



Spacecraft Plasma Interaction Analysis and Simulation Toolkit

Gmsh User Tutorial for CAD and Meshing

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Abstract

The Gmsh User Tutorial introduces the main features of this software with simple and progressive examples. This tutorial will be completed and improved on the basis of users comments and questions.

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A. Introduction

I. Purpose of this document

This document introduces the main features of Gmsh with simple and progressive examples. It mainly focuses on the use of the Geometry and Mesh modules.

II. Gmsh

Gmsh is an automatic 3D finite element mesh generator with build-in CAD and post-processing facilities. Its primal design goal is to provide a simple meshing tool with parametric input and up to date visualization capabilities. One of its strengths is the ability to respect a characteristic length field for the generation of adapted meshes on lines, surfaces and volumes, and to mix these meshes with simple structured grids.

Gmsh is built around four modules: geometry, mesh, solver and post-processing. The specification of any input to these modules is done either interactively using the graphical user interface or in ASCII text files using Gmsh's own scripting language.

This software is available under GPL license at <http://www.geuz.org/gmsh>

B. General presentation

I. Launch Gmsh

To run Gmsh:

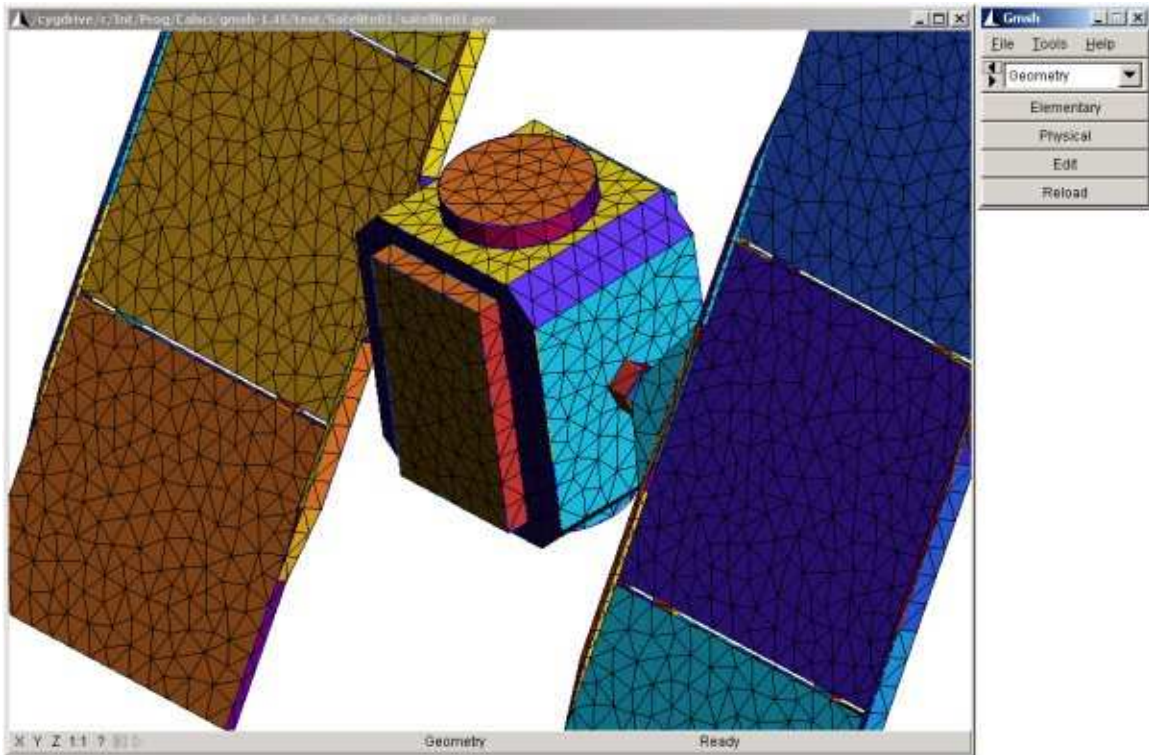
- type `gmsh` at your UNIX/Linux prompt
- double-click on the Gmsh icon for Window

II. The user interface

Upon running Gmsh, two windows are displayed:

These two windows are:

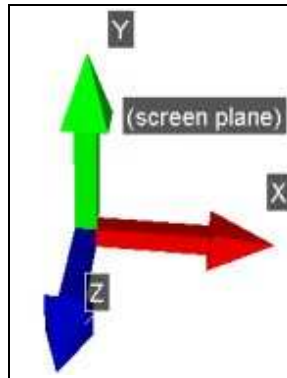
- the visualization window
- the menus window



III. Visualization window

III.1. Introduction

The visualization window is the area where the model (geometry or mesh) is displayed. We can interact (rotate, move, zoom in/out, selection, ...) on it with the mouse. By default the 3D axes orientation is defined such as the *Z axis* is perpendicular to the screen in the direction of the user.



III.2. The tools and status bar

The *tools and status bar* gathers some visualization icons and displays some useful informations when you load a file or when the mouse is upon entities (vertices, curves, surfaces, nodes, elements, etc ...). For instance, the coordinates and the ID of a point could be displayed here.



- **X** : Displays in the YZ plane ($Y=Z=0$)
- **Y** : Displays in the ZX plane ($Z=X=0$)
- **Z** : Displays in the XY plane ($X=Y=0$)
- **1:1** : The display is set to unit scale
- **?** : Shows current selected options in a separate window

III.3. Basic mouse interaction

<i>Mouse button</i>	<i>Result</i>
Left	Rotation
Ctrl+Left	Zoom window selection
Middle	Zoom in/out
Rigth	Pan
Ctrl+Right	Reset the default viewpoint

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Note: the others mouse interaction functionality will be introduced in the examples.

