

Automesh-2D (Multi-Domains) Manual

Example Study

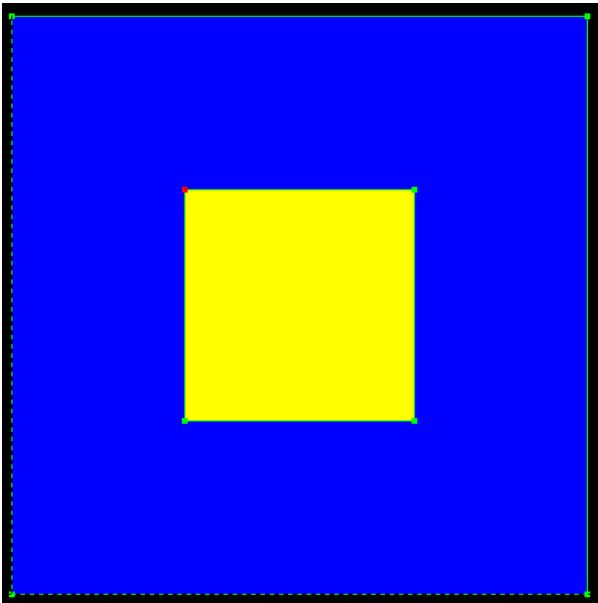


Fig. 1 The geometry with two domains

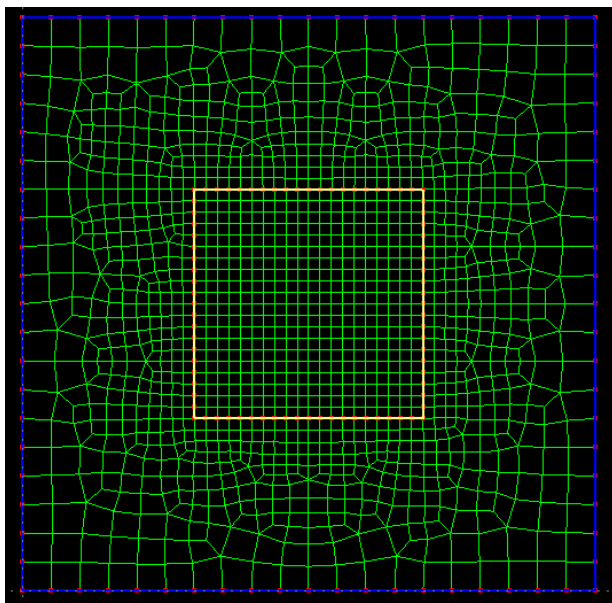


Fig. 2 The mesh

We will show you the usage of the program with an example. In this example, the geometry has two domains (colored with blue and yellow respectively). The generated mesh for the geometry is shown in Fig. 2.

Step 1 Define Points

Define the eight points by typing the coordinates in the textbox on the left dialog or by clicking in the graphics window with the mouse.

Points Definition

☒ By mouse click on screen

No (x, y, size)

3	(100, 100, 0)
4	(0, 100, 0)
5	(30, 30, 0)
6	(70, 30, 0)
7	(70, 70, 0)
8	(30, 70, 0)

☐ As Constraint Point

Input X,Y and Mesh Size:

30.000000 70.000000

Change Size: 0.000000

Add Remove

☐ Show points no.

☒ Show Grid

Setting

Points list. Select one point to change its Mesh Size or remove it. You can also select one point by clicking on it in the graphic window.

