Gmsh GUI Tutorial I: How to Create a Simple 2D Model?
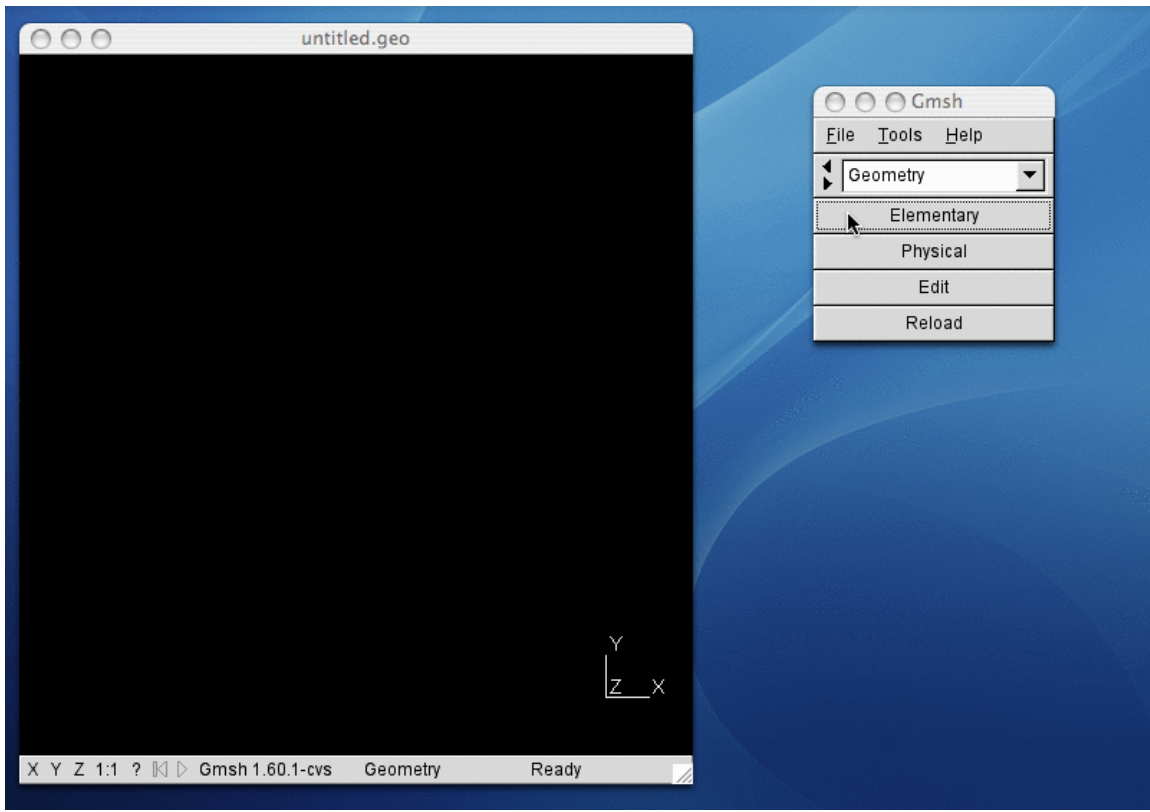
Christophe Geuzaine and Jean-François Remacle
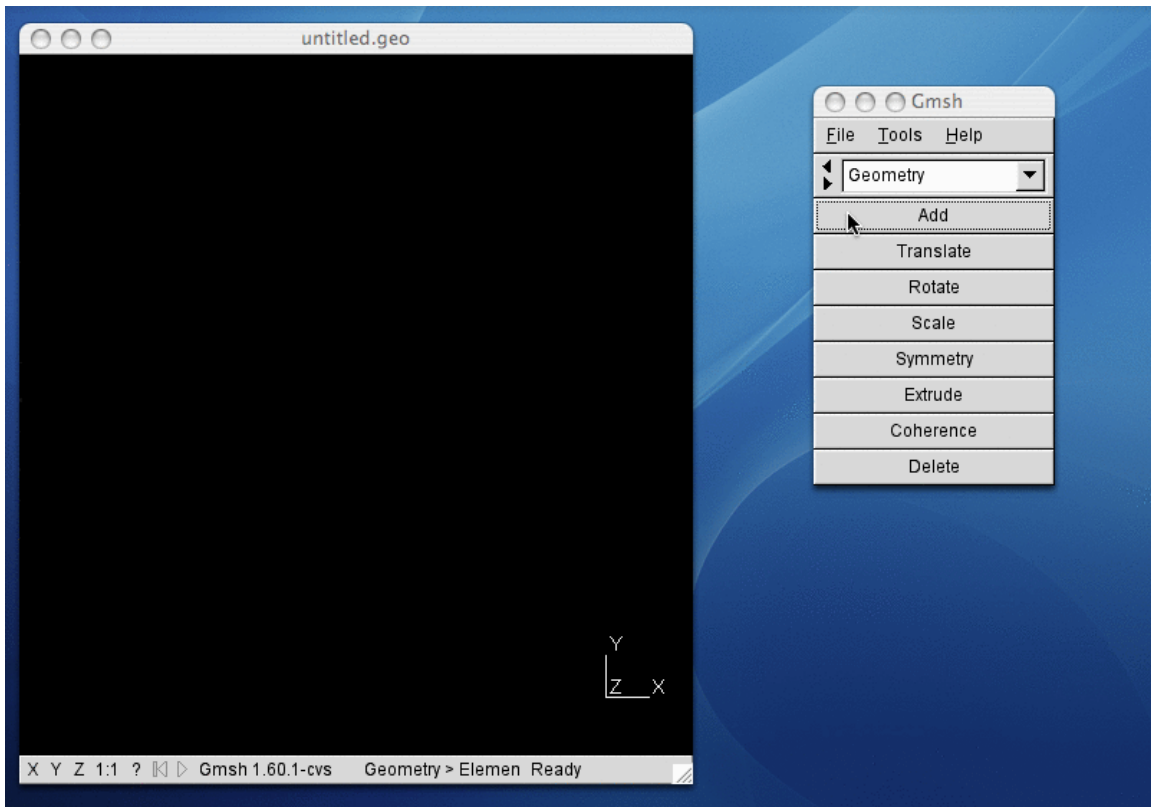
January 4, 2006

This tutorial shows all the steps involved in the creation of a simple two-dimensional geometry using Gmsh's graphical user interface.

We start by launching Gmsh, then we click on 'Elementary' to create new elementary geometrical entities.

We click on 'Add' to add an entity. This is the only sensible choice since there are no other elementary entities to manipulate at the moment.
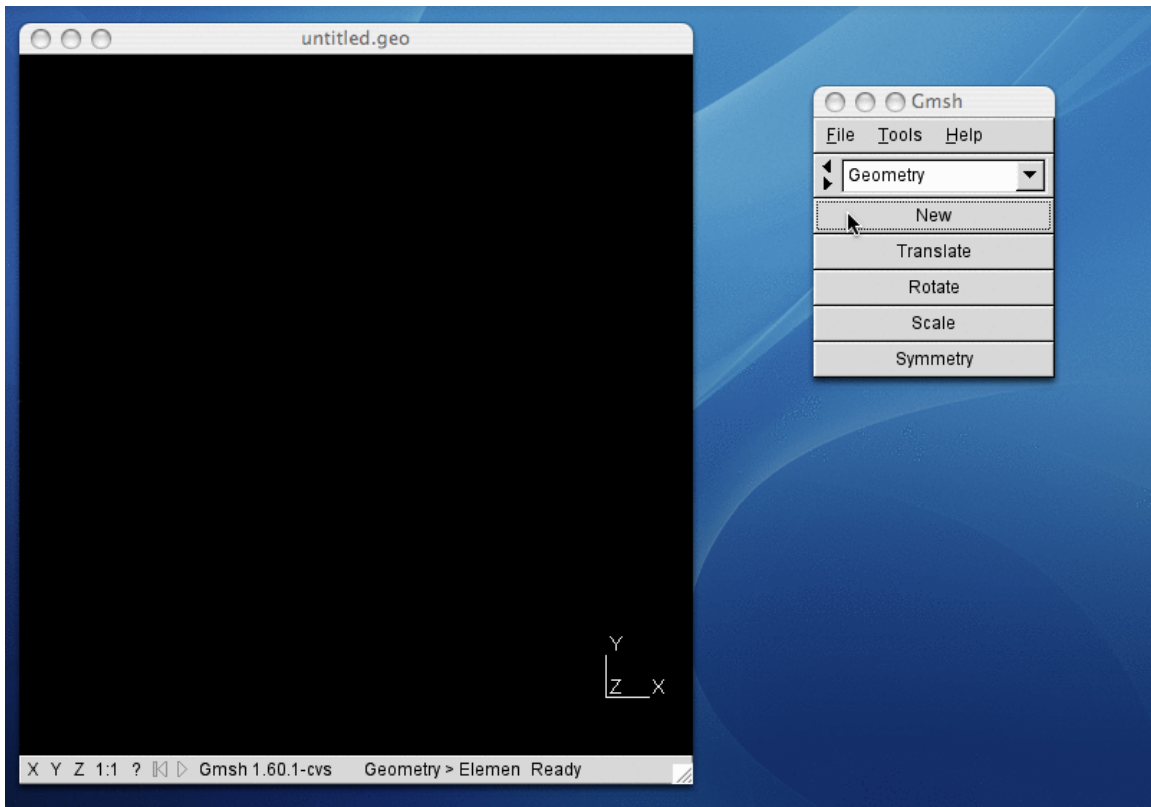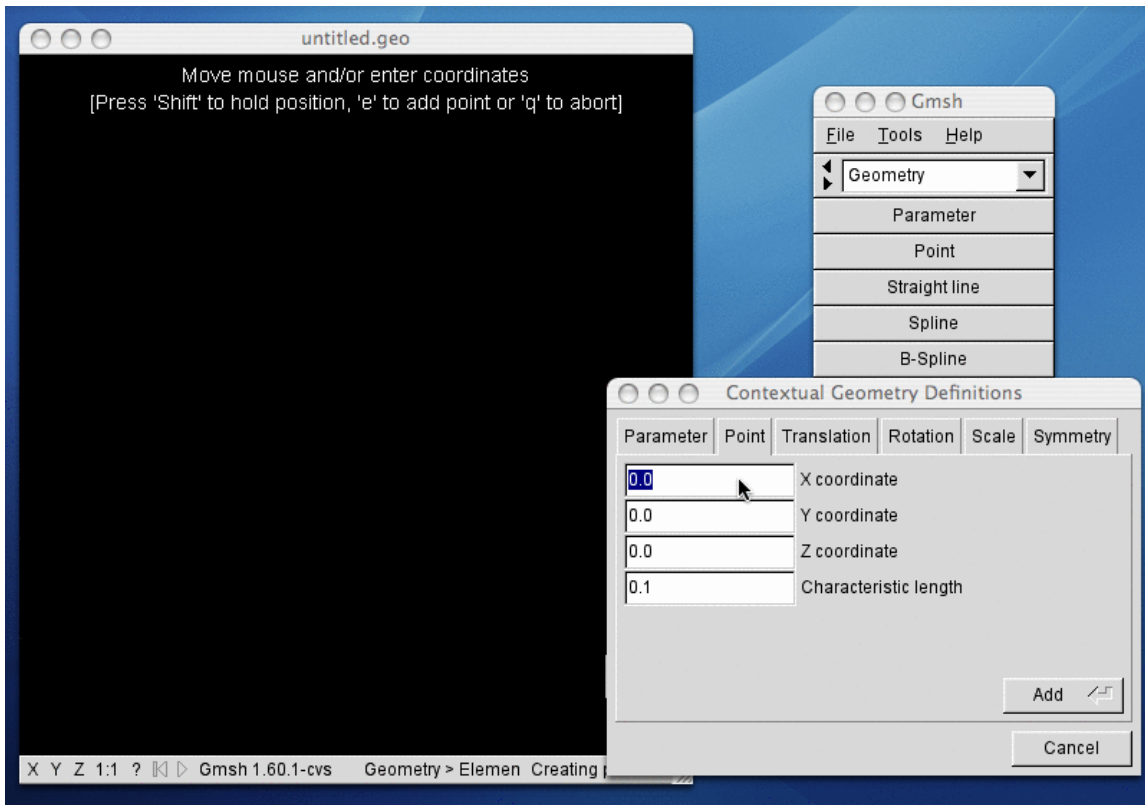
We click on 'New' to add a new entity. If other entities were present, we could transform them using the 'Translate', 'Rotate', ..., buttons.
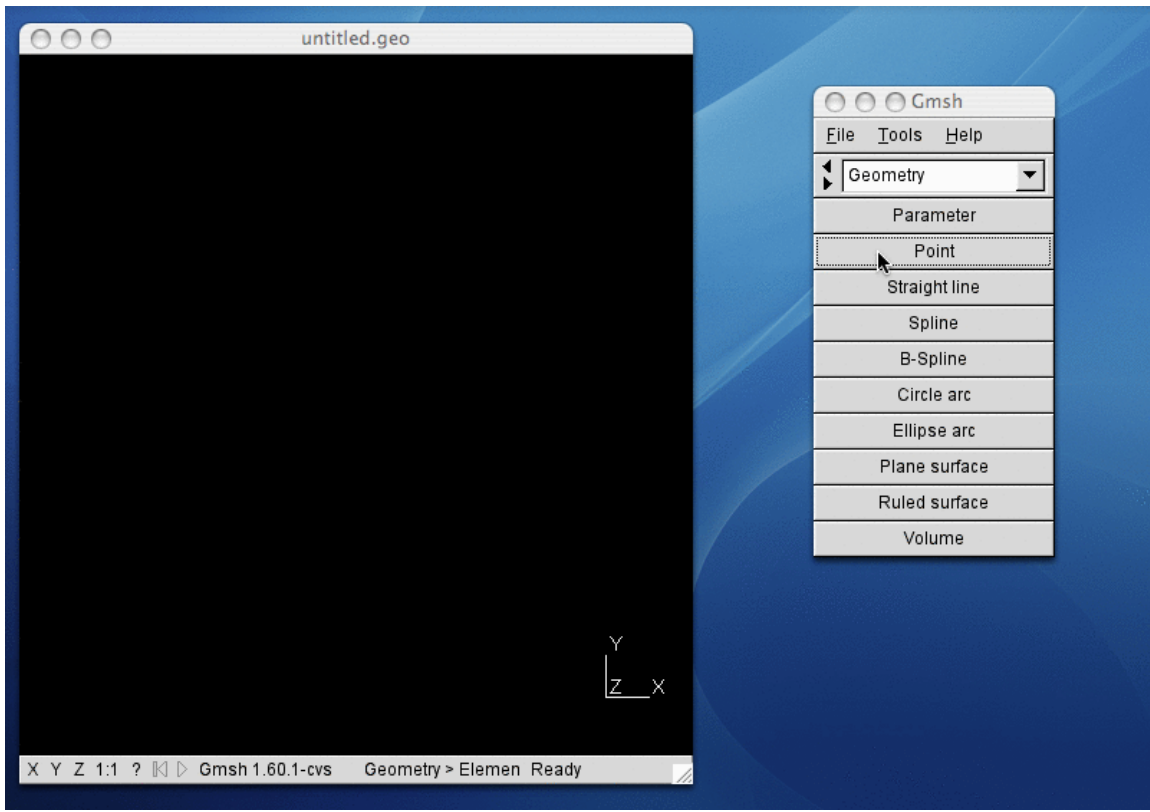
Gmsh's CAD modeler works by using boundary representations of geometrical entities, so if we want to define a 2D model (i.e., surfaces) we first need to create points and curves.

So we start by clicking on 'Point' to add new points.

A contextual dialog appears, where we can directly enter the X, Y, Z coordinates of a new point (as well as the target mesh size at this point).

Later we will see that, instead of entering the point coordinates by hand, we can also create points using the mouse.
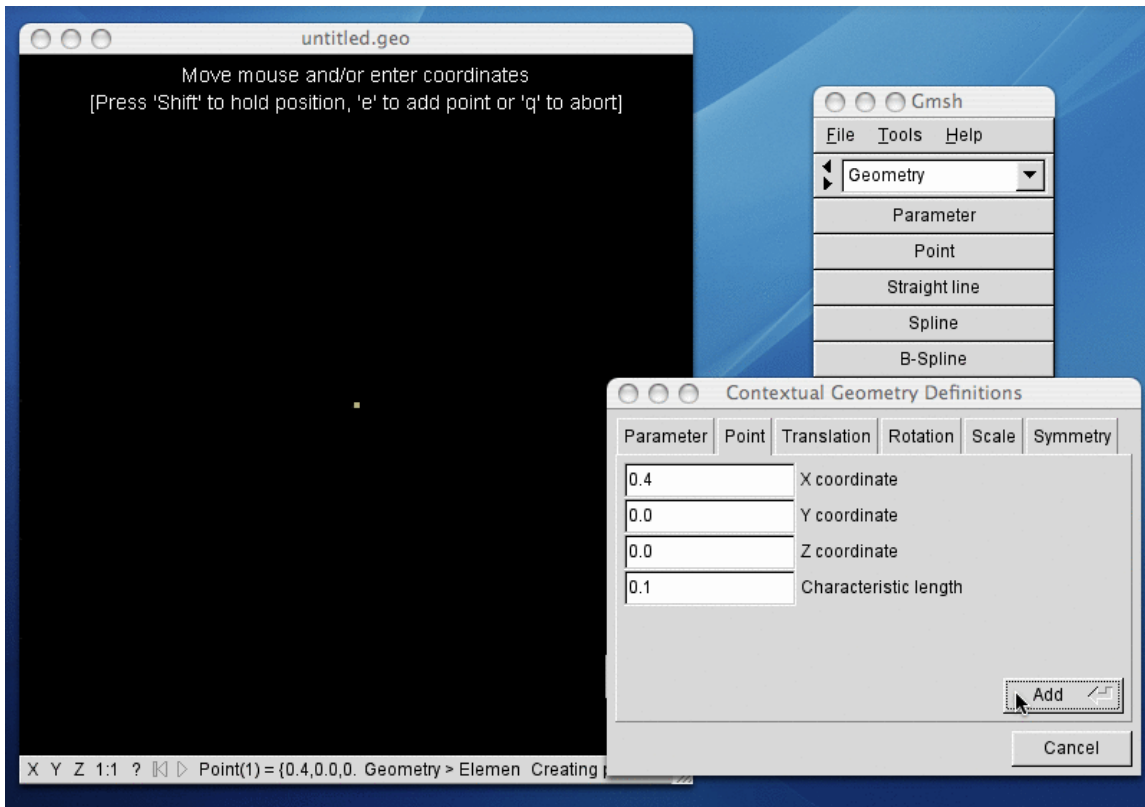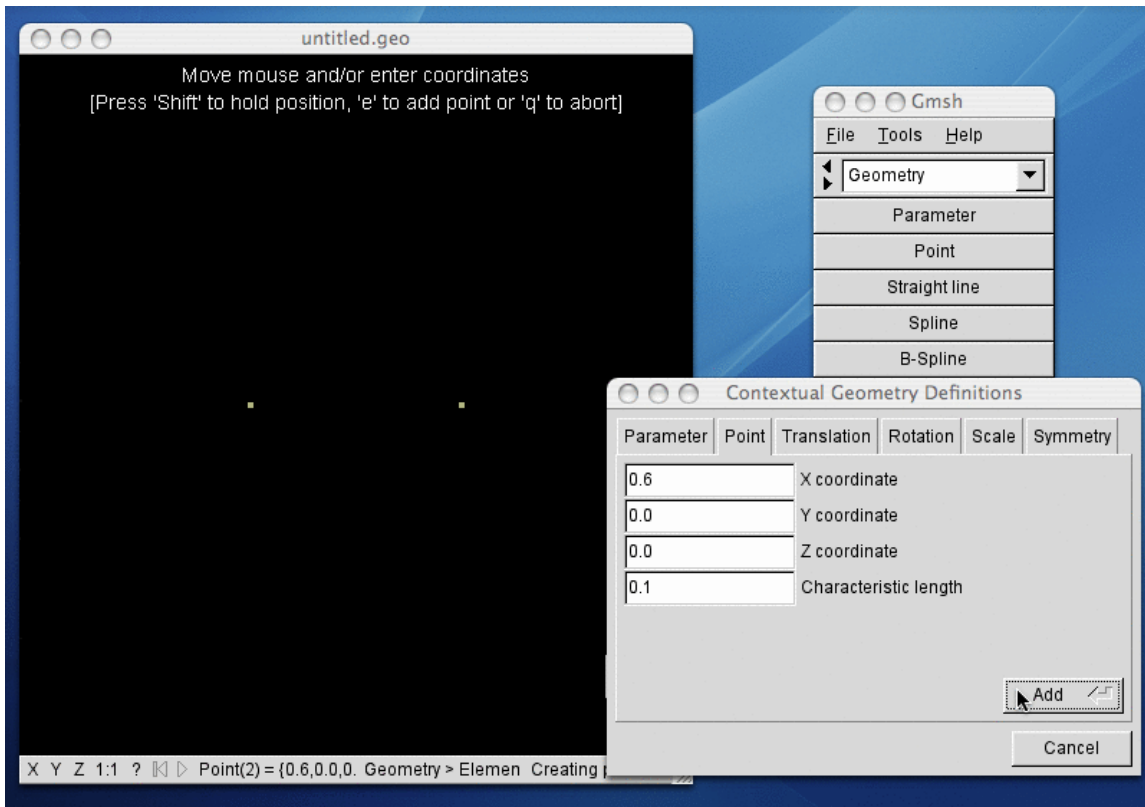
We want to create our first point at the position (0.4, 0.0, 0.0), with a target mesh characteristic length of 0.1.