IV. Menus

IV.1. Files

Allow to import and export geometry or mesh files, fixing appropriate options or format.

IV.2. Tools

By default, three sub menus (Options, visibility and statistics) allow you to

Option : access and change Gmsh options (inculding rendering option of the visualisation windows),

Visibility : Define and manipulate the list of mesh/geometrical elements and group (called Physical in Gmsh), you want to visualize.

Statistics : Which give a reduce set of statistics, mainly number of geometrical and mesh element of a given type.

IV.3. Helps

This menu give you acces to windows where list of mains short cut and options are detailed.

C. Fundamental examples

I. Breps approach

For the CAD part, Gmsh uses Breps approach. Breps means Boundary Representations. In this approach a geometric model is defined and stored as a set of vertices, edges, faces wich completely enclose its volume.

So the construction of a geometry is always based on a PLSV (Point, Line, Surface, Volume) hierarchy coupled with operations like translation, rotation or extrusion.



II. 2D examples

II.1. Square

The first example shows how to build a square on the XY plane and how to mesh the surface. This first example is more detailed than the others to help the user to get used to Gmsh. Read it carefully if you want to understand the other examples.

II.1.a. Geometry

The obvious way

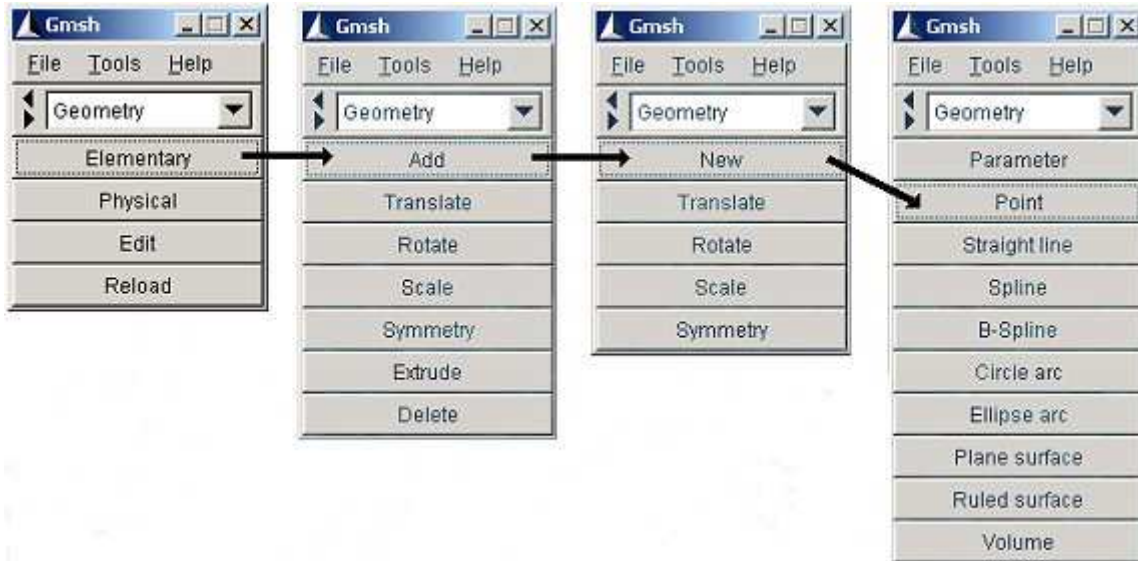
If you have understood the simple philosophy of Breps approach, the creation of a square will consist on :

- Entering 4 points
- Create 4 lines
- Declare a surface

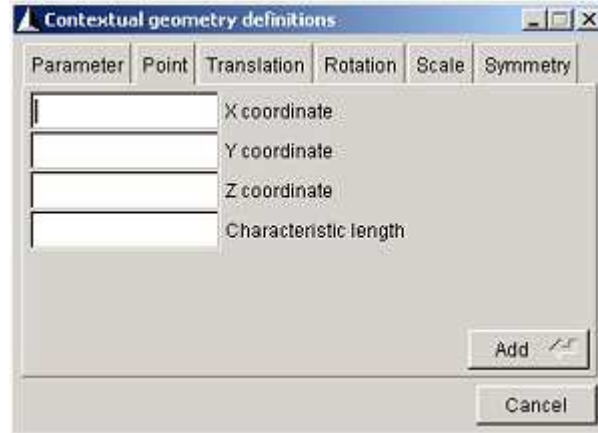
So let's go !

STEP 1 : Entering point coordinates

Click on [Elementary] > [Add] > [New] > [Point] :



A contextual geometry window appears with the *Point* notebook selected:



Enter the coordinates of Point 1 :

- X coordinate: 0
- Y coordinate: 0
- Z coordinate: 0
- Characteristic length: 1.0

The *Characteristic length* will define the size of the mesh around the point.

Click on **[Add]** and a point will be plotted on the visualization window.



Tips & Tricks

- The **[Return]** key (keyboard) do the same thing that the **[Add]** button.
- If you put the mouse cursor over a point in the visualization window, its coordinates and ID are displayed in the status bar.
- After pressing **[Add]**, the geometry window is always here ready to get the next coordinate.