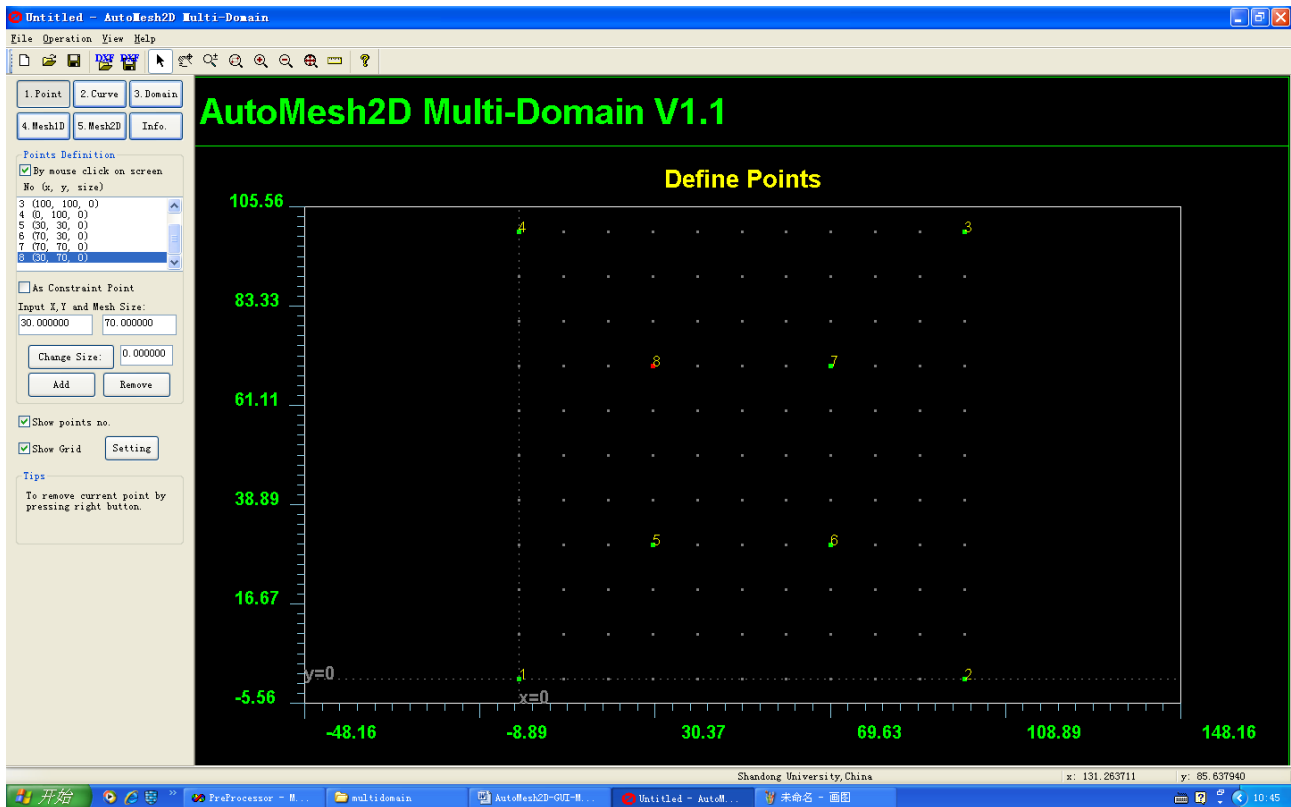
You can define constraint points inside the geometry that need apply load on points or refine the mesh at the points. You need define the element size at the points.

Type point coordinates here