So we enter X=0.4, Y=0.0 and Z=0.0 and we click on 'Add': this creates our first point.
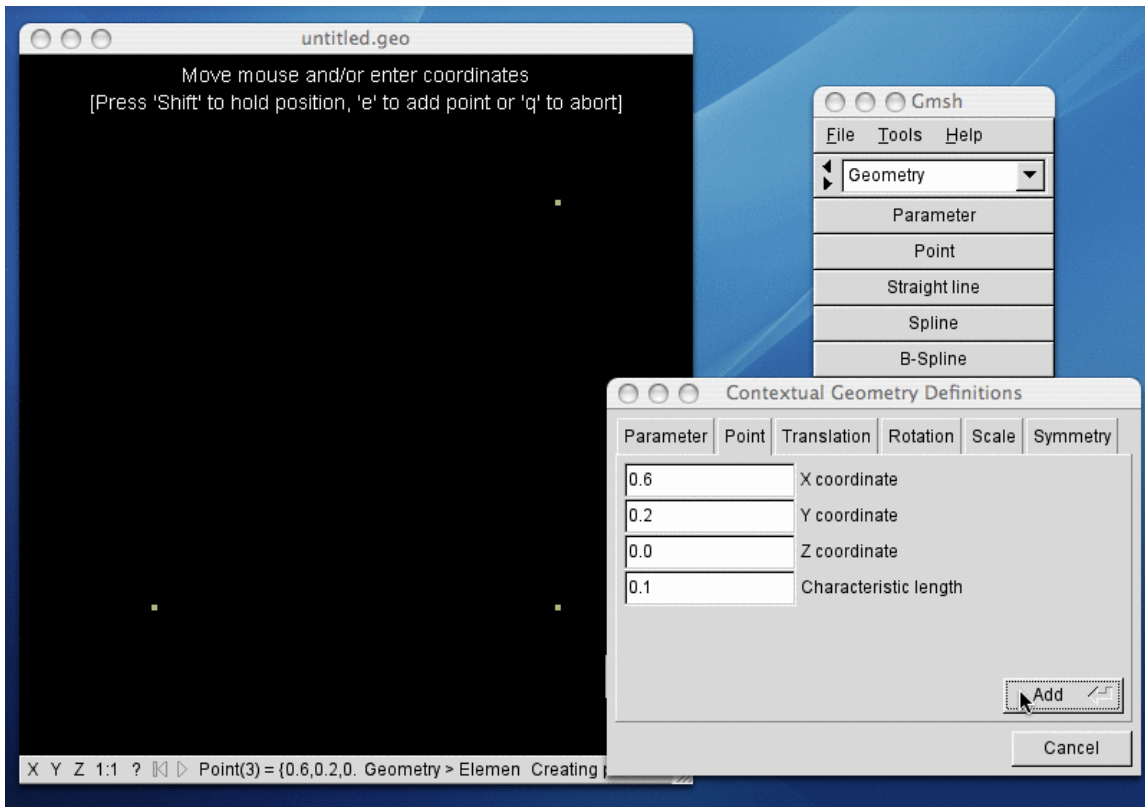
If we opened the project file ('untitled.geo') with a text editor, we would see that Gmsh has added the line

```
Point(1) = {0.4,0.0,0.0,0.1};
```
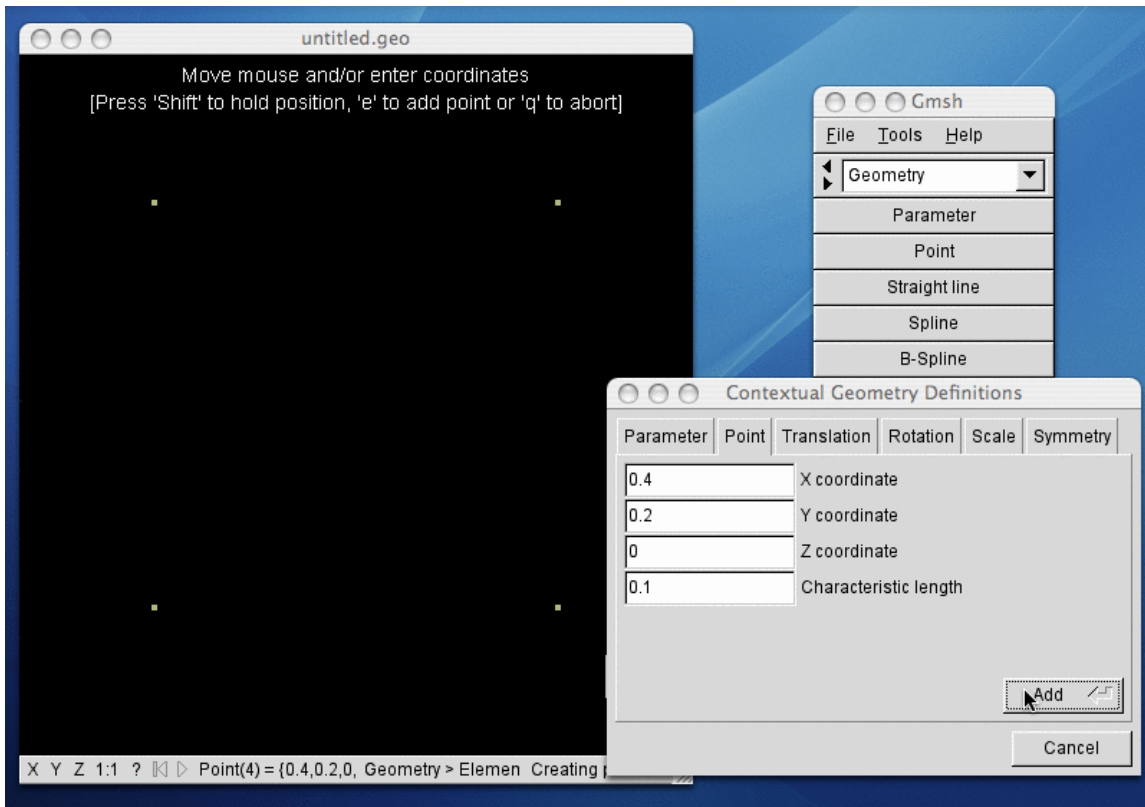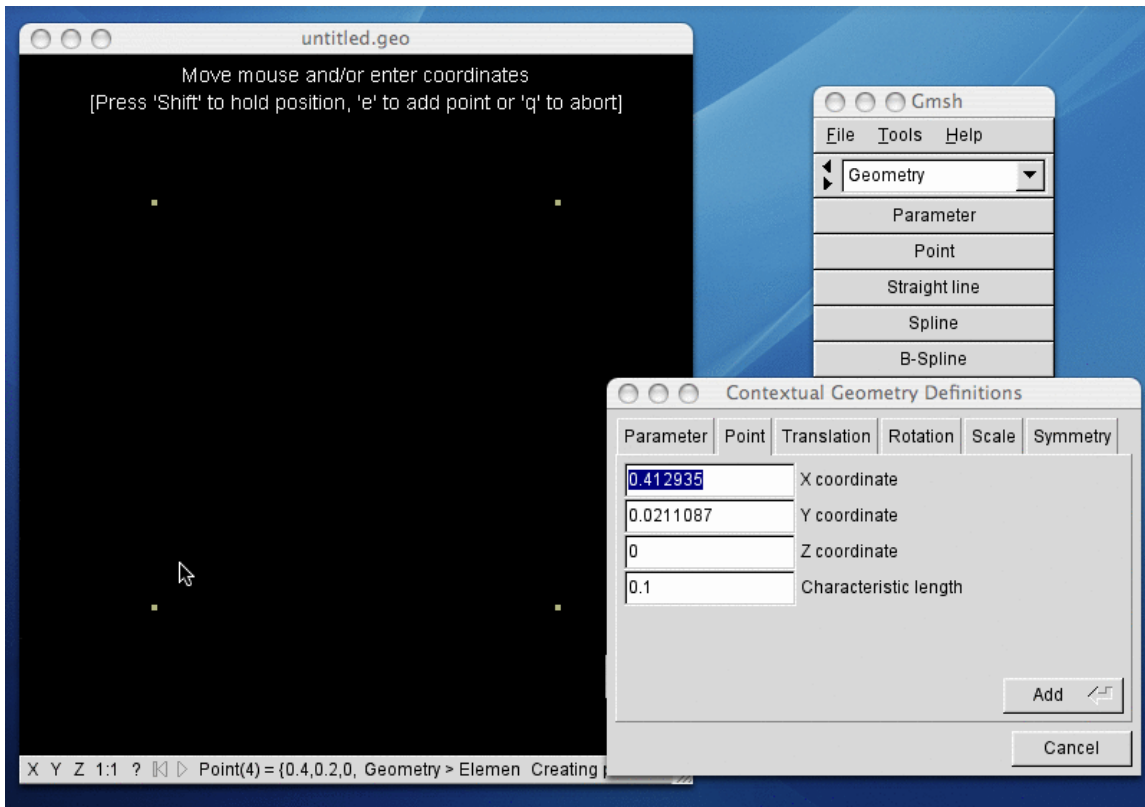
at the end of the file.

We add a second point at (X,Y,Z) = (0.6,0.0,0.0).
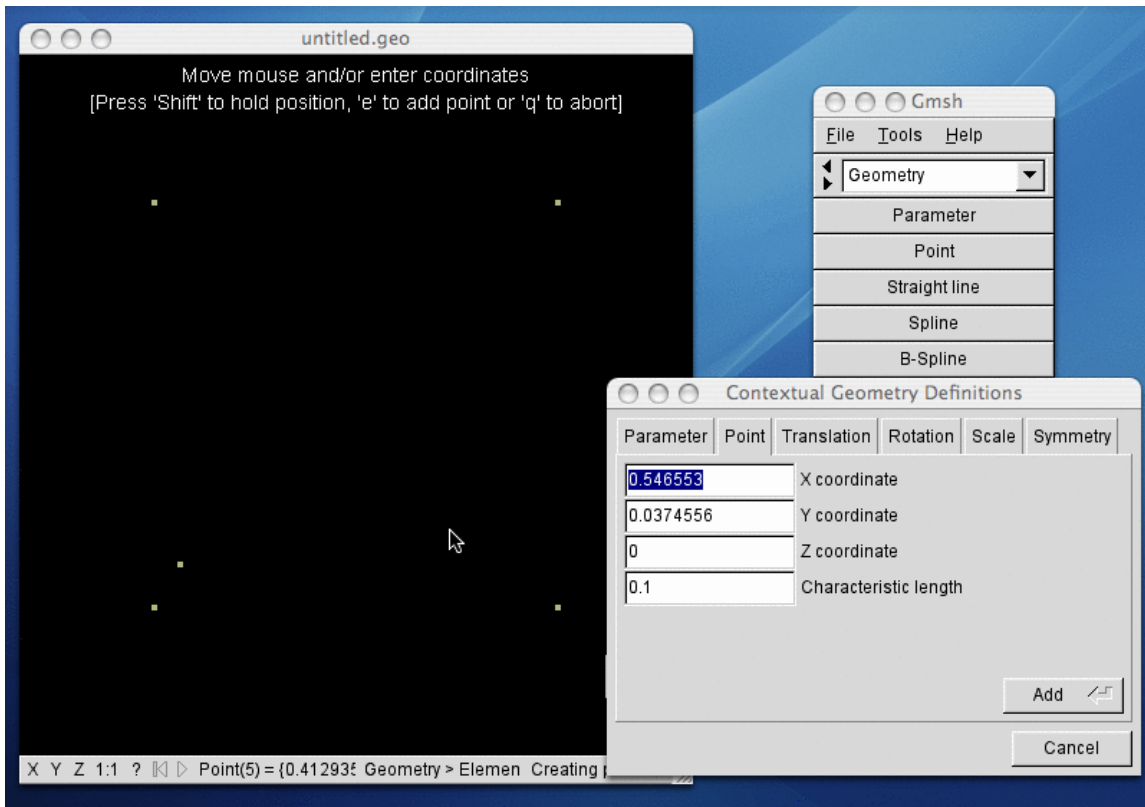
We add a third point at (X,Y,Z) = (0.6,0.2,0.0).

We add a fourth point at (X,Y,Z) = (0.4,0.2,0.0).

Instead of entering the point coordinates in the dialog window, we can also create points directly with the mouse.

For this, we just position the mouse at the desired location in the graphic window, and we press 'e' when we are done. This adds the fifth point in our model.

We also add the sixth point using the mouse. To fine-tune the position, we can hold the 'Shift' key and edit the value in the dialog.

We add the points 7, 8, 9 and 10 in the same way, then press 'q'.
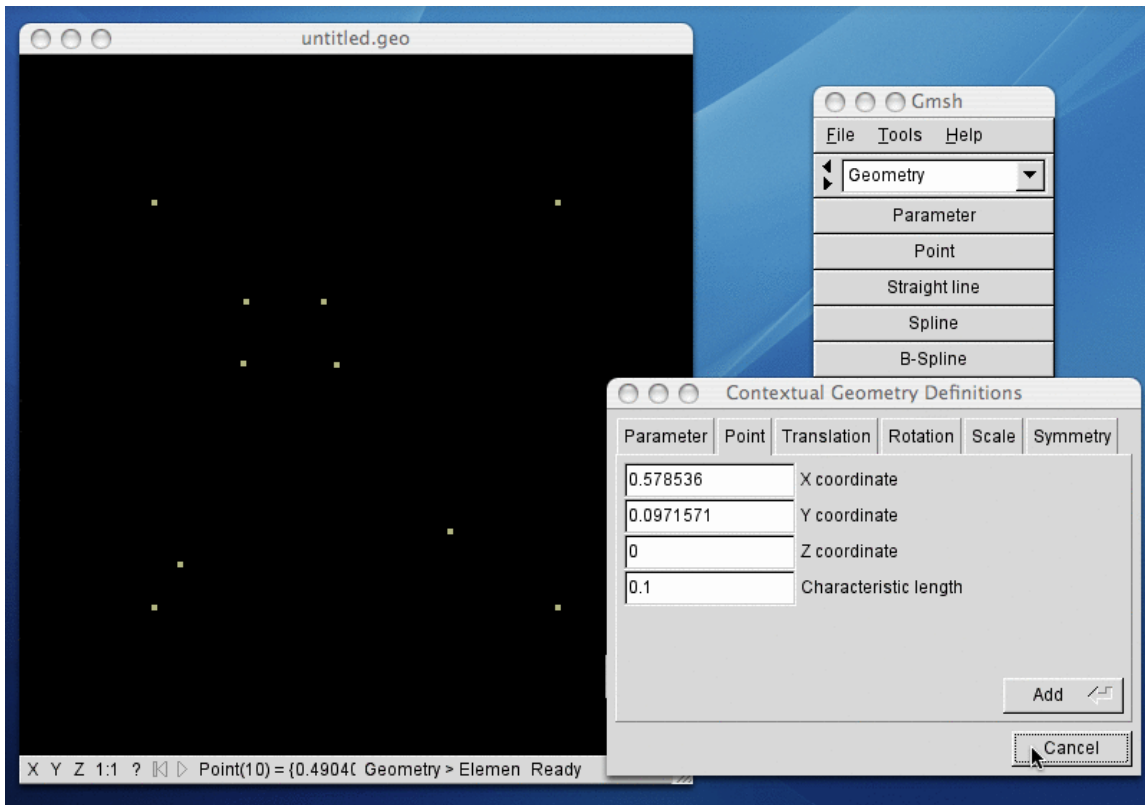
If we opened the project file with a text editor, we would see that the file now contains the definition of all ten points:

```
Point(1)  = {0.4,0.0,0.0,0.1};
Point(2)  = {0.6,0.0,0.0,0.1};
Point(3)  = {0.6,0.2,0.0,0.1};
Point(4)  = {0.4,0.2,0,0.1};
Point(5)  = {0.412935,0.0211087,0,0.1};
Point(6)  = {0.546553,0.0374556,0,0.1};
Point(7)  = {0.445629,0.151173,0,0.1};
Point(8)  = {0.484009,0.151173,0,0.1};
Point(9)  = {0.444208,0.120611,0,0.1};
Point(10) = {0.490405,0.1199,0,0.1};
```