Do the same for Points 2, 3 and 4 by introducing their coordinates in the geometry window :

- Point 2: (10, 0, 0, 1.0)
- Point 3: (10, 10, 0, 1.0)
- Point 4: (0, 10, 0, 1.0)

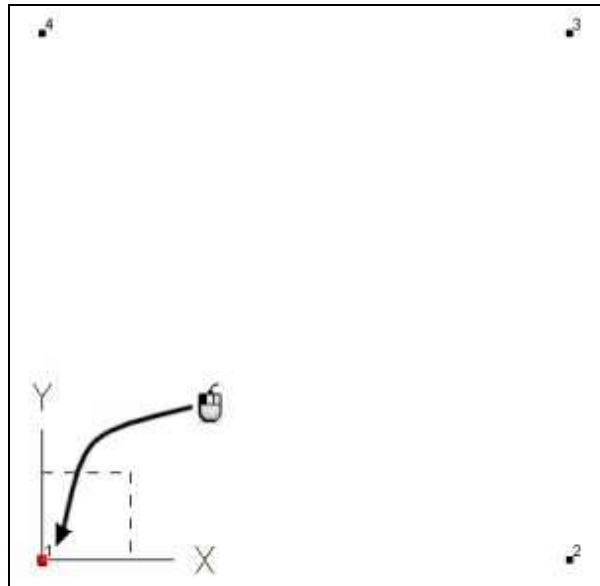
STEP 2 : Drawing lines

Now, we are going to create lines between each points.

Click on **[Straight line]** (the button just under **[Point]**).

A message appears in the status bar: *Select start point ('q'=quit).*

In order to draw the first line, select Point 1 by clicking on it with the first button of the mouse.

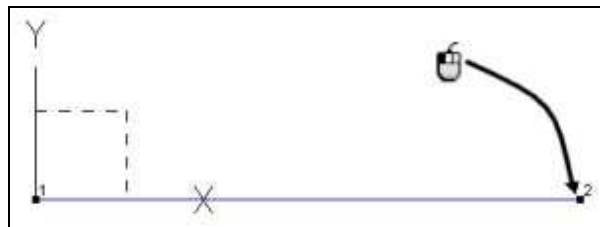


It becomes highlighted in red color.

A message appears in the status bar: *Select end point ('q'=quit)*.

So, always with the first button of the mouse, select Point 2 (the end point).

A blue straight line is drawn.



A well known message appears in the status bar: *Select start point ('q'=quit)*.

We proceed as previously seen for lines between (Point 2 - Point 3), (Point 3 – Point 4) and (Point 4 – Point 1).

Once the square is drawn, just type 'q' for quitting the *Straight line* selection mode.

STEP 3 : Declaring surface

So, we have a square but for Gmsh, it's only four lines.

To declare a surface, click on [**Plane surface**].

A message appears in the status bar: *Select exterior boundary ('q'=quit)*.

With the first button of the mouse select the first line (between Point 1 and Point 2).

The four lines become highlighted in red color and a message appears in the status bar:

Select hole boundary ('q'=quit).

Since there is no hole in our simple square, just type 'q' for quitting the *Plane surface* selection. The status bar shows that a plane surface was created.

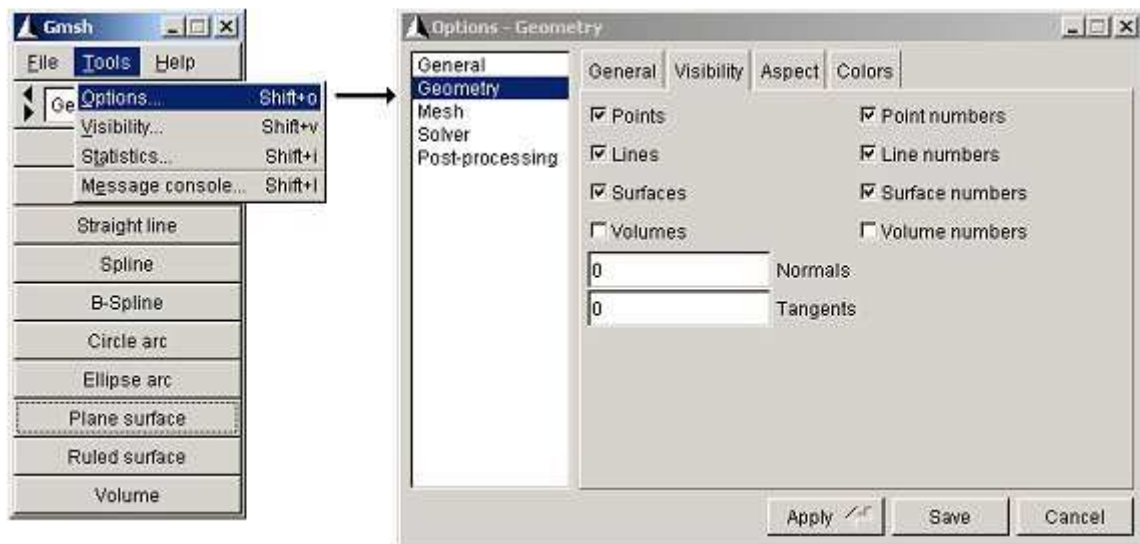
Note: in our simple example, Gmsh has found the red closed curve alone. In more complex cases, the user must select all the line one by one to create the *line loop*.

STEP 4: Visibility options

To ease the visualization of this surface, it is useful to change some visibility options.

In the Gmsh menus, select **[Tools] > [Options]**. An option window appears.

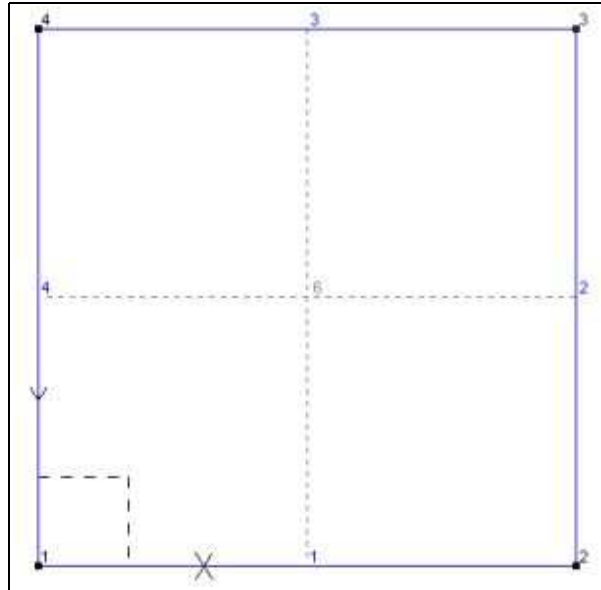
From this option window, select *Geometry* in the left field and after click on the *Visibility* notebook.