Type Mesh Size here and Change it

Add: Add one point.
Remove: Remove the point from the list.

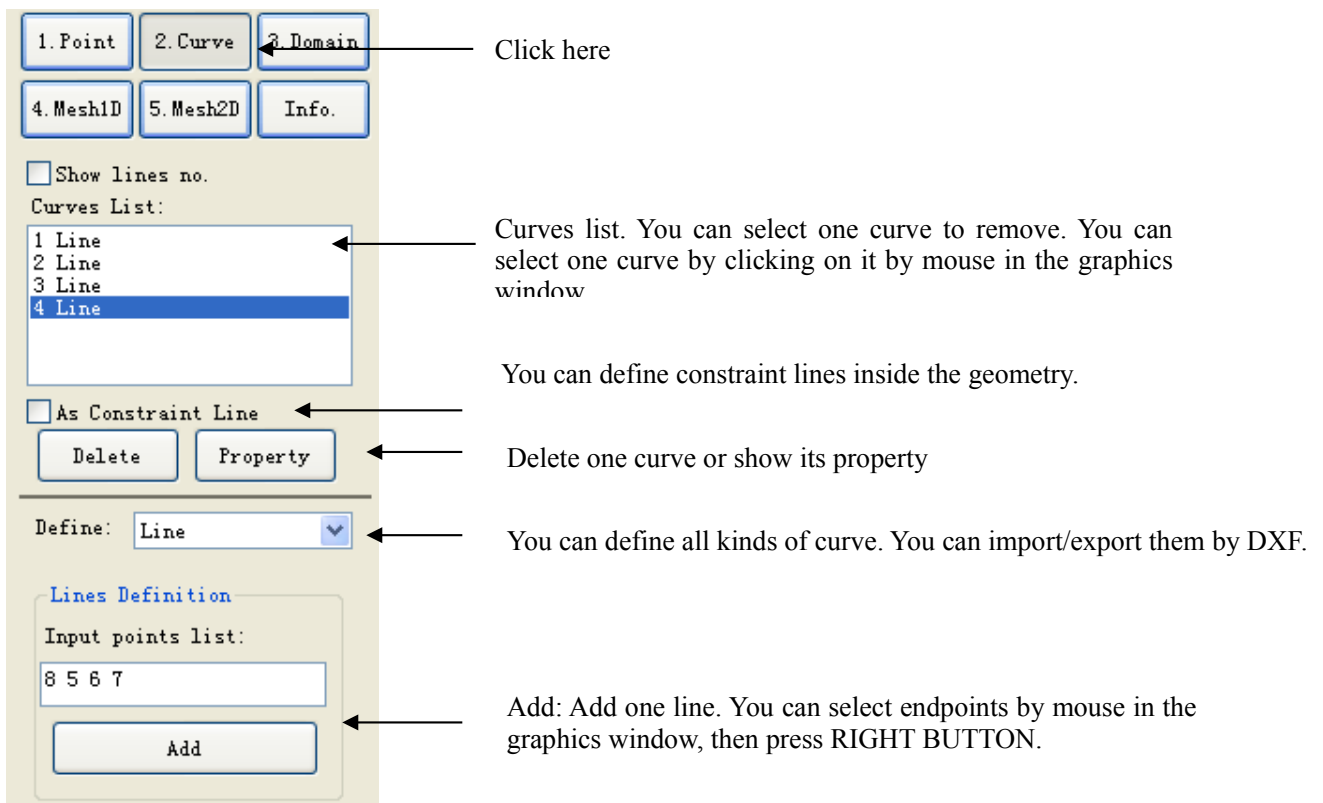
You can define a point on the grid by the mouse clicking on the point.

