Size of computational domain
coast_line.dat	Information about land-sea/land-lake boundaries
control.dat	Parameters controlling mesh generation
observing_site.dat	Mesh size around observation sites

File format of analysis_domain.dat

Minimum of X coordinate value (km)	*Maximum of X coordinate value (km)*
Minimum of Y coordinate value (km)	*Maximum of Y coordinate value (km)*
Minimum of Z coordinate value (km)	*Maximum of Z coordinate value (km)*

Example

```
-50.0 50.0
-50.0 50.0
-50.0 50.0
```



File format of coast_line.dat

Number of boundaries			
X coordinate value (km)	*Y coordinate value (km)*	*End flag*	0

⋮

End flag

[*End flag*] = 0

- ➢ The point is NOT the end point of a boundary

[*End flag*] = 1

- ➢ The point is the end point of a closed boundary

[*End flag*] = -1

- ➢ The point is the end point of an unclosed boundary

Example of coast_line.dat

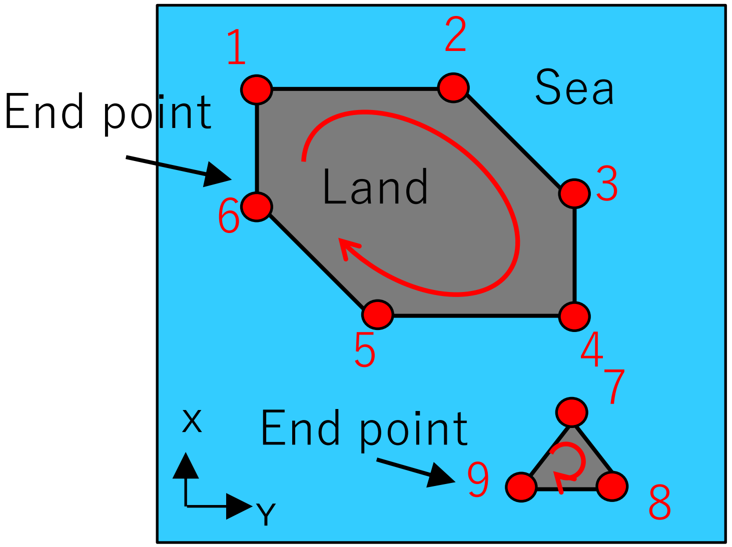
```
10014
353.479913264 174.431 0 0
353.492502288 174.418 0 0
353.495529099 174.413 0 0
353.49608366 174.409 0 0
353.499731761 174.394 0 0
353.510112063 174.392 0 0
353.510340906 174.386 0 0
353.512989179 174.381 0 0
353.518524188 174.378 0 0
353.522898605 174.377 0 0
353.527148244 174.378 0 0
```

If you do not include the sea in the computational domain, please define a boundary which covers whole of the computational domain.

If there is no land area in the computational domain, please set [*Number of boundaries*] to zero.

File format of coast_line.dat

You must define the points of land-sea or land-lake boundaries in the order that land locates on the right-hand side.



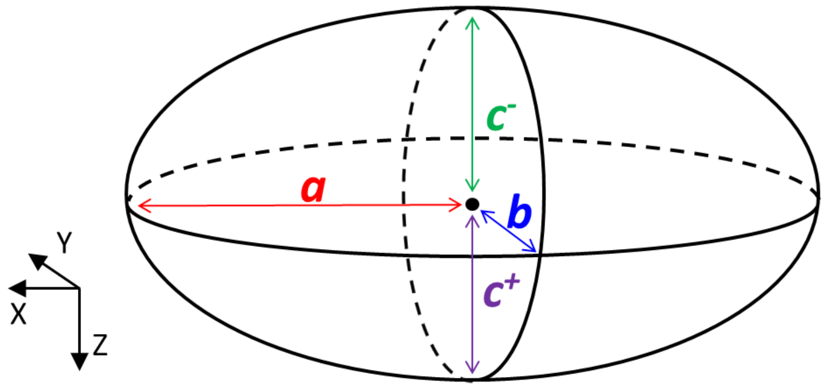
2
X_1 Y_1 0 0
X_2 Y_2 0 0
X_3 Y_3 0 0
X_4 Y_4 0 0
X_5 Y_5 0 0
X_6 Y_6 1 0
X_7 Y_7 0 0
X_8 Y_8 0 0
X_9 Y_9 1 0

File format of ‘control.dat ‘ (1/2)

Keyword	Content	Data type	Option	Default	Example	Step *1)			
						1	2	3	4
CENTER	Coordinate values (x,y,z) of the ellipsoids to specify edge lengths. The length scale is kilo-meter.	Three real values			CENTER 0.0 0.0 0.0	○	○		○
ROTATION	Rotation angle (deg.) of the ellipsoids to specify edge lengths around the x-y plane	Real value		0.0	ROTATION 30.0	○	○		○
ELLIPSOIDS	Information about the ellipsoids to control edge lengths	*Shown in the next slide*			*Shown in the next slide*	○	○		○
NUM_THREADS	Number of threads used in parallel processing	Positive integer		1	NUM_THREADS 3			○	
SURF_MESH	Keyword for making surface mesh. This keyword is always required				SURF_MESH				○

*1) Circle is drawn for the steps where the setting of the keywords is used.

Keyword ‘ELLIPSOIDS’



ELLIPSOIDS

Number of ellipsoids (N_e)

Information about the 1st ellipsoid

⋮

Information about the N_e-th ellipsoid

$a \quad len \quad f_h \quad f_v{}^+ \quad f_v{}^-$

[NOTE] Subsequent ellipsoids must cover the formers

a : *Length along x axis (km)*

len : *Upper limit of edge lengths within the ellipsoid (km)*

f_h : *Oblateness on the X-Y plane*

$f_v{}^+$: *Oblateness on the Z-X plane (Upper side)*

$f_v{}^-$: *Oblateness on the Z-X plane (Lower side)*

$$\frac{b}{a} = 1 - f_h$$

$$\frac{c^+}{a} = 1 - f_v{}^+$$

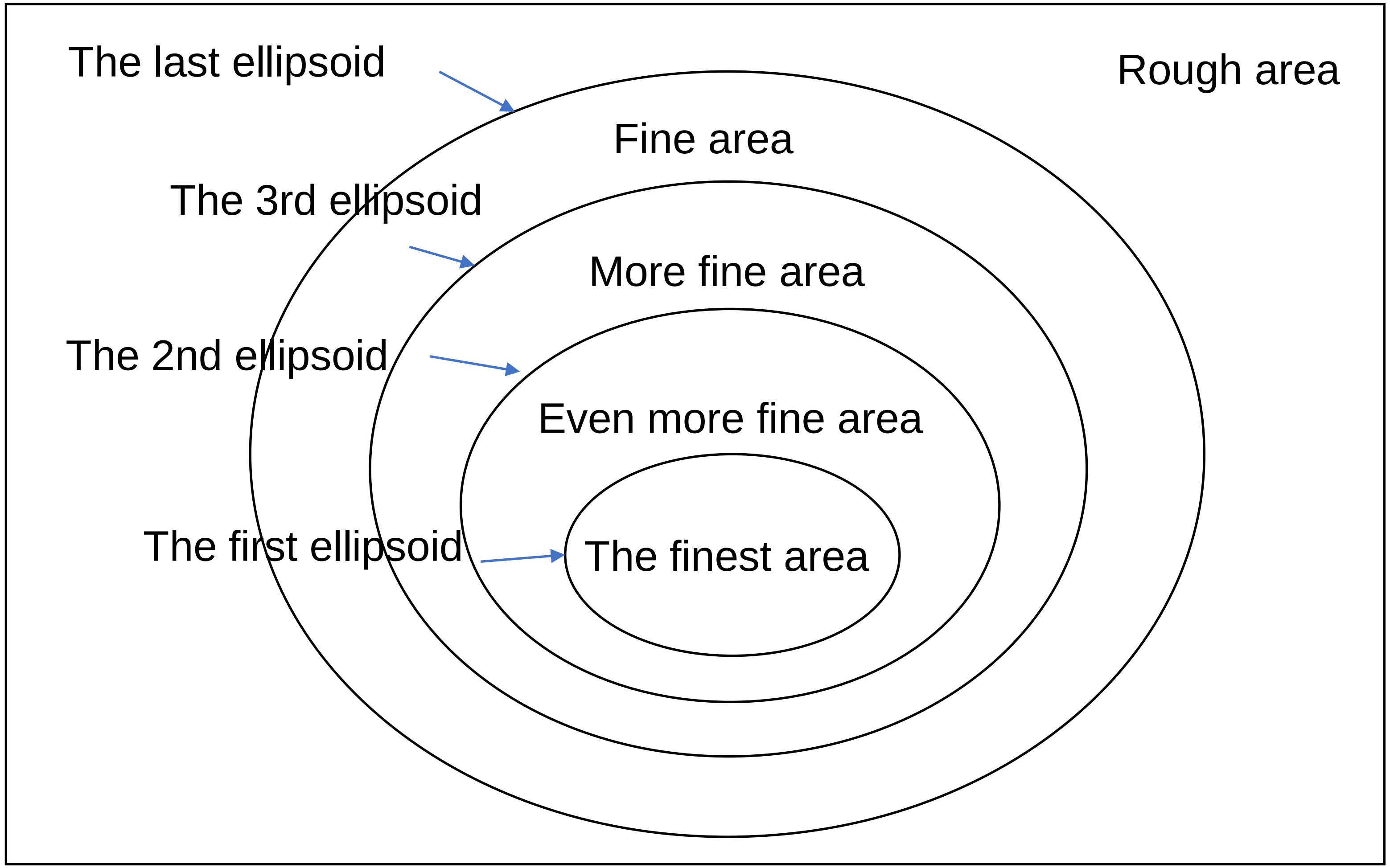
$$\frac{c^-}{a} = 1 - f_v{}^-$$

Example

```
ELLIPSOIDS
6
  40.0   1.0  0.5  0.5  0.7
  60.0   5.0  0.3  0.3  0.5
 100.0  10.0  0.2  0.1  0.3
 200.0  20.0  0.0  0.0  0.0
 300.0  30.0  0.0  0.0  0.0
 500.0  50.0  0.0  0.0  0.0
```

Keyword ‘ELLIPSOIDS’

Ellipsoids of control.dat (and spheres of observing_site.dat) are used to change the fineness of mesh (in other words, size of triangles) hierarchically.