Select *Surfaces* (by default *Points* and *Lines* are already selected).

Select *Point numbers*, *Line numbers* and *Surface numbers*.

Click on **[Apply]** button and we may get something like that:



The created surface is represented by a cross with dotted lines.
We can see the point ID, the line ID and the surface ID.

Note: In this example we have declared one surface but its ID is 6. Don't panic, you haven't created 5 invisible surfaces ! It's only related to the internal numbering of geometrical entities in Gmsh.

Tips & Tricks



If you want to keep your favorite options, just press the **[Save]** button in the option window. These options will be saved in a configuration file.

STEP 5: Saving the geometry

To save your masterpiece, select **[File]** > **[Save as]** in Gmsh menus and give it a name without forgetting the *.geo* extension.

Here is the content of this file:

```
square.geo
```

```
Point(1) = {0, 0, 0, 1};
Point(2) = {10, 0, 0, 1};
Point(3) = {10, 10, 0, 1};
Point(4) = {0, 10, 0, 1};
Line (1) = {1, 2};
Line (2) = {2, 3};
Line (3) = {3, 4};
Line (4) = {4, 1};
Line Loop (1000006) = {2, 3, 4, 1};
Plane Surface (6) = {1000006};
```

If you quit Gmsh without saving your work, the current file is saved as *unnamed.geo* .

The translation function

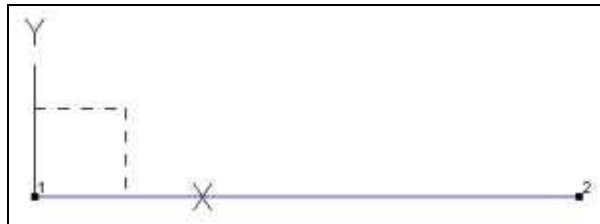
With CAD tools, there is always several ways to draw something.
For example our square could be built using translation function.

As previously seen, create two points:

- Point 1: (0, 0, 0, 1.0)
- Point 2: (10, 0, 0, 1.0)

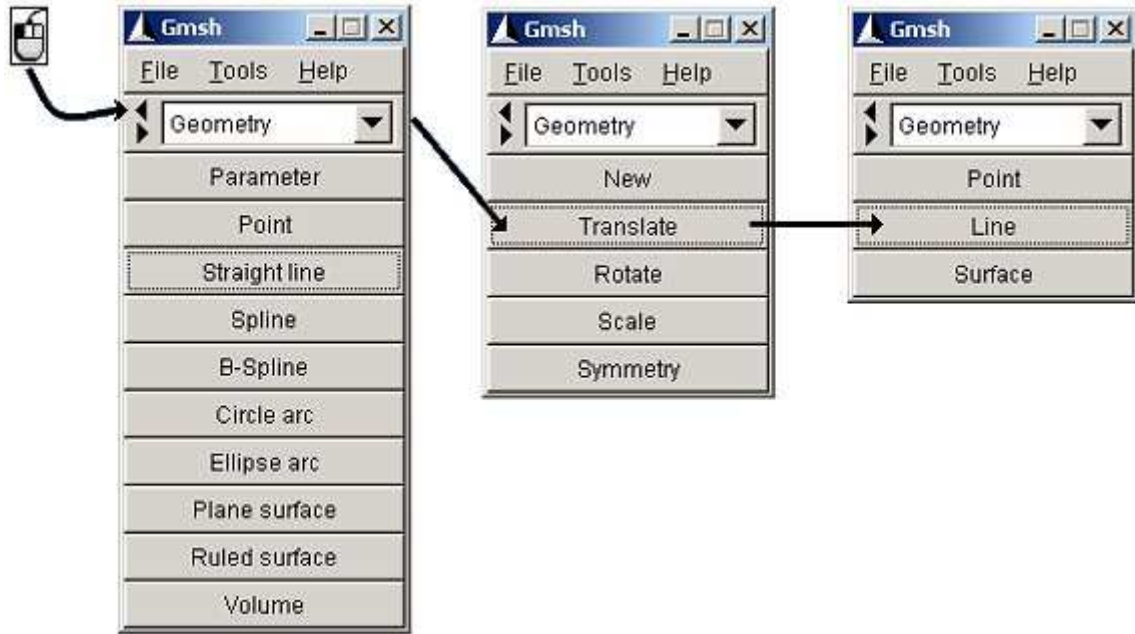
(with **[Elementary]** > **[Add]** > **[New]** > **[Point]**)

Create a *Straight line* between them by clicking on **[Straight line]** :

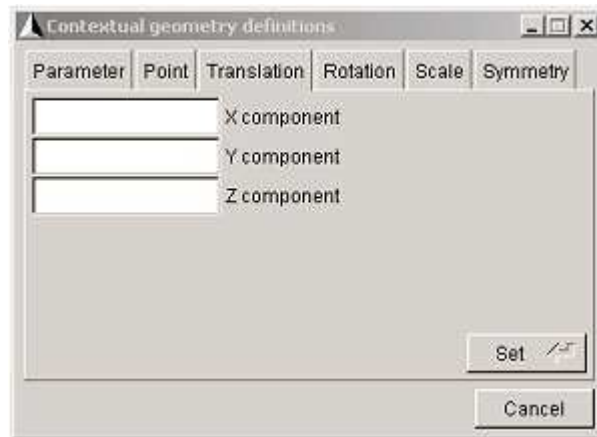


Click on the backward arrow at the left of *Geometry*.

