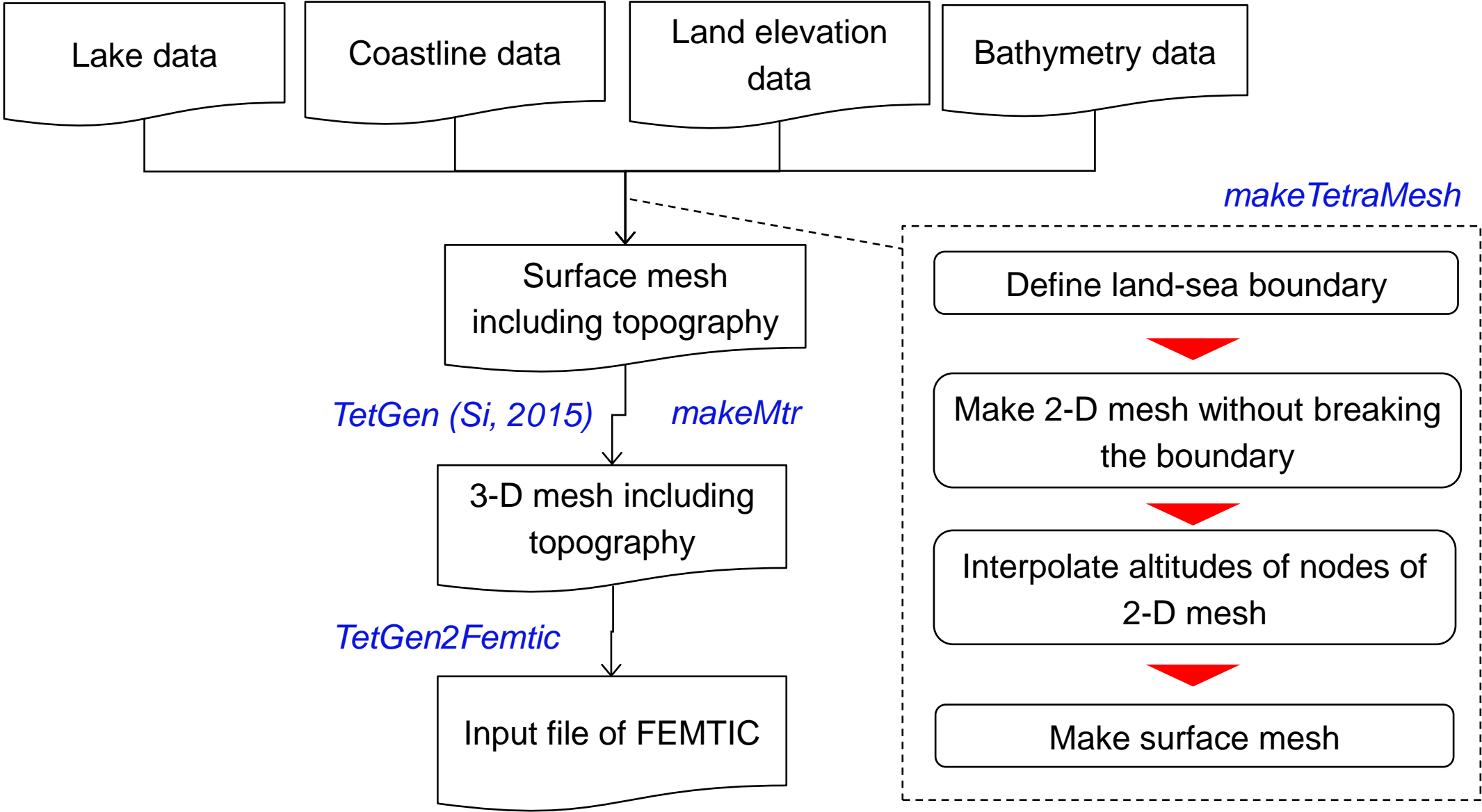


# How to make 3-D mesh including topography




# Procedures of makeTetraMesh

## *makeTetraMesh*


**Step 1**

Define land-sea/land-lake  
boundaries




**Step 2**

Make 2-D mesh without breaking  
the boundaries



**Step 3**

Interpolate altitudes of nodes of  
2-D mesh



**Step 4**

Make surface mesh

## Execution commands

`makeTetraMesh -stp 1`

`makeTetraMesh -stp 2`

`makeTetraMesh -stp 3`

`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

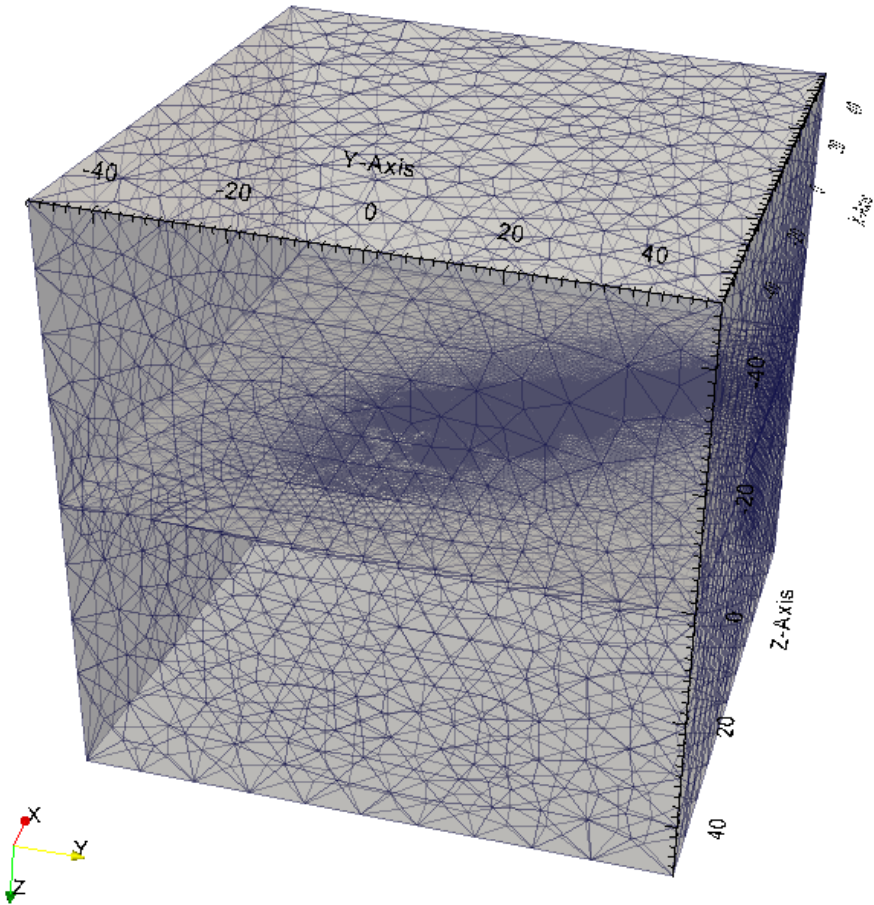
File name	Contents
analysis_domain.dat	Size of computational domain
coast_line.dat	Information about land-sea/land-lake boundaries
control.dat	Parameters controlling mesh generation
observing_site.dat	Mesh size around observation sites

# File format of analysis\_domain.dat

<i>Minimum of X coordinate value (km)</i>	<i>Maximum of X coordinate value (km)</i>
<i>Minimum of Y coordinate value (km)</i>	<i>Maximum of Y coordinate value (km)</i>
<i>Minimum of Z coordinate value (km)</i>	<i>Maximum of Z coordinate value (km)</i>

## Example

```
-50.0 50.0↓  
-50.0 50.0↓  
-50.0 50.0↓
```



# File format of coast\_line.dat

<i>Number of boundaries</i>			
<i>X coordinate value (km)</i>	<i>Y coordinate value (km)</i>	<i>End flag</i>	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