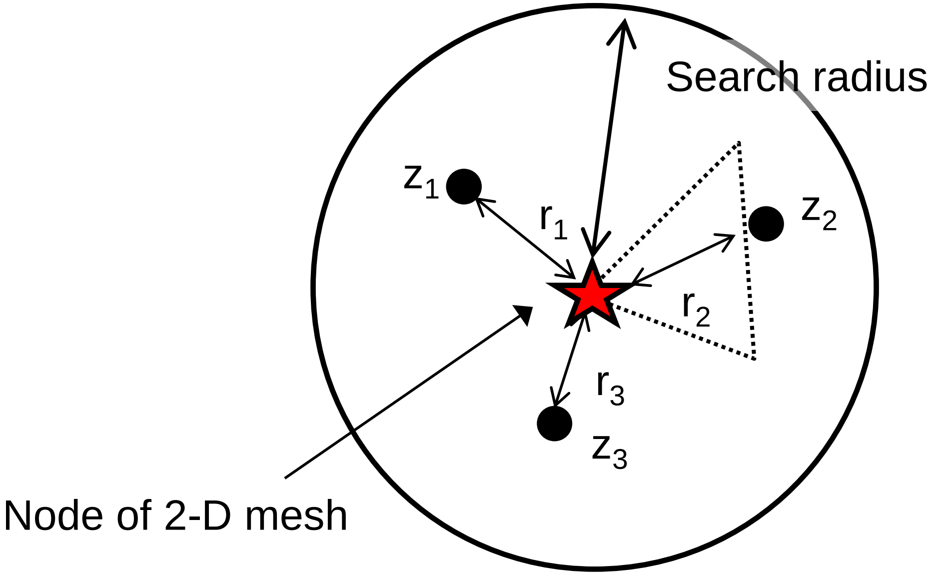
File format of ‘control.dat’ (2/2)

Keyword	Content	Data type	Option	Default	Example	Step *1)			
						1	2	3	4
INTERPOLATE	Search radius (km)	Positive real value		100.0	INTERPOLATE 10.0 4 1.0e-6			○	
	Maximum number of points used for the interpolation of topography data	Positive integer		3					
	Small number to avoid zero divide (km)	Positive real value		1.0e-6					
ALTITUDE	Name of the file including land topography (Altitudes)	Character string			ALTITUDE altitude.txt 0.0 1.0e20			○	
	Minimum altitude (km)	Real value		0.0					
	Maximum altitude (km)	Real value		1.0e10					
SEA_DEPTH	Name of the file including bathymetry data (depths of the sea floor)	Character string			SEA_DEPTH Sea_depth.txt 0.01 1.0e20			○	
	Minimum sea depth (km)	Real value		0.01					
	Maximum sea depth (km)	Real value		1.0e10					
END	Indication of the end of controlling parameters				END	○	○	○	○

*1) Circle is drawn for the steps where the setting of the keywords is used.

Interpolation method

Inverse distance weighting method is used for interpolation of the altitude and sea depths.



$$z = \frac{\sum_{i=1}^{N} w_i z_i}{\sum_{i=1}^{N} w_i}$$

$$w_i = \frac{1}{(r_i + \varepsilon)}$$

N: Maximum number of the points used for interpolation

ε : Small number to avoid zero divide (km)

Format of the files including topography/bathymetry data

Land topography (Altitudes)

X coordinate value (km)	*Y coordinate value (km)*	*Altitude (km)*

⋮

Bathymetry (Depths of the sea floor)

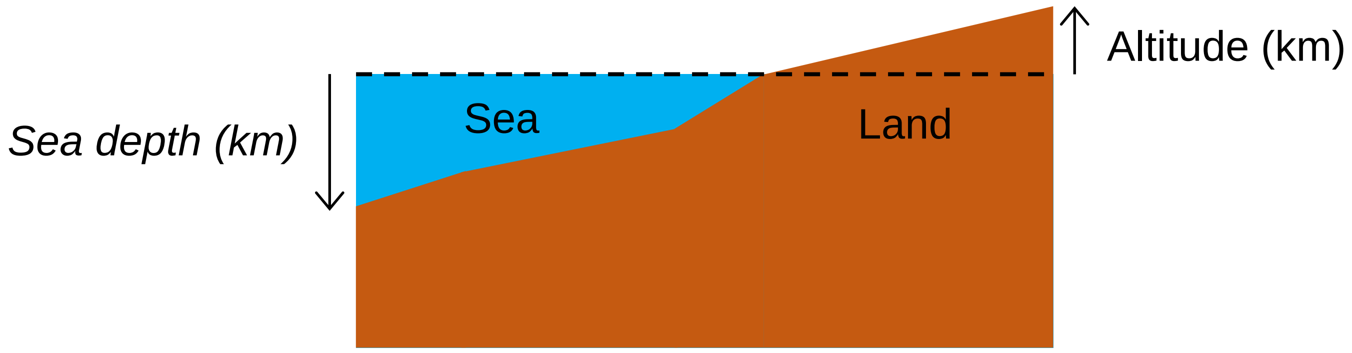
X coordinate value (km)	*Y coordinate value (km)*	*Sea depth (km)*

⋮

Example

-106.824031915	76.2046151595	2.100000e-003
-106.824276161	76.1651038245	6.000000e-003
-106.824459262	76.1354703228	8.500000e-003
-106.824642292	76.1058368206	7.400000e-003
-106.824886221	76.0663254836	9.000000e-003
-106.825069084	76.0366919804	9.300000e-003
-106.825251877	76.0070584767	8.600000e-003
-106.825495489	75.9675471378	8.600000e-003
-106.825678115	75.9379136331	9.400000e-003
-106.82586067	75.908280128	1.040000e-002
-106.826103966	75.8687687872	1.070000e-002
-106.826286354	75.839135281	9.300000e-003
-106.826468671	75.8095017744	1.110000e-002
-106.82671165	75.7699904316	1.390000e-002

Altitudes are positive in the upward direction while sea depths are positive in the downward direction.



File format of 'observing_site.dat'

Number of observation sites (N_{site})

X coordinate value (km) | *Y coordinate value (km)*

Information about the edge length around the 1st site

Number of spheres defining edge length (N_{sph})

Radius of sphere of the 1st sphere (km) | *Upper limit of edge lengths within the 1st sphere (km)*

⋮

Radius of sphere of the N_{sph}-th sphere (km) | *Upper limit of edge lengths within the N_{sph}-th sphere (km)*

⋮

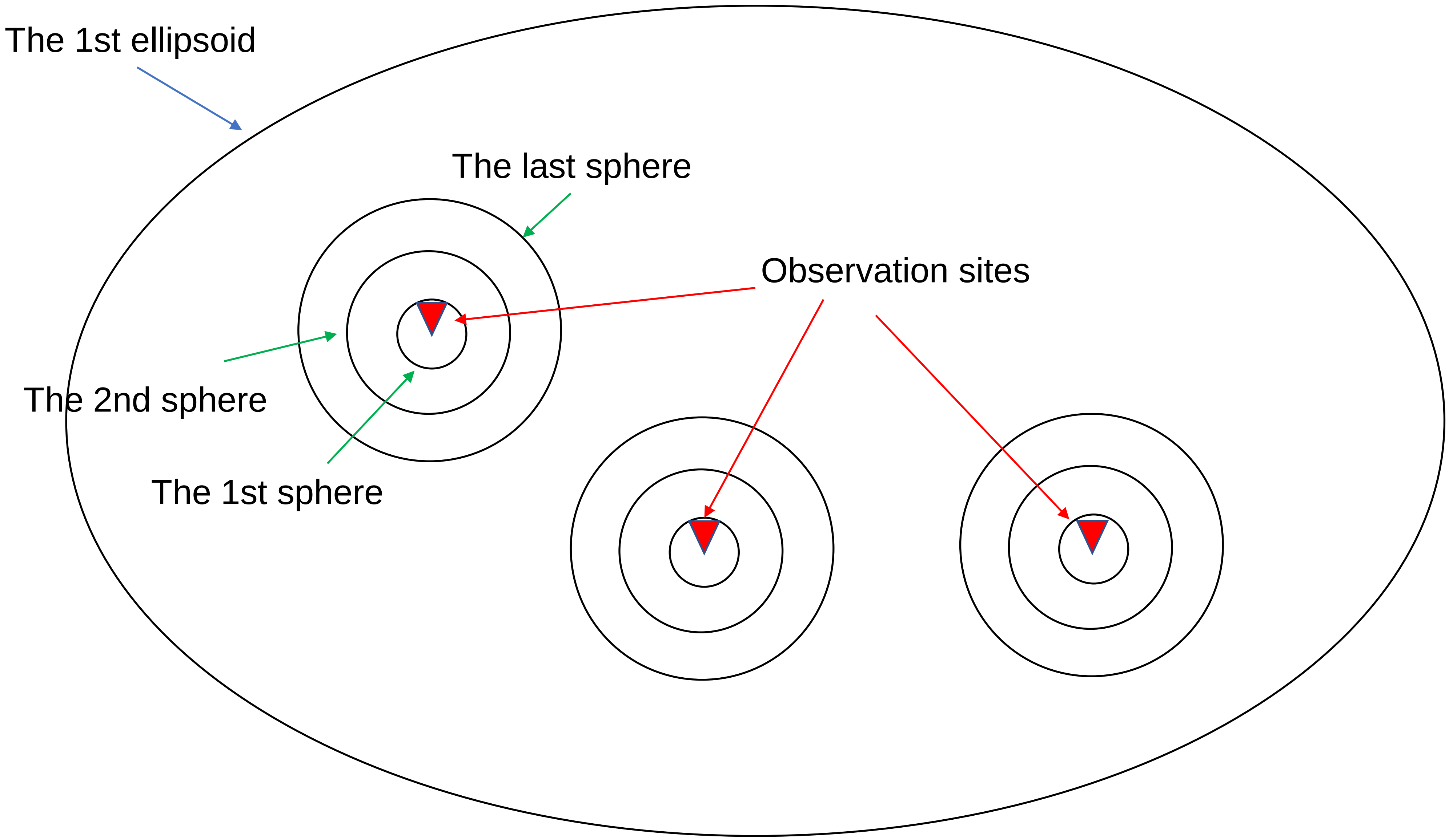
Information about the edge length around the N_{site}-th site

*Subsequent spheres must cover the formers

Example

```
28
  0.000000000     0.000000000
5
0.1  0.02
0.3  0.05
1.0  0.10
3.0  0.30
5.0  0.50
```

The spheres at the observing sites are used to define fineness around each observation site and to make the area around observation sites even finer.