We are done dealing with points for now, so we can close the dialog.
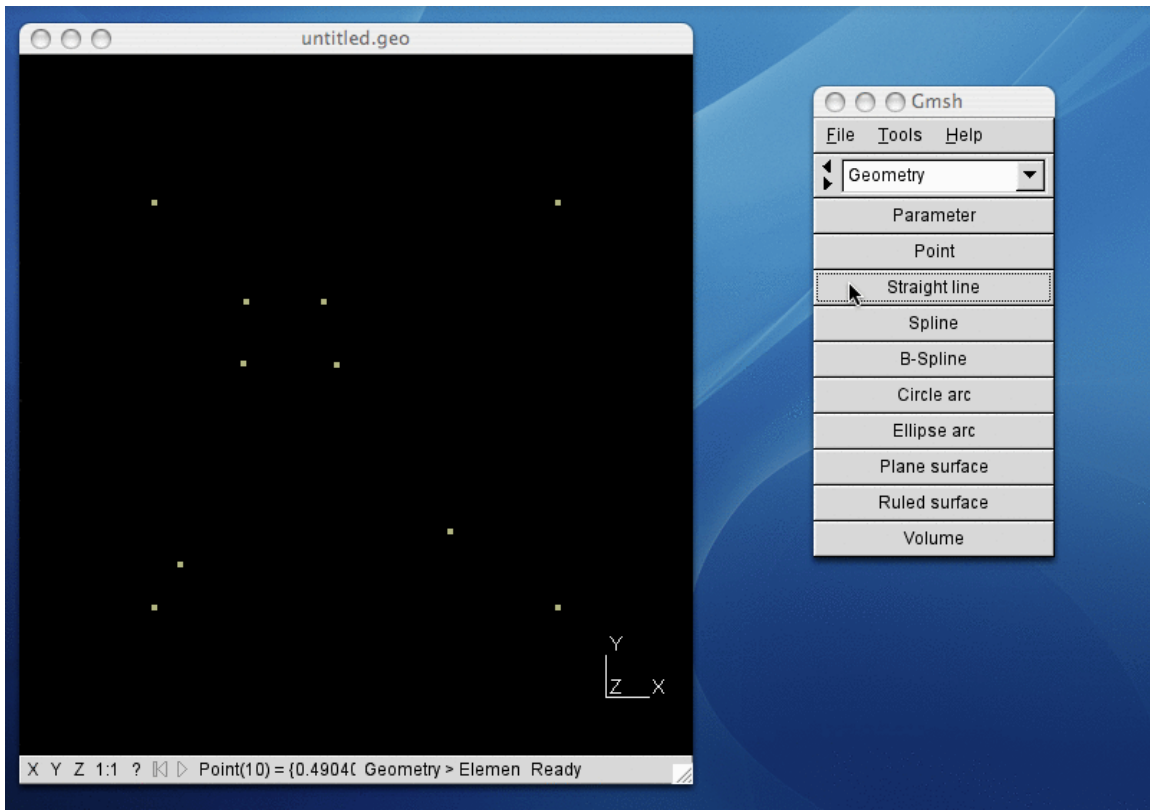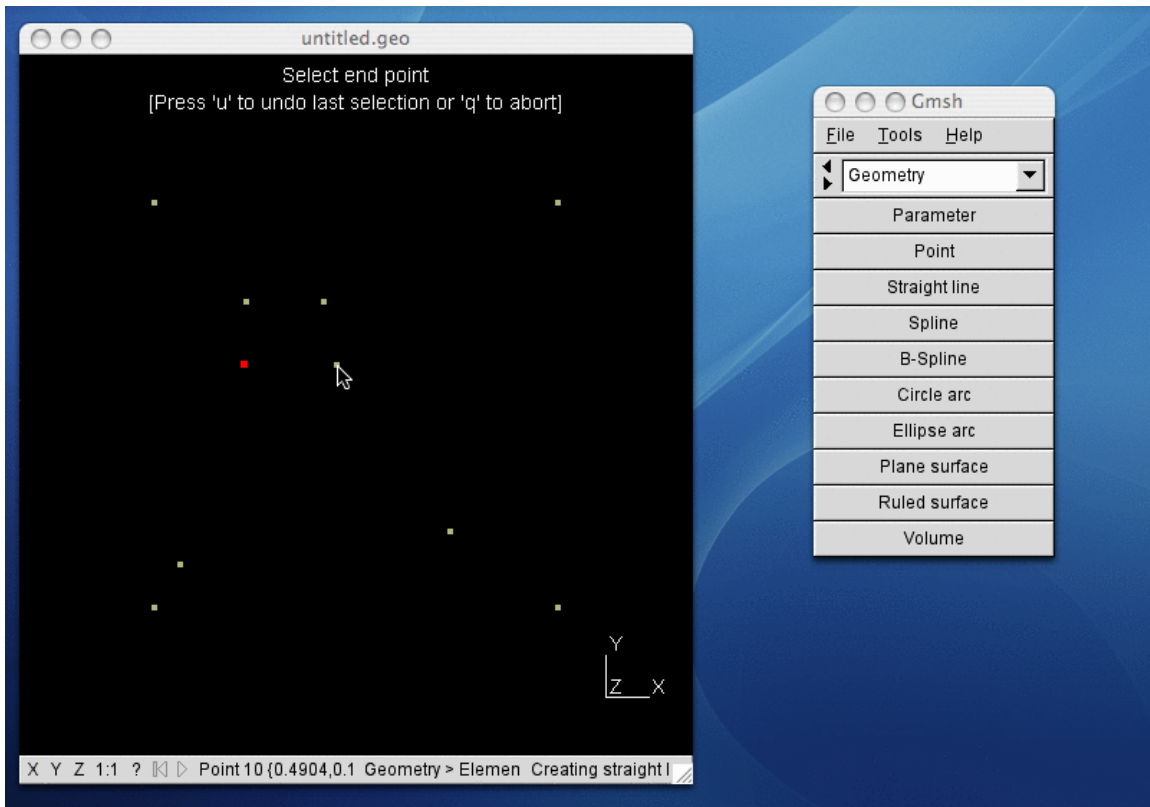
Now we need to create some curves, and we start by creating a series of straight line segments.

In order to create straight lines, we click on the 'Straight line' button.
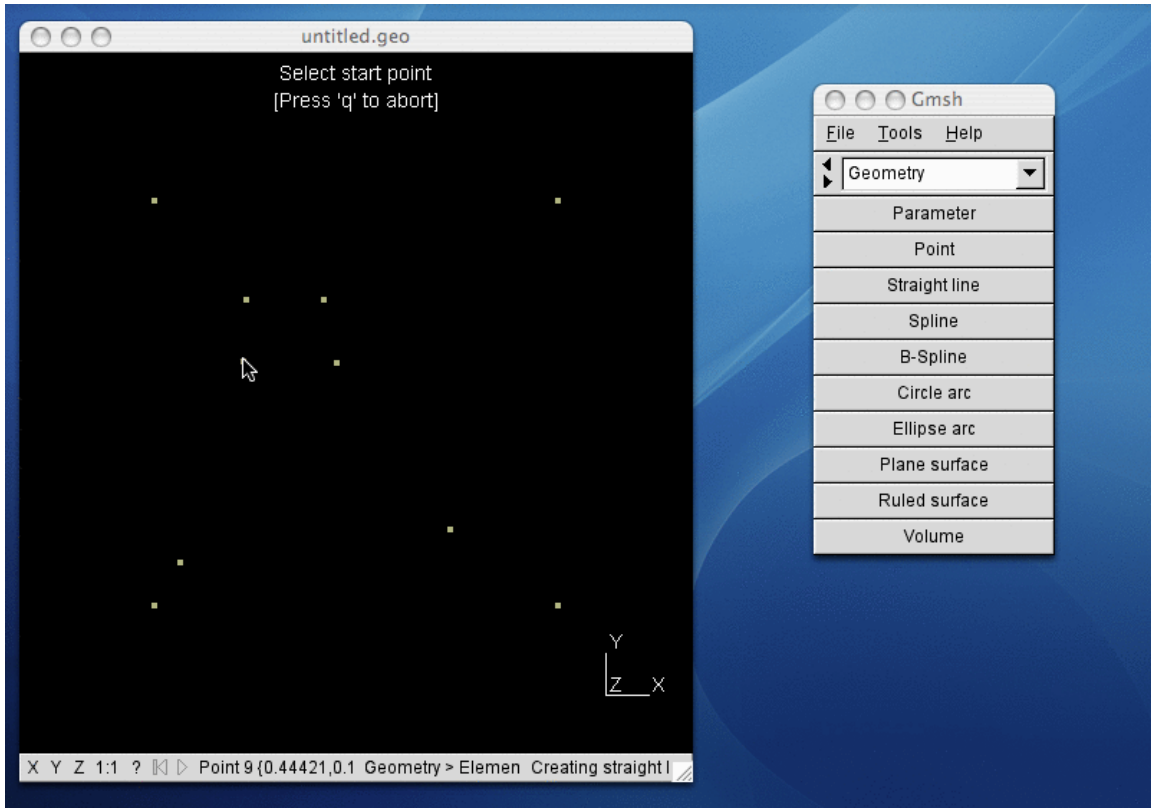
A straight line segment is defined by a start point and an end point. We select the start point by clicking on it (here we click on point 9).

The start point is now highlighted (and we could undo our selection by pressing 'u').

To complete the first line segment, we select the end point (point 10) by clicking on it.
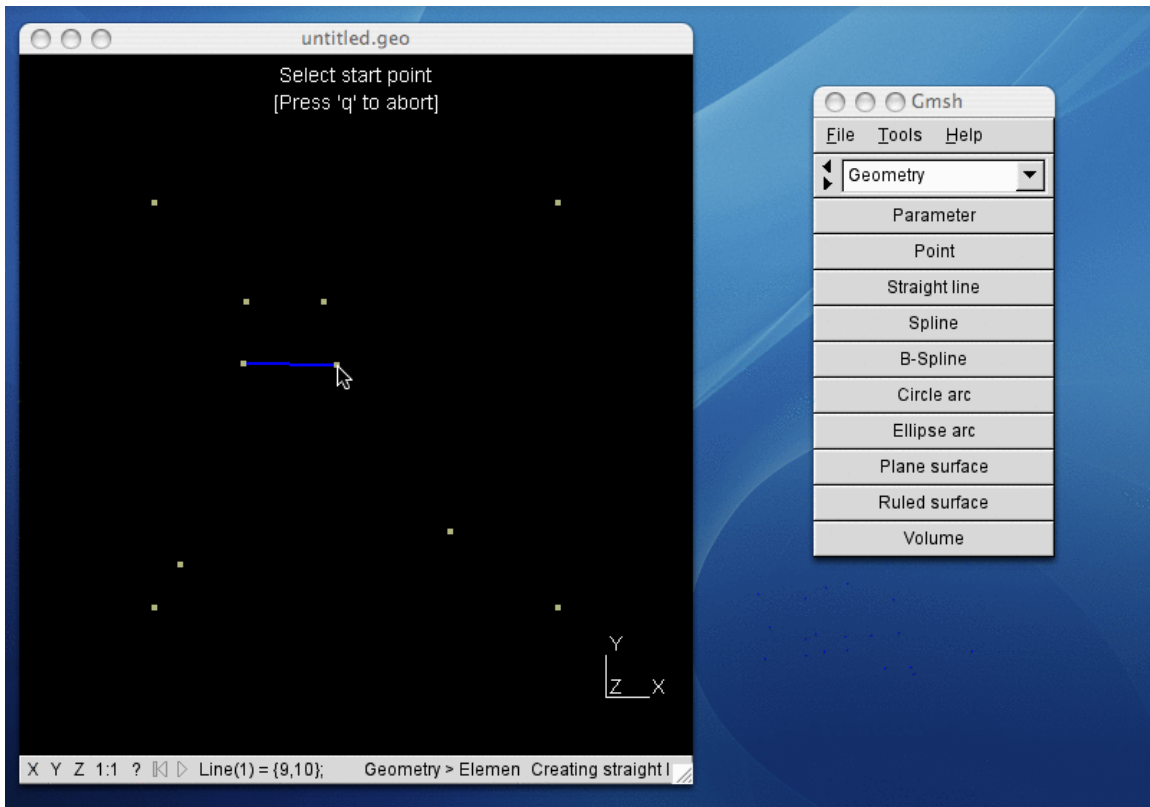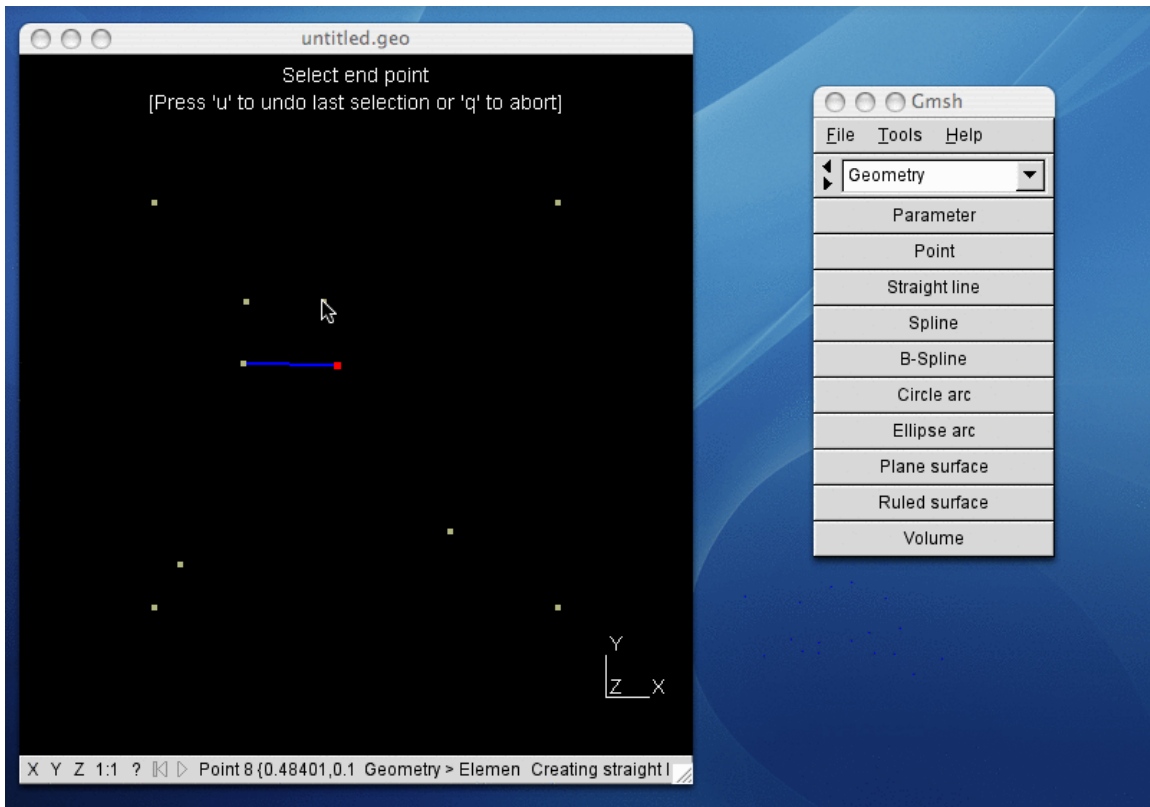
This has the effect to create a new line segment (the first line) that joins the points 9 and 10.
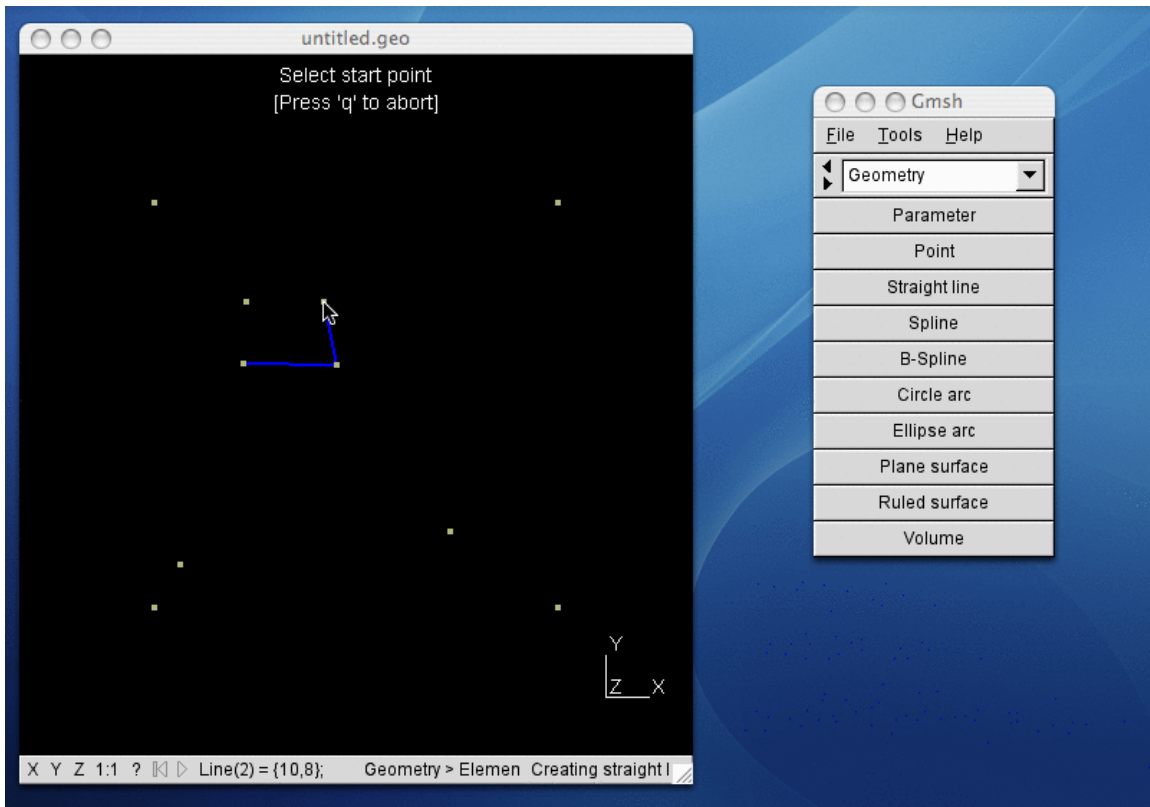
If we opened the project file with a text editor, we would see that Gmsh has added the line

```
Line(1)={9,10};
```
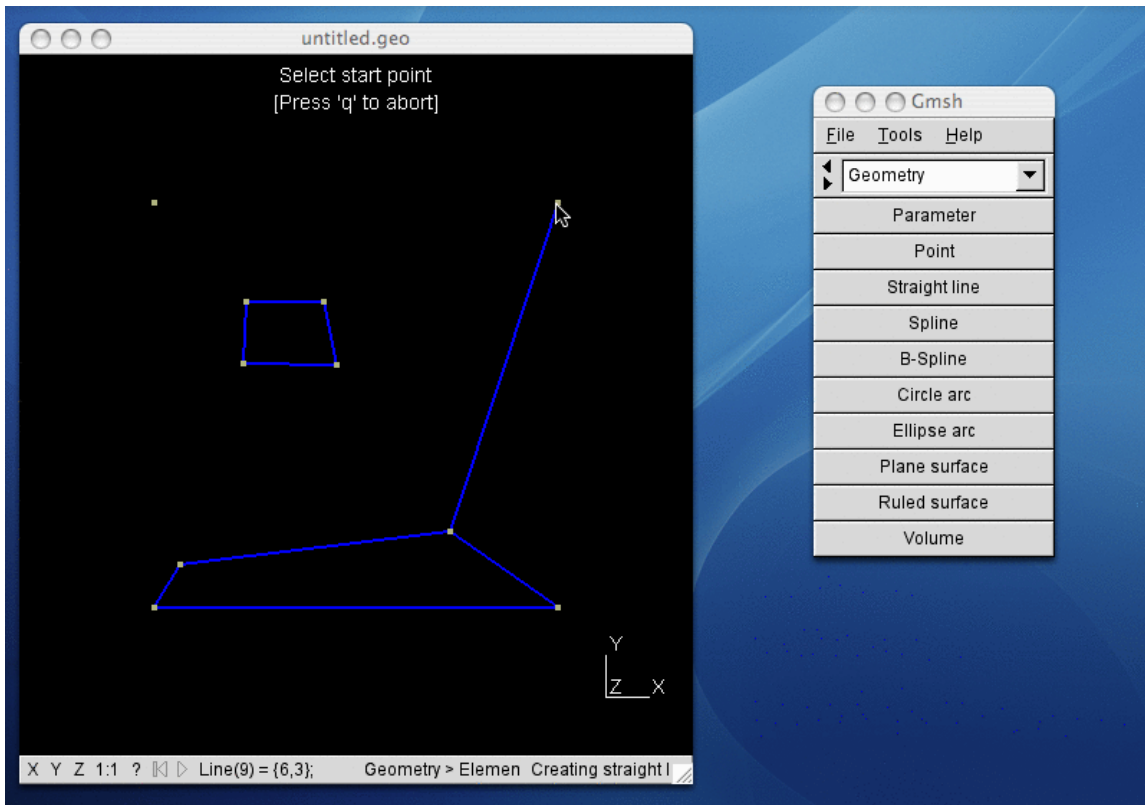
at the end of the file.