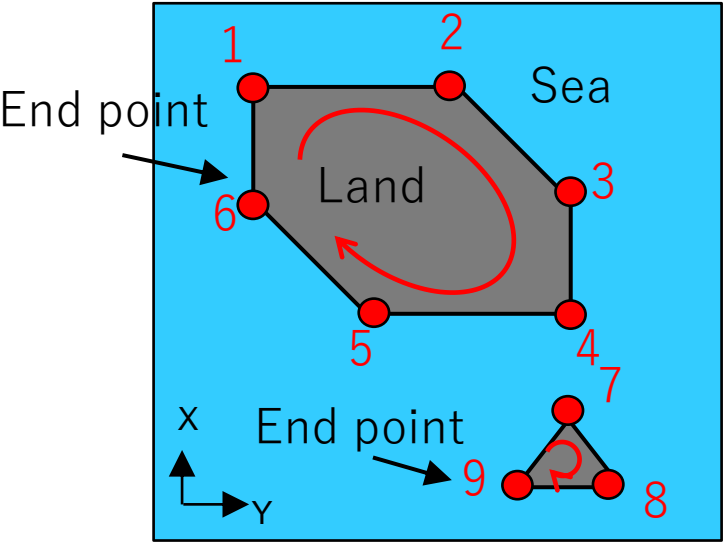
```
10014↓
353.479913264 174.431 0 0↓
353.492502288 174.418 0 0↓
353.495529099 174.413 0 0↓
353.49608366 174.409 0 0↓
353.499731761 174.394 0 0↓
353.510112063 174.392 0 0↓
353.510340906 174.386 0 0↓
353.512989179 174.381 0 0↓
353.518524188 174.378 0 0↓
353.522898605 174.377 0 0↓
353.527148244 174.378 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 <sub>1</sub>	Y <sub>1</sub>	0	0	
X <sub>2</sub>	Y <sub>2</sub>	0	0	
X <sub>3</sub>	Y <sub>3</sub>	0	0	
X <sub>4</sub>	Y <sub>4</sub>	0	0	
X <sub>5</sub>	Y <sub>5</sub>	0	0	
X <sub>6</sub>	Y <sub>6</sub>	1	0	
X <sub>7</sub>	Y <sub>7</sub>	0	0	
X <sub>8</sub>	Y <sub>8</sub>	0	0	
X <sub>9</sub>	Y <sub>9</sub>	1	0	

# File format of ‘control.dat ‘ (1/2)

Keyword	Content	Data type	Option	Default	Example	Step <sup>*1)</sup>			
						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	<input type="radio"/>	<input type="radio"/>		<input type="radio"/>
ROTATION	Rotation angle (deg.) of the ellipsoids to specify edge lengths around the x-y plane	Real value		0.0	ROTATION 30.0	<input type="radio"/>	<input type="radio"/>		<input type="radio"/>
ELLIPSOIDS	Information about the ellipsoids to control edge lengths	<i>Shown in the next slide</i>			<i>Shown in the next slide</i>	<input type="radio"/>	<input type="radio"/>		<input type="radio"/>
NUM_THREADS	Number of threads used in parallel processing	Positive integer		1	NUM_THREADS 3			<input type="radio"/>	
SURF_MESH	Keyword for making surface mesh. <u>This keyword is always required</u>				SURF_MESH				<input type="radio"/>