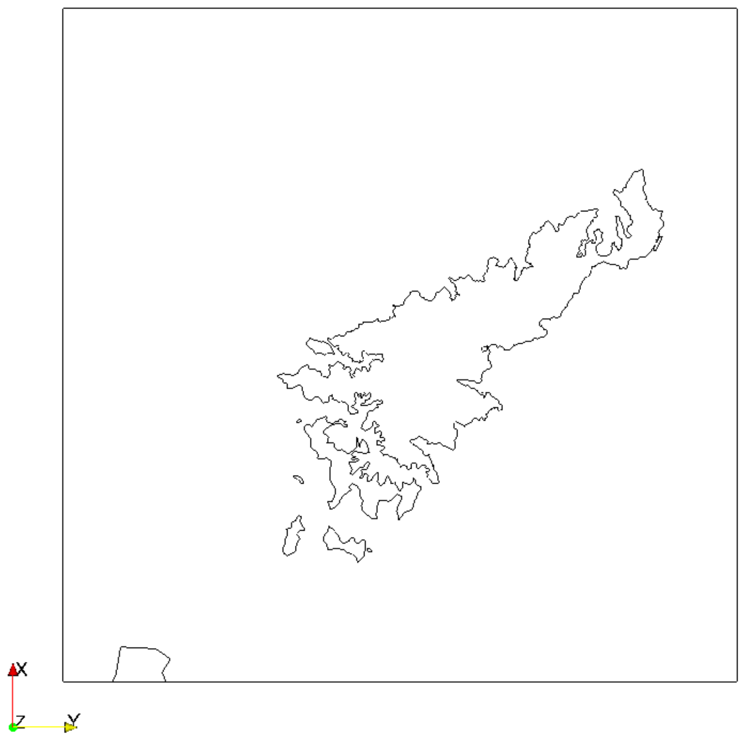


Output files of Step 1

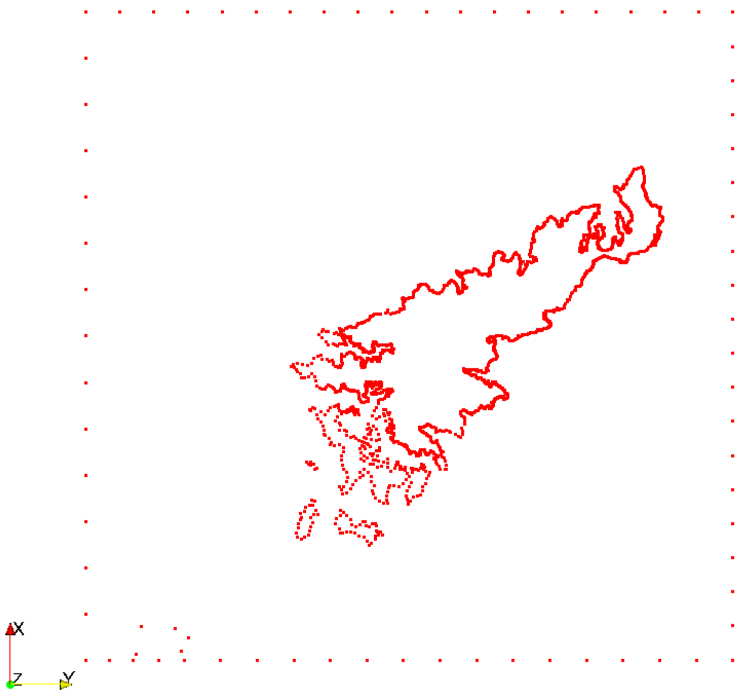
File name	Contents
makeTetraMesh.log	Log output
node_bound_curve.stp1	Coordinate values of the points on inner/outer boundaries (Input file of Step 2)
boundary_curves.stp1	Connections of the points of inner/outer boundaries (Input file of Step 2)
boundary_curves_relation.stp1	Relationship between inner/outer boundaries (Input file of Step 2)
boundary_curve_node_stp1.vtk	Coordinate values of the points of inner/outer boundaries (Input file of ParaView)
boundary_curve_stp1.vtk	Connections of the points of inner/outer boundaries (Input file of ParaView)

Visualization by ParaView (Step 1)

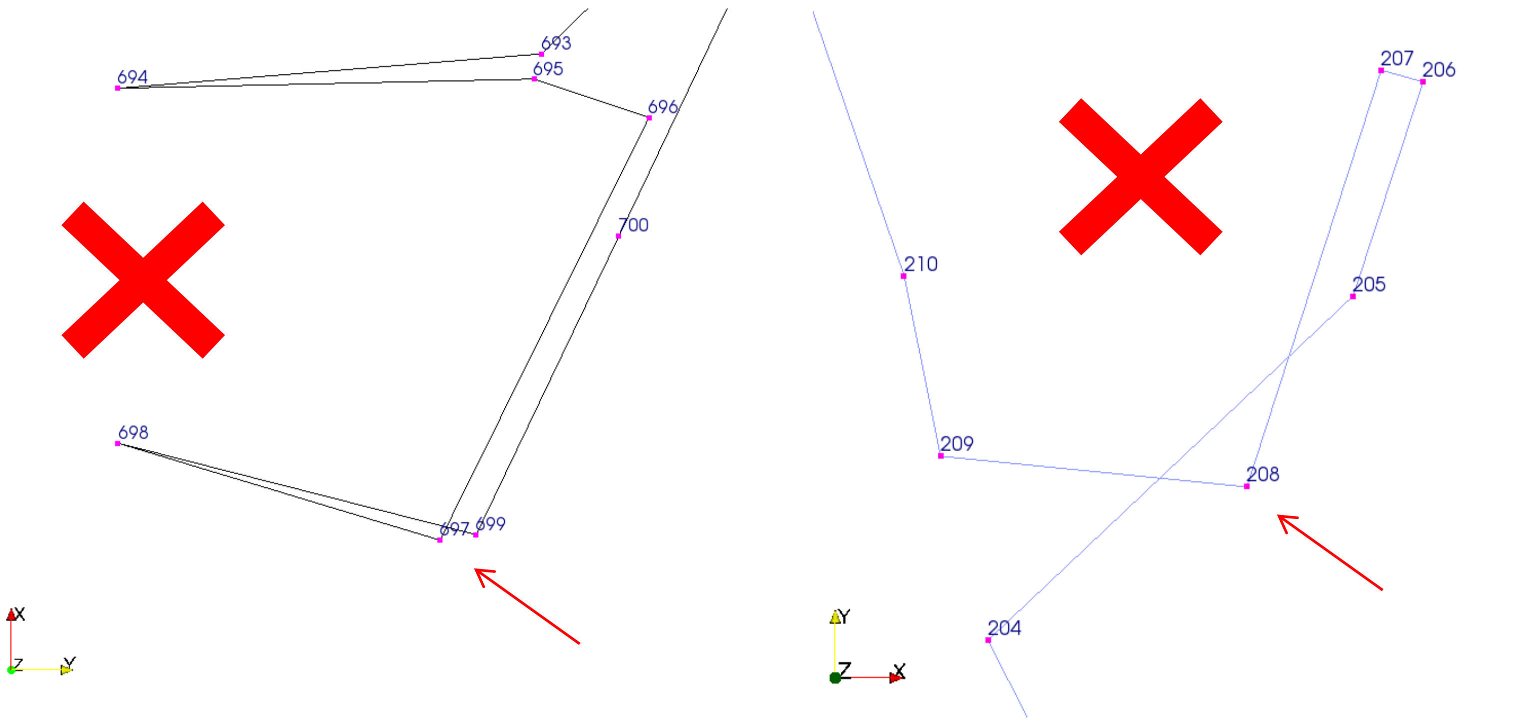
boundary_curve_stp1.vtk



boundary_curve_node_stp1.vtk



Bad boundary curve



- Boundary curves must not be crossed.
- It causes error at Step 2.
- If there are such boundaries, please execute Step 1 by different parameters or edit ‘node_bound_curve.stp1’ directly.

Step 2: Make 2-D mesh without breaking the boundary

Input files of Step 2

File name	Contents
analysis_domain.dat	Size of computational domain
control.dat	Parameters controlling mesh generation
observing_site.dat	Mesh size around observation sites
node_bound_curve.stp1	Coordinate values of the points of inner/outer boundaries (Output file of step 1)
boundary_curves.stp1	Connections of the points of inner/outer boundaries (Output file of step 1)
boundary_curves_relation.stp1	Relationship between inner/outer boundaries (Output file of step 1)

Output files of Step 2

File name	Contents
makeTetraMesh.log	Log output
node_bound_curve.stp2	Coordinate values of the points of inner/outer boundaries (Input file of Step 3)
boundary_curves.stp2	Connections of the points of inner/outer boundaries (Input file of Step 3)
boundary_curves_relation.stp2	Relationship between inner/outer boundaries (Input file of Step 3)
node_mesh.stp2	Coordinates of the points of 2D mesh (Input file of Step 3)
triangle_list.stp2	Connections of the points of 2D mesh (Input file of Step 3)
node_curves2mesh.stp2	Relation of the points of boundaries and the ones of 2D mesh (Input file of Step 3)
surface_triangle.laplacian_last.vtk	Information of 2D mesh (Input file of ParaView)

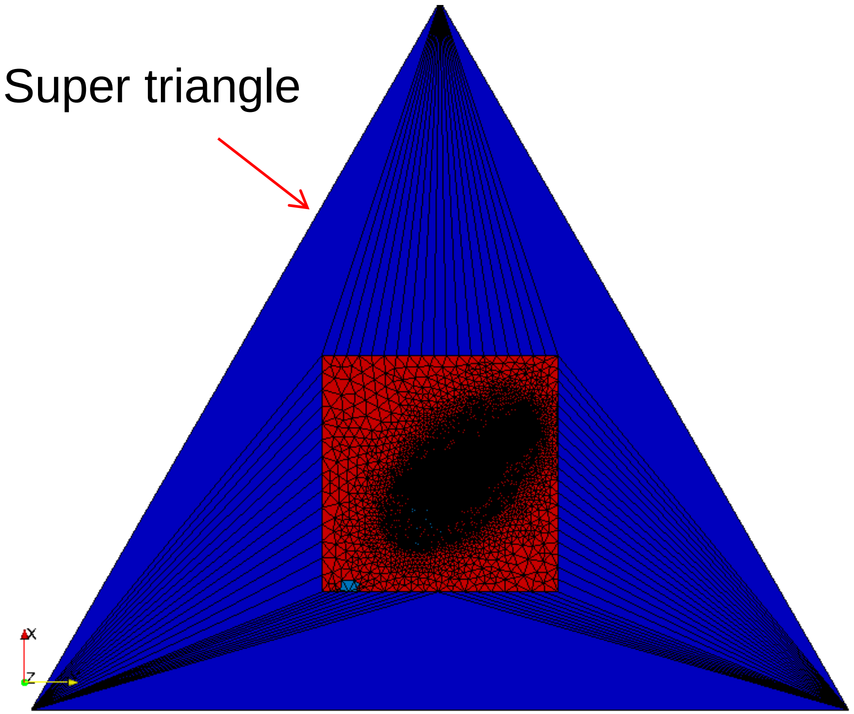
In addition, some intermediate file is outputted. However, you can omit these files.