We proceed by creating other line segments in exactly the same way, first selecting the start point, then the end point.

The second line segment is created, joining the points 10 and 8.

We proceed by creating seven more line segments, joining the points 8 and 7, 7 and 9, 5 and 6, 1 and 2, 2 and 6, 1 and 5 and, finally, 6 and 3.

We now have 9 straight lines.

If we opened the project file with a text editor, we would see that the file now contains the definition of all ten points and all nine lines:

```
Point(1) = {0.4,0.0,0.0,0.1};
Point(2) = {0.6,0.0,0.0,0.1};
Point(3) = {0.6,0.2,0.0,0.1};
Point(4) = {0.4,0.2,0,0.1};
Point(5) = {0.412935,0.0211087,0,0.1};
Point(6) = {0.546553,0.0374556,0,0.1};
Point(7) = {0.445629,0.151173,0,0.1};
Point(8) = {0.484009,0.151173,0,0.1};
Point(9) = {0.444208,0.120611,0,0.1};
Point(10) = {0.490405,0.1199,0,0.1};
Line(1) = {9,10};
Line(2) = {10,8};
Line(3) = {8,7};
Line(4) = {7,9};
Line(5) = {5,6};
Line(6) = {1,2};
Line(7) = {2,6};
Line(8) = {1,5};
Line(9) = {6,3};
```
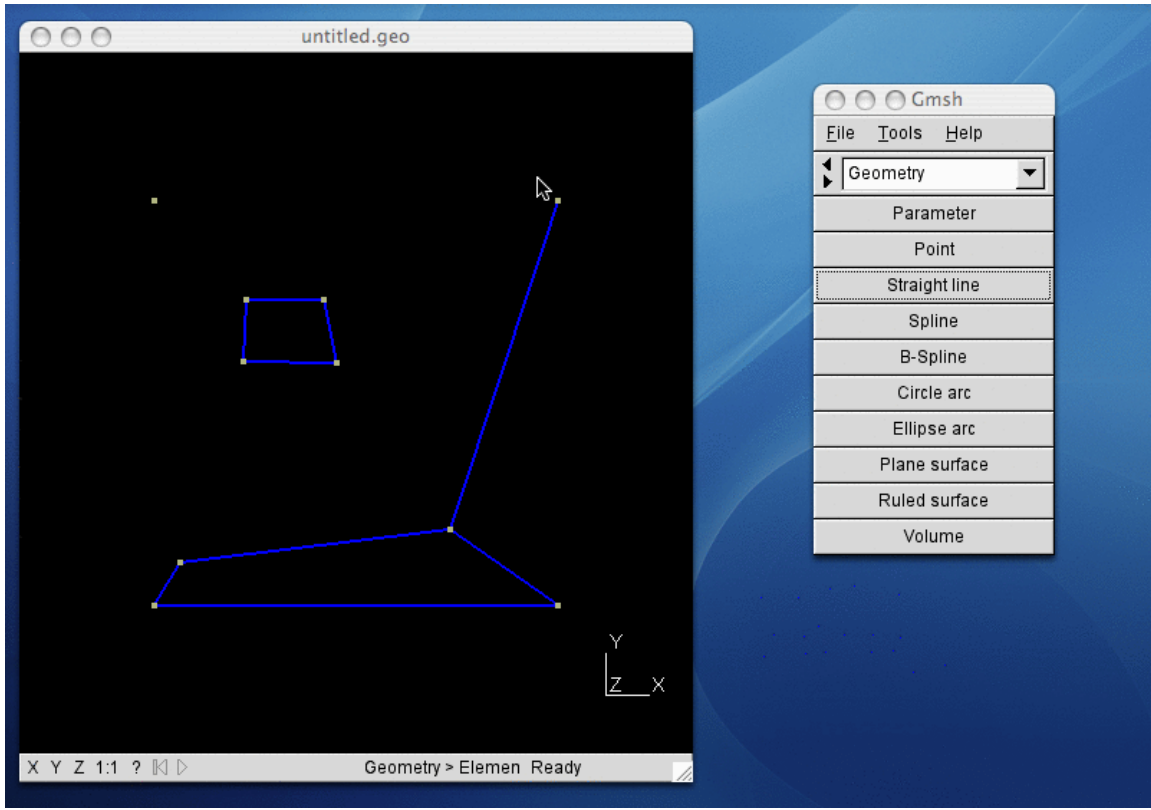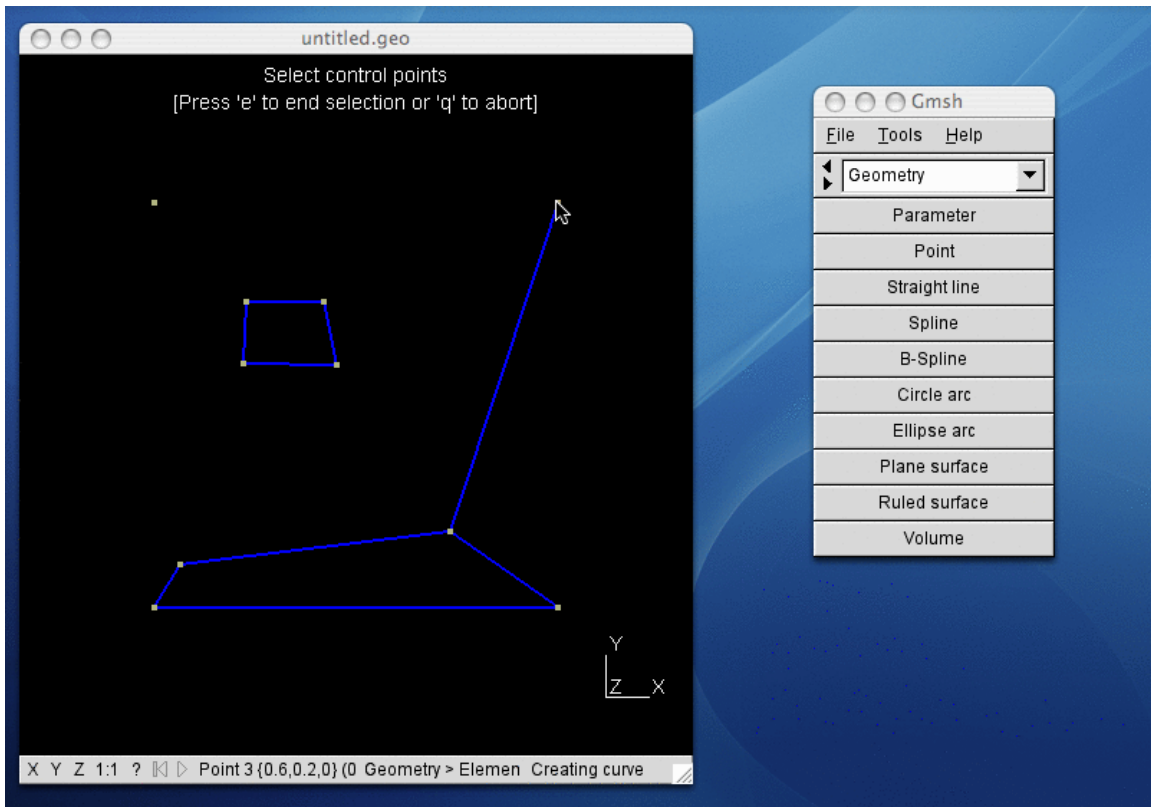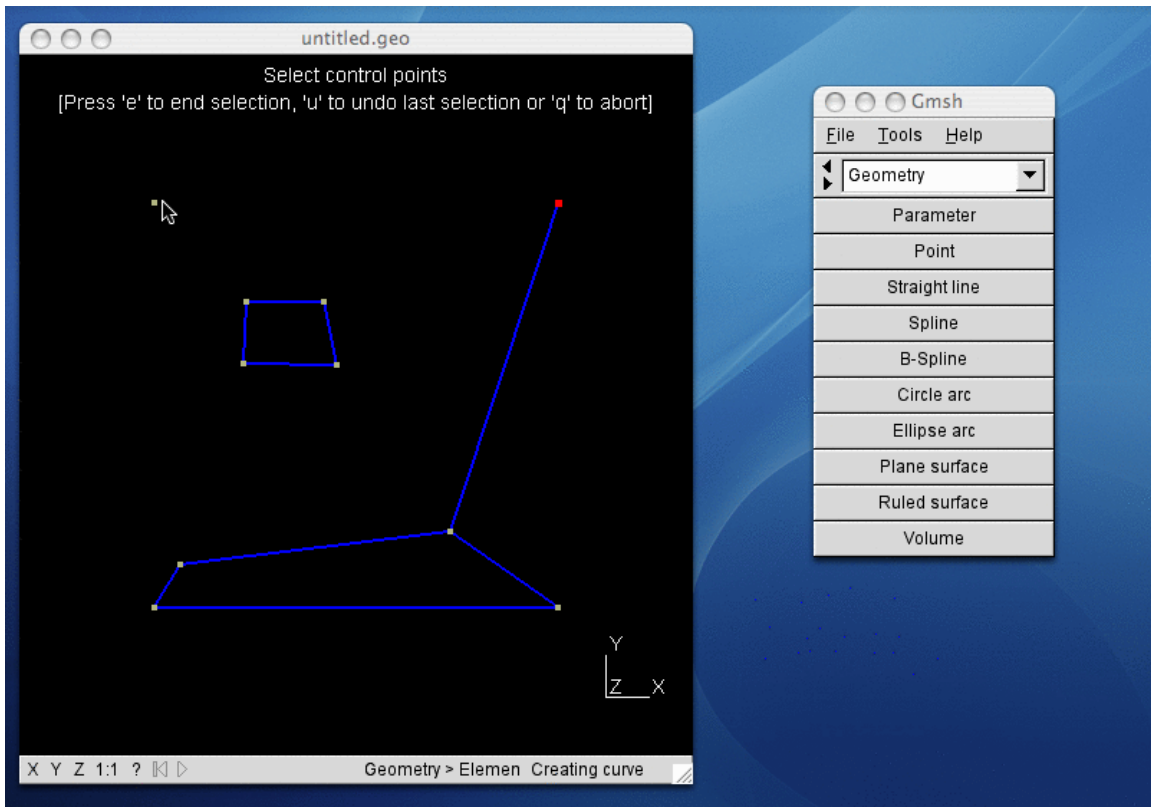
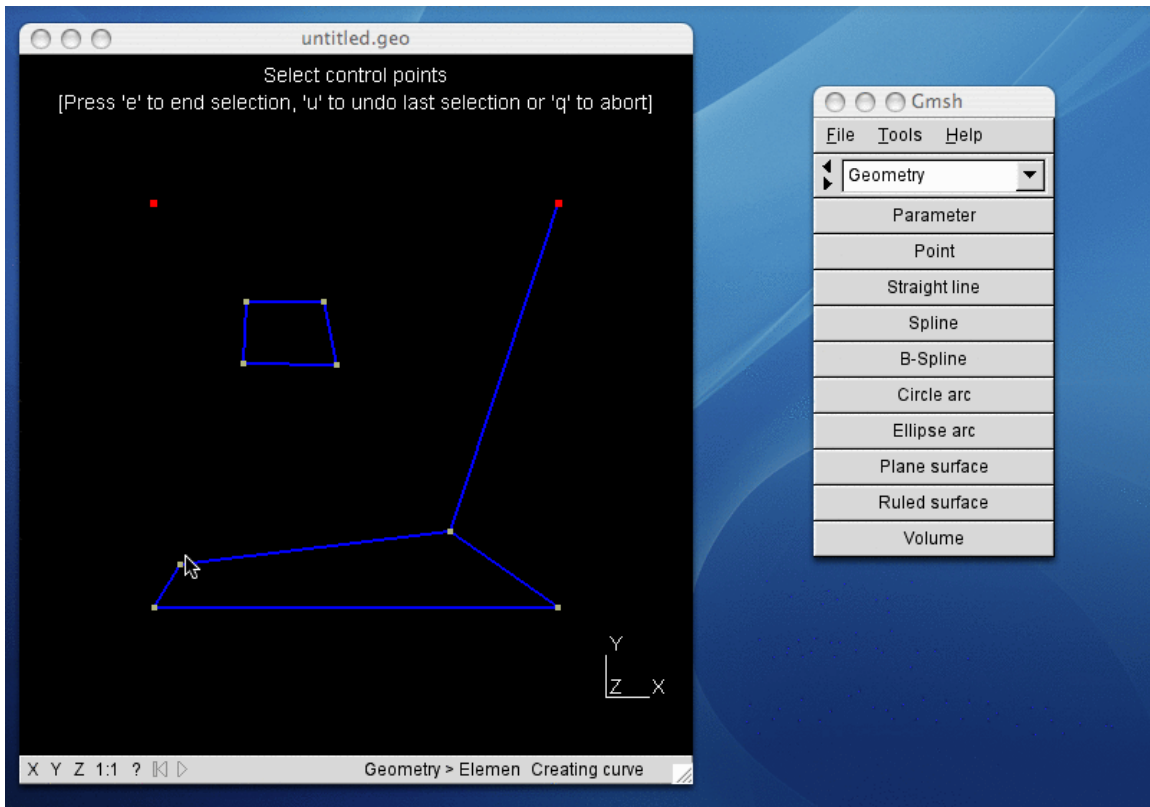Since we are done with the straight lines, we can press 'q'.
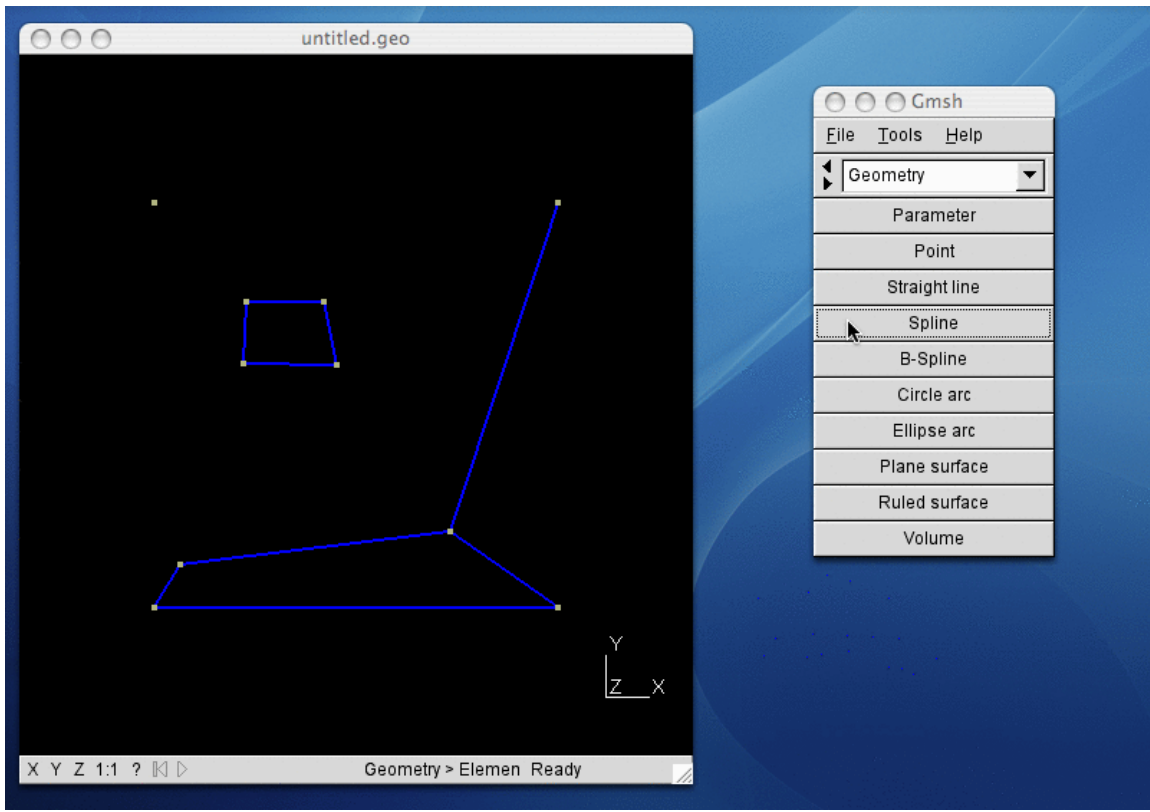
The last curve we want to create is a spline. So we click on the 'Spline' button in the menu.
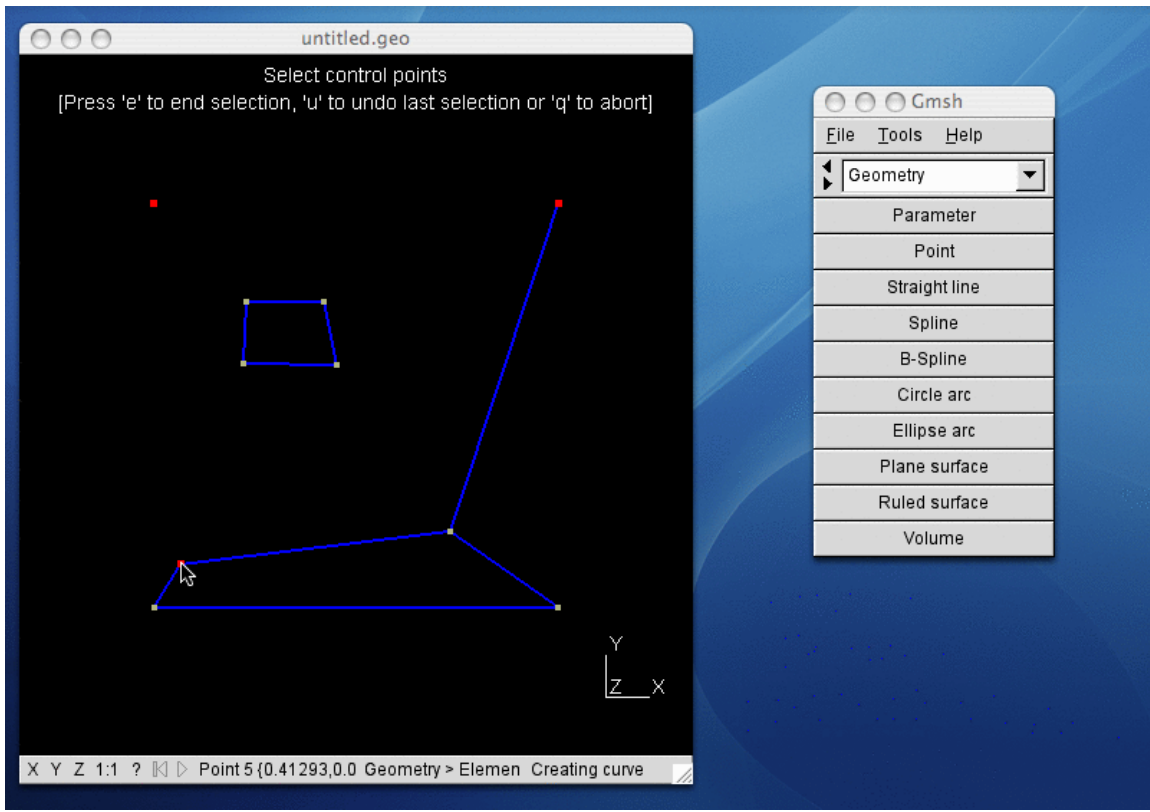
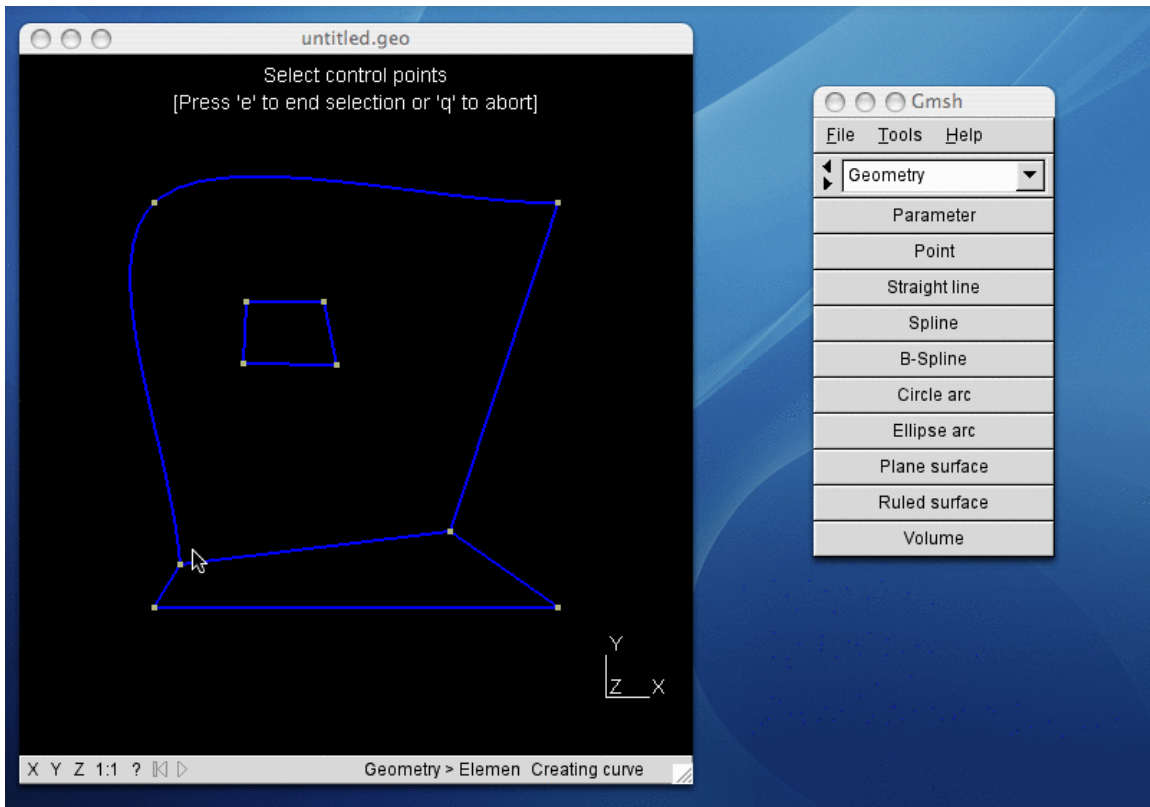We select the first control point by clicking on it.