\*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

$a$     $len$     $f_h$     $f_v^+$     $f_v^-$

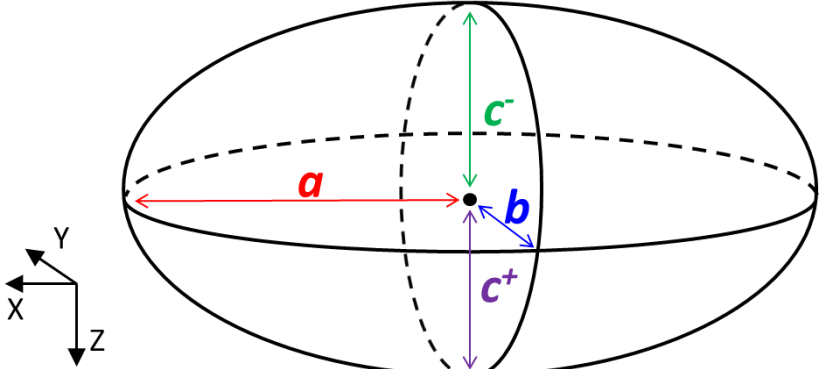
$a$  : Length along x axis (km)

$len$  : 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)



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

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

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

[NOTE] Subsequent ellipsoids must cover the formers

## 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	



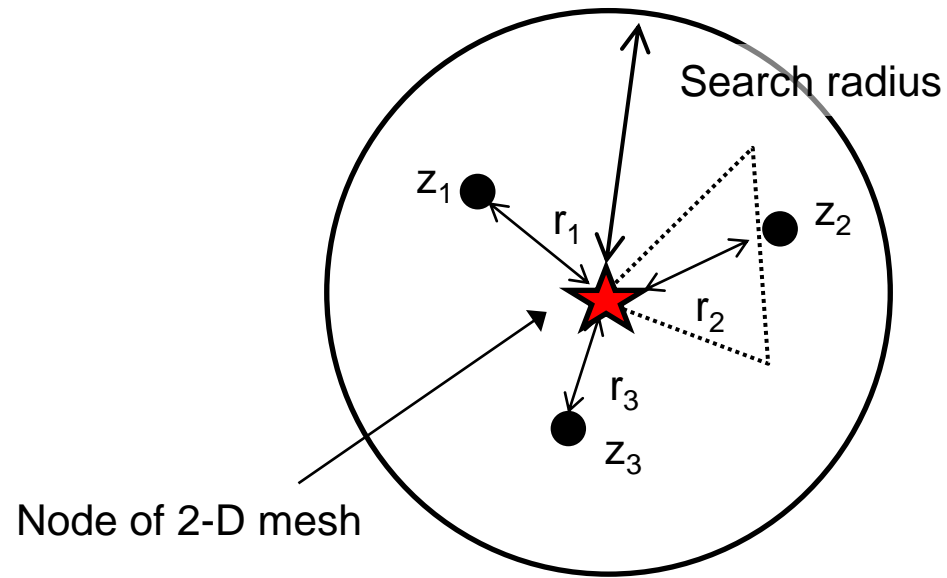
# File format of ‘control.dat’ (2/2)

Keyword	Content	Data type	Option	Default	Example	Step <sup>*1)</sup>			
						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

$\varepsilon$  : Small number to avoid zero divide (km)

# Format of the files including topography/bathymetry data

## Land topography (Altitudes)

<i>X coordinate value (km)</i>	<i>Y coordinate value (km)</i>	<i>Altitude (km)</i>
--------------------------------	--------------------------------	----------------------

⋮

### Example

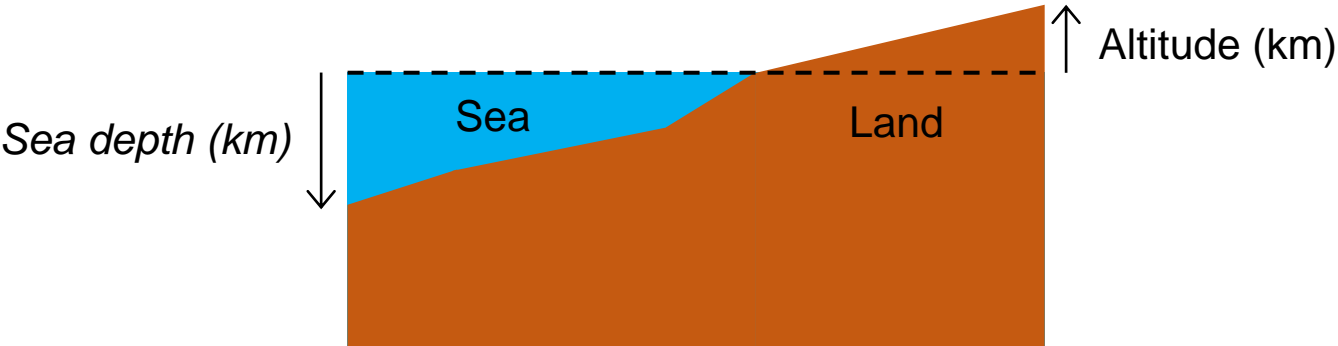
-106.824031915	76.2046151595	2.100000e-003↓
-106.824276161	76.1651038245	6.000000e-003↓
-106.824459262	76.1354703228	8.500000e-003↓
-106.824642292	76.1058368206	7.400000e-003↓
-106.824886221	76.0663254836	9.000000e-003↓
-106.825069084	76.0366919804	9.300000e-003↓
-106.825251877	76.0070584767	8.600000e-003↓
-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↓