Click on **[Translate]** > **[Line]**



A contextual geometry definition window appears with the *Translation* notebook selected:



Enter the vector components of the translation :

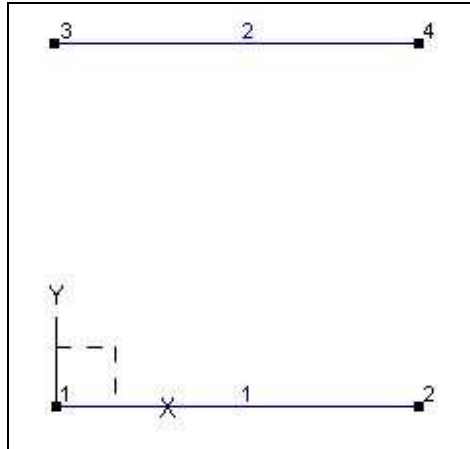
- X component: 0
- Y component: 10
- Z component: 0

Click on **[Set]** to validate this vector.

Follow the advice in the status bar: *Select line ('q'=quit)*, by clicking on the line with the left button of the mouse.

We get another line translated of 10 units in the XY plane and to the Y direction.


Once the translation is performed, just type 'q' for quitting the *Translate* selection mode.



To finish your work, click twice on the little backward arrow at the left of *Geometry* to change tools. Click on **[Add] > [New] > [Straight line]** and create lines between (Point 2 - Point 4), and (Point 3 - Point 1).

For the last step, declare your surface by clicking on **[Plane surface]** and select it. And don't forget to save your masterpiece by selecting **[File] > [Save as]** in Gmsh menus. The saved file looks like this:

<i>square.geo</i>
<pre> Point(1) = {0,0,0,1.0}; Point(2) = {10,0,0,1.0}; Line(1) = {1,2}; Translate {0,10,0} { Duplicata { Line{1}; } } Line(3) = {2,4}; Line(4) = {3,1}; Line Loop(5) = {3,-2,4,1}; Plane Surface(6) = {5}; </pre>

 Tips & Tricks
<ul style="list-style-type: none"> - If you want to generate a flat geometry only with simple entities without transformations command, you can use in the [File] menu, [Save as] and choose in the [Format] combobox: <i>Gmsh unrolled geometry</i>.

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A square with a hole

Now we are going to examine the case of a square with a triangular hole. The major interest of this example is the declaration of the surface between these two polygons.

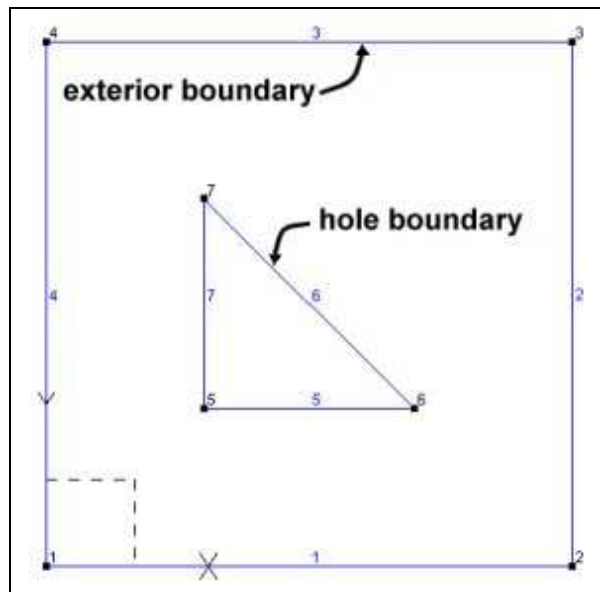
This task begins by creating a square. Now, it must be easy for you.

As previously done, just create a square with side length of 10 units without declaring the surface (just points and lines).

After, create a triangle by entering those three points:

- Point 5: (3, 3, 0, 1.0)
- Point 6: (7, 3, 0, 1.0)
- Point 7: (3, 7, 0, 1.0)

And create the associated lines but not the surface. The final results looks like this:



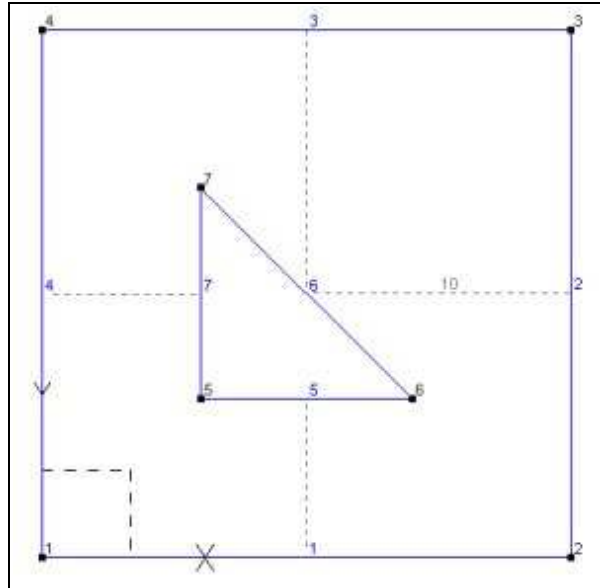
Click on [**Plane surface**] and select the *exterior boundary* (as it is mentioned in the status bar).

Just pick with your mouse a side of the square, then the *line loop* becomes highlighted in red color.

Another message appears in the status bar: *Select hole boundary ('q'=quit)*.

Just pick with your mouse a side of the triangle, then the *line loop* of the triangle becomes highlighted in red color.

Since there is no other hole in our square, just type '**q**' for quitting the *hole boundary* selection and '**q**' again for quitting the *Plane surface* selection.



Finally, save the geometry (*square-hole.geo*) by selecting **[File]** > **[Save as]** in Gmsh menus.