We do the same for the second control point.

And we finally also select the third control point by clicking on it.
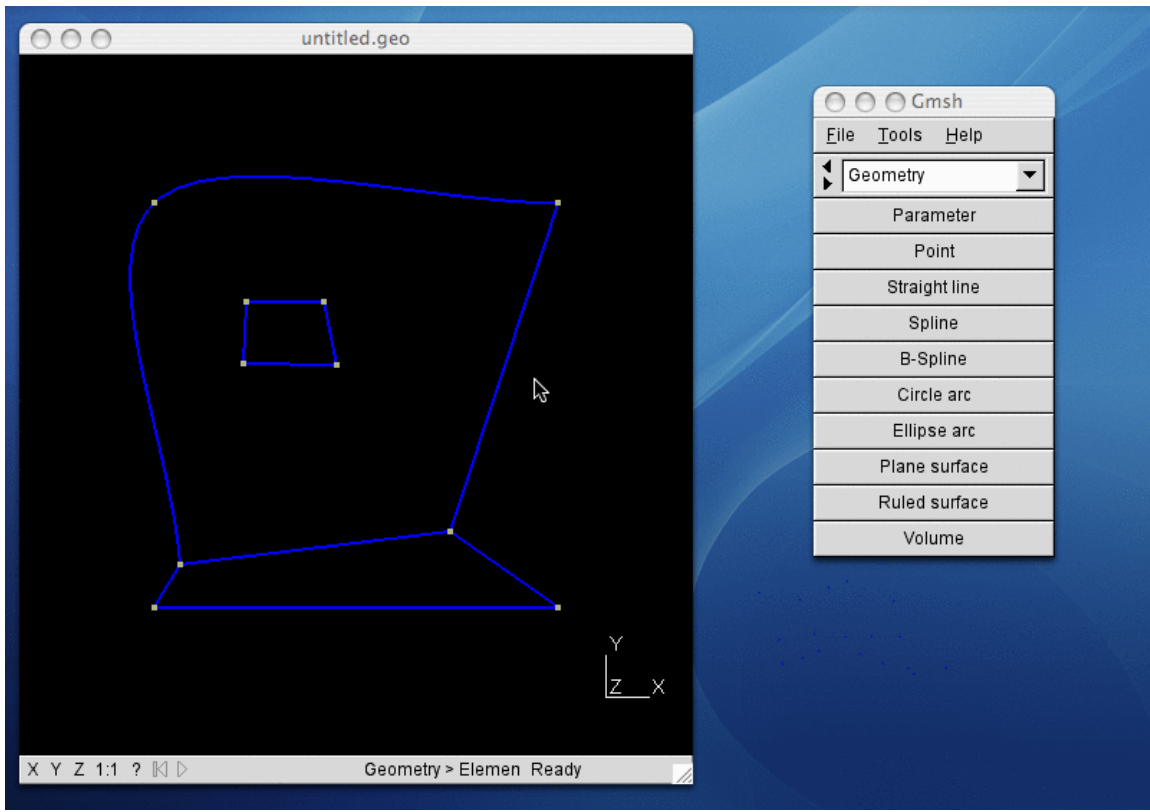
All the control points being selected, we can end the spline definition by pressing 'e'.
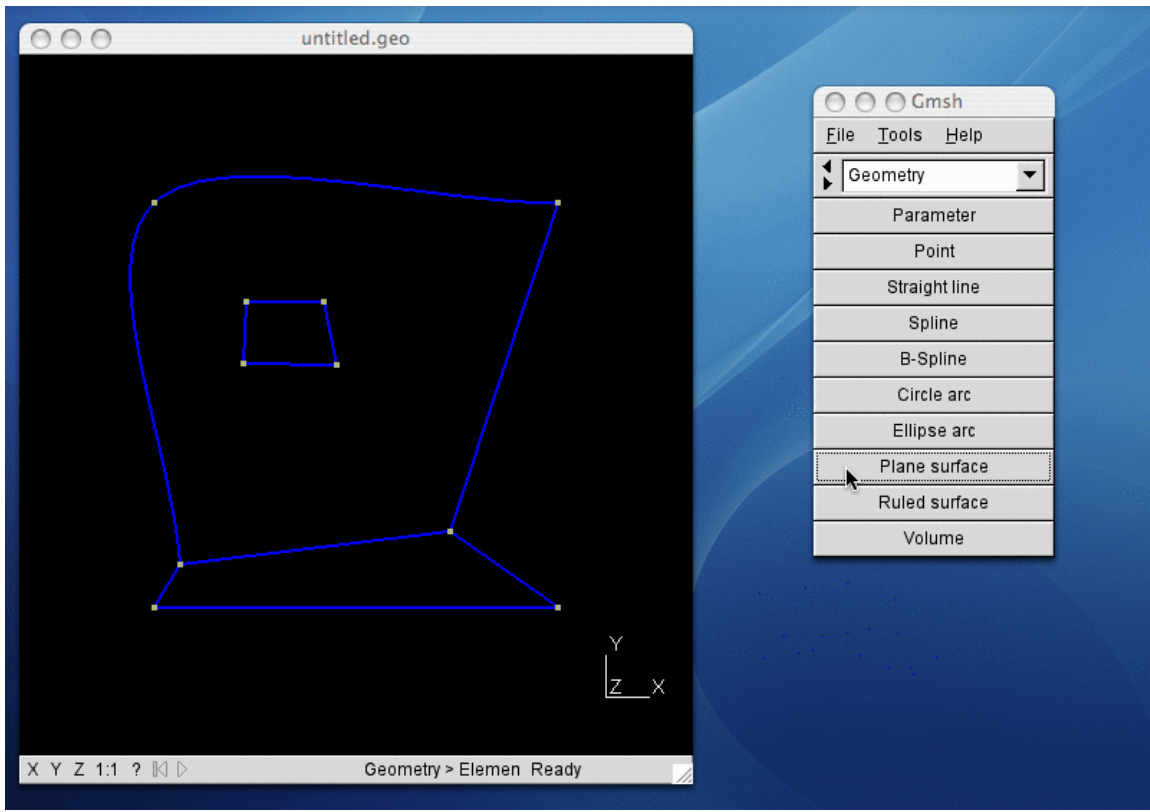
This creates the spline.

If we opened the project file with a text editor, we would see that Gmsh has added the following line at the end of the file:

```
CatmullRom(10) = {3,4,5};
```

Since we are done with the spline definition, we press 'q'.

We are now ready to create some surfaces. Since this is a planar 2D geometry, we click on the 'Plane surface' button in the menu to create some planar surfaces.