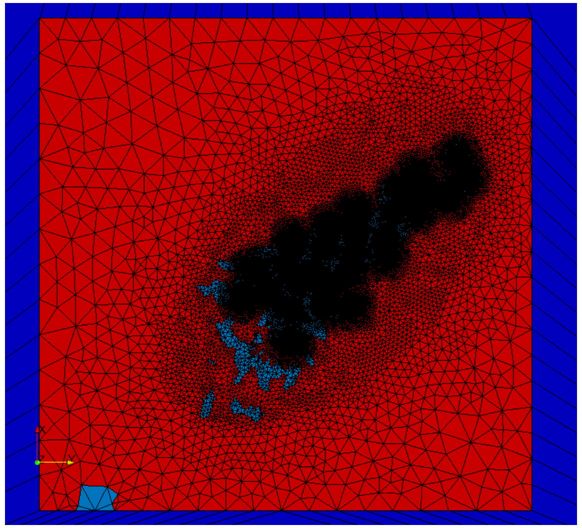
Visualization by ParaView (Step 2)

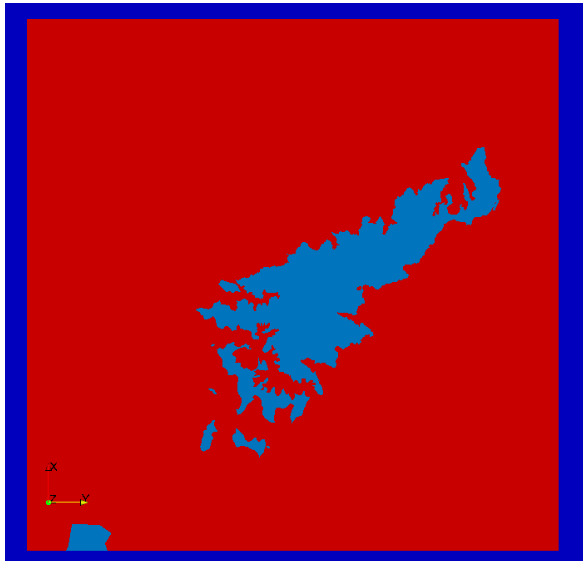
surface_triangle.laplacian_last.vtk

Contour of domain types

Super triangle







Domain type

Sea : 0

Land : 1

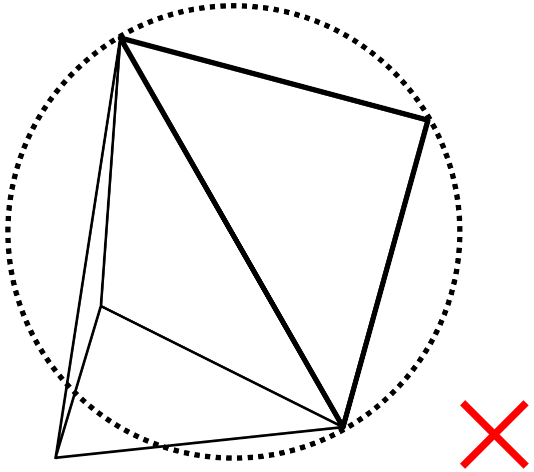
Lake : 2

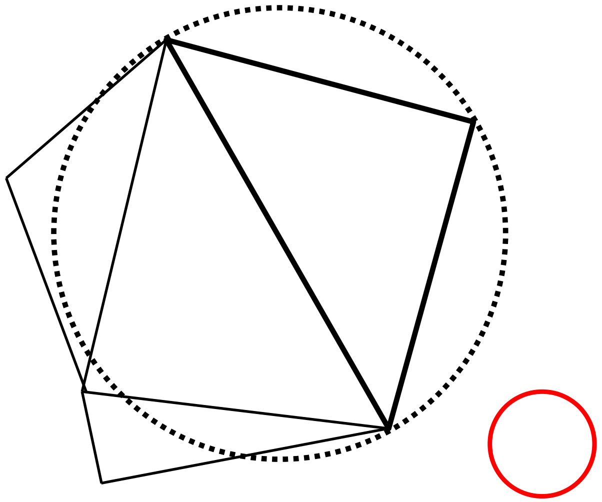
Delaunay triangulation method

The Delaunay triangulation method with boundary constraints is used to make 2-D mesh

Delaunay triangulation method

- ✓ In the context of a finite point set *S*, a triangle is *Delaunay* if its vertices are in *S* and its open circumdisk is empty.
- ✓ *A Delaunay triangulation* of *S* is a triangulation in which every triangle is *Delaunay.*





Step 3: Interpolate altitudes of the nodes of 2-D mesh

Input files of Step 3

File name	Contents
analysis_domain.dat	Size of computational domain
control.dat	Parameters controlling mesh generation
observing_site.dat	Mesh size around observation sites
node_bound_curve.stp2	Coordinate values of the points of inner/outer boundaries (Output file of Step2)
boundary_curves.stp2	Connections of the points of inner/outer boundaries (Output file of Step2)
boundary_curves_relation.stp2	Relationship between inner/outer boundaries (Output file of Step2)
node_mesh.stp2	Coordinate values of the points of 2D mesh (Output file of Step2)
triangle_list.stp2	Connections of the points of 2D mesh (Output file of Step2)
node_curves2mesh.stp2	Relation of the points of boundaries and the ones of 2D mesh (Output file of Step2)
Defined in 'control.dat'	Land topography (altitudes)
Defined in 'control.dat'	Bathymetry (depth of the sea floor)
lake.dat	Information about lakes. This file is required only if you incorporate lakes in the model.

File format of ‘lake.dat’

Number of lakes (N_l)

X coordinate value (km)	*Y coordinate value (km)*	*Z coordinate value (km)*

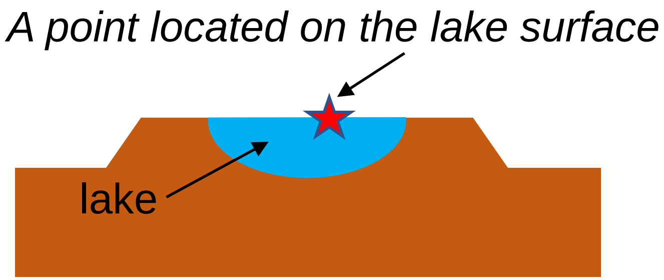
Coordinate values of a point located on the surface of the 1st lake

⋮

X coordinate value (km)	*Y coordinate value (km)*	*Z coordinate value (km)*

Coordinate values of a point located on the surface of the N_l-th lake

The depths of the lake bottoms are interpolated from the land topography data.



Output files of Step 3

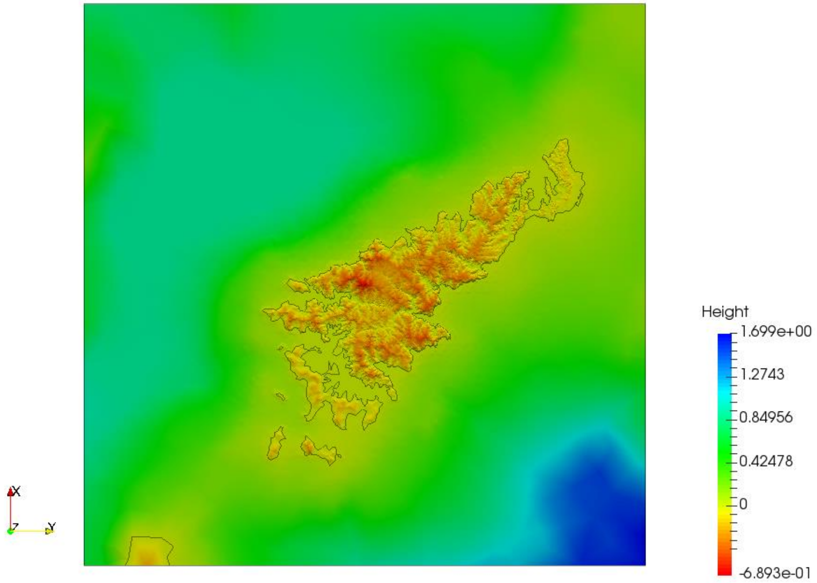
Output files of Step 3

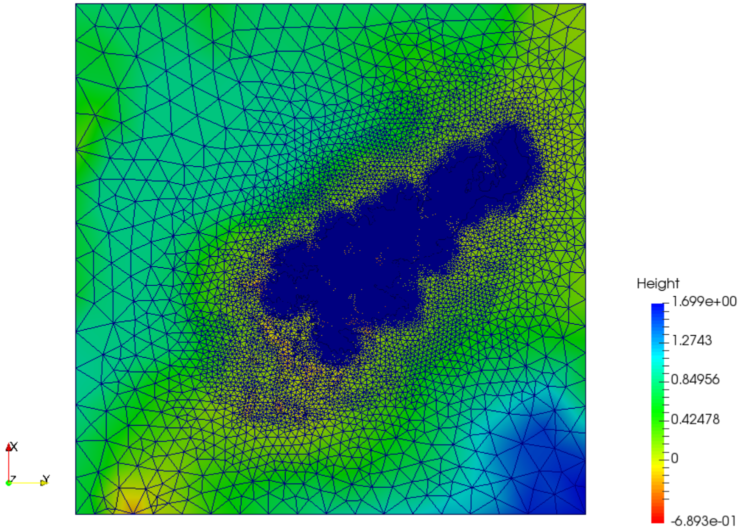
File name	Contents
makeTetraMesh.log	Log output
node_bound_curve.stp3	Coordinate values of the points of inner/outer boundaries (Input file of Step4)
boundary_curves.stp3	Connections of the points of inner/outer boundaries (Input file of Step4)
boundary_curves_relation.stp3	Relationship between inner/outer boundaries (Input file of Step4)
node_mesh.stp3	Coordinate values of the points of 2D mesh (Input file of Step4)
triangle_list.stp3	Connections of the points of 2D mesh (Input file of Step4)
node_curves2mesh.stp3	Relation of the points of boundaries and the ones of 2D mesh (Input file of Step4)
triangles_with_height.vtk	Earth surface’s mesh with topography (Input file of ParaView)