<i>square-hole.geo</i>
<pre> Point(1) = {0, 0, 0, 1}; Point(2) = {10, 0, 0, 1}; Point(3) = {10, 10, 0, 1}; Point(4) = {0, 10, 0, 1}; Point(5) = {3, 3, 0, 1}; Point(6) = {7, 3, 0, 1}; Point(7) = {3, 7, 0, 1}; Line (1) = {1, 2}; Line (2) = {2, 3}; Line (3) = {3, 4}; Line (4) = {4, 1}; Line (5) = {5, 6}; Line (6) = {6, 7}; Line (7) = {7, 5}; Line Loop (1000010) = {2, 3, 4, 1, 6, 7, 5}; Plane Surface (10) = {1000010}; </pre>

II.2. Arcs

These examples shows how to build arcs on the XY plane

II.2.a. Circle arcs

Drawing a circle arc

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A *circle arc* is defined by three points :

- The *start point* of the arc
- The *center point* of the arc
- The *end point* of the arc

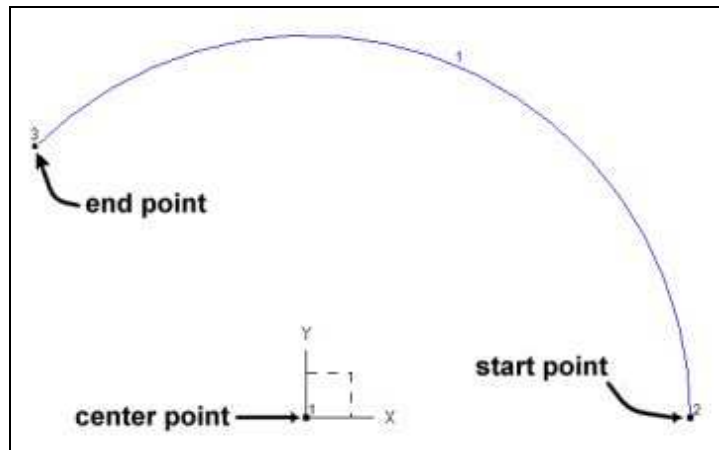
The angular sector must be strictly smaller than π .

Click on **[Elementary]** > **[Add]** > **[New]** > **[Point]**.

Enter the following points in the contextual geometry window :

- Point 1: (0, 0, 0, 1.0)
- Point 2: (10, 0, 0, 1.0)
- Point 3: (-7.07, 7.07, 0, 1.0)

Click on **[Circle arc]**



A message appears in the status bar: *Select start point ('q'=quit)*
 Select the start point (Point 2) with the mouse.
 Another message appears in the status bar: *Select center point ('q'=quit)*
 Select the center point (Point 1) with the mouse.
 Another message appears in the status bar: *Select end point ('q'=quit)*
 Select the end point (Point 3) with the mouse and your arc is drawn.
 Finally, type 'q' for quitting the selection mode.

Drawing a circle

The circle geometrical entity doesn't exist in Gmsh. A circle must be drawn with piece of arcs. At least with 3 arcs, but preferably with 4 arcs (it is easier to manage 90° angles).

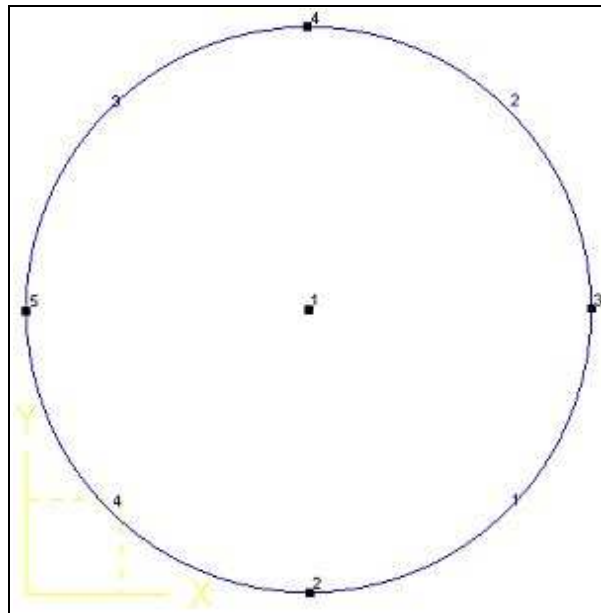
To draw a circle in the XY plane, click on **[Elementary]** > **[Add]** > **[New]** > **[Point]**.
 Enter the following points in the contextual geometry window :

- Point 1: (2, 2, 0, 1.0)
- Point 2: (2, 0, 0, 1.0)
- Point 3: (4, 2, 0, 1.0)
- Point 4: (2, 4, 0, 1.0)
- Point 5: (0, 2, 0, 1.0)

Click on **[Circle arc]** and select the points with this order.

<i>Start point</i>	<i>Center point</i>	<i>End point</i>
Point 2	Point 1	Point 3
Point 3	Point 1	Point 4
Point 4	Point 1	Point 5
Point 5	Point 1	Point 2

The result must look like this:



```

circle.geo

Point(1) = {2, 2, 0, 1};
Point(2) = {2, 0, 0, 1};
Point(3) = {4, 2, 0, 1};
Point(4) = {2, 4, 0, 1};
Point(5) = {0, 2, 0, 1};
Circle (1) = {2, 1, 3} Plane{0, 0, 1};
Circle (2) = {3, 1, 4} Plane{0, 0, 1};
Circle (3) = {4, 1, 5} Plane{0, 0, 1};
Circle (4) = {5, 1, 2} Plane{0, 0, 1};

```

II.2.b. Ellipse arcs

Drawing an ellipse arc

An *ellipse arc* is defined by four points :

- The *start point* of the arc
- The *center point* of the ellipse
- The *major axis point* of the ellipse (a point belonging to the major axis)
- The *end point* of the arc

The eccentric angle (the angular sector measured from the center) must be strictly smaller than Pi.