In Gmsh surfaces are defined by their boundaries, so we start by selecting the exterior boundary of our first surface (the "small one") by clicking on one of the segments on its boundary.
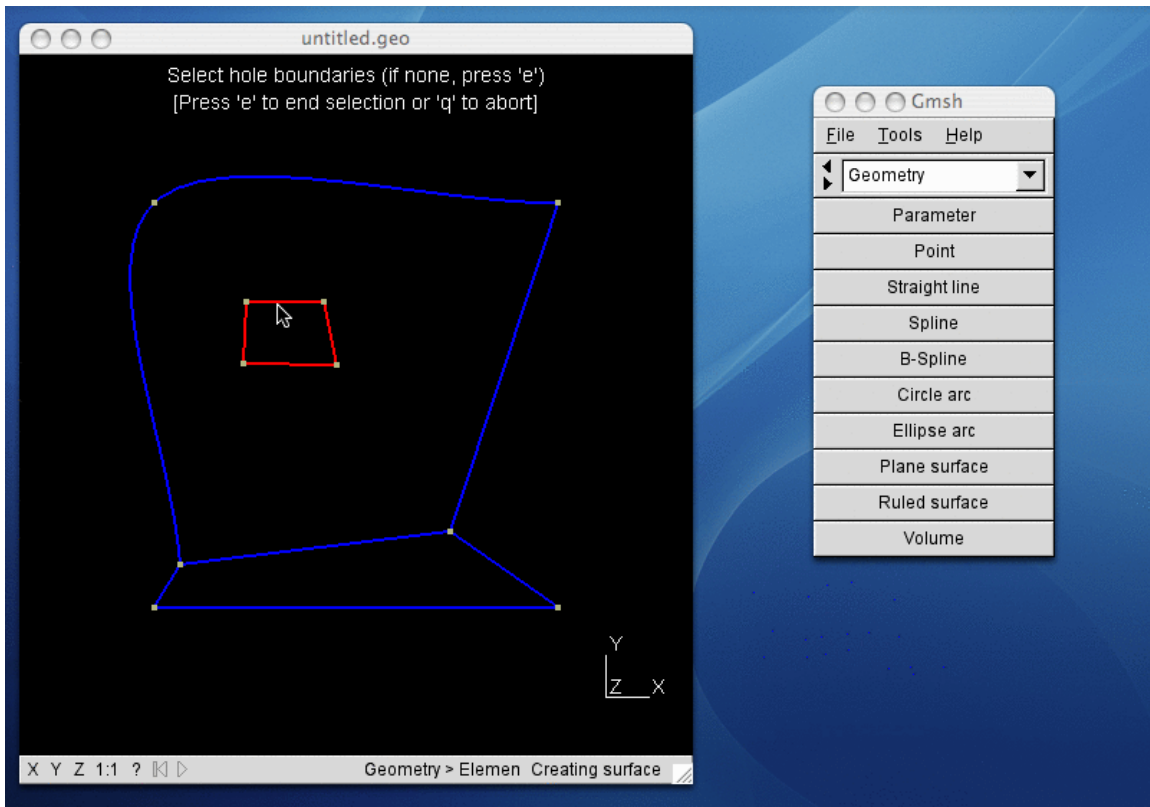
Gmsh completes the line loop automatically. And since there are no holes in this first surface, we press 'e'.
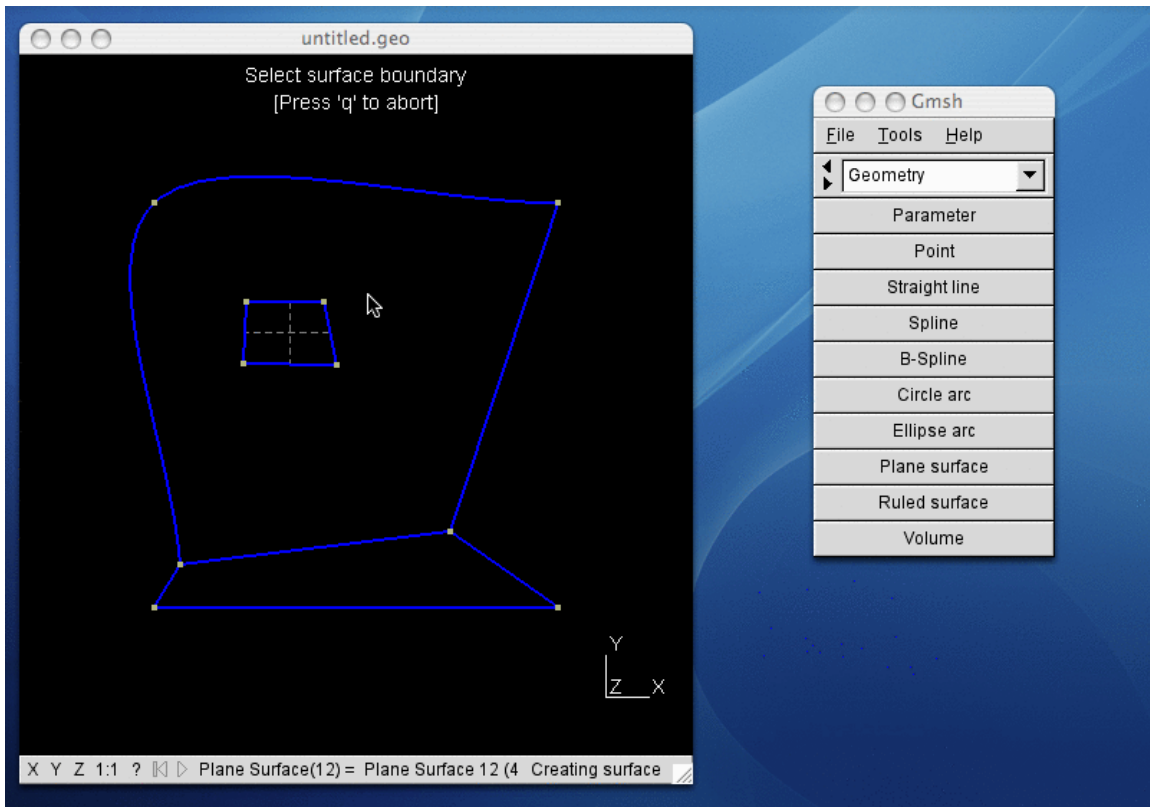
This has the effect to create the first surface in our model. The surface is represented on screen by the dashed cross.
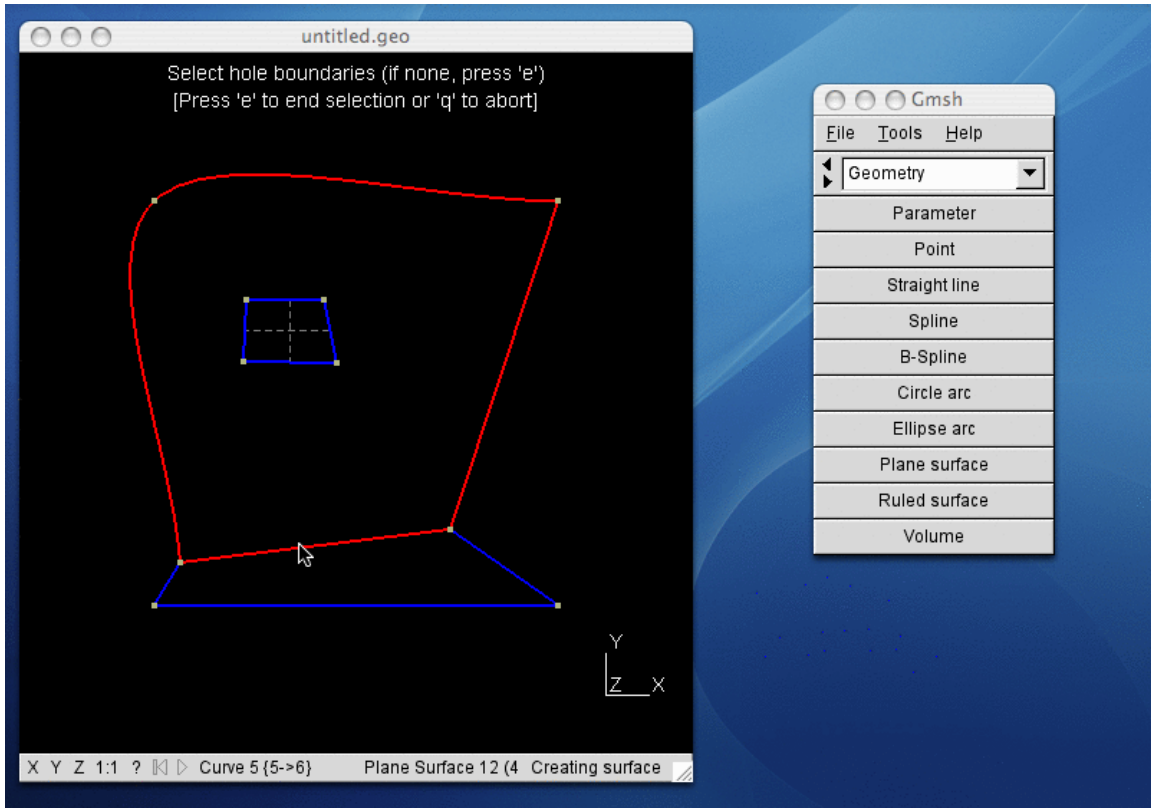
If we opened the project file with a text editor, we would see that Gmsh has added the two lines

```
Line Loop(11) = {3,4,1,2};
Plane Surface(12) = {11};
```
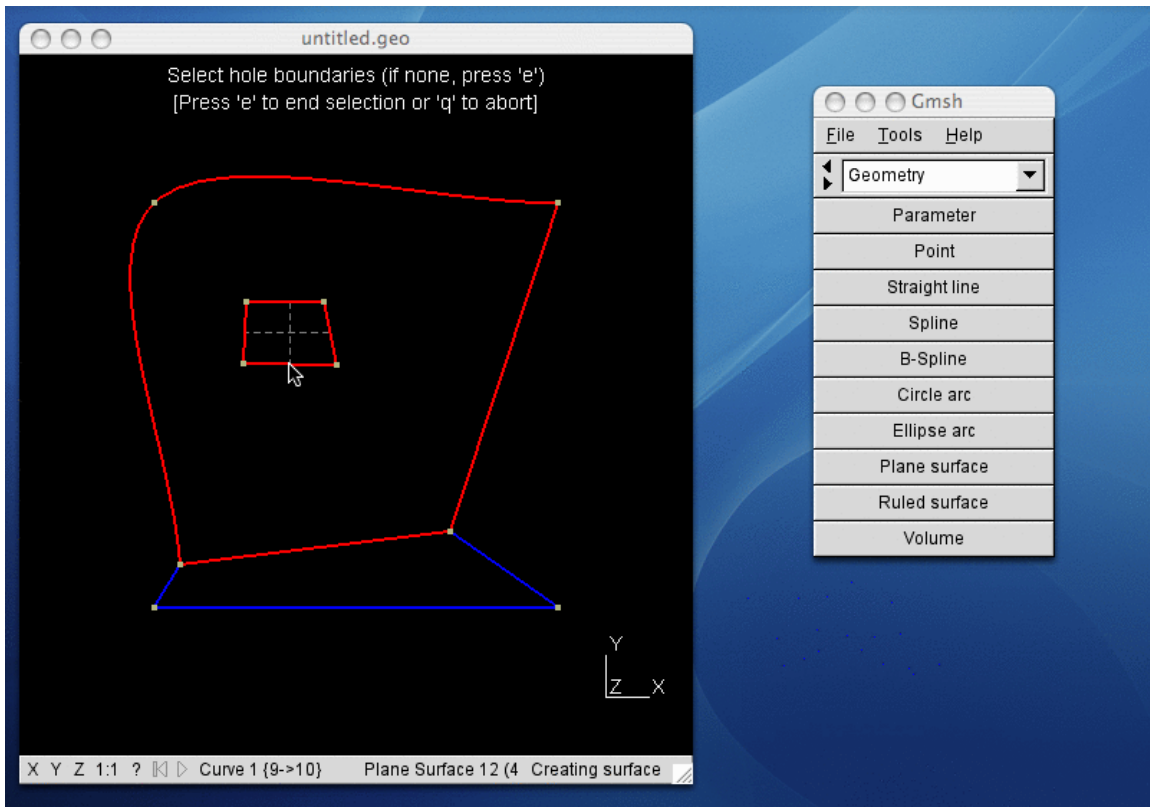
at the end of the file. The first line defines the line loop (a closed path representing the exterior boundary of the surface) and the second line defines the surface as being composed of this single line loop.
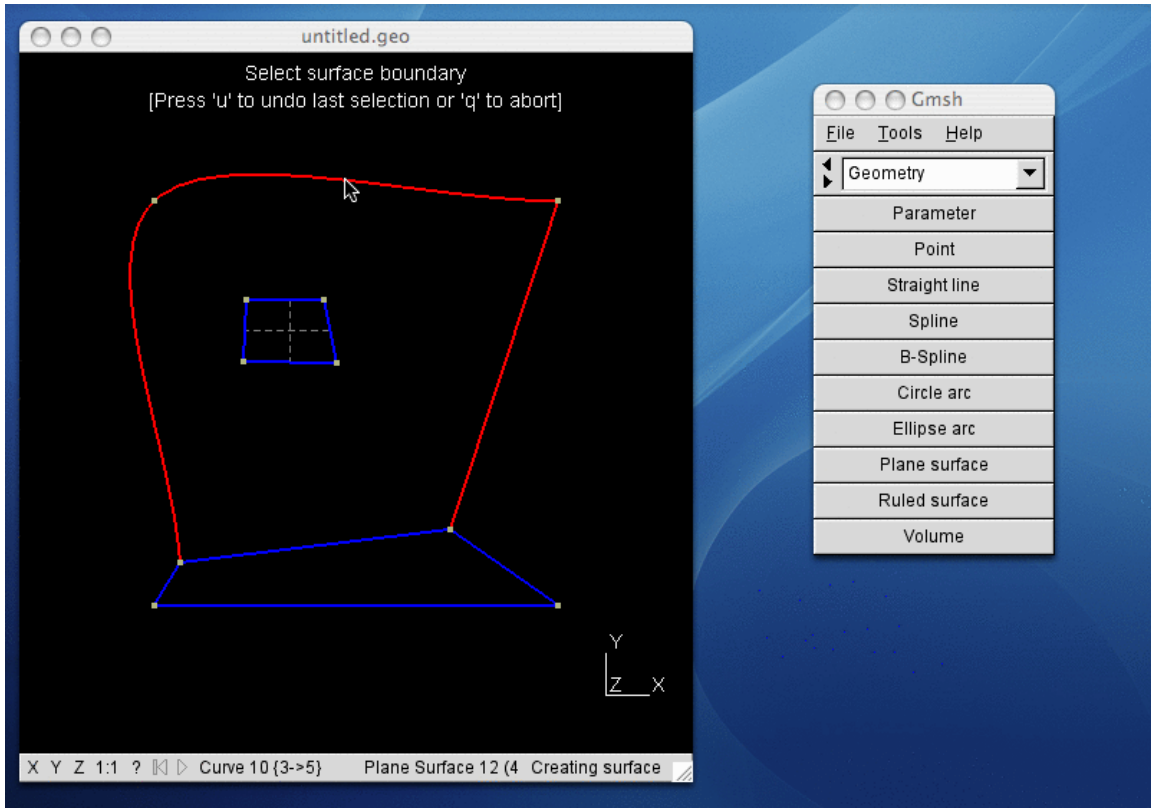
We proceed in the same manner to define our second surface, by selecting one curve on its exterior boundary. This time, Gmsh cannot close the line loop automatically.

We select the missing line in order to close the loop. But we are not done yet: this new surface has a hole in the middle, which we need to select, too.

We select the hole boundary by clicking on one of the segments on its boundary. Then we can press 'e' to end the surface definition.
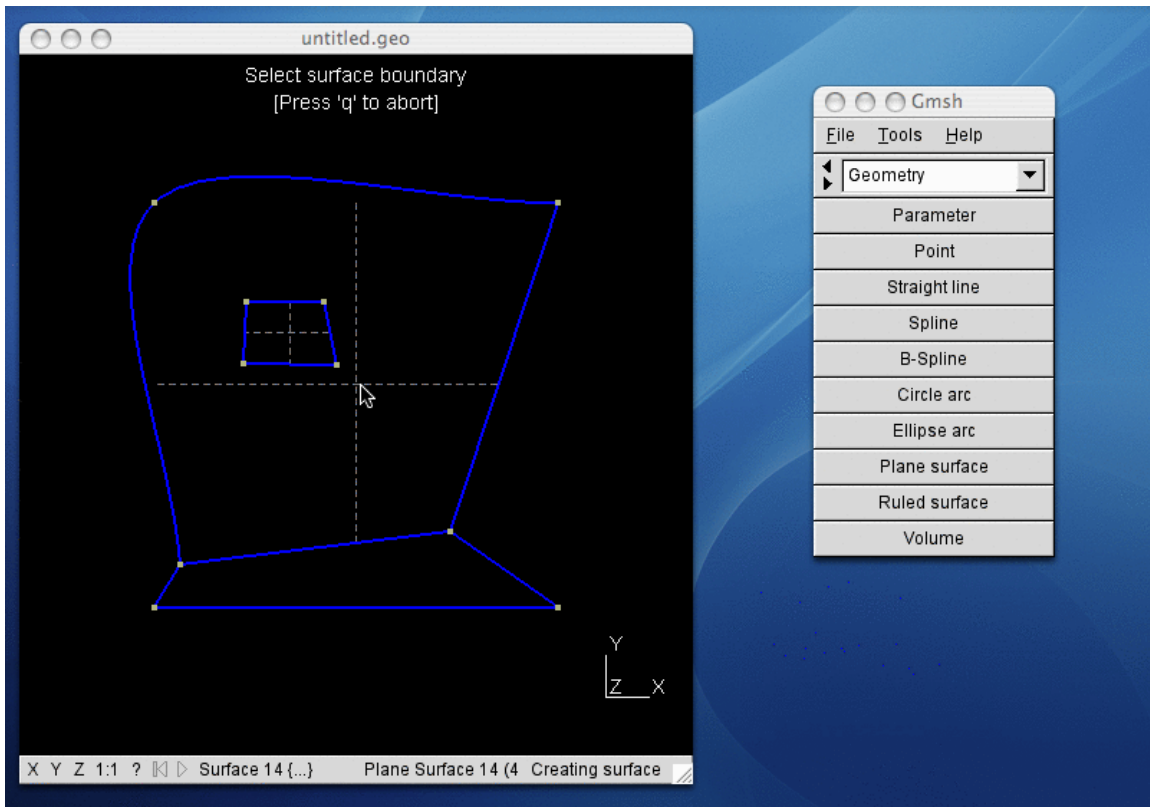
And we have our second surface (with a hole inside).

If we opened the project file with a text editor, we would see that Gmsh has added the two lines

```
Line Loop(13) = {10,5,9};
Plane Surface(14) = {13,11};
```
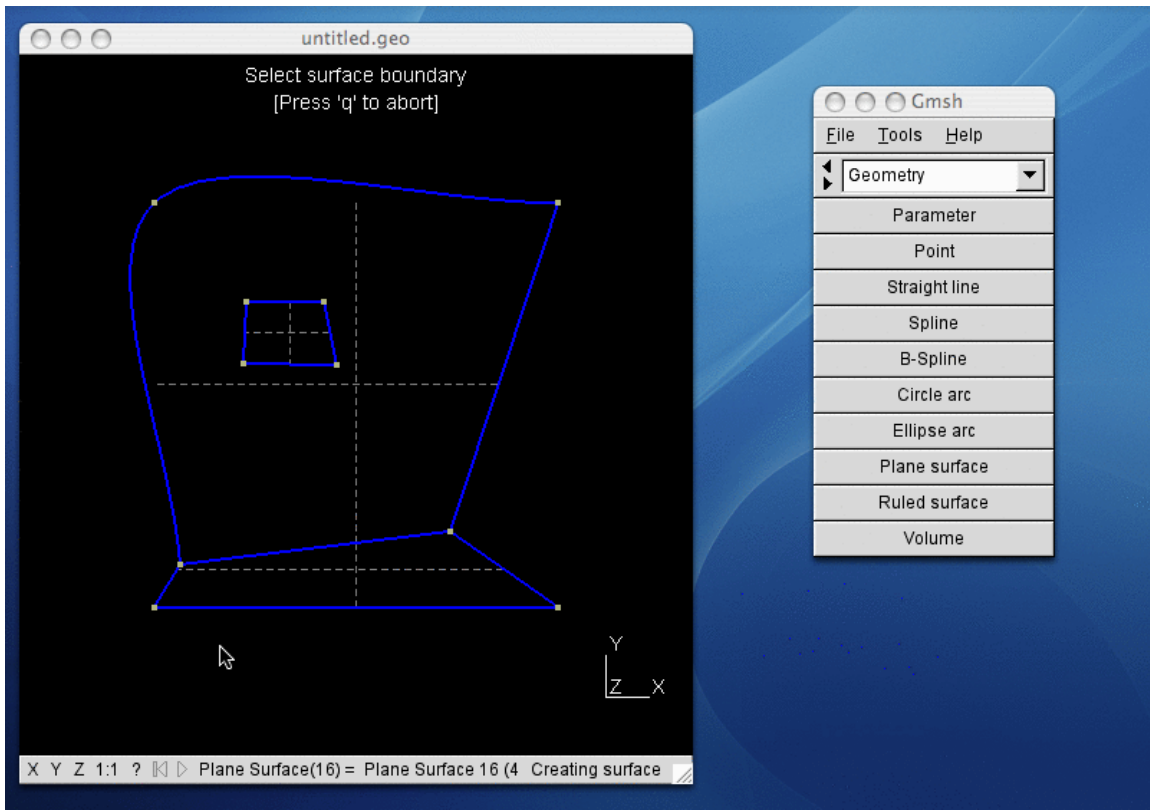
at the end of the file. The first line defines the exterior line loop and the second line defines the surface by a list of two line loops: the exterior boundary first, then the boundary of the hole.
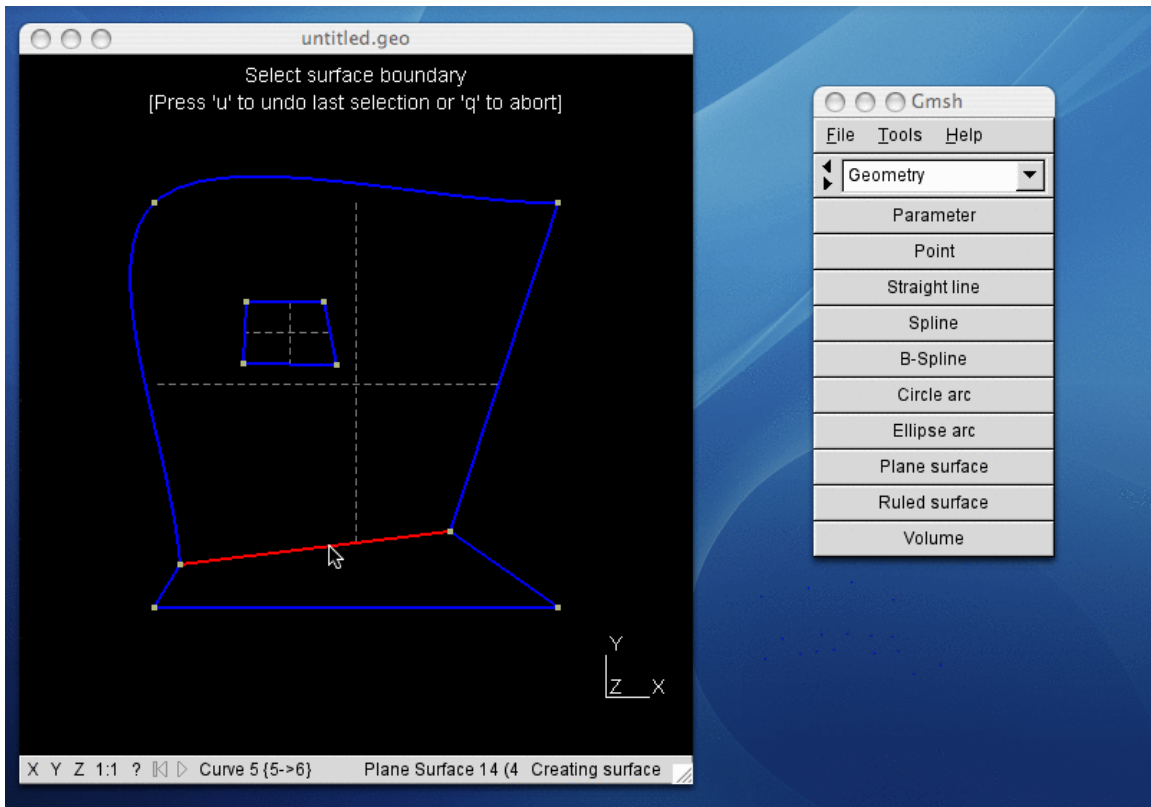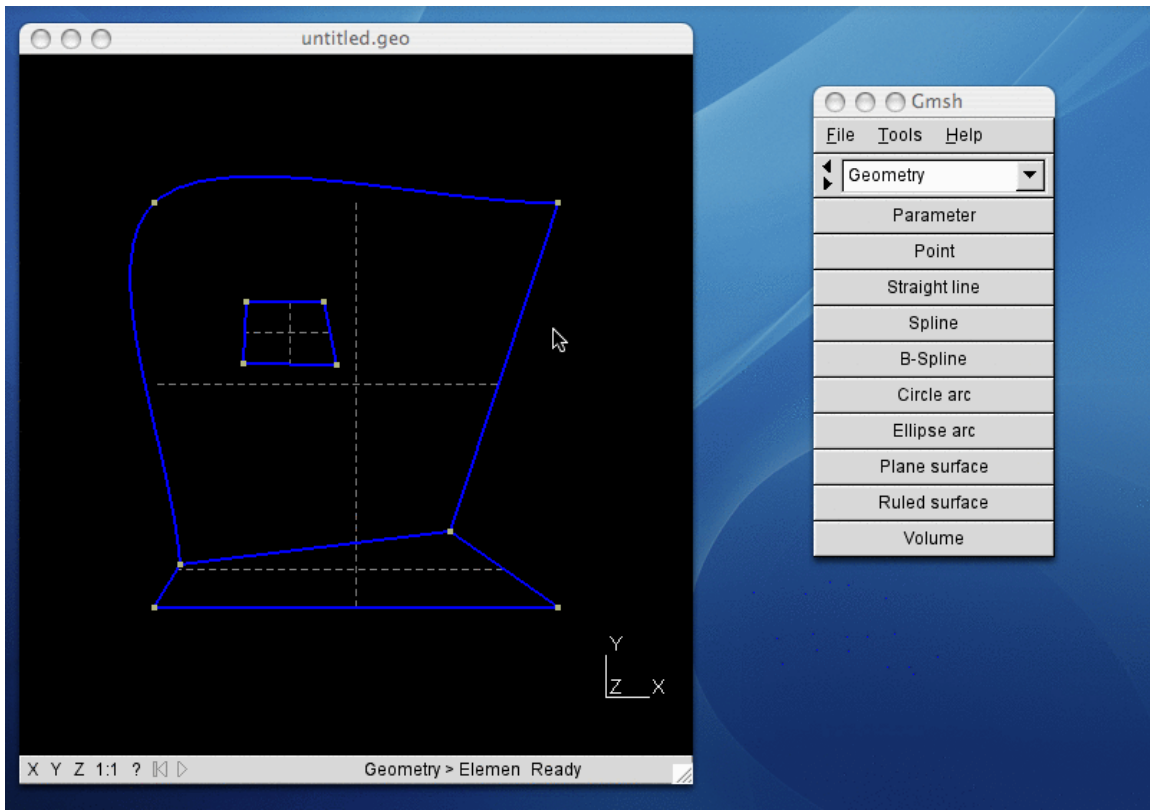
We finally select the boundary of the third surface by clicking on the line segment 5.
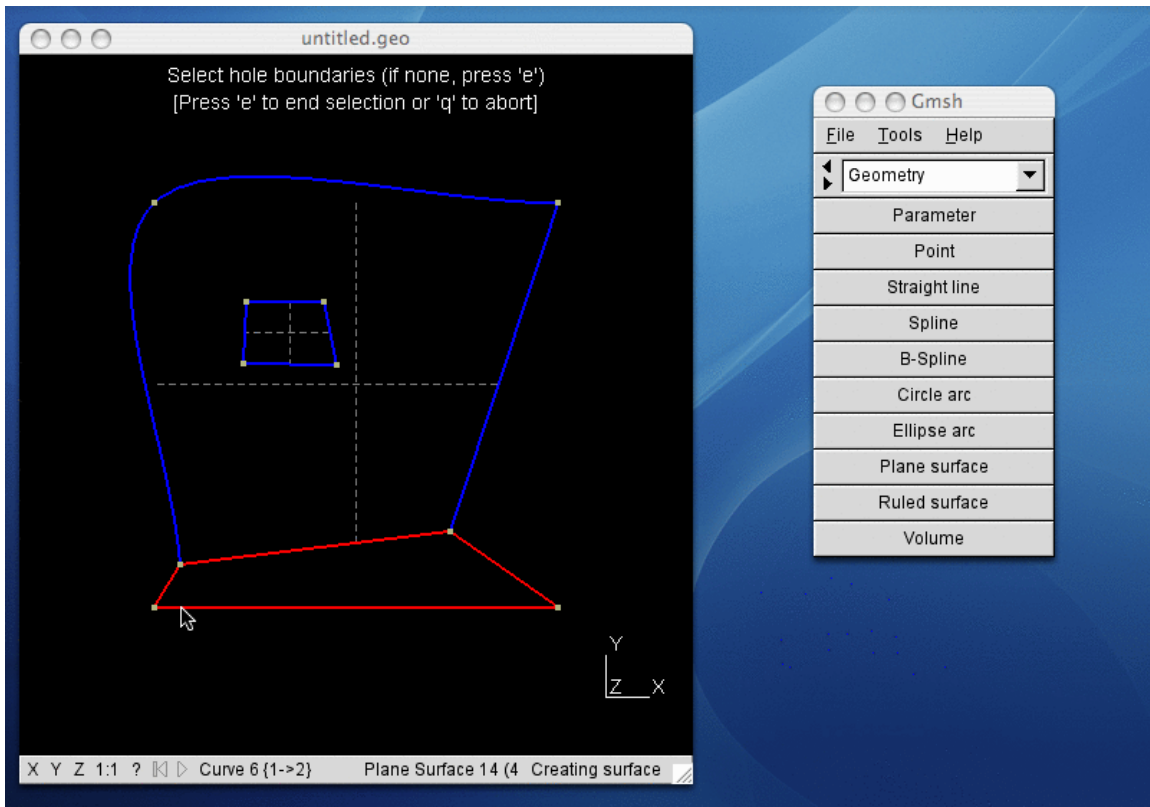
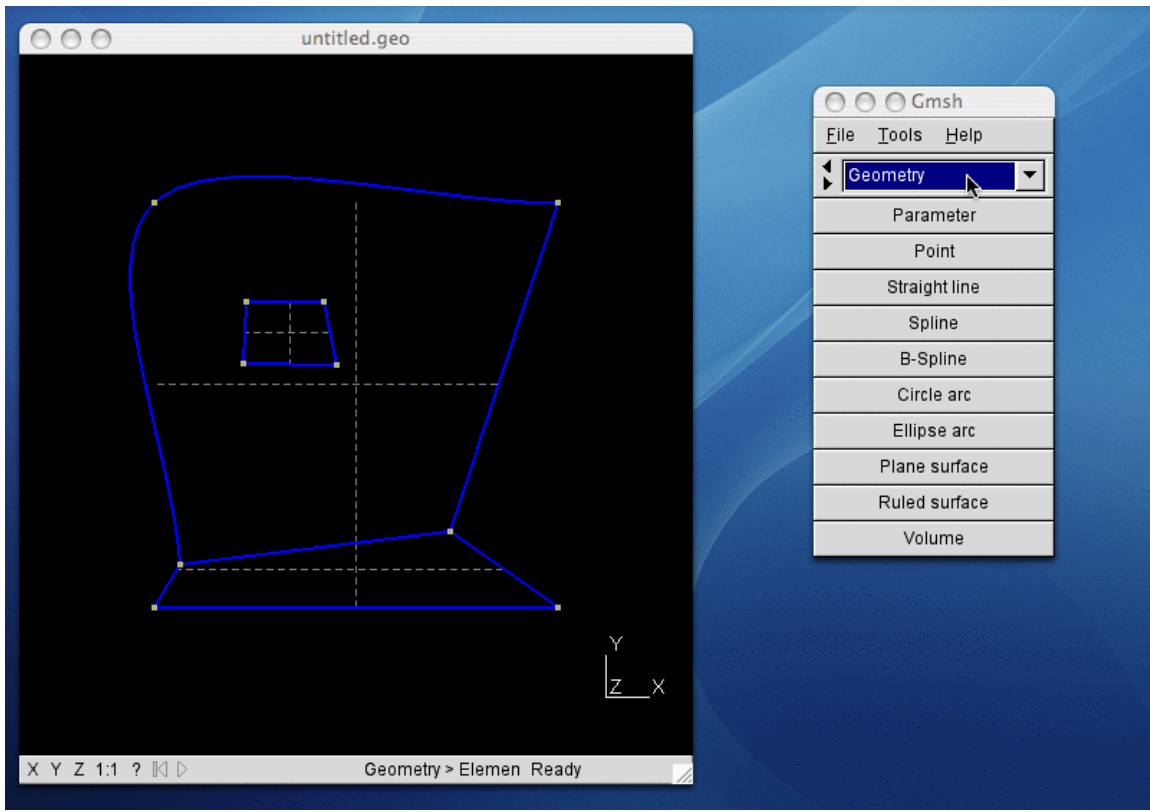And we close the loop by selecting one of the remaining segments in the loop.