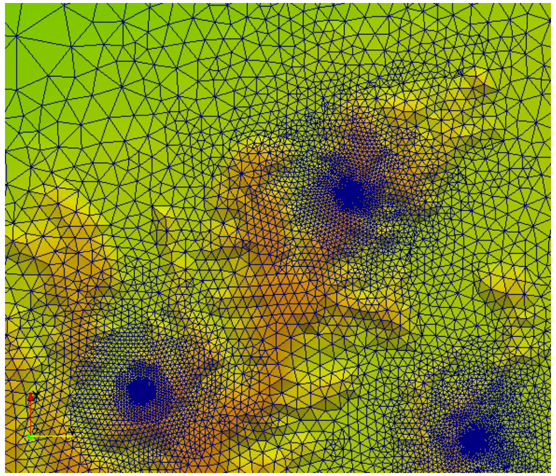
In addition, some intermediate file is outputted.

Visualization by ParaView (Step 3)

triangles_with_height.vtk







Step 4: Make surface mesh

Input files of Step 4

File name	Contents
analysis_domain.dat	Size of computational domain
control.dat	Parameters controlling mesh generation
observing_site.dat	Mesh size around observation sites
node_bound_curve.stp3	Coordinates of the points of inner/outer boundaries (Output file of Step3)
boundary_curves.stp3	Connections of the points of inner/outer boundaries (Output file of Step3)
boundary_curves_relation.stp3	Relationship between inner/outer boundaries (Output file of Step3)
node_mesh.stp3	Coordinates of the points of 2D mesh (Output file of Step3)
triangle_list.stp3	Connections of the points of 2D mesh (Output file of Step3)
node_curves2mesh.stp3	Relation of the points of boundaries and the ones of 2D mesh (Output file of Step3)
lake.dat	Information about lakes. This file is required only if you incorporate lakes in the model.

Output files of Step 4

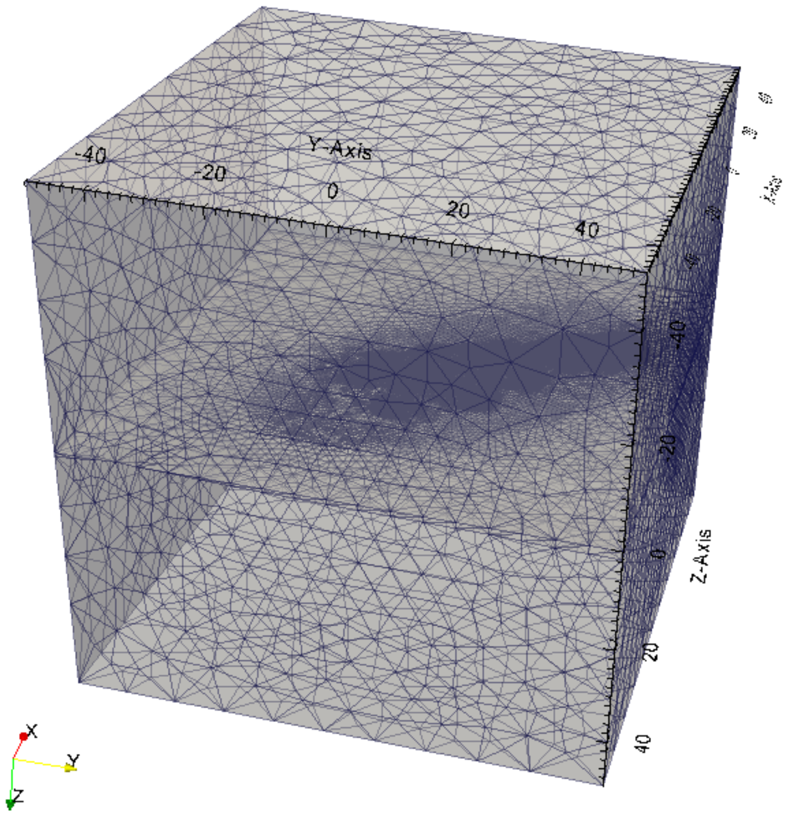
Output files of Step 4

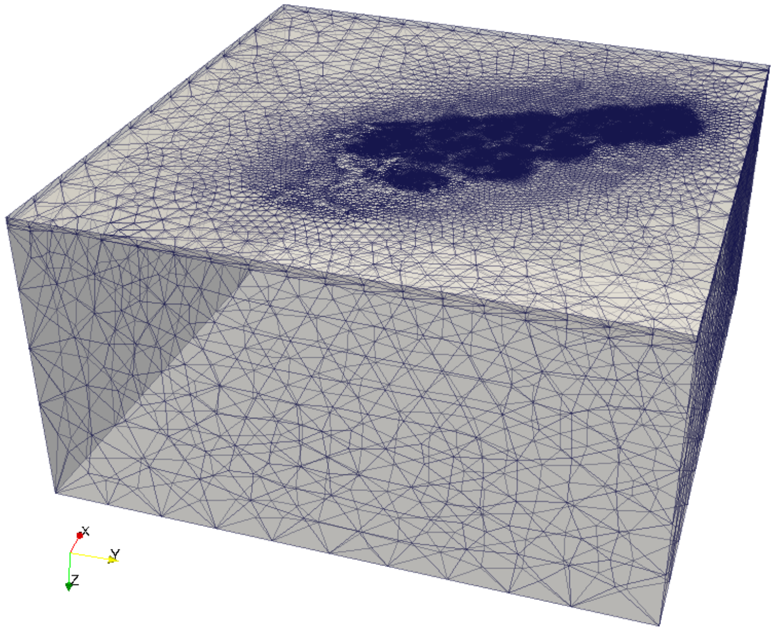
File name	Contents
makeTetraMesh.log	Log output
output.poly	Surface mesh covering the computational domain (Input file of TetGen)
plc.vtk	Surface mesh covering the computational domain (Input file of ParaView)

In addition, some intermediate file is outputted.

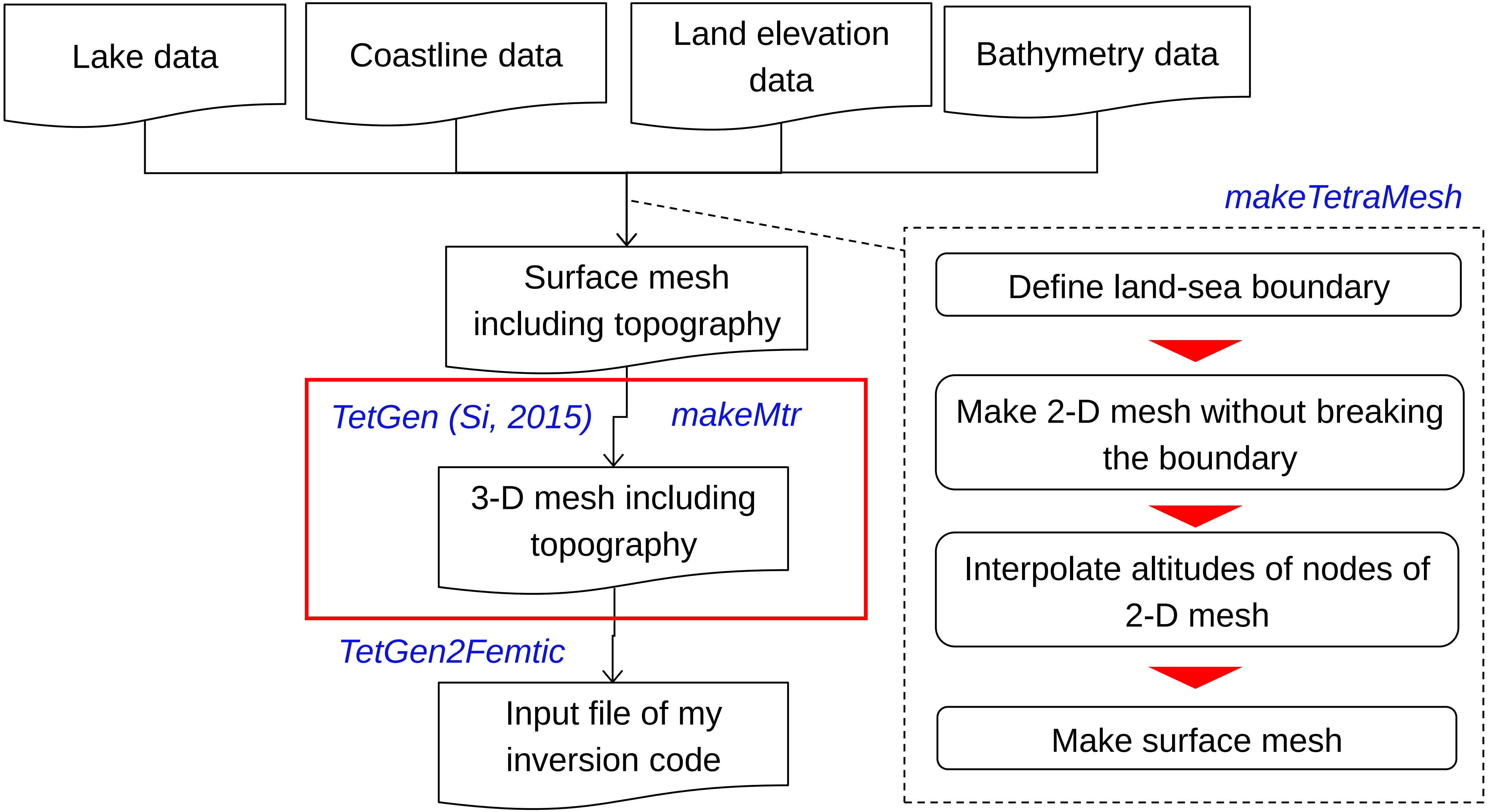
Visualization by ParaView (Step 4)