...	...	...

## Bathymetry (Depths of the sea floor)

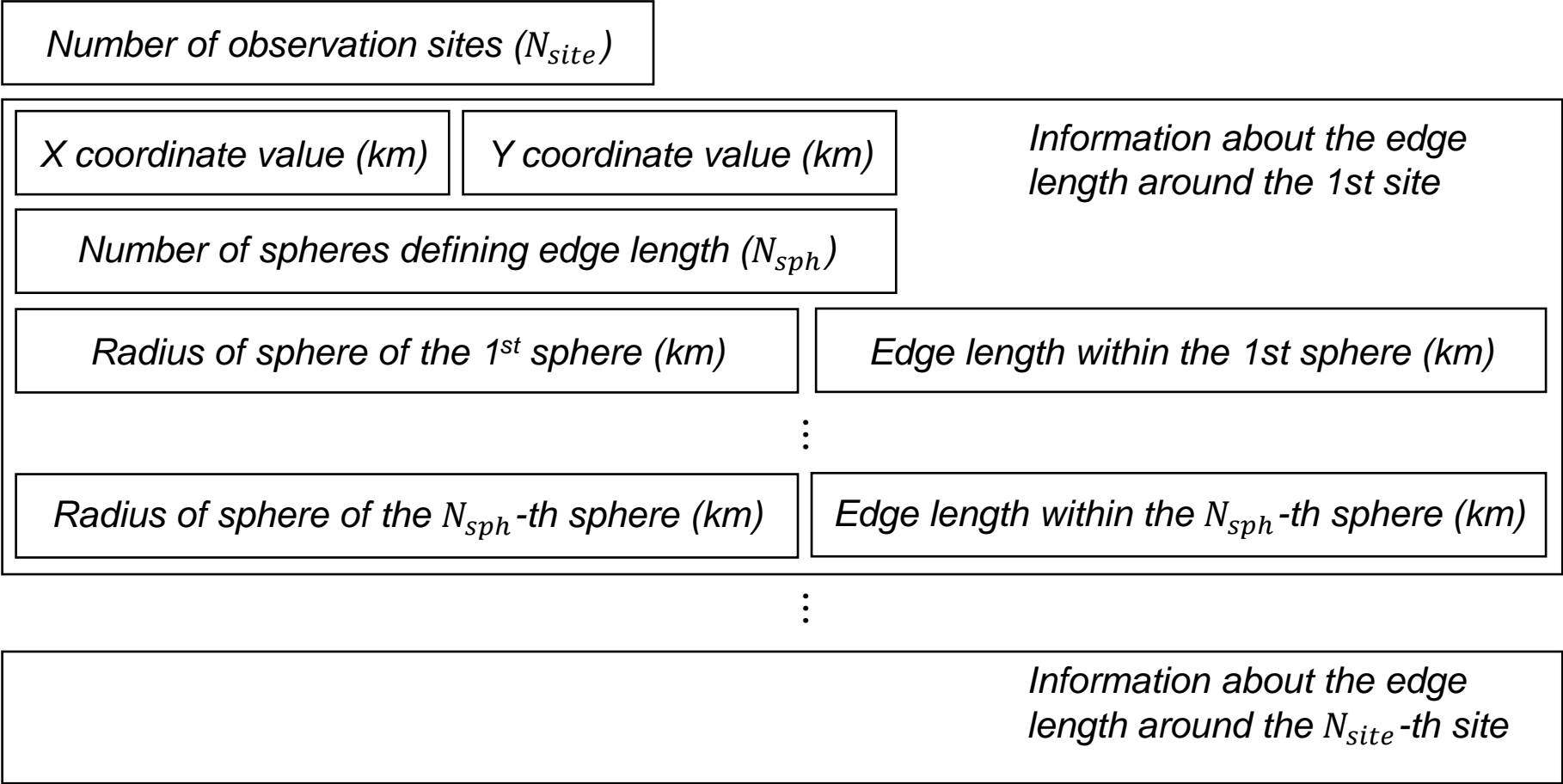
<i>X coordinate value (km)</i>	<i>Y coordinate value (km)</i>	<i>Sea depth (km)</i>
--------------------------------	--------------------------------	-----------------------

⋮

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



# File format of ‘observing\_site.dat’



\*Subsequent spheres must cover the formers

## Example

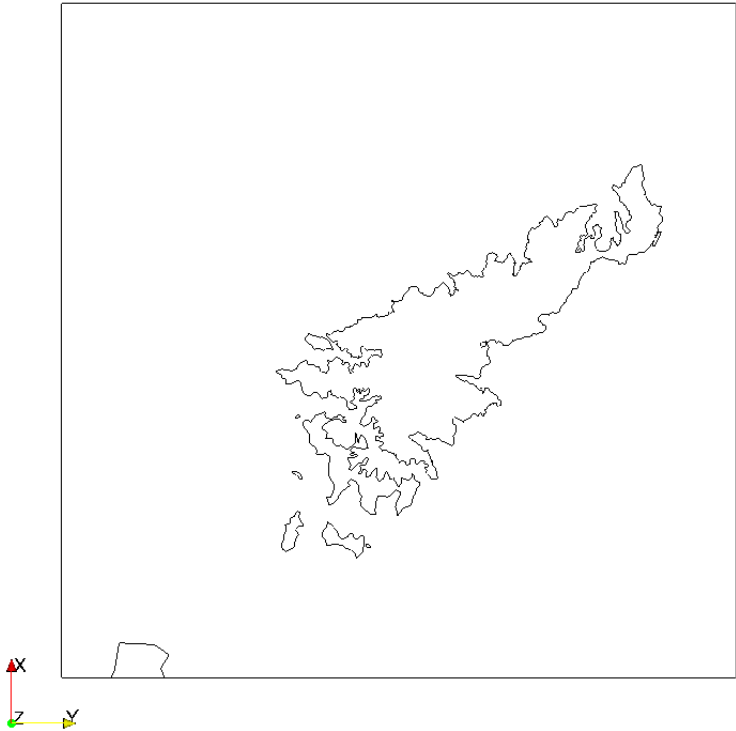
```
28↓
0.0000000000 0.0000000000↓
5↓
0.1 0.02↓
0.3 0.05↓
1.0 0.10↓
3.0 0.30↓
5.0 0.50↓
```

# Output files of Step 1

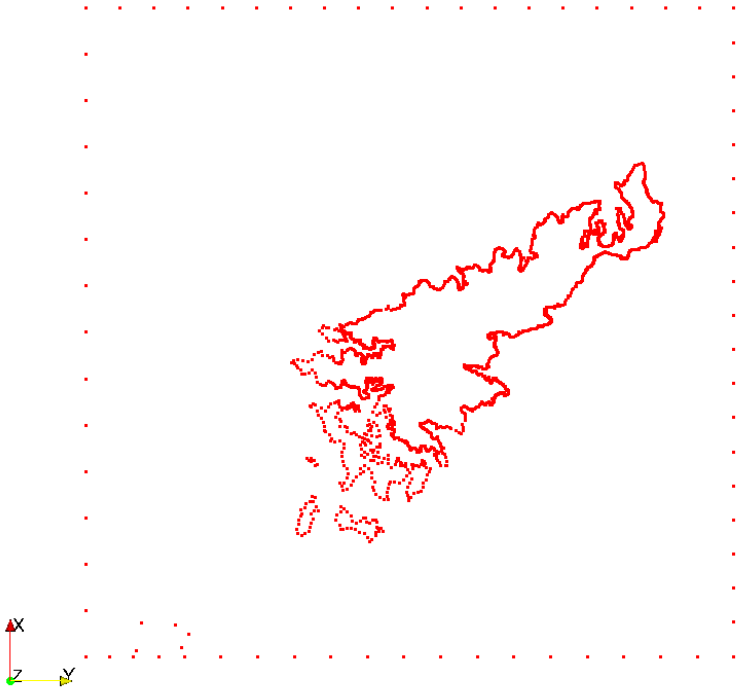
File name	Contents
makeTetraMesh.log	Log output
node_bound_curve.stp1	Coordinate values of the points on inner/outer boundaries (Input file of Step 2)
boundary_curves.stp1	Connections of the points of inner/outer boundaries (Input file of Step 2)
boundary_curves_relation.stp1	Relationship between inner/outer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

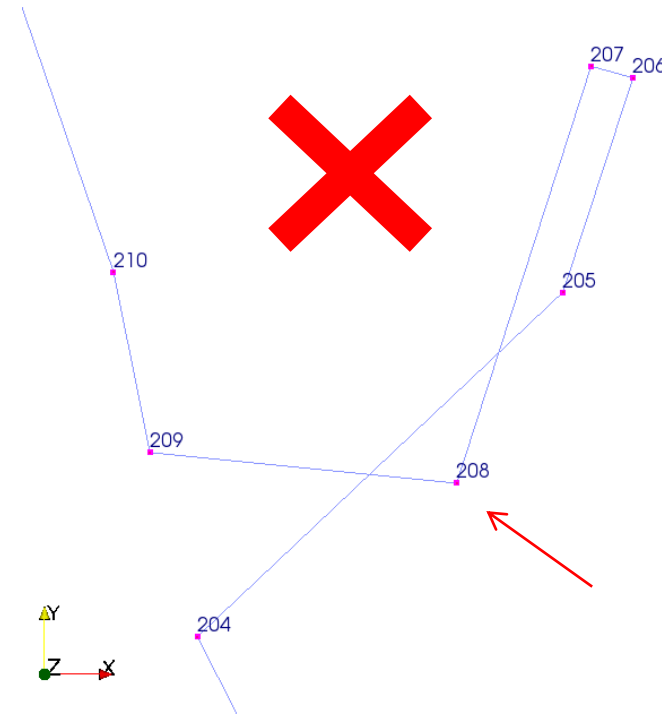
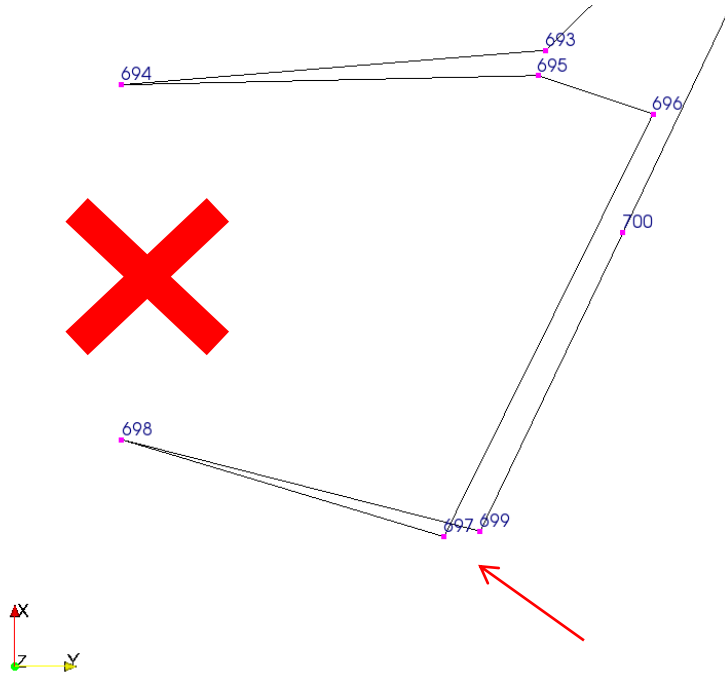
boundary\_curve\_stp1.vtk



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.

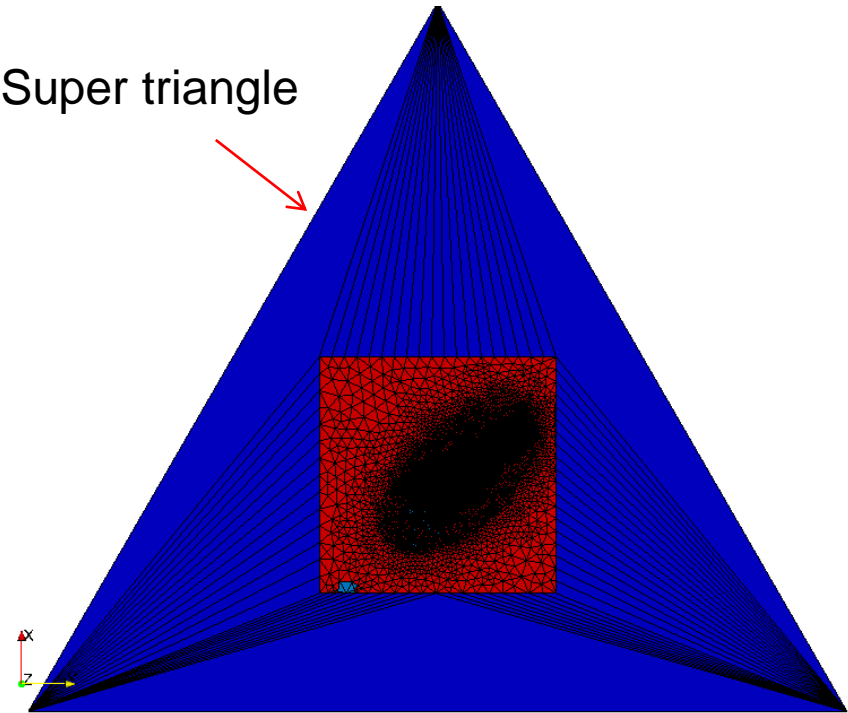
# Visualization by ParaView (Step 2)

surface\_triangle.laplacian\_last.vtk



Contour of domain types

Super triangle

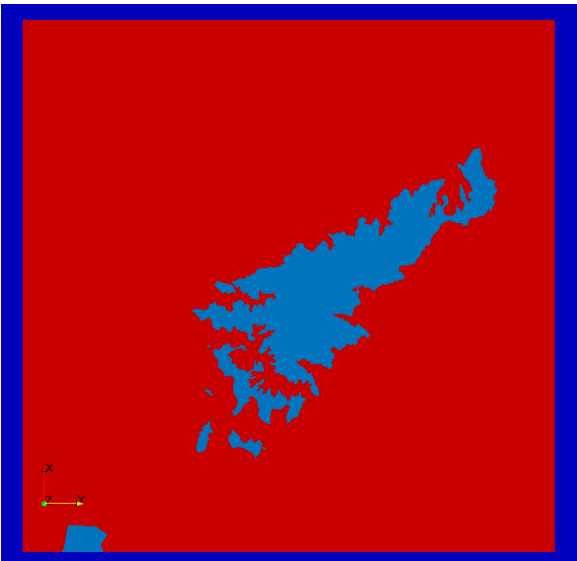
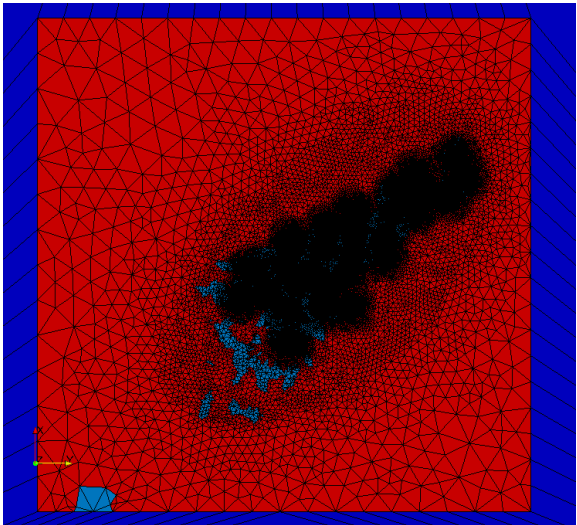


Domain type

Sea : 0

Land : 1

Lake : 2

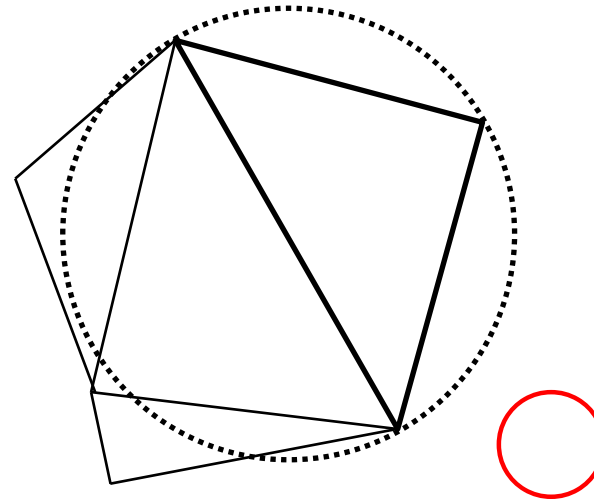
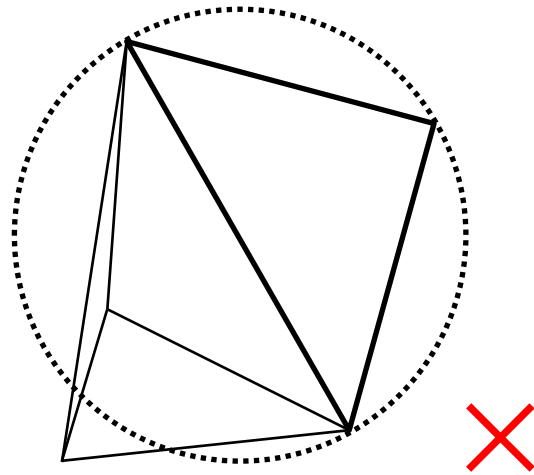