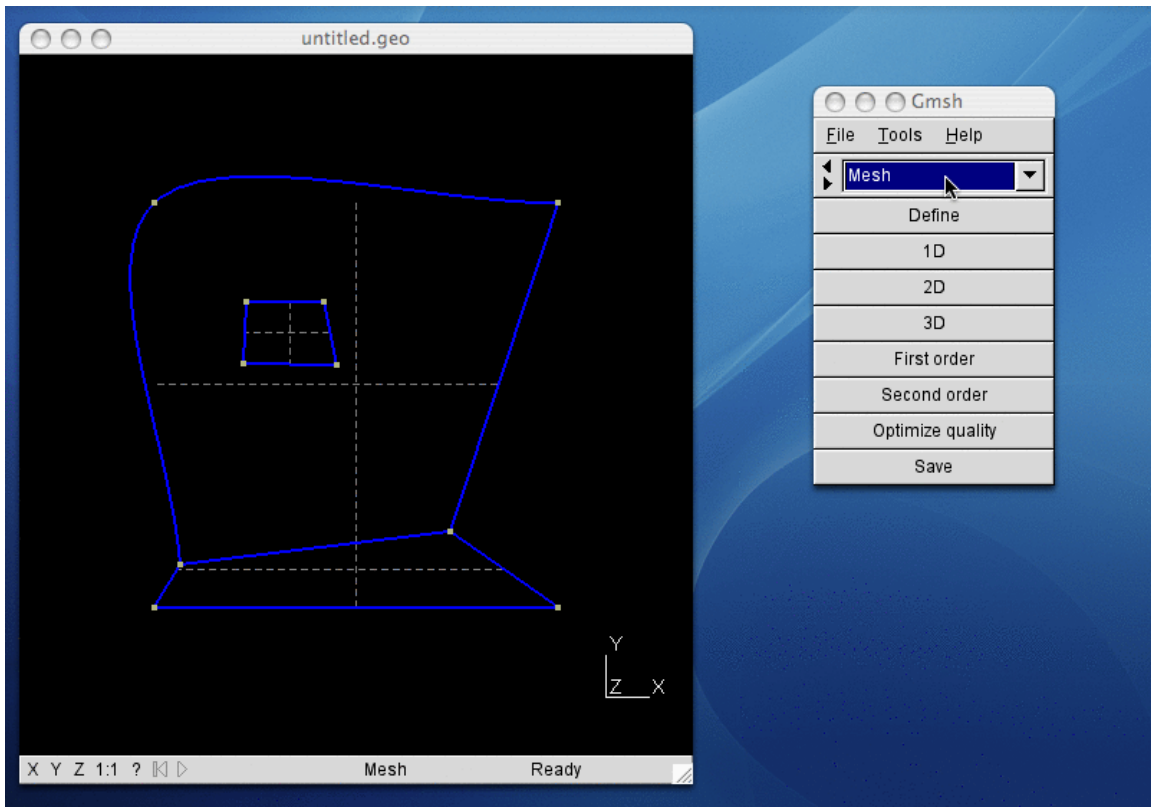As with the other two, the third and last surface is displayed using a dashed cross.
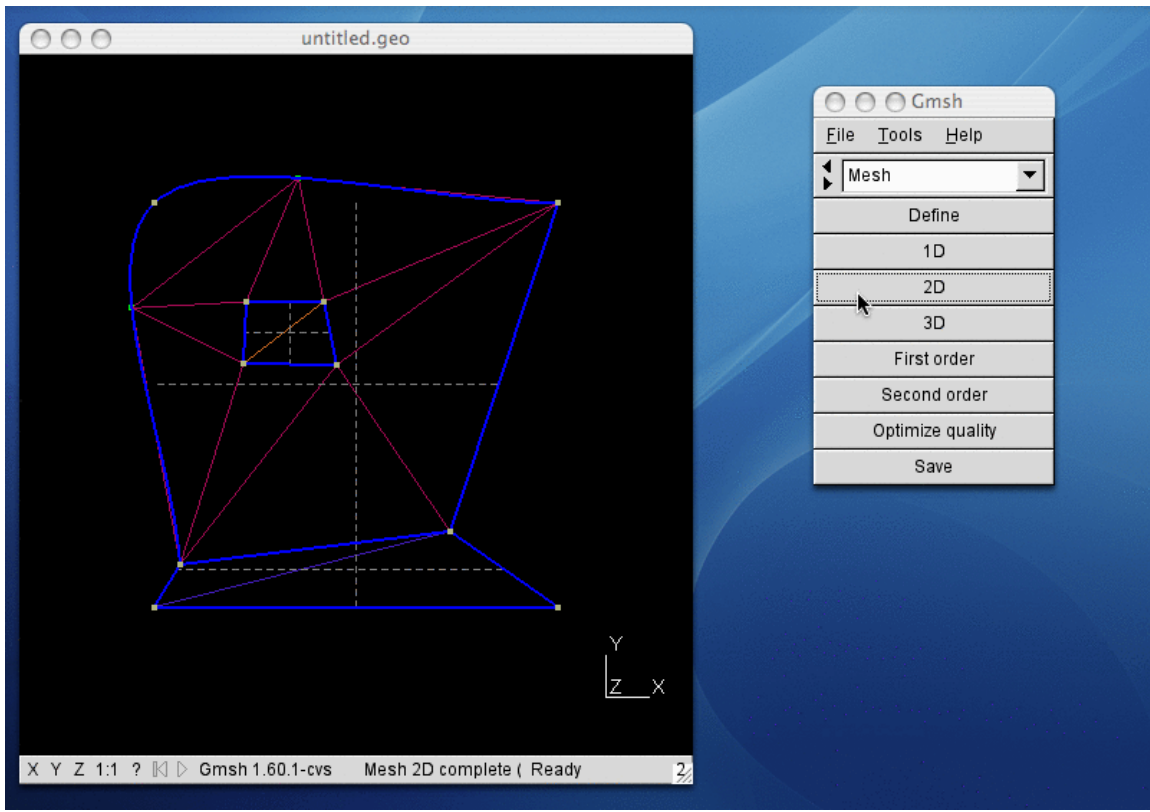
We are done creating surfaces, so we can press 'q'.

Our geometrical model is complete: 'untitled.geo'.

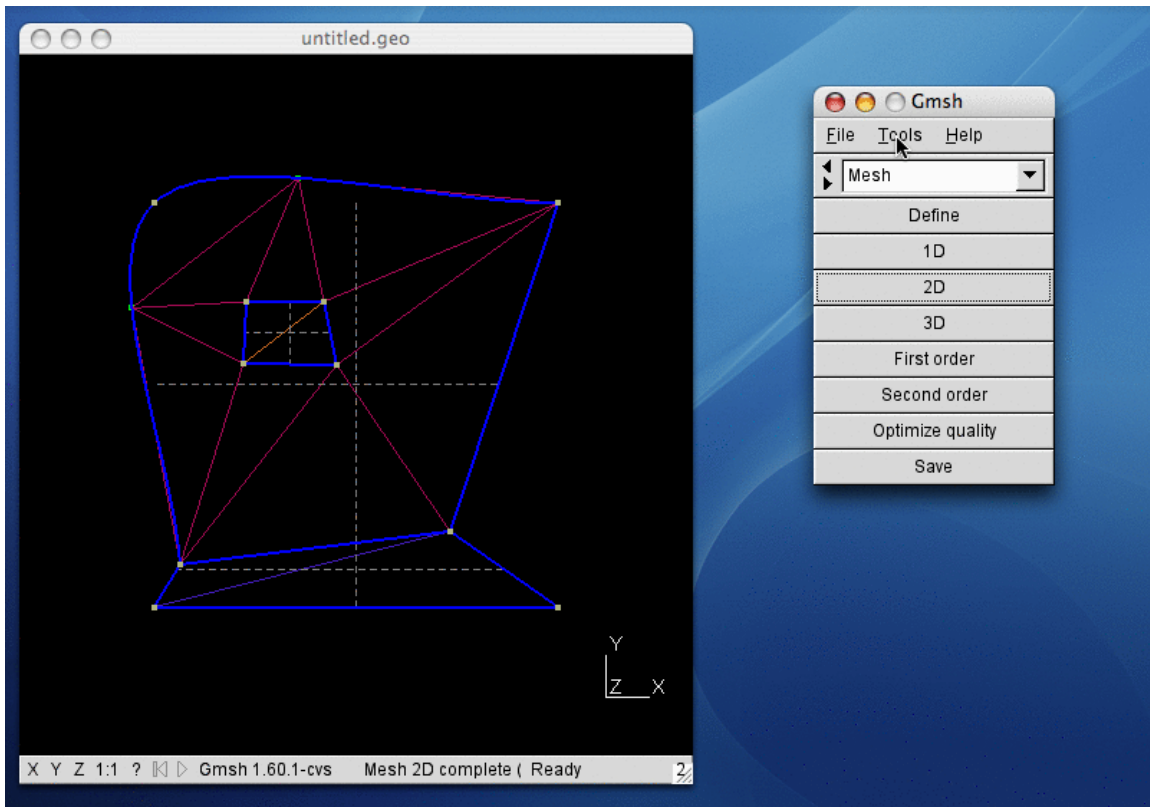We now want to mesh our model, so we go to the module menu.

We select 'Mesh' in the module menu.

We then click on '2D' to generate the 2D mesh.