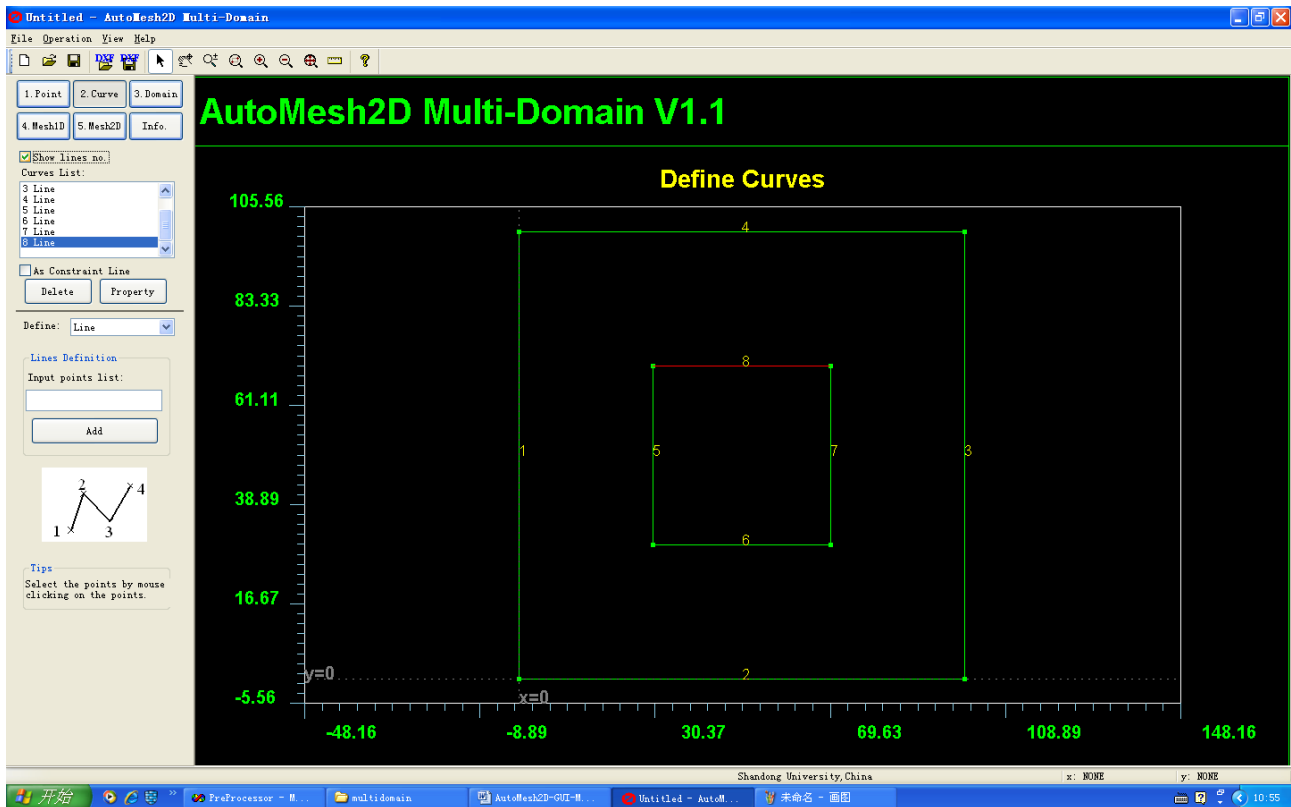


Step 2 Define Lines

Click on the button **Curve** on the dialog.

Select the points with the mouse to define lines. Press right button after selecting. Eight lines are defined.