# Delaunay triangulation method

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*.

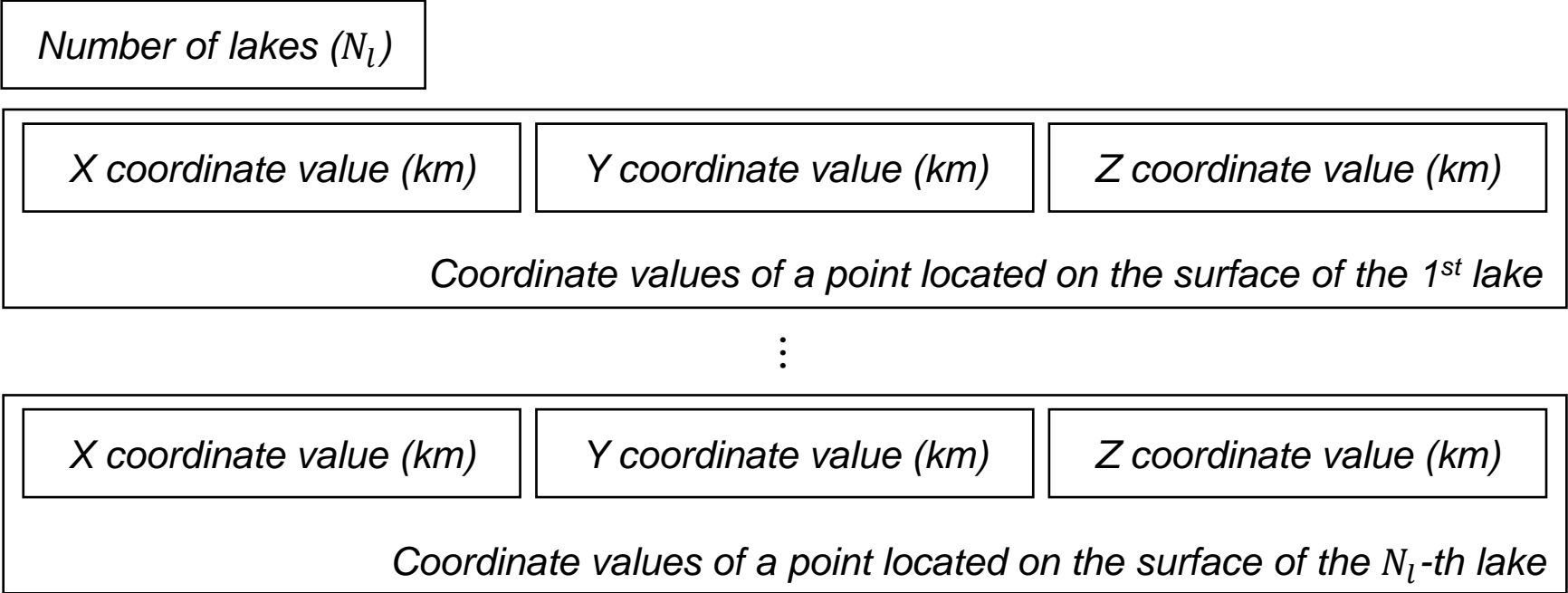


# Step 3: Interpolate altitudes of the nodes of 2-D mesh

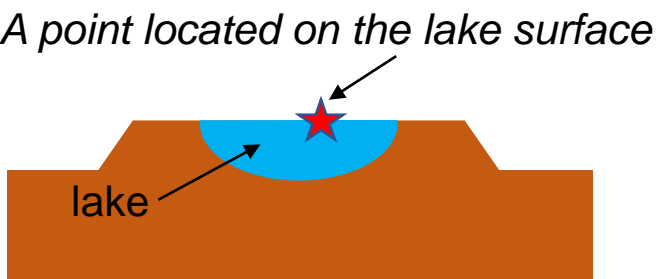
## Input files of Step 3

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)
<i>Defined in 'control.dat'</i>	Land topography (altitudes)
<i>Defined in 'control.dat'</i>	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' (2/2)



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



# Output files of Step 3

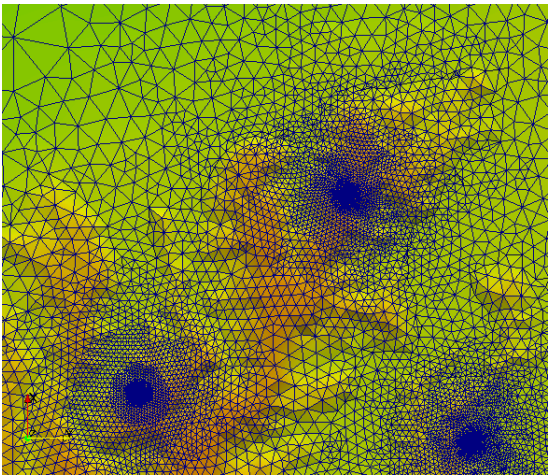
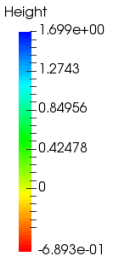
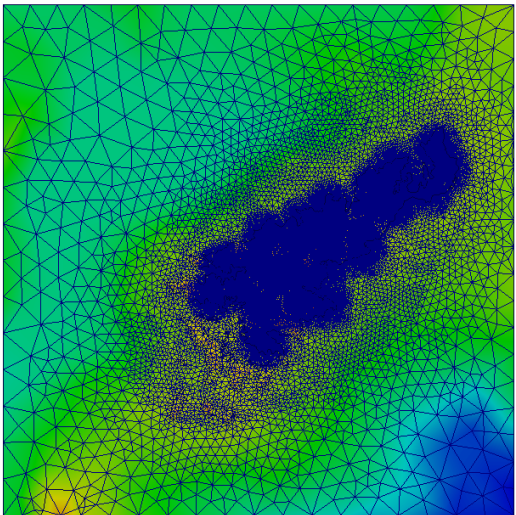
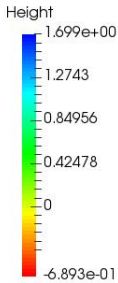
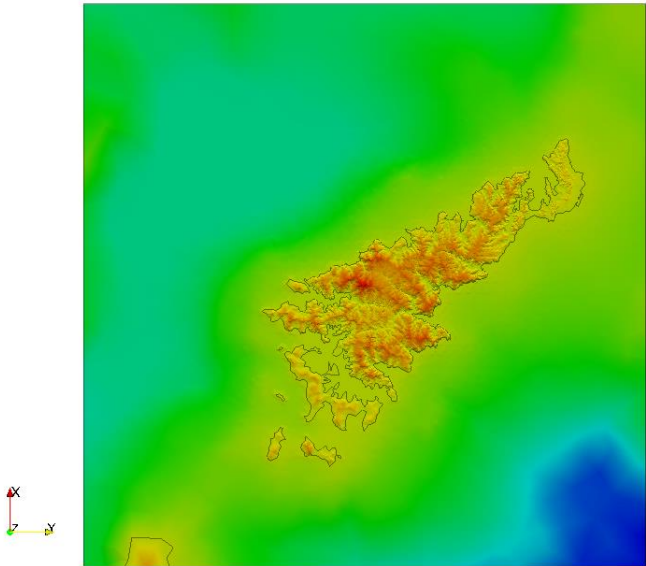
## Output files of Step 3

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. However, you can omit these files.

# 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

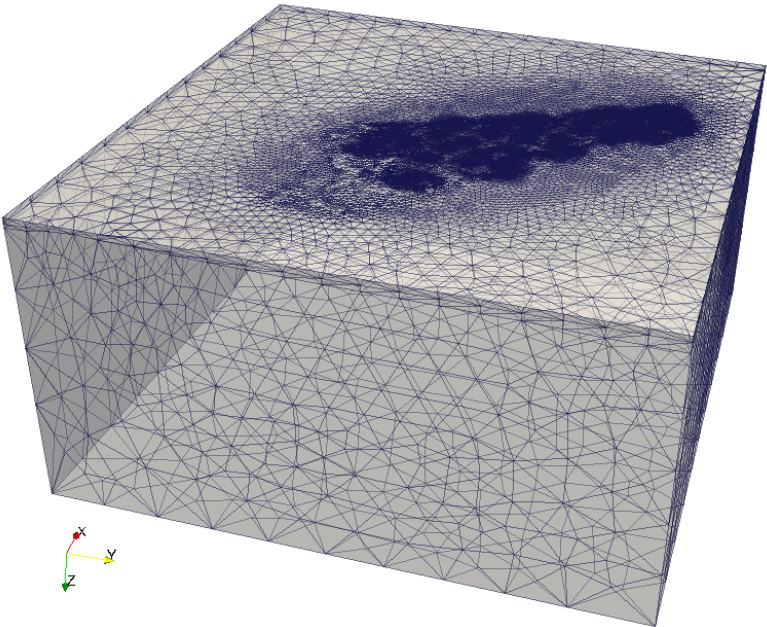
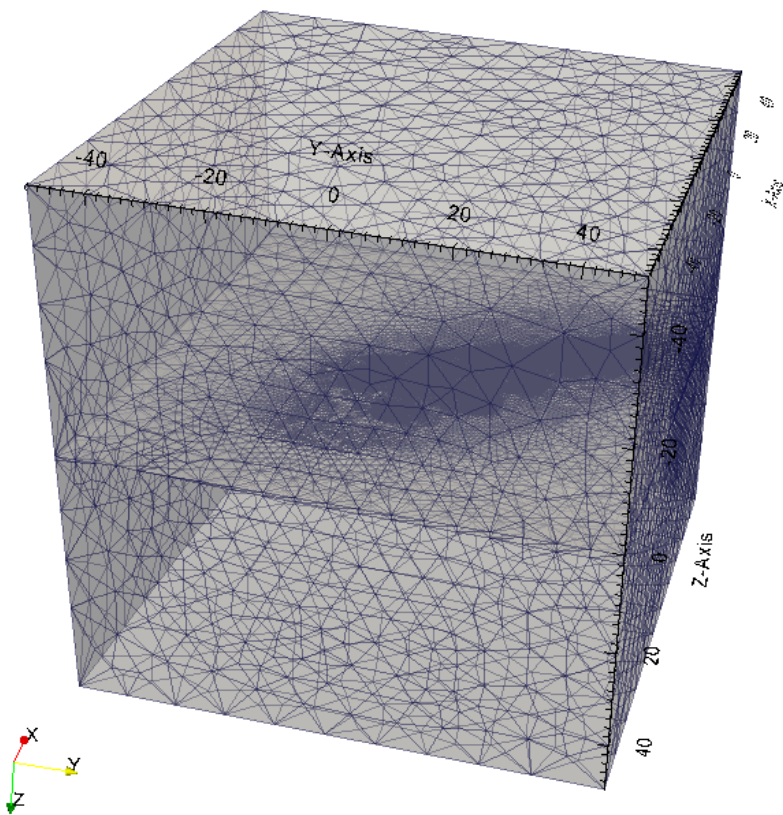
## Output files of Step 4

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)

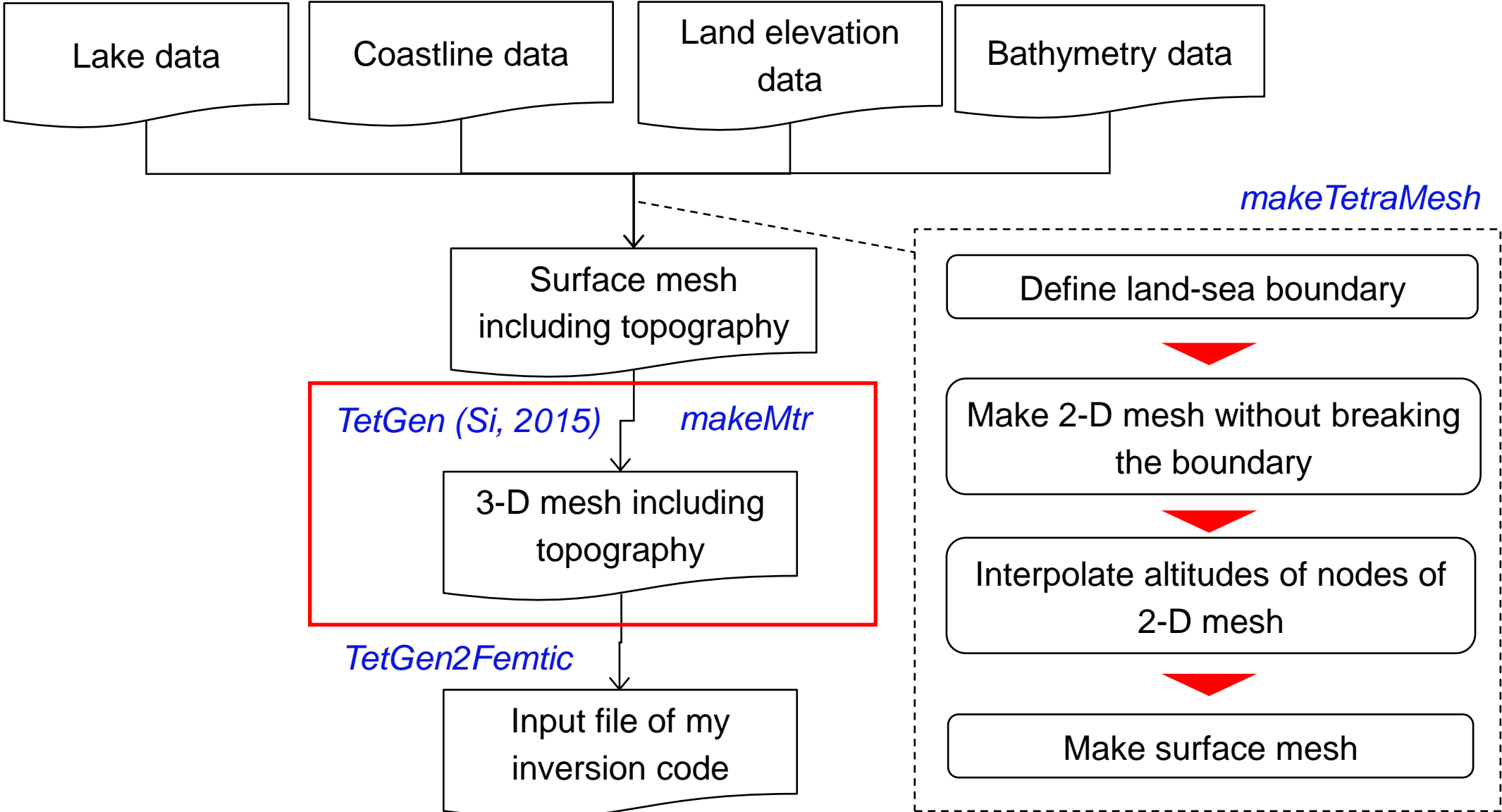
In addition, some intermediate file is outputted. However, you can omit these files.

# Visualization by ParaView (Step 4)

plc.vtk

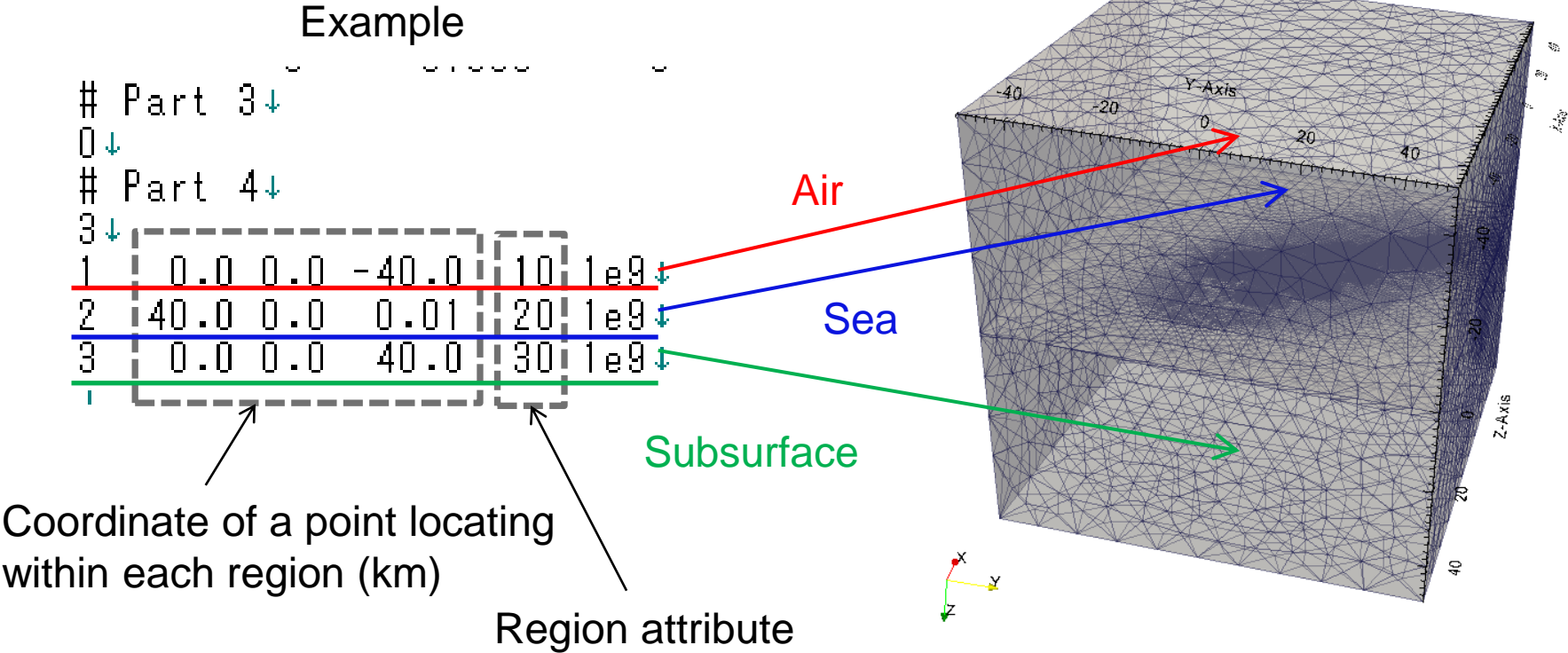


# How to make 3-D mesh including topography



# Region attribute in 'output.poly'

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



# 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
```