Since we asked for characteristic lengths of 0.1 and the model's width and height are about 0.2, the resulting mesh is very coarse.
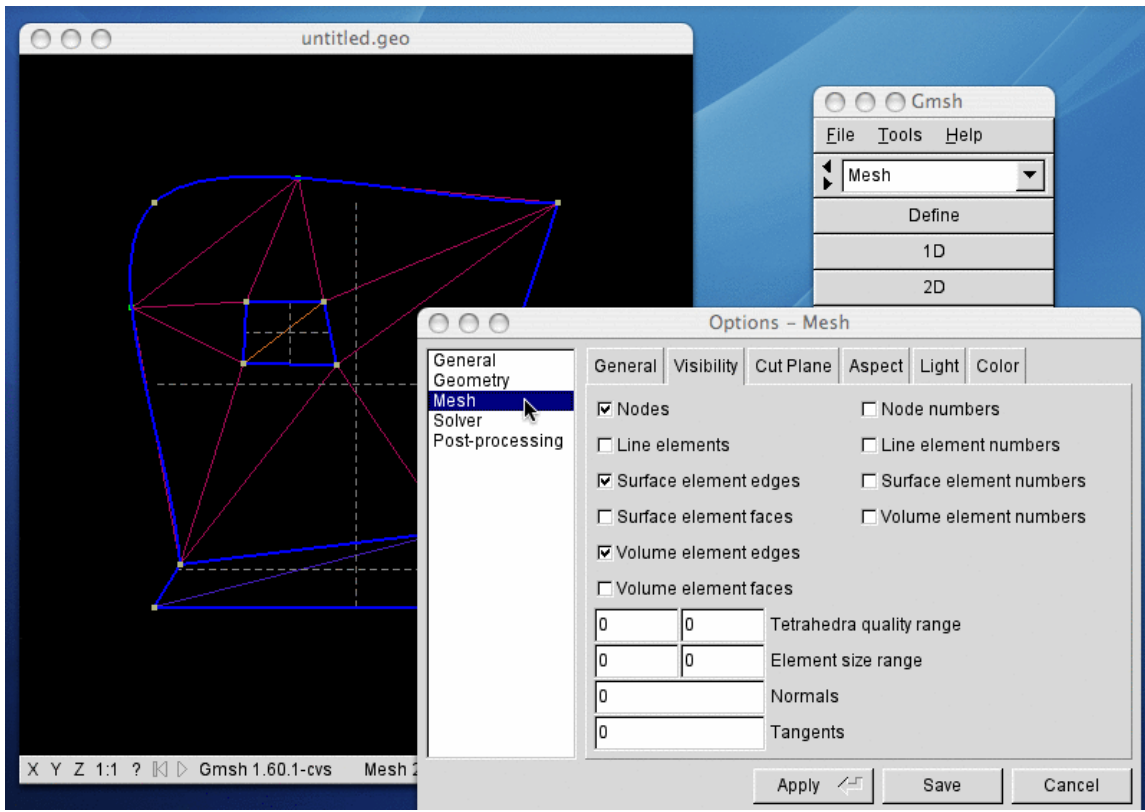
In order to generate a more refined mesh, we could edit the project file ('untitled.geo') and change the characteristic lengths associated with the points.
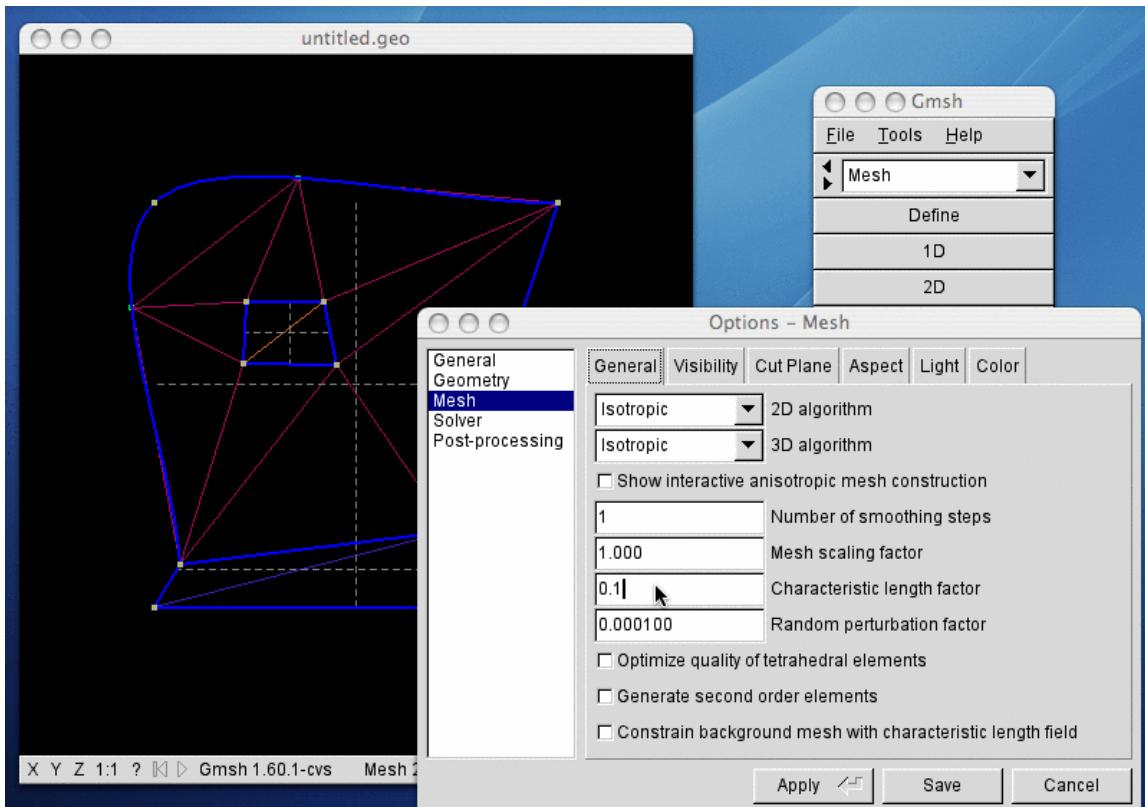
But we can also change the characteristic length globally in the graphical interface. In order to do so, we open the 'Tools' menu, and select 'Options'.
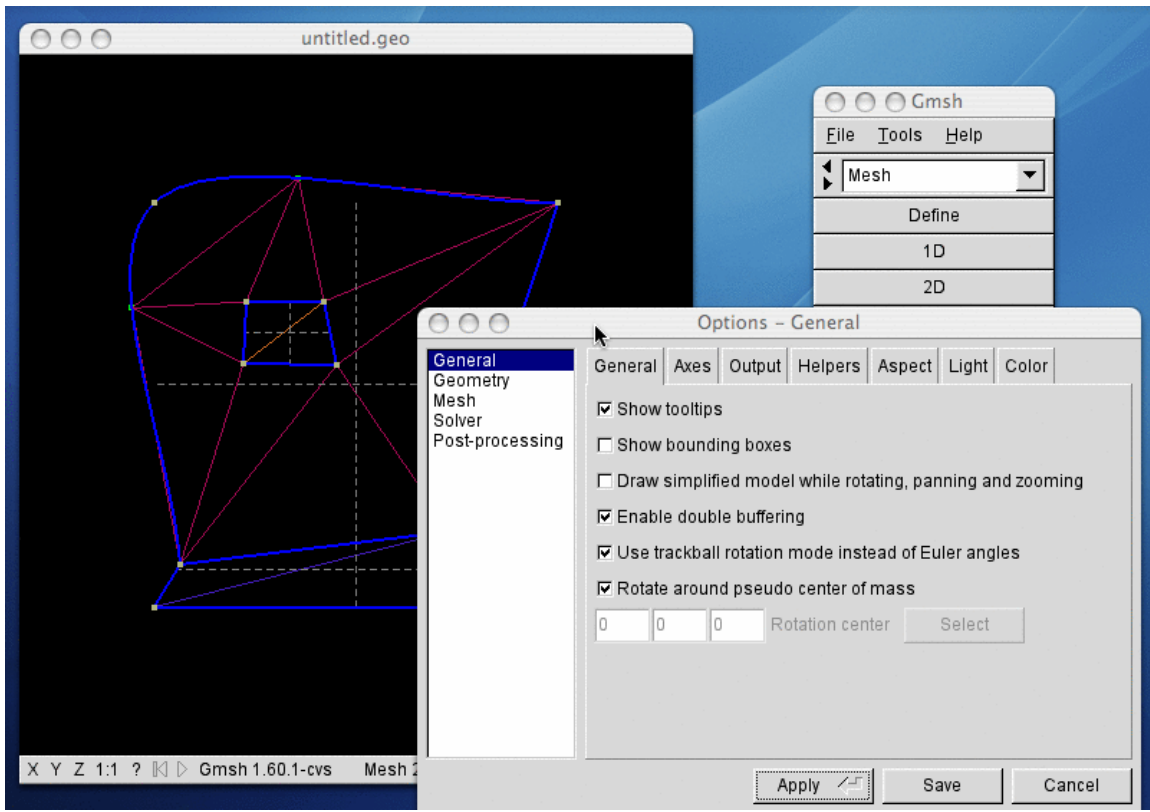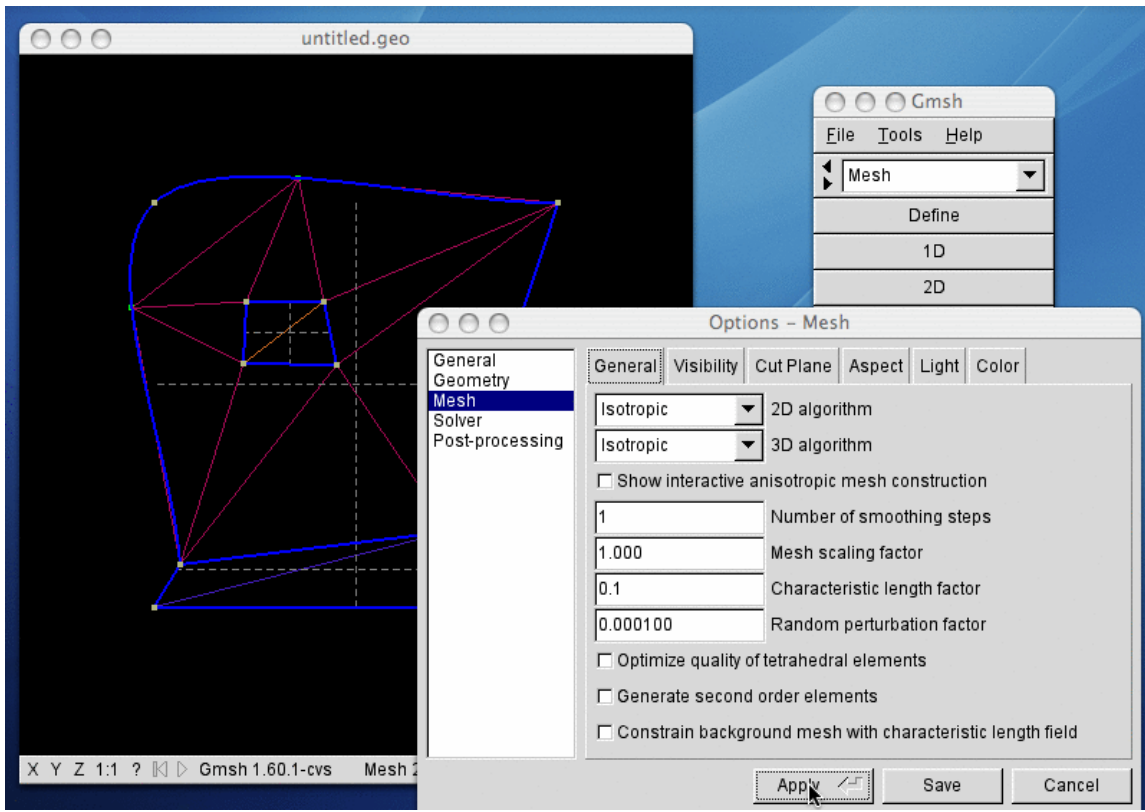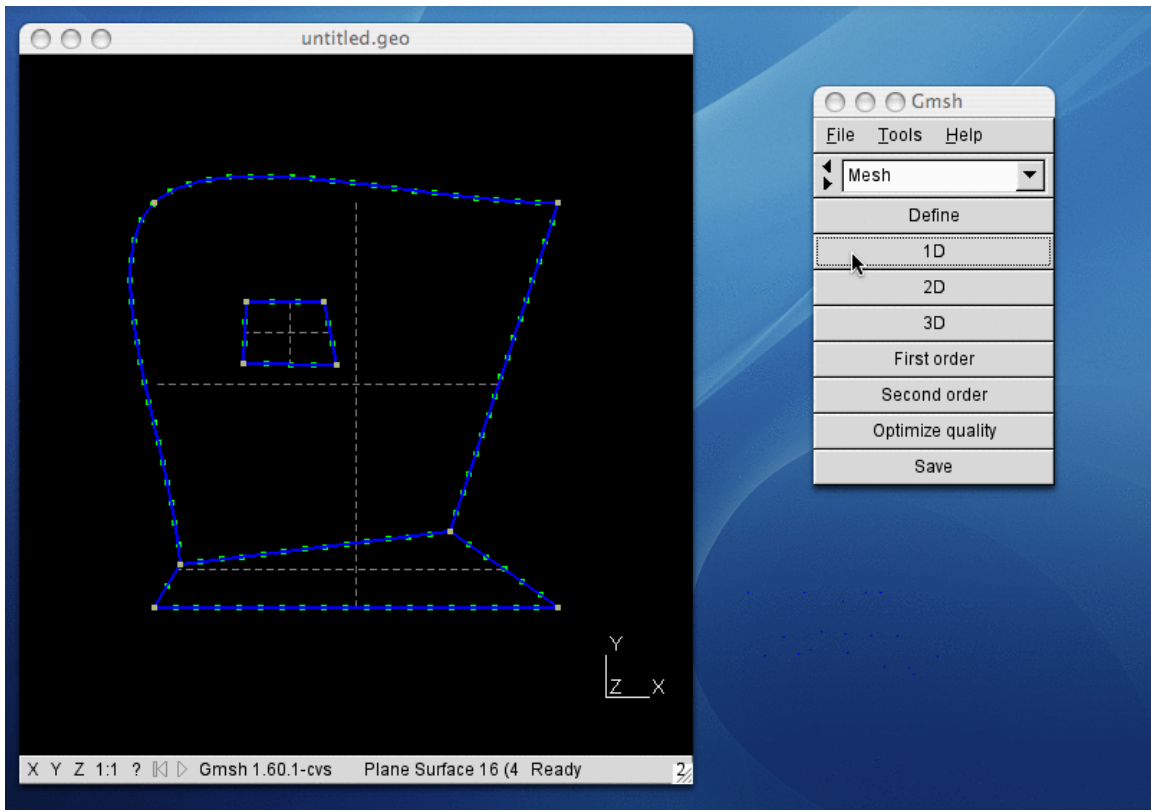
The option dialog pops up.

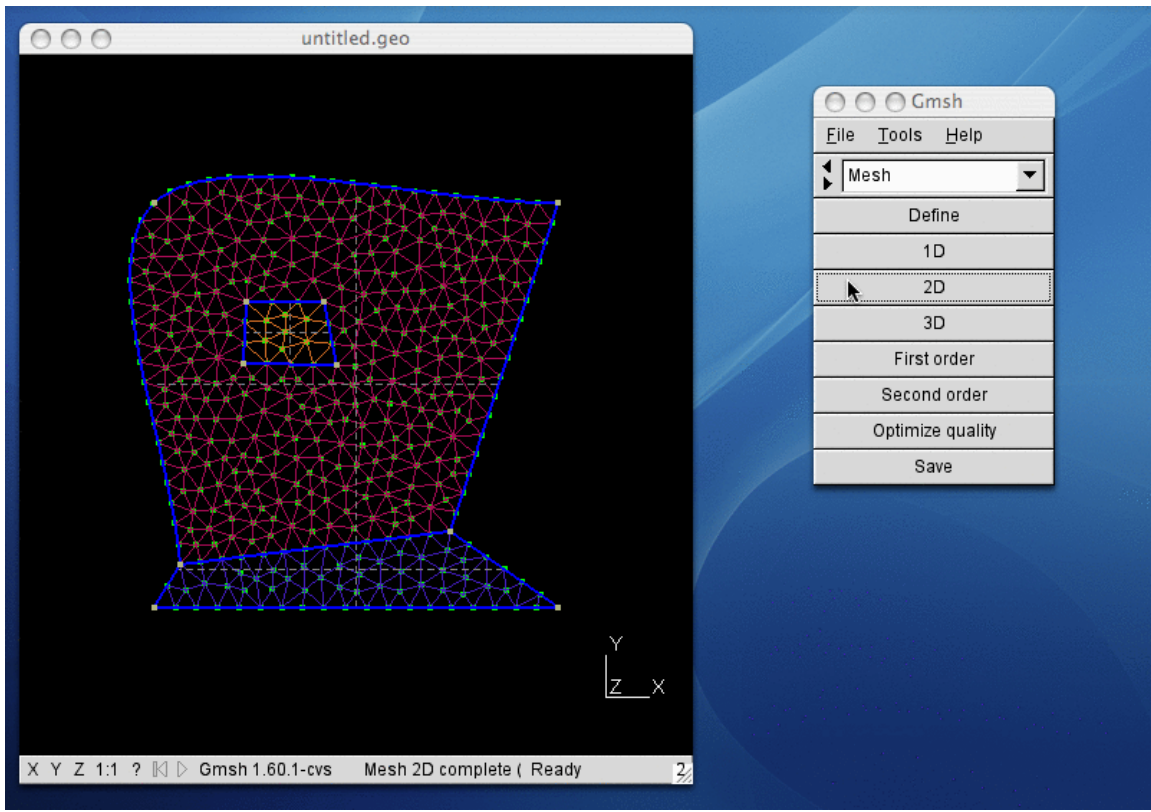We select 'Mesh' to access the mesh options.

We select the 'General' mesh options panel, and we specify a global mesh characteristic length factor of 0.1.
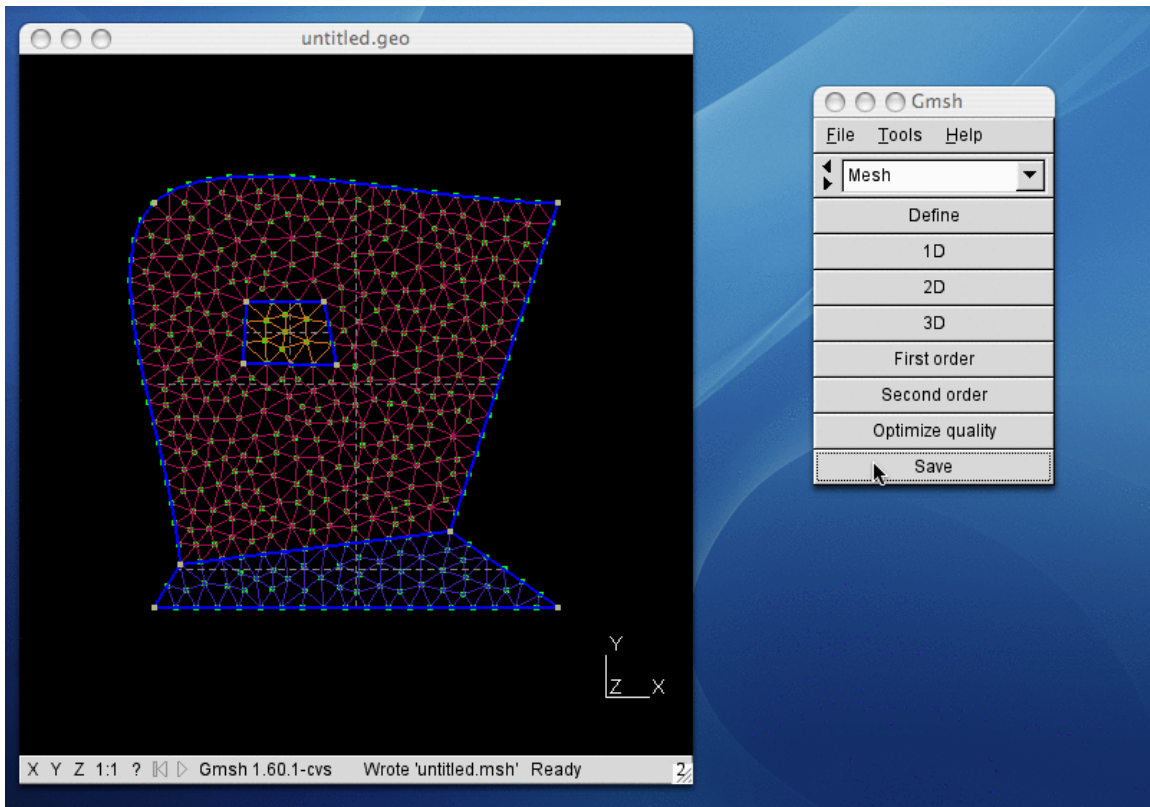
We apply the change by clicking on 'Apply'.

We can then for example remesh the lines with the new characteristic length by pressing on '1D', which results in a much finer grid (about 20 points per side, instead of 2).

We then remesh the surfaces with the new characteristic length by pressing the '2D' button.

Finally, since this mesh suits our needs, we can save it by clicking on the 'Save' button. This creates the following file: 'untitled.msh'.

To choose a different file format, we could open the 'File' menu, choose 'Save As', and select the appropriate format in the dialog.

That's all, folks!