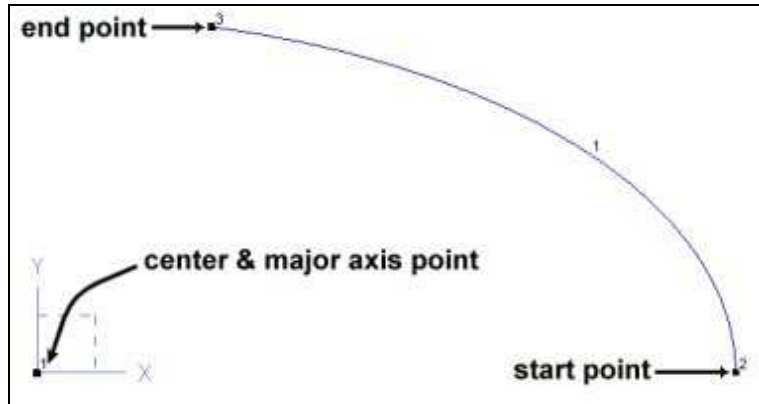
Click on **[Elementary]** > **[Add]** > **[New]** > **[Point]**.

Enter the following points in the contextual geometry window :

- Point 1: (0, 0, 0, 1.0)
- Point 2: (4, 0, 0, 1.0)
- Point 3: (1, 2, 0, 1.0)

Note: center point and the major axis point are the same but you can use four points if you want.

Click on **[Ellipse arc]**



A message appears in the status bar: *Select start point ('q'=quit)*
 Select the start point (Point 2) with the mouse.
 Another message appears in the status bar: *Select center point ('q'=quit)*
 Select the center point (Point 1) with the mouse.
 Another message appears in the status bar: *Select major axis point ('q'=quit)*
 Select the major axis point (Point 1 again) with the mouse.
 Another message appears in the status bar: *Select end point ('q'=quit)*
 Select the end point (Point 3) with the mouse and your ellipse arc is drawn.

Finally, type '**q**' for quitting the selection mode.

Here is the source code:

```
ellipse-arc.geo
```

```
Point(1) = {0, 0, 0, 1};
Point(2) = {2, 0, 0, 1};
Point(3) = {0.5, 1, 0, 1};
Ellipse (1) = {2, 1, 1, 3} Plane{0, 0, 1};
```

Drawing an ellipse

As for the circle, the ellipse geometrical entity doesn't exist in Gmsh. An ellipse must be drawn with piece of arcs. At least with 3 arcs, but preferably with 4 arcs (it is easier to manage 90° angles).

To draw an ellipse in the XY plane, click on **[Elementary]** > **[Add]** > **[New]** > **[Point]**.
 Enter the following points in the contextual geometry window :

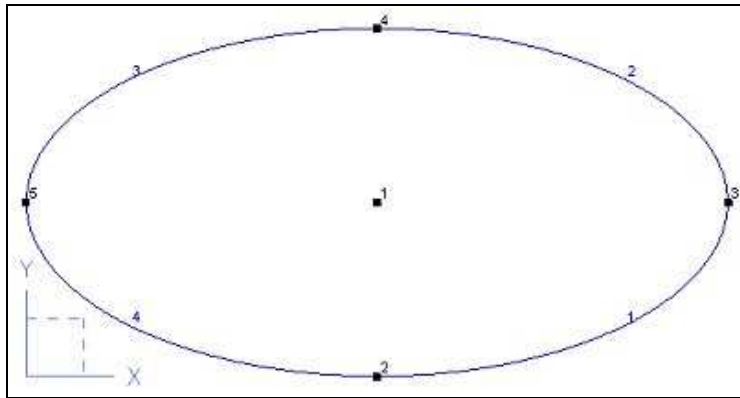
- Point 1: (2, 1, 0, 1.0)
- Point 2: (4, 2, 0, 1.0)
- Point 3: (4, 2, 0, 1.0)
- Point 4: (2, 4, 0, 1.0)

- Point 5: (0, 2, 0, 1.0)

Click on [**Ellipse arc**] and select the points with this order.

<i>Start point</i>	<i>Center point</i>	<i>Major axis point</i>	<i>End point</i>
Point 2	Point 1	Point 1	Point 3
Point 3	Point 1	Point 1	Point 4
Point 4	Point 1	Point 1	Point 5
Point 5	Point 1	Point 1	Point 2

The result must look like this:



<i>ellipse.geo</i>	
Point(1) =	{ 2, 1, 0, 1 };
Point(2) =	{ 2, 0, 0, 1 };
Point(3) =	{ 4, 1, 0, 1 };
Point(4) =	{ 2, 2, 0, 1 };
Point(5) =	{ 0, 1, 0, 1 };
Ellipse (1) =	{ 2, 1, 1, 3 } Plane{ 0, 0, 1 };
Ellipse (2) =	{ 3, 1, 1, 4 } Plane{ 0, 0, 1 };
Ellipse (3) =	{ 4, 1, 1, 5 } Plane{ 0, 0, 1 };
Ellipse (4) =	{ 5, 1, 1, 2 } Plane{ 0, 0, 1 };

II.3. Spline

REMARK : IN THE CURENT VERSION OF SPIS/UI, SPLINE ARE NOT SUPPORTED. DO NOT TRY TO IMPORT THEM IN THE CAD DATA STRUCTURE.