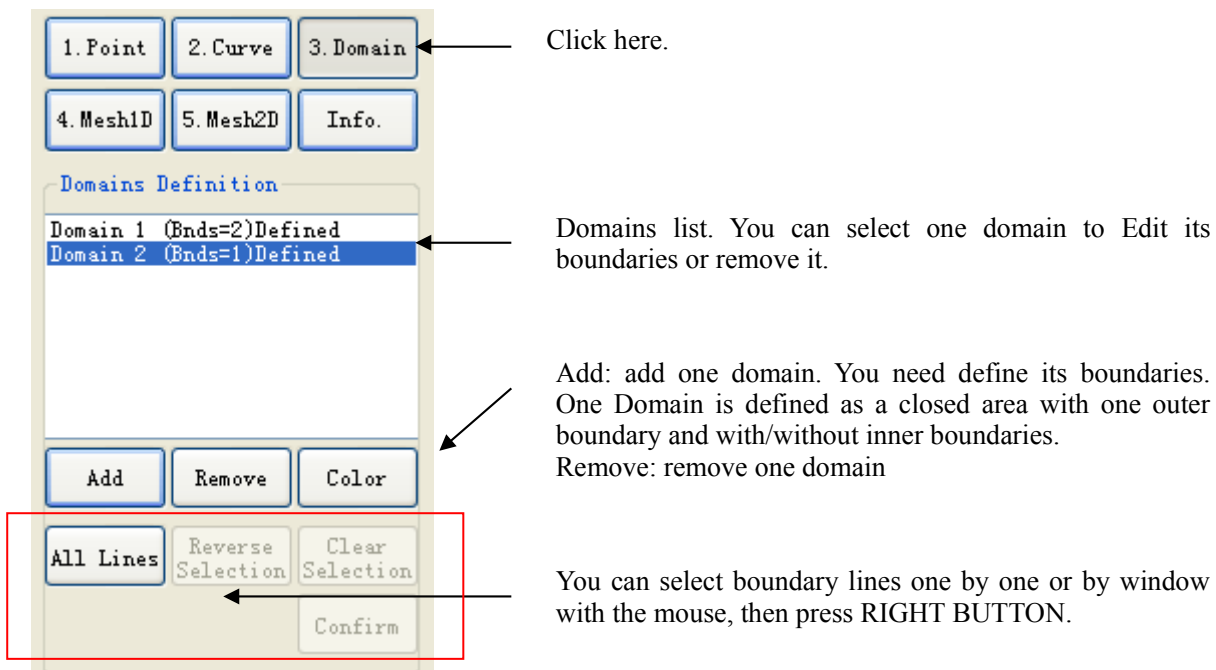


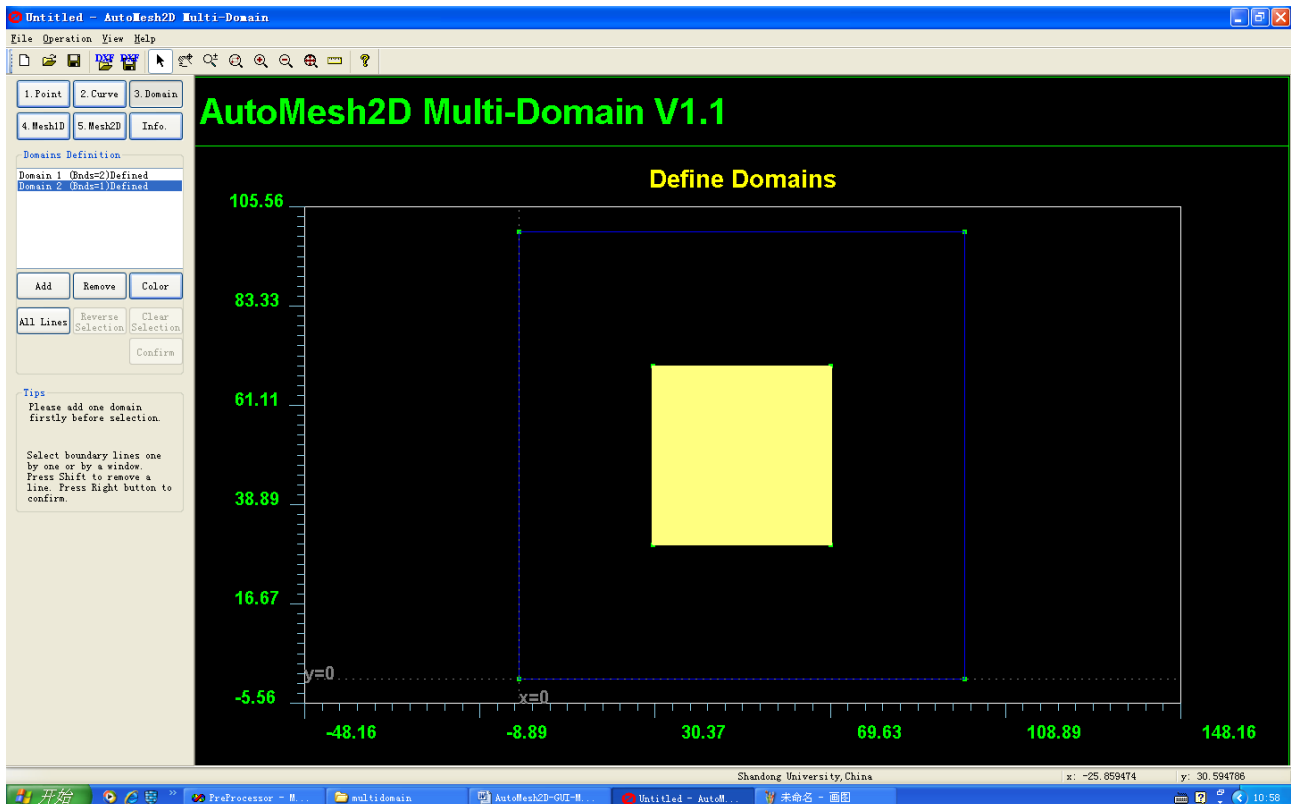


Step 3 Define Domains

Click on the button **Domains** on the dialog.

We will define two domains. First click on the button **Add** to add a new domain. Select all the lines with the mouse. After that, press the RIGHT BUTTON and the first domain is defined. Then define the second domain. Click on the button **Add** to add a new domain. Select four lines of the second domain boundary. After that, press the RIGHT BUTTON and the second domain is defined.