plc.vtk





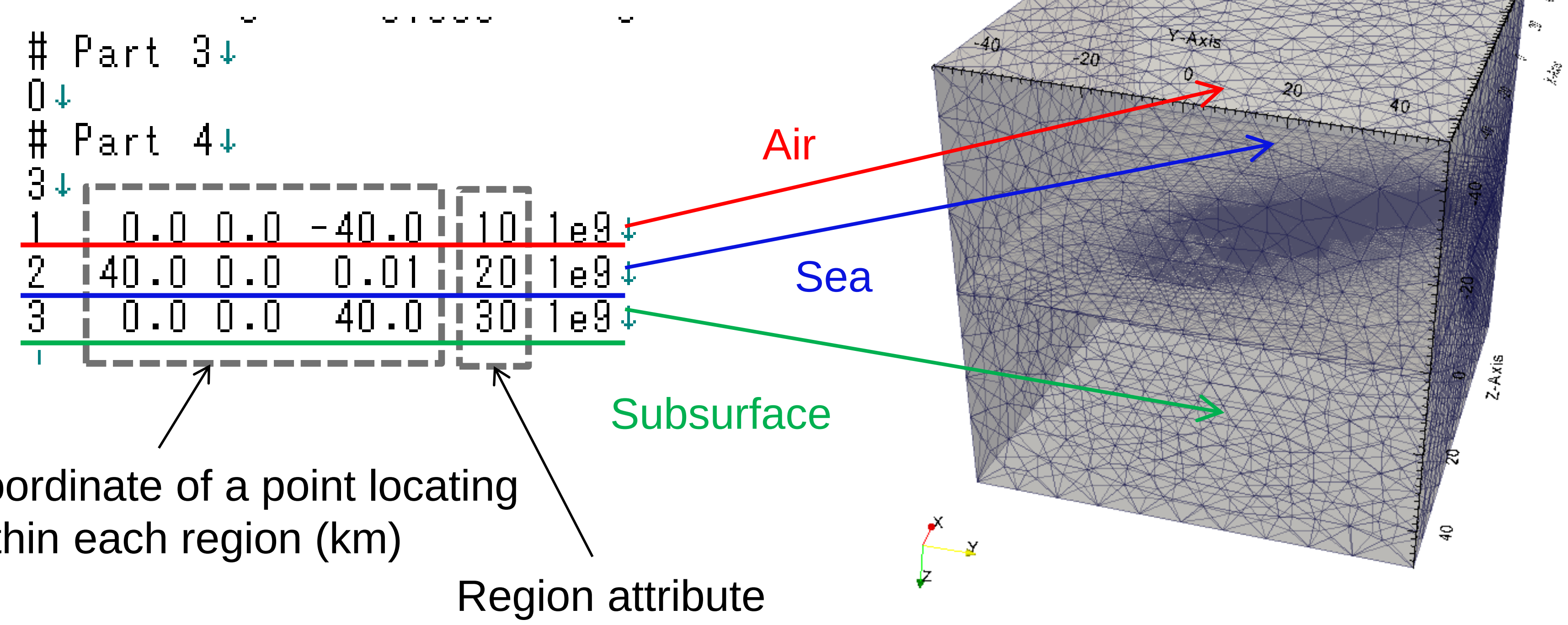
How to make 3-D mesh including topography



Region attribute in ‘output.poly‘

Please give unique region attribute to the air, the sea, each lake and the subsurface, respectively.

Example



How to make 3D mesh from poly file

You need to execute the following command in the directory where ‘output.poly’ exist.

tetgen –nVpYAakq3.0/0 output.poly

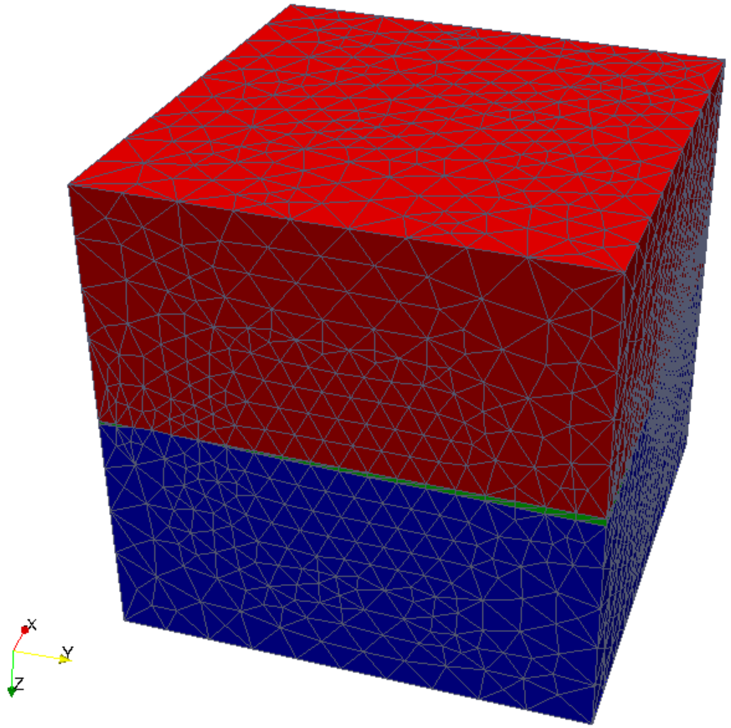
Output files of TetGen

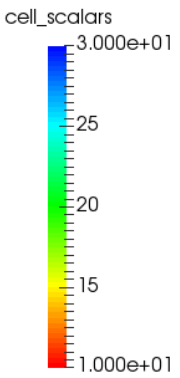
File name	Contents
output.1.node	Information of nodes (Input file of iterative refinement)
output.1.ele	Information of elements (Input file of iterative refinement)
output.1.face	Information of element faces
output.1.edge	Information of element edges
output.1.neigh	Information of adjacency relationship between elements (Input file of iterative refinement)
output.1.vtk	3D mesh (Input file of ParaView)

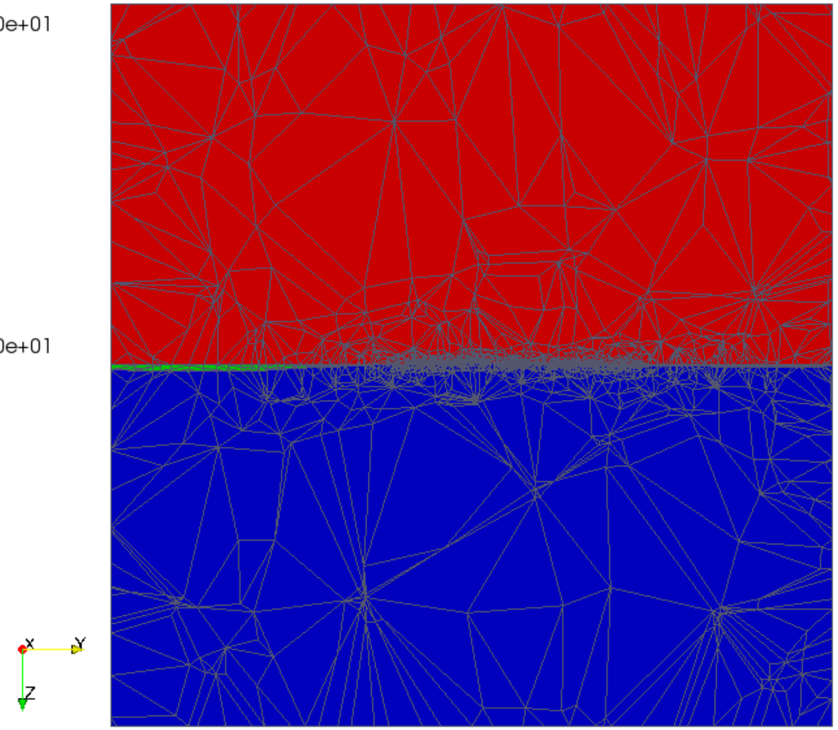
More information about the command and the format of the output files are available in the manual of TetGen.

Visualization by ParaView (3D mesh)

output.1.vtk



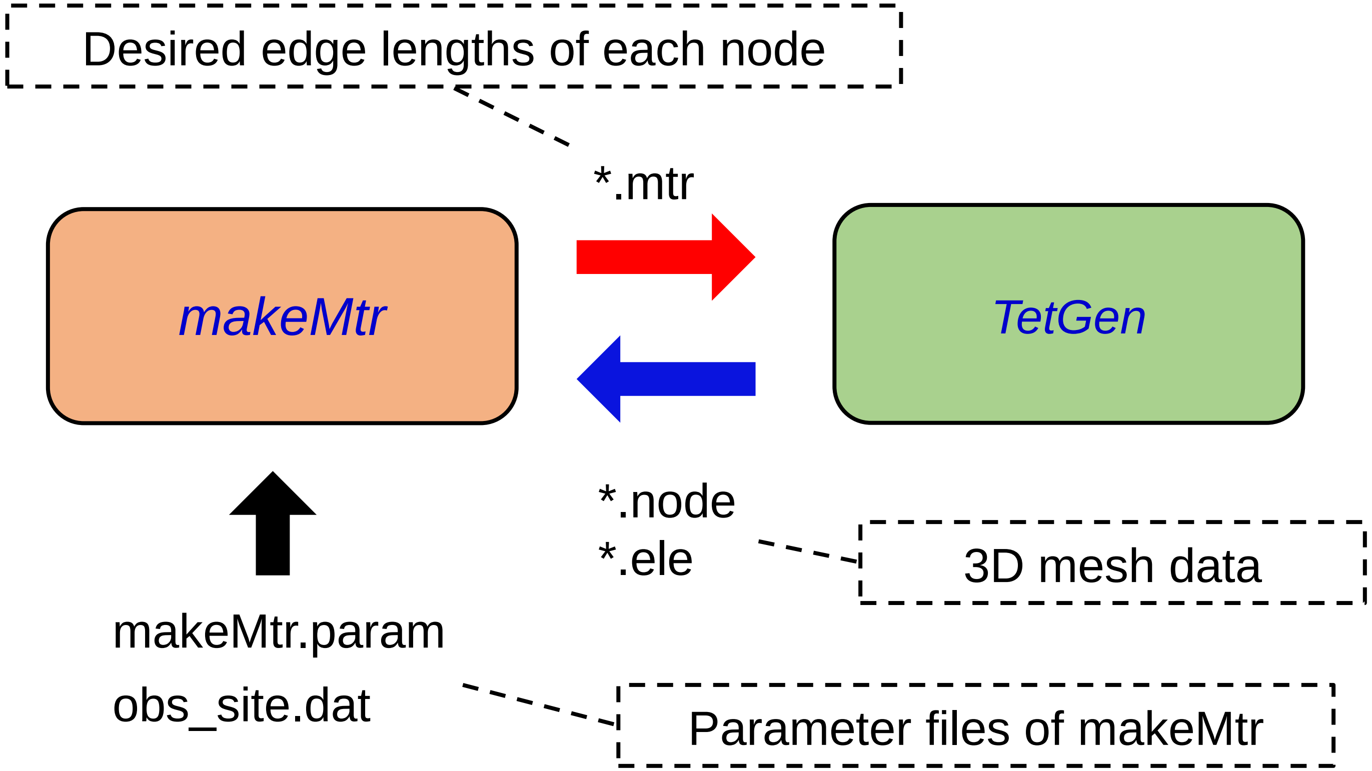




Contour of region attributes

Iterative refinement of mesh

The 3-D mesh should be refined iteratively to make element sizes sufficiently small in all parts of the model.