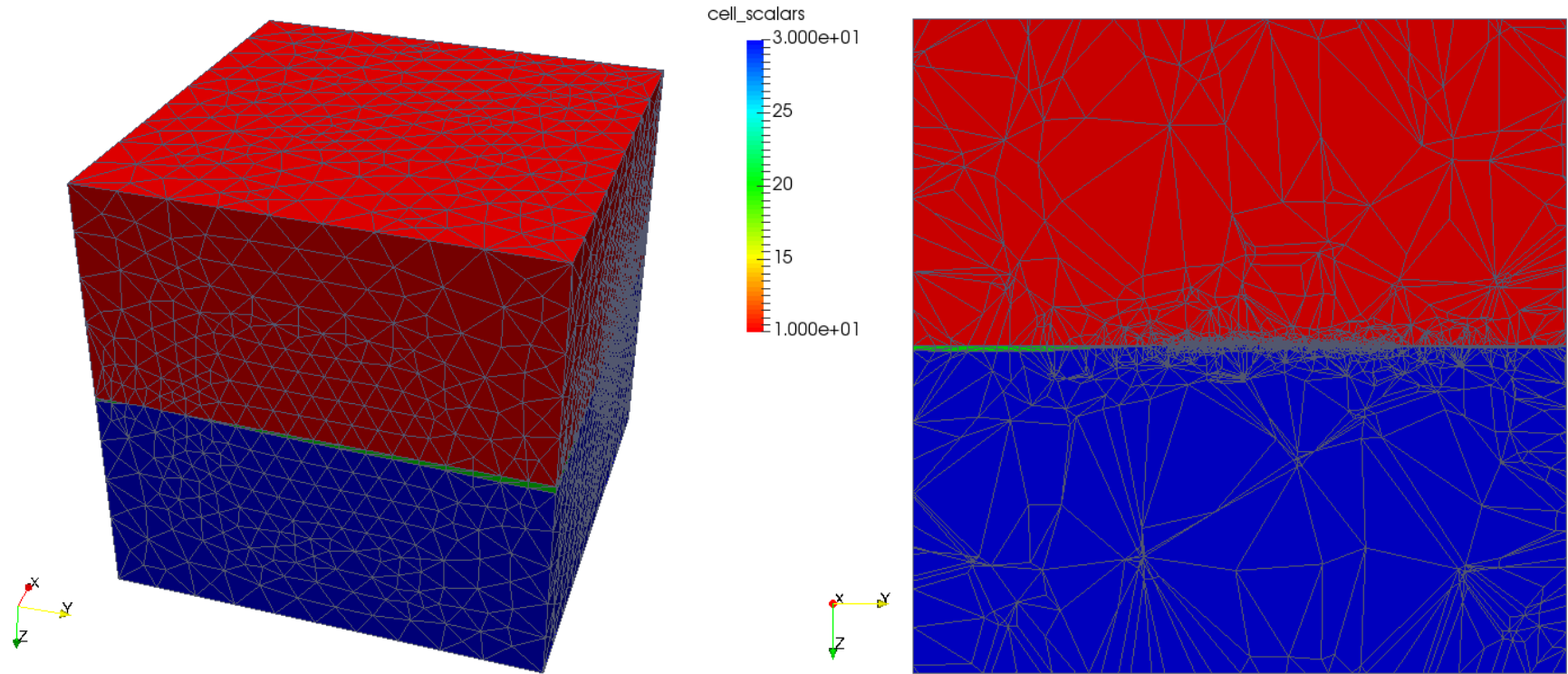
## Output files of TetGen

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 )

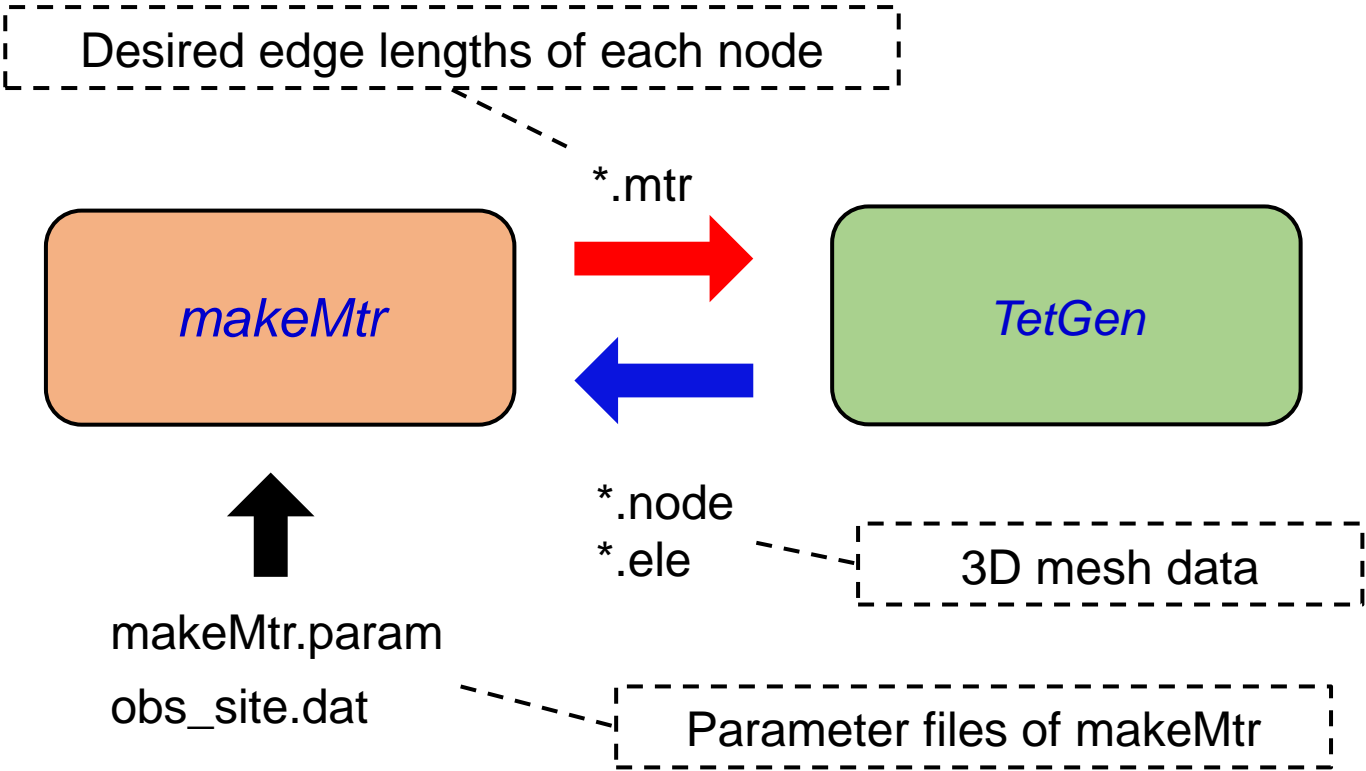
output.1.vtk



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 <sup>*1)</sup>	Information about the elements of 3-D mesh (Output file of TetGen).
XXX.node <sup>*1)</sup>	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 <sup>*1)</sup>	Desired edge lengths of each node of 3-D mesh. (Input file of TetGen)

<sup>\*1)</sup> In the files ‘XXX’ indicates the main part of the name of TetGen output files, which is specified as the execution argument of makeMtr.

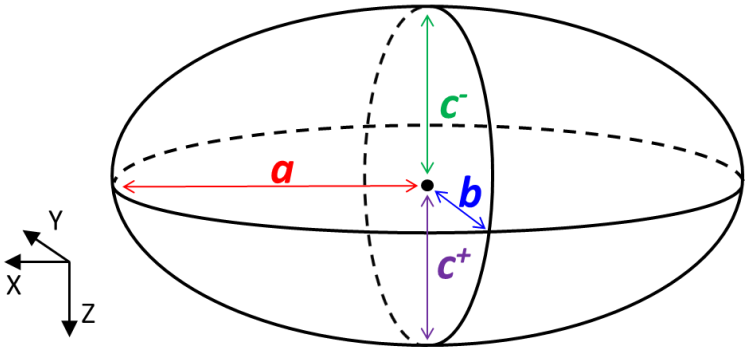


# File format of 'makeMtr.param'

<i>X coordinate value (km)</i>	<i>Y coordinate value (km)</i>	<i>Z coordinate value (km)</i>
<i>Coordinate values of the center of the ellipsoids defining desired edge lengths</i>		
<i>Rotation angle (deg.) of the ellipsoids around the x-y plane</i>		
<i>Number of the ellipsoids (<math>N_e</math>)</i>		
<i>Information of the 1st ellipsoid</i>	<div><math>a</math>   <math>len</math>   <math>f_h</math>   <math>f_v^+</math>   <math>f_v^-</math></div>	
$\vdots$		
<i>Information of the <math>N_e</math>-th ellipsoid</i>		

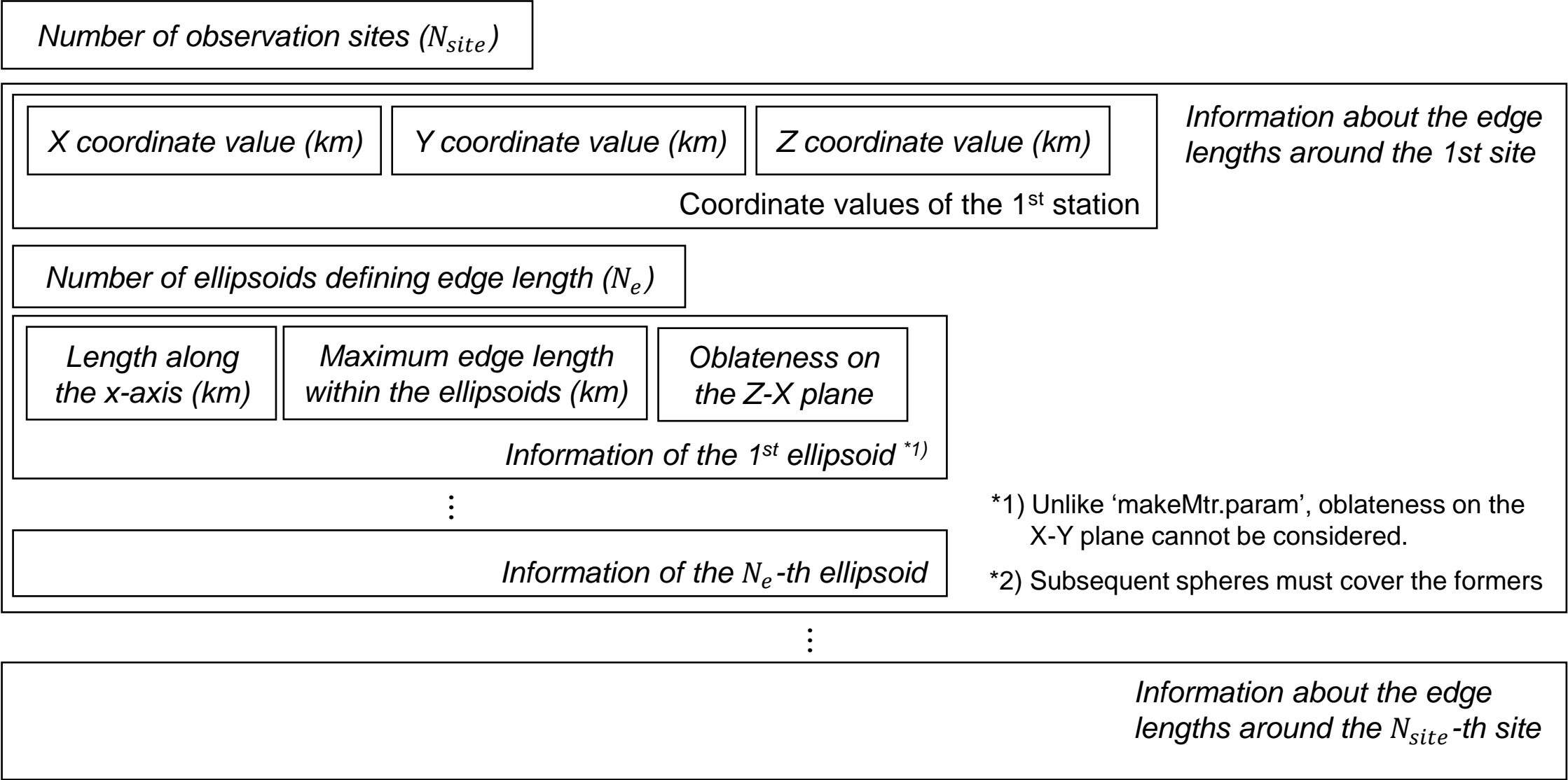
\*Subsequent ellipsoids must cover the formers