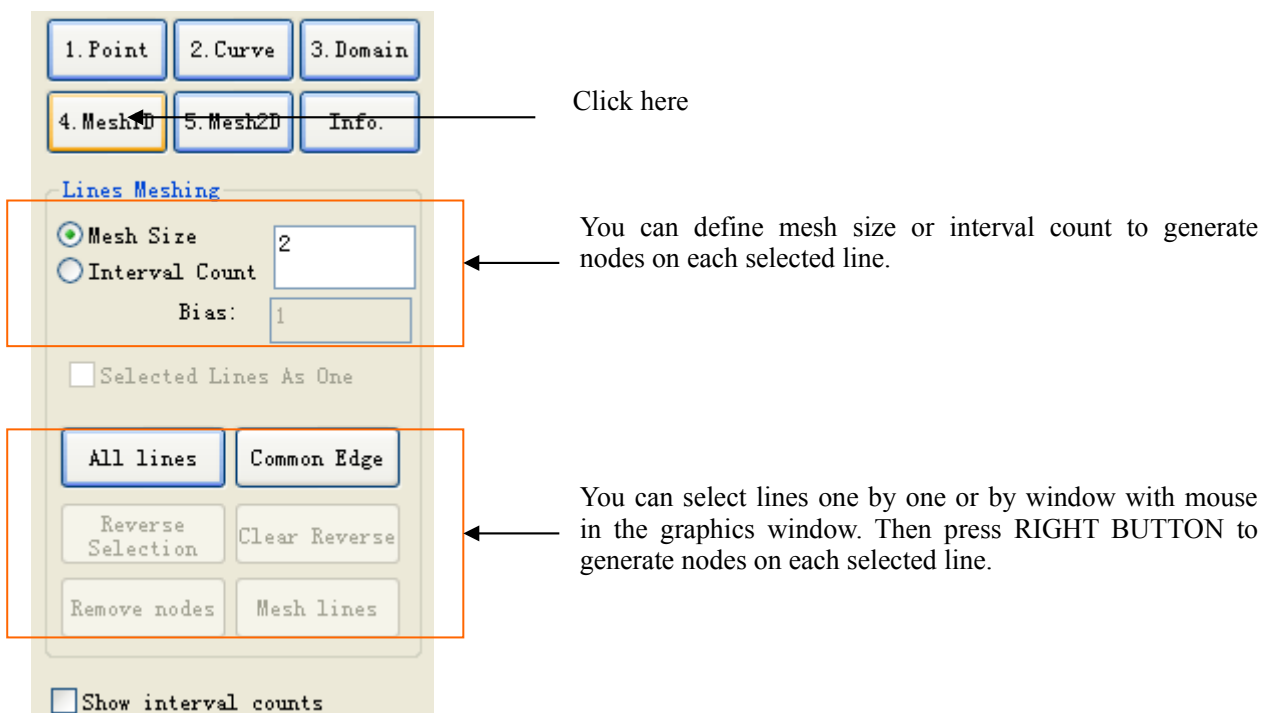


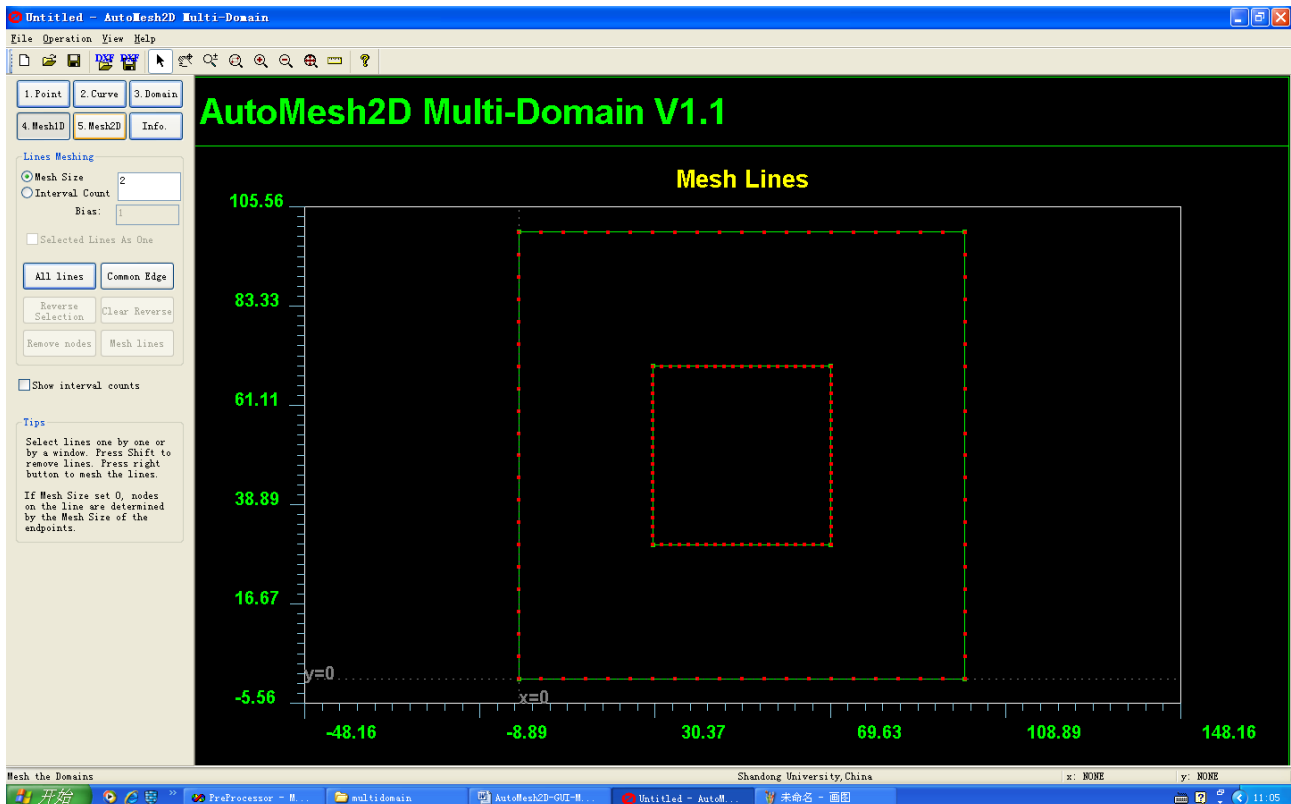
Step 4 Mesh Lines

Click on the button **Mesh1D** on the dialog.