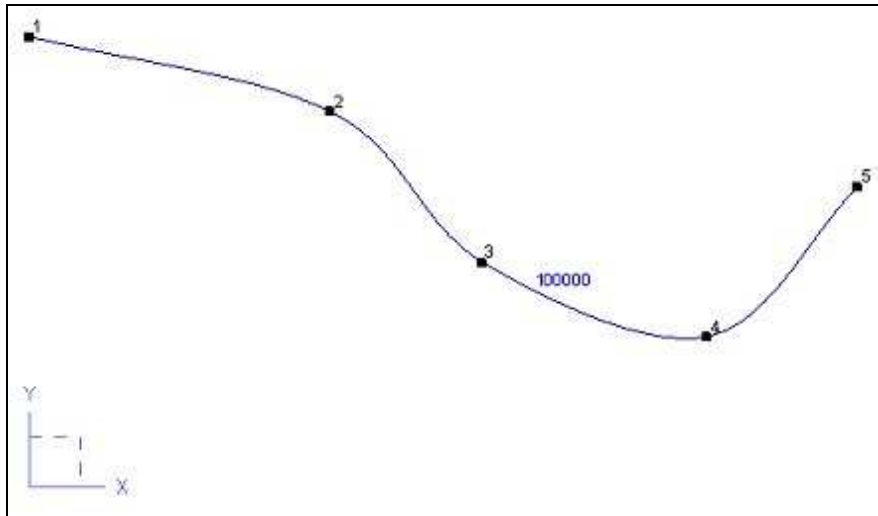
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Spline curves implemented in Gmsh are Catmull-Rom splines. That is to say the curve will pass through all of the *control points*.

To draw an spline curve, click on **[Elementary]** > **[Add]** > **[New]** > **[Point]**.
Enter the *control points* in the contextual geometry window :

- Point 1: (0, 6, 0, 1.0)
- Point 2: (4, 5, 0, 1.0)
- Point 3: (6, 3, 0, 1.0)
- Point 4: (9, 2, 0, 1.0)
- Point 5: (11, 4, 0, 1.0)

Click on **[Spline]** .



A message appears in the status bar: *Select point ("e'=end, q'=quit)*
 Select the first point (Point 1) with the mouse.
 The same message appears again in the status bar: *Select point ("e'=end, q'=quit)*
 Select the second point (Point 2) with the mouse.
 The same message appears again.
 Etc
 Finally, type 'e' for ending the list of *control points* and 'q' for quitting the selection mode.

Note: In this example we have declared one spline curve but its ID is 100000. Don't panic, you haven't created 99999 other invisible curves ! It's only related to the internal numbering of geometrical entities in Gmsh.

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II.4. B-spline

REMARK : IN THE CURENT VERSION OF SPIS/UI, B-SPLINE ARE NOT SUPPORTED. DO NOT TRY TO IMPORT THEM IN THE CAD DATA STRUCTURE.

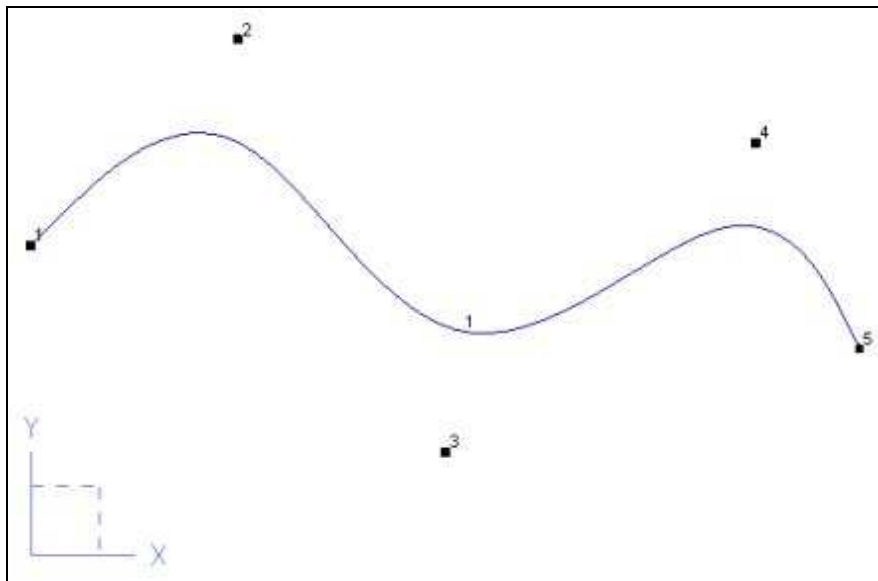
For the B-Spline, the curve doesn't pass through the *control points* by default. If you want to force a curve to pass trough or nearer a *control point*, you must repeat its selection.

To draw an B-spline curve, click on **[Elementary]** > **[Add]** > **[New]** > **[Point]**.
Enter the *control points* in the contextual geometry window :

- Point 1: (0, 3, 0, 1.0)
- Point 2: (2, 5, 0, 1.0)
- Point 3: (4, 1, 0, 1.0)
- Point 4: (7, 4, 0, 1.0)
- Point 5: (2, 2, 0, 1.0)

In the example below, we force the curve to pass trough the first and the last point by repeating three time their selection.

Click on **[B-Spline]**.



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A message appears in the status bar: *Select point ('e'=end, q'=quit)*
 Select the first point (Point 1) with the mouse.
 The same message appears again in the status bar: *Select point ('e'=end, q'=quit)*
 Select the first point (Point 1) again and another one with the mouse.
 The same message appears again in the status bar: *Select point ('e'=end, q'=quit)*
 Select the second point (Point 2) with the mouse.
 The same message appears again.
 Etc
 Select the last point (Point 5) with the mouse.
 And select it again and another one.

Finally, type 'e' for ending the list of *control points* and 'q' for quitting the selection mode.

D. Annexe A: Acronyms

ASCII	American Standard Code for Information Interchange
Breps	Boundary representations
CAD	Computer Aided Design
GPL	Gnu General Public License
PLSV	Point Line Surface Volume