- $a$  : 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)



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

# File format of ‘obs\_site.dat’



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

\*2) Subsequent spheres must cover the formers

# How to perform iterative refinement

Example of execution commands

```
for I in 1 2 3 4 5
```

```
do
```

```
makeMtr output.$i
```

```
tetgen -nmpYVrAakq3.0/0 output.$i
```

```
done
```

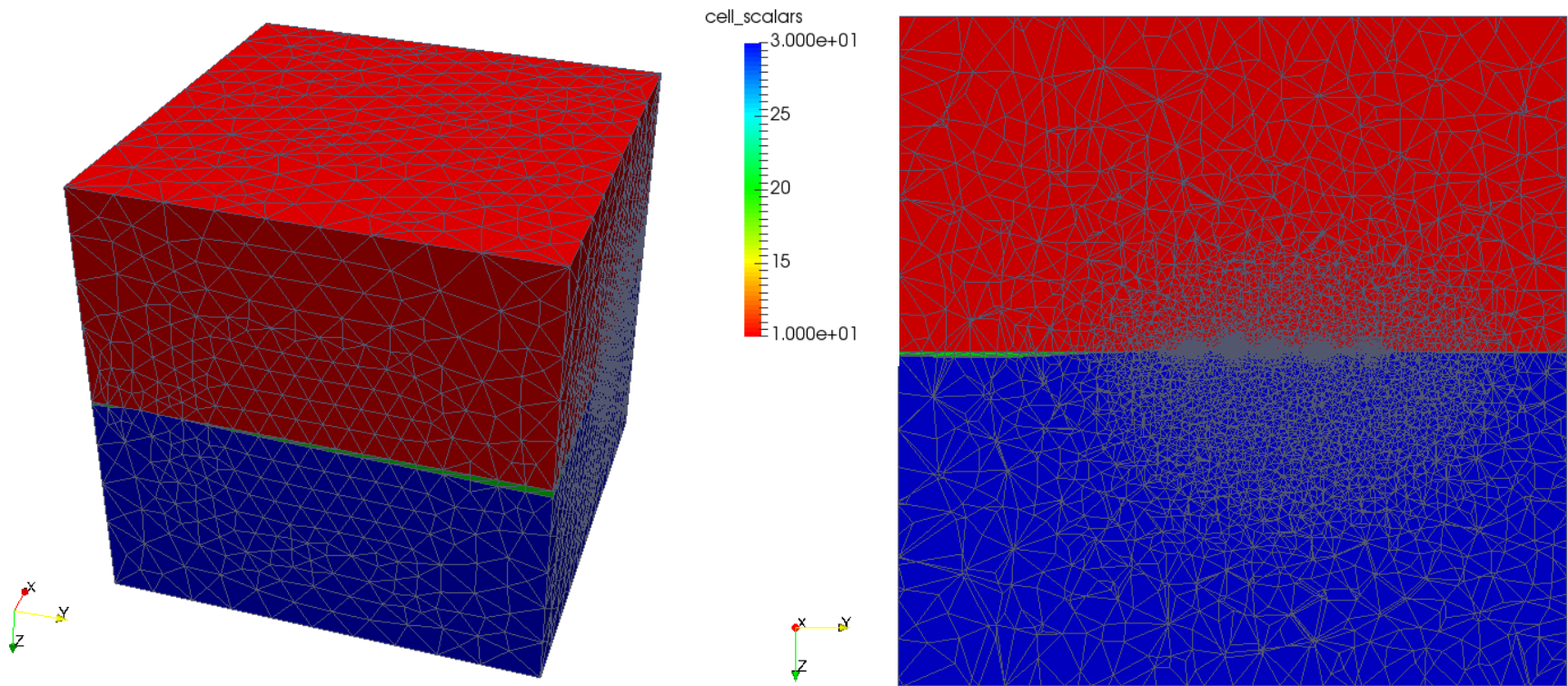
# Output files of the iterative refinement

Output files of the X-th iteration

<i>File name</i>	<i>Contents</i>
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)

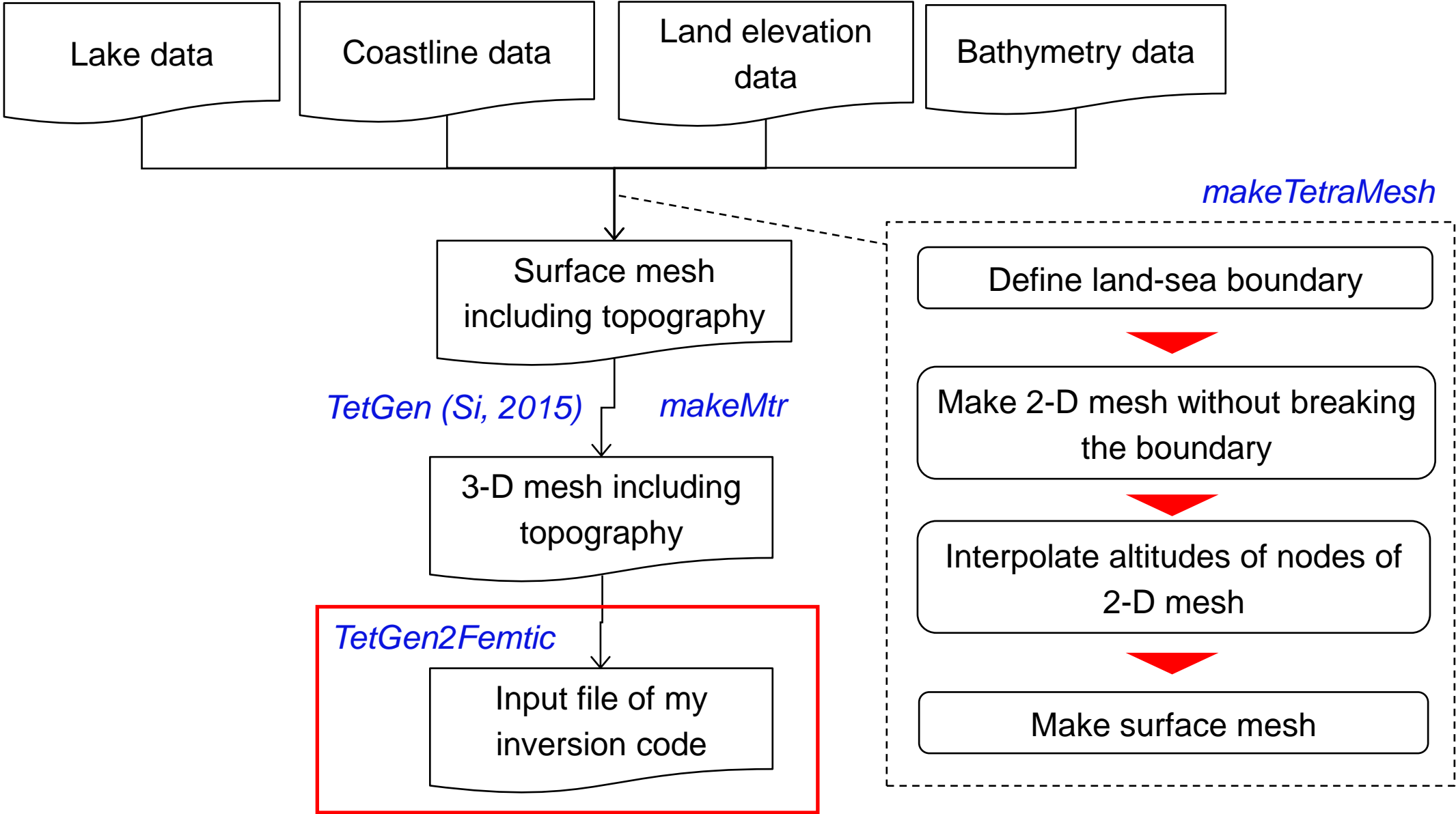
# Visualization by ParaView

output.6.vtk



Contour of region attributes

# How to make 3-D mesh including topography



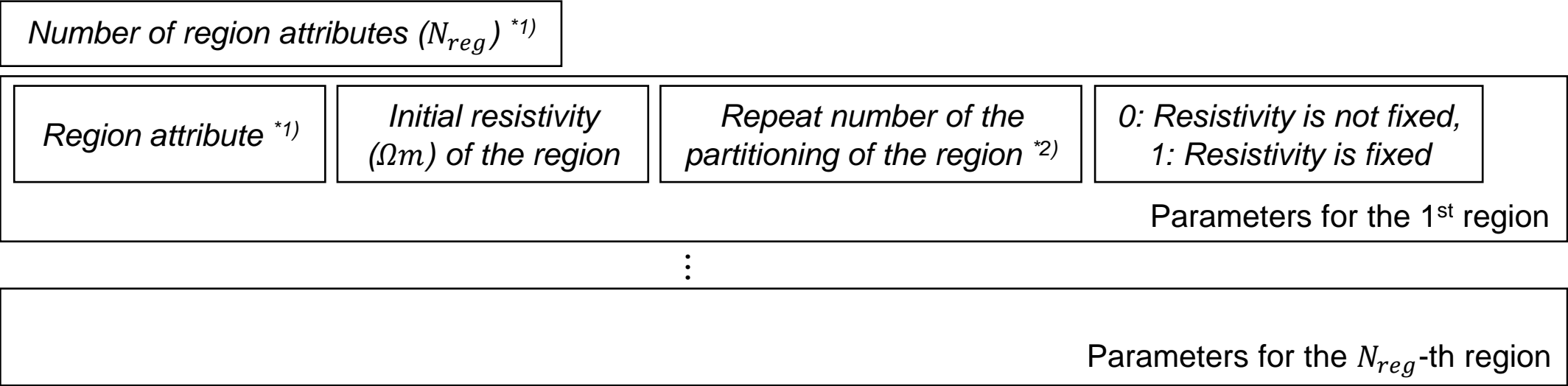
# Input files of TetGen2Femtic

## 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)



\*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 ( $N_e$ )