Input the mesh size or the number of the nodes on every line. (Set mesh size 5 and 2 for external an internal boundary lines respectively.)

Select the lines with the mouse to be meshed. After that, press the RIGHT BUTTON to mesh the selected lines.

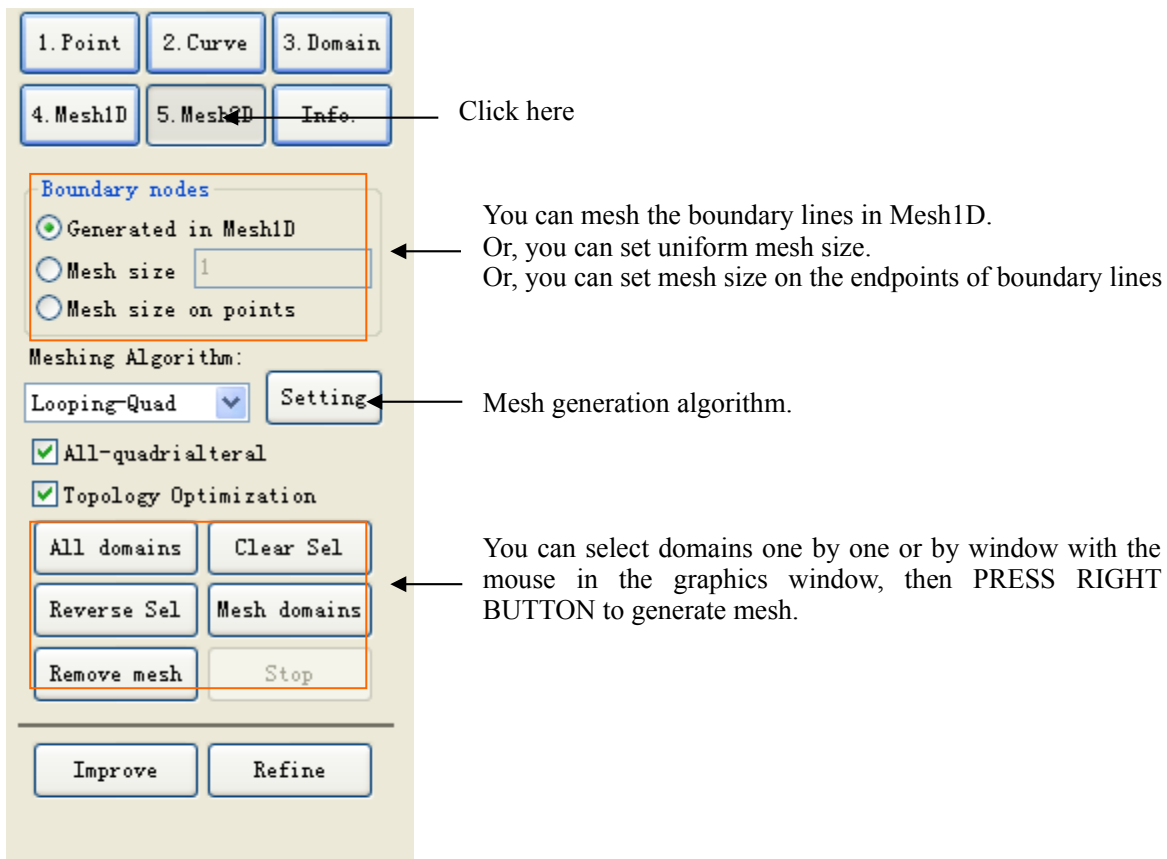


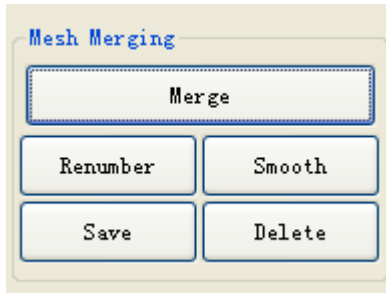


Step 5 Mesh Domains

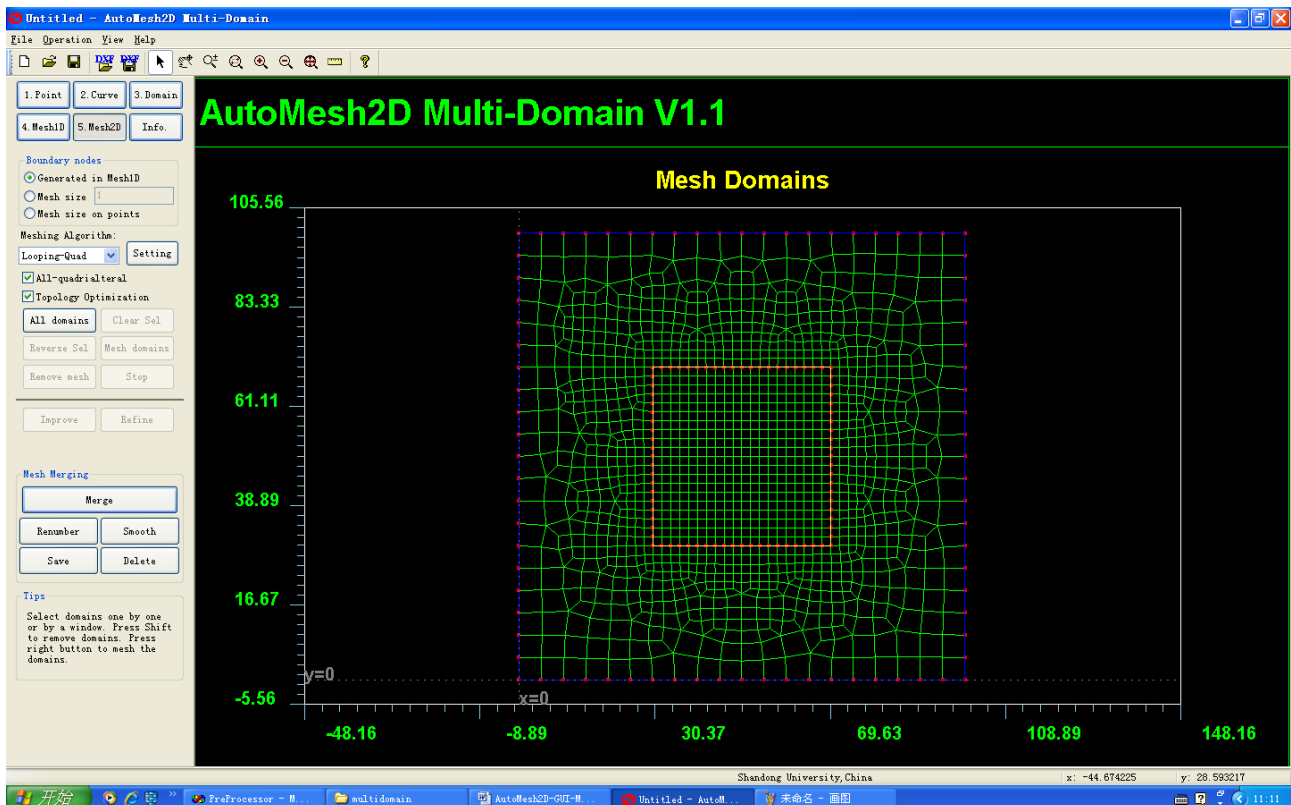
Click on the button **Mesh-2D** on the dialog.

Select the domains with the mouse. After that, press the right button.





Merge: Merge mesh of all domains into one mesh.
 Renumber: Bandwidth Optimization
 Smooth: Laplace Smoothing
 Save: Save the mesh to the file.
 Delete: Delete the merged mesh



Click on the button **Merge** to merge all the mesh.

Click on the button **Save** to write the mesh to a file.

If you have difficulty in using this program or need new function, Please contact me!

If the mesh generation fails, Please send the 'send_to_me.msdx' file to me: maxinwu@sdu.edu.cn . Thanks!

Please refer this if you publish your paper or thesis:

Ma XW, Zhao GQ, Sun L. AUTOMESH - 2D/3D: robust automatic mesh generator for metal forming simulation. Materials Research Innovations, 2011, 15(s1): s482-s486