Input/Output files of makeMtr

Input files

File name	Contents
XXX.ele *1)	Information about the elements of 3-D mesh (Output file of TetGen).
XXX.node *1)	Information about the nodes of 3-D mesh (Output file of TetGen). The unit of coordinate values are kilo-meter.
makeMtr.param	Desired edge lengths for the entire region of the model
obs_site.dat	Desired edge lengths around observation sites. The unit of coordinate values are kilo-meter.

Output files

File name	Contents
XXX.mtr *1)	Desired edge lengths of each node of 3-D mesh. (Input file of TetGen)

*1) In the files 'XXX' indicates the main part of the name of TetGen output files, which is specified as the execution argument of makeMtr.

File format of ‘makeMtr.param’

X coordinate value (km)	*Y coordinate value (km)*	*Z coordinate value (km)*

Coordinate values of the center of the ellipsoids defining desired edge lengths

Rotation angle (deg.) of the ellipsoids around the x-y plane

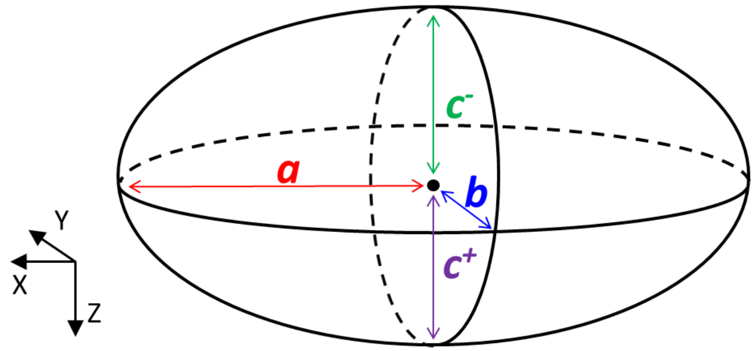
Number of the ellipsoids (N_e)

Information of the 1st ellipsoid → $a \quad len \quad f_h \quad f_v{}^{+} \quad f_v{}^{-}$

⋮

Information of the N_e-th ellipsoid

*Subsequent ellipsoids must cover the formers



$$\frac{b}{a} = 1 - f_h$$

$$\frac{c^{+}}{a} = 1 - f_v{}^{+}$$

$$\frac{c^{-}}{a} = 1 - f_v{}^{-}$$

a : *Length along x axis (km)*

len : *Maximum edge length within the ellipsoid (km)*

f_h : *Oblateness on the X-Y plane*

$f_v{}^{+}$: *Oblateness on the Z-X plane (Upper side)*

$f_v{}^{-}$: *Oblateness on the Z-X plane (Lower side)*

File format of ‘obs_site.dat’

Number of observation sites (N_{site})

X coordinate value (km) | *Y coordinate value (km)* | *Z coordinate value (km)*

Coordinate values of the 1st station

Information about the edge lengths around the 1st site

Number of ellipsoids defining edge length (N_e)

Length along the x-axis (km) | *Maximum edge length within the ellipsoids (km)* | *Oblateness on the Z-X plane*

*Information of the 1st ellipsoid *1)*

⋮

Information of the N_e-th ellipsoid

*1) Unlike ‘makeMtr.param’, oblateness on the X-Y plane cannot be considered.

*2) Subsequent spheres must cover the formers

⋮

Information about the edge lengths around the N_{site}-th site

How to perform iterative refinements

Example of execution commands

```
for  I   in   1 2 3 4 5
do
makeMtr   output.$i
tetgen   –nmpYVrAakq3.0/0   output.$i
done
```

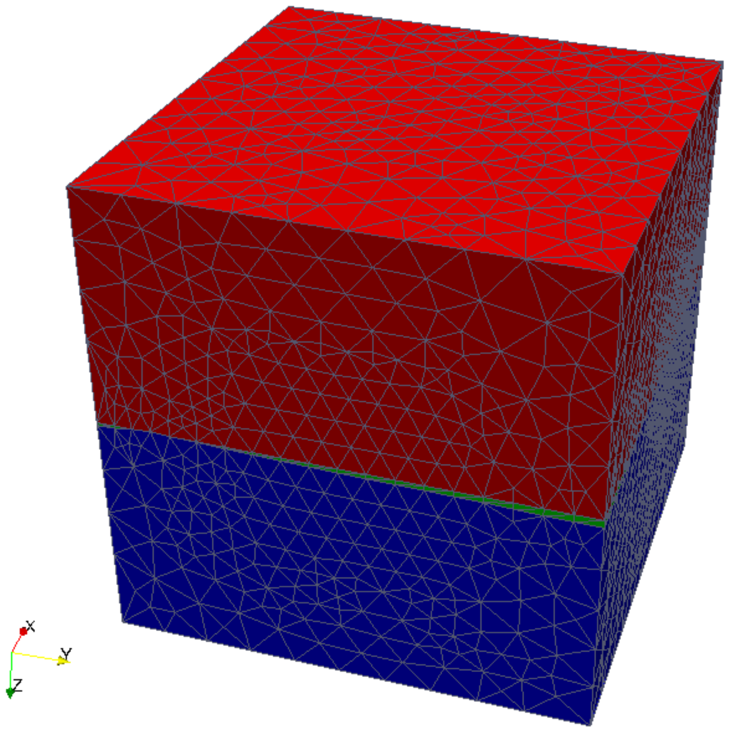
Output files of the iterative refinement

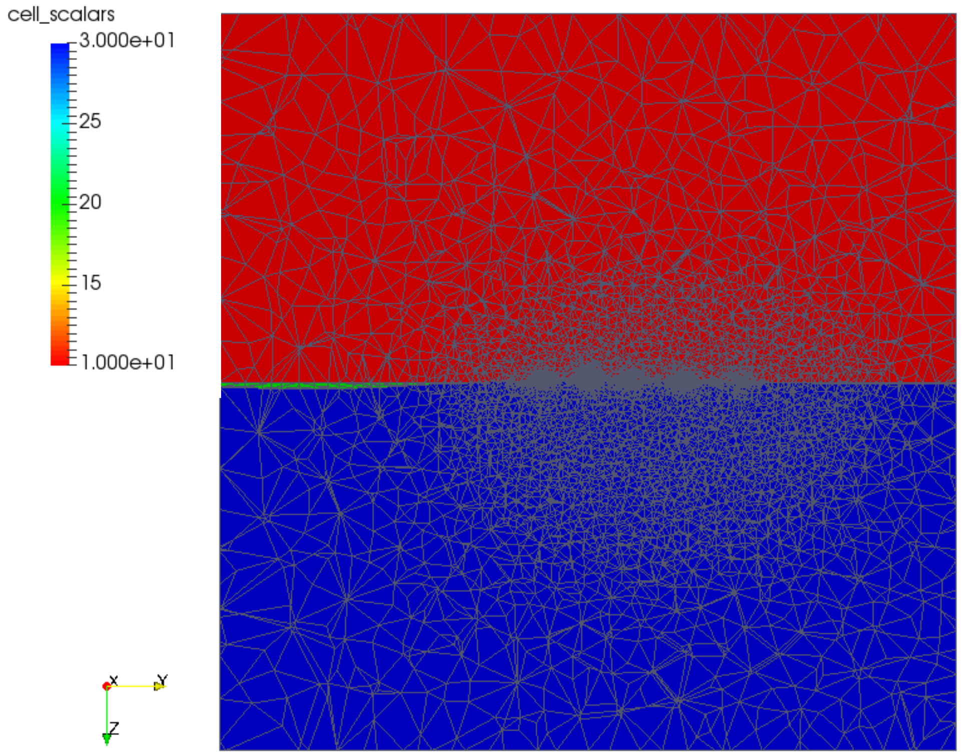
Output files of the X-th iteration

File name	***Contents***
output.X.node	Information of nodes of the X-th iteration
output.X.ele	Information of elements of the X-th iteration
output.X.face	Information of element faces of the X-th iteration
output.X.edge	Information of element edges of the X-th iteration
output.X.neigh	Information of adjacency relationship between elements of the X-th iteration
output.X.vtk	3D mesh of the X-th iteration (Input file of ParaView)