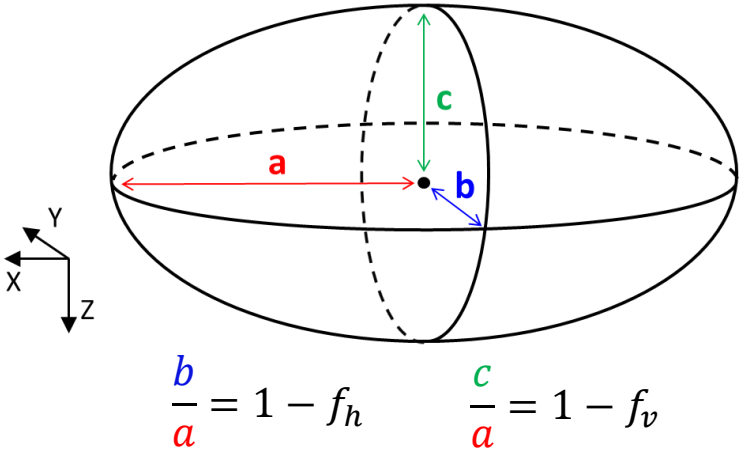
Information of the 1st ellipsoid

⋮

Information of the  $N_e$ -th ellipsoid

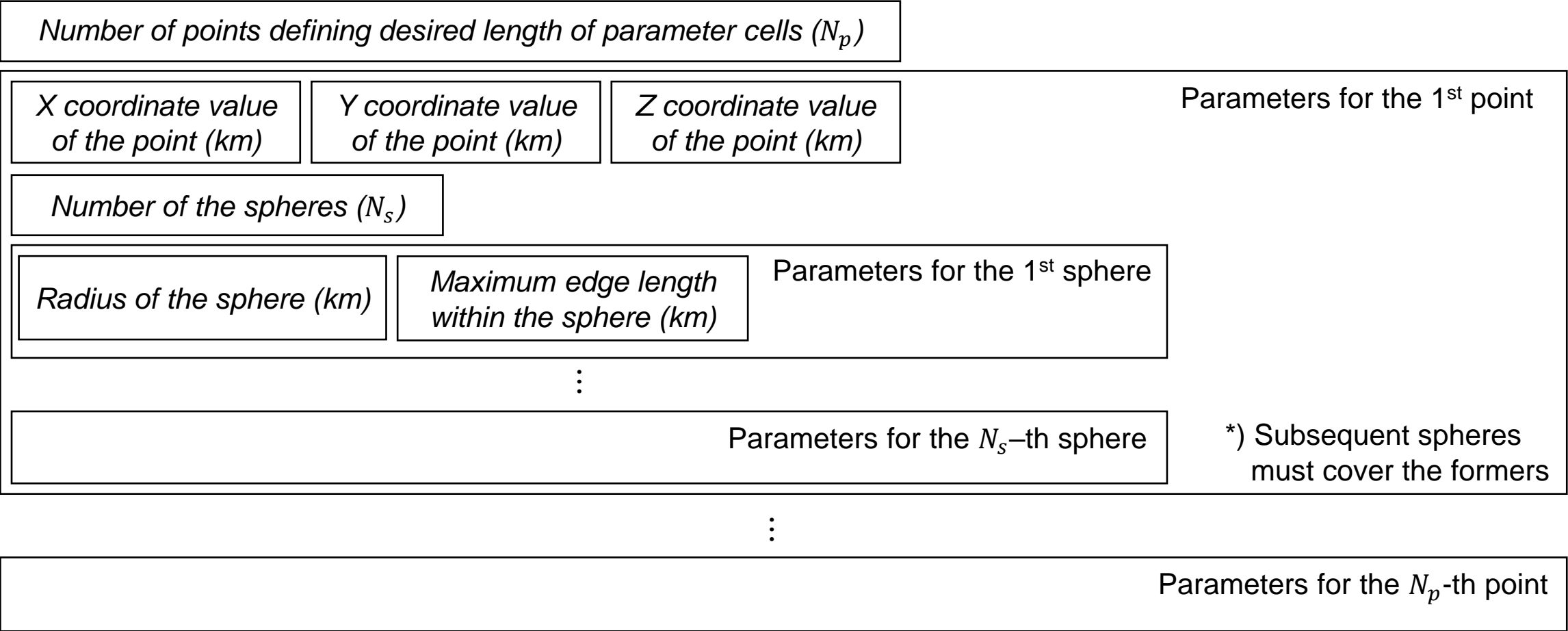
$a$     $len$     $f_h$     $f_v$

$a$  : 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



\*) Subsequent spheres  
must cover the formers

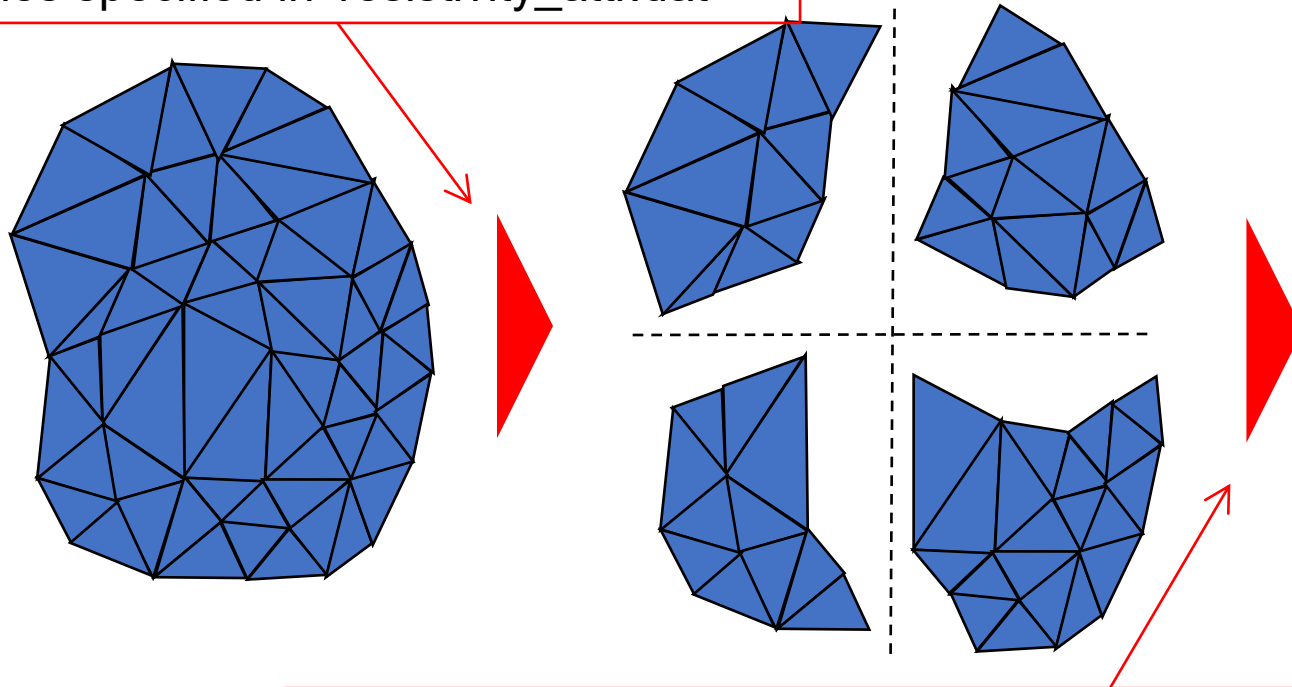
# File format of 'resistivity\_attr.dat' (3/3)



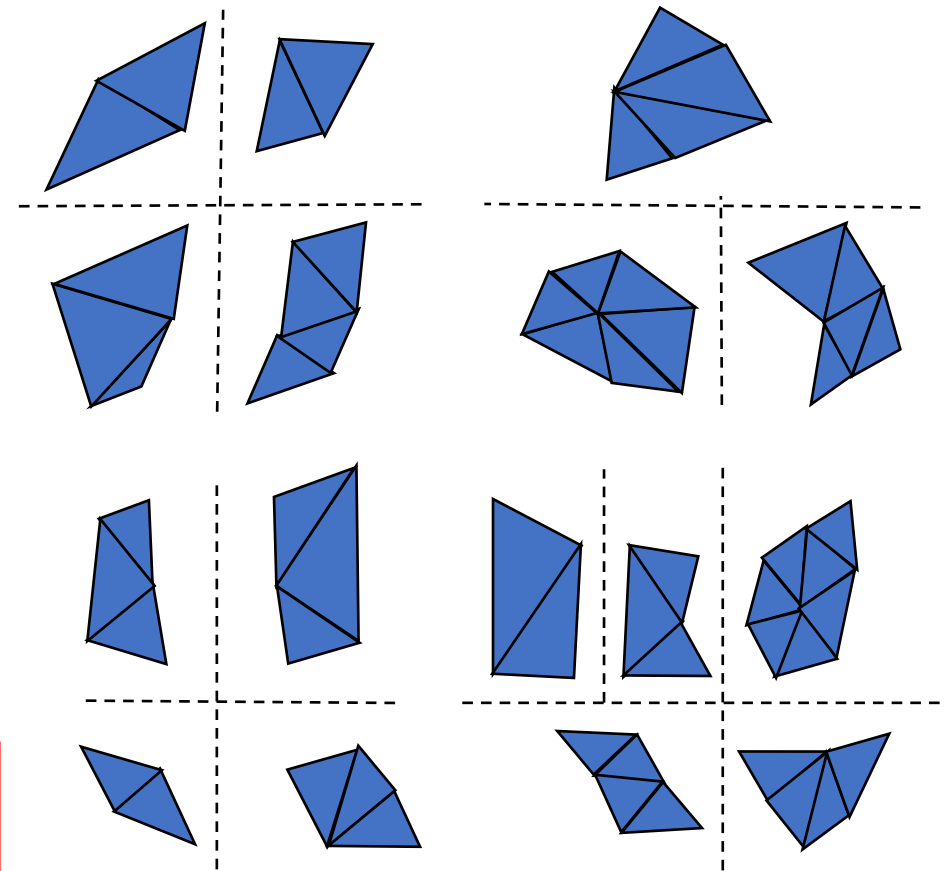
# Partitioning algorithm

- 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 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