Visualization by ParaView

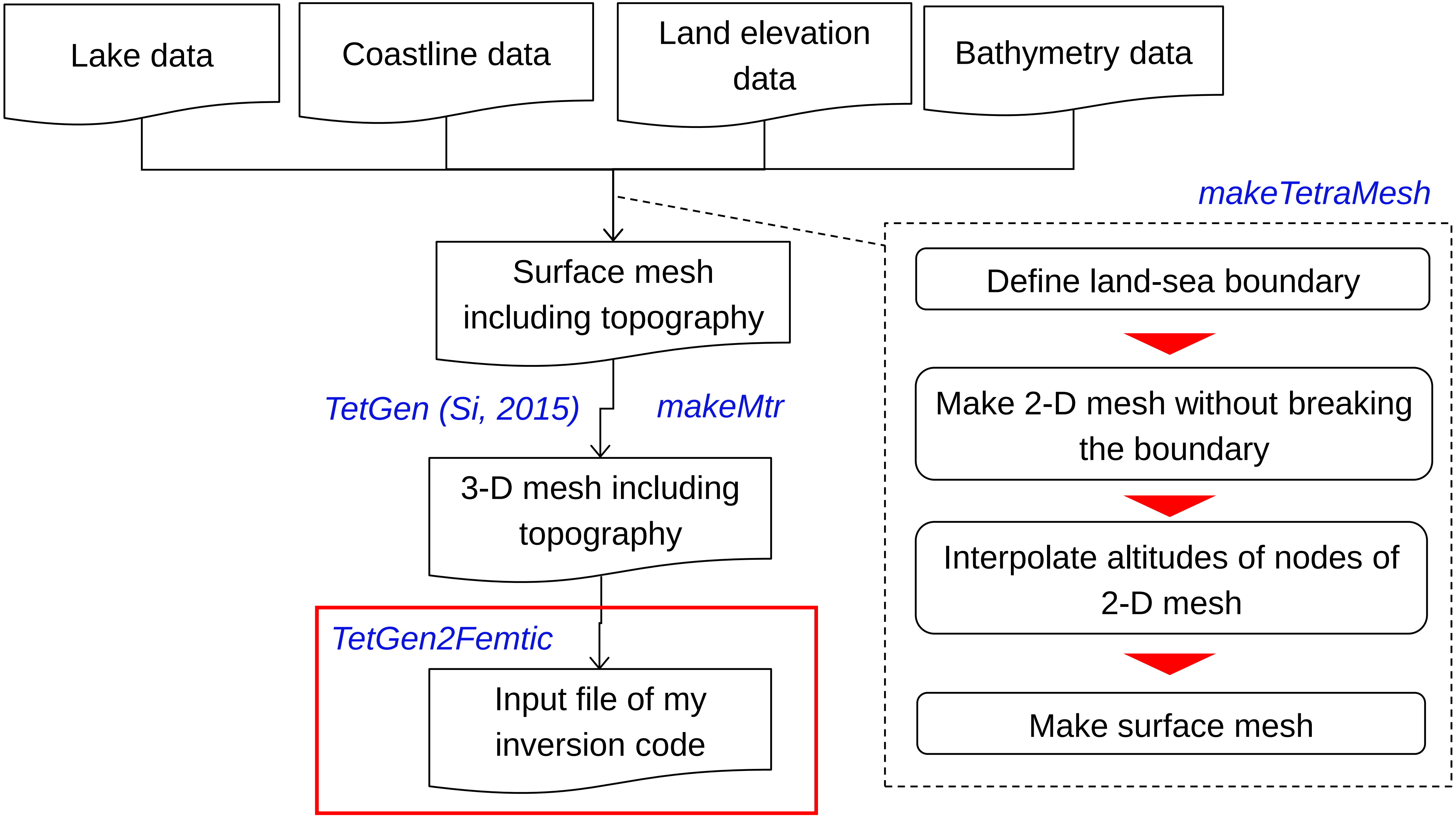
output.6.vtk





Contour of region attributes

How to make 3-D mesh including topography



Input files of TetGen2Femtic

Input files of TetGen2Femtic

File name	Contents
output.X.node	Information of nodes (output file of TetGen)
output.X.ele	Information of elements (output file of TetGen)
output.X.face	Information of element faces (output file of TetGen)
output.X.neigh	Information of adjacency relationship between elements (output file of TetGen)
resistivity_attr.dat	Parameter file for TetGen2Femtic. Unit of lengths in this file is kilo-meter.

* In the file names, ‘X ‘ indicates the iteration number of the iterative refinement.

File format of ‘resistivity_attr.dat’ (1/3)

Number of region attributes (N_{reg}) *1)

Region attribute *1)	*Initial resistivity (Ωm) of the region*	*Repeat number of the partitioning of the region* *2)	*0: Resistivity is not fixed, 1: Resistivity is fixed*

Parameters for the 1st region

⋮

Parameters for the N_{reg}-th region

*1) Region attributes are specified in ‘output.poly’.

*2) Repeat number must be -1 for the region corresponding to the sea, lakes or the air.

File format of ‘resistivity_attr.dat’ (2/3)

X coordinate value (km) | *Y coordinate value (km)* | *Z coordinate value (km)*

Coordinate values of the center of the ellipsoids defining desired length of parameter cells

Rotation angle (deg.) of the ellipsoids around the x-y plane

Number of the ellipsoids (N_e)

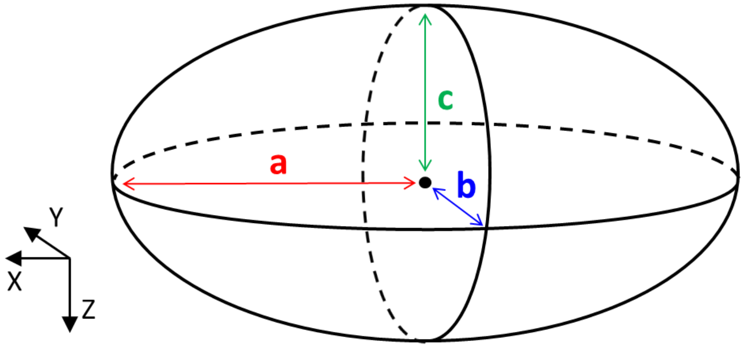
Information of the 1st ellipsoid

$a \quad len \quad f_h \quad f_v$

⋮

Information of the N_e-th ellipsoid

a : *Length along x axis (km)*
len : *Maximum edge length within the ellipsoid (km)*
f_h : *Oblateness on the X-Y plane*
f_v : *Oblateness on the Z-X plane*



$$\frac{b}{a} = 1 - f_h \qquad \frac{c}{a} = 1 - f_v$$

*) Subsequent spheres must cover the formers

File format of ‘resistivity_attr.dat’ (3/3)

Number of points defining desired length of parameter cells (N_p)

Parameters for the 1st point

X coordinate value of the point (km) | *Y coordinate value of the point (km)* | *Z coordinate value of the point (km)*

Number of the spheres (N_s)

Parameters for the 1st sphere

Radius of the sphere (km) | *Maximum edge length within the sphere (km)*

⋮

Parameters for the N_s–th sphere

*) Subsequent spheres must cover the formers

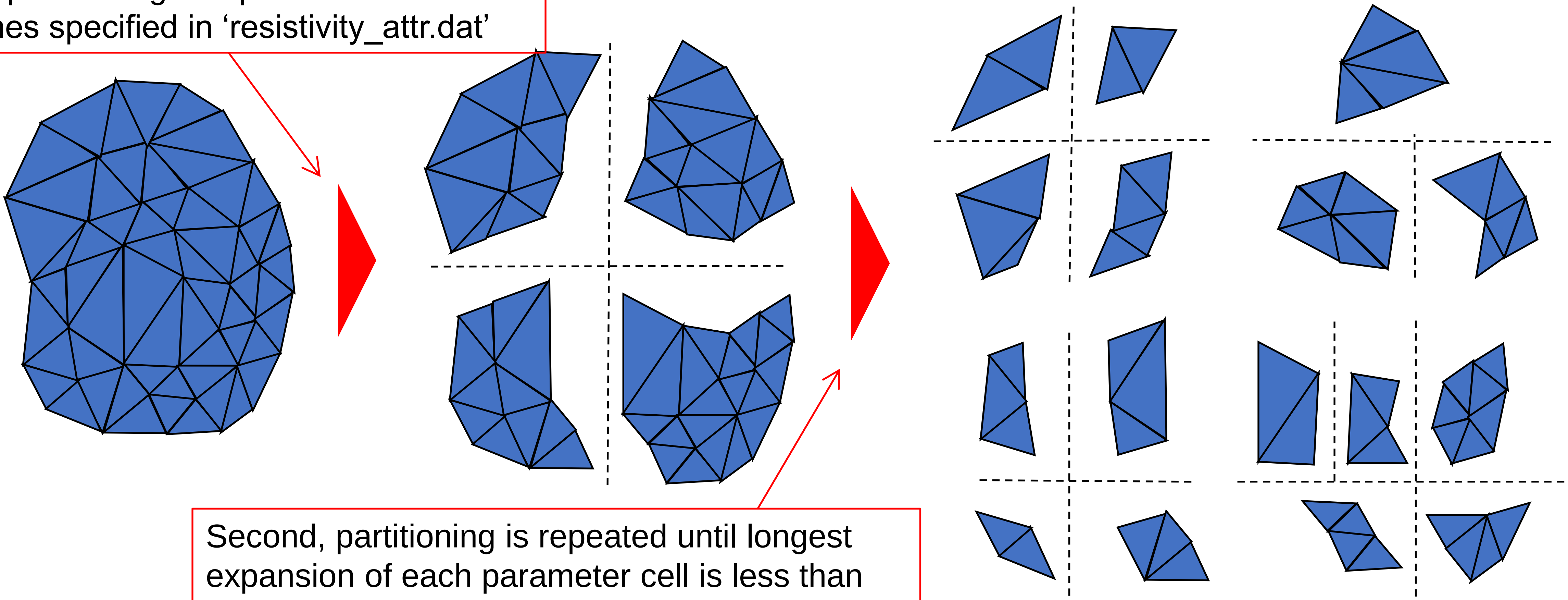
⋮

Parameters for the N_p-th point

Partitioning algorithm

- Each region is partitioned by an algorithm such as the Recursive Coordinate Bisection (Simon 1991) as shown in Usui (2015).
- Region are partitioned into haves according to the coordinate values in its longest expansion (x, y or z direction).

First, partitioning is repeated the number of times specified in 'resistivity_attr.dat'



Second, partitioning is repeated until longest expansion of each parameter cell is less than the desired length defined in 'resistivity_attr.dat'

How to run TetGen2Femtic

You need to execute the following command in the directory where input files exist.

TetGen2Femtic   output.X

* X in the execution argument indicates the iteration number of the iterative refinement.

If you add '-div_all' option in executing TetGen2Femtic, every individual subsurface element become a different model parameter in 'resistivity_block_iter0.dat'.

TetGen2Femtic   output.X -div_all

If you use this option, the number of model parameters become very large because each parameter cell consists of a finite element.
Therefore, you need to use data-space method (model-space method cannot be used for the problems with large unknowns).

Output files of TetGen2Femtic

File name	Contents
mesh.dat	Data of computational mesh (Input file of FEMTIC)
resistivity_block_iter0.dat	Initial resistivity values (Input file of FEMTIC)
output.X.femtic.vtk	Data of computational mesh (Input file of ParaView)

* X in the above file indicates the iteration number of the iterative refinement.

Visualization by ParaView

output.6.femtic.vtk