Imperial College London

Meshing

Fluidity training

Applied Modelling and Computation Group

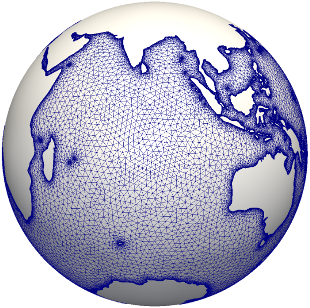
Alexandros Avdis

Department of Earth Science and Engineering, Imperial College London

4-6 November 2013

Tutorial overview

- What is a mesh.
- What is Gmsh.
- Viewing and meshing a 3–D geometry.
- Generating and meshing a 2–D geometry.
- Meshing realistic domains.



but, before we start...

- The presents slides are freely available, as well as a more detailed tutorial we have prepared.
- Open a terminal and issue the following commands:

mkdir meshingTutorial
(to create a directory for this tutorial)
cd meshingTutorial
bzr cat lp:fluidity/training/gmsh_tutorial_pres.pdf > gmsh_tutorial_pres.pdf
(to fetch a pdf copy of the present slides)
bzr cat lp:fluidity/training/gmsh_tutorial.pdf>gmsh_tutorial.pdf
(to fetch a pdf copy of the tutorial document)
evince gmsh_tutorial_pres.pdf &
evince gmsh_tutorial.pdf &

- Keep this terminal open, as we will come back to it.
- Fell free to consult the slides (and/or the document) at any time!

What is a mesh?

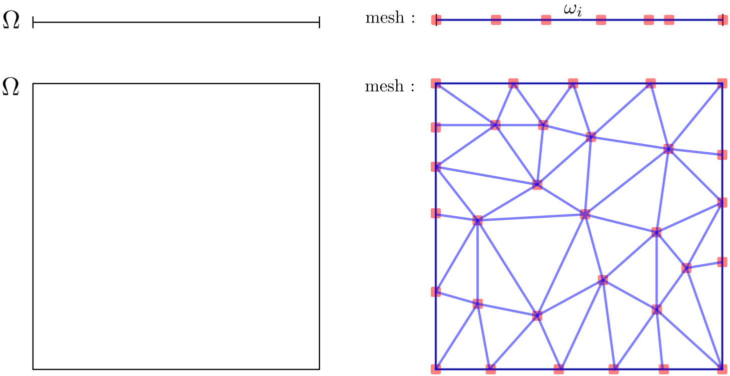
A mesh can be qualitatively thought of as the tessellation of a domain Ω into a set of non-overlapping sub-domains ω_i:

$$\begin{aligned} \Omega &= \cup\{\omega_i \,|\, i = 1, 2, \ldots ele\} \\ 0 &= \cap\{\omega_i \,|\, i = 1, 2, \ldots ele\} \end{aligned} \tag{1}$$

where *ele* is the number of elements in the tessellation.

What is a mesh?

A mesh can be qualitatively thought of as the tessellation of a domain Ω into a set of non-overlapping sub-domains ω_i:



What is Gmsh?

- It is the role of the mesh (or grid) generator to scatter the points and generate the mesh, whilst ensuring high quality elements.
- Gmsh is a "3D finite element grid generator with a build-in CAD engine and post-processor. Its design goal is to provide a fast, light and user-friendly meshing tool with parametric input and advanced visualization capabilities."[1]. Furthermore, Gmsh can be used as a 1–, 2– and 3– dimensional mesh generator for use with the Fluidity CFD code.
- Not the only mesh generator that can be used with Fluidity.
  - Fluidity can also read meshes in ExodusII format.
- Distributed under the GNU General Public License, available for Linux, Windows and Mac OS.

[1] from `http://www.geuz.org/gmsh/`

Starting Gmsh

- Go back to your terminal and issue the following commands:

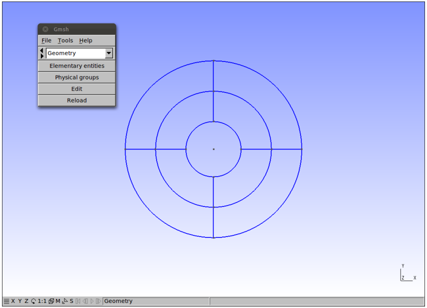
bzr cat lp:fluidity/training/gmsh/annulus.geo>annulus.geo

(to fetch an example-file)

gmsh annulus.geo &

(To open the example-file with Gmsh, the ampersand is important.)

- Gmsh will open annulus.geo, two windows will appear:
  - The Menu window (smaller).
  - The Graphic window (larger).



Viewing and meshing a 3D geometry: Navigating menus.

- Gmsh's architecture is centred around four modules, this is reflected in the menu window.
- The menu widow can be used to switch between the different modules, but you are encouraged to switch between modules by using the keyboard (letters in parentheses below.)
  - Geometry (*G*): For defining domain geometry.
  - Mesh (*M*): For building the mesh.
  - Solver (*S*).
  - Post–Processing (*P*).
  - Practical: Try switching between different modules.

Viewing and meshing a 3D geometry: Manipulating the view

- Panning : Hold right button down and move cursor.
- Zooming : Scroll or hold middle button down and move cursor.
- Rotating : Hold left button down and move cursor.
- Practical : Try modifying the view.

Viewing and meshing a 3D geometry: Getting and saving the mesh

- Mesh the annulus
  - Mesh (M) $>$ 3D
  - Once again, try modifying the view
- To save the mesh click on File (menu window) and select Save Mesh.
  - This creates a file “annulus.msh”, storing the mesh.
- More information on the annulus is available in the tutorial document fetched earlier.
  - Including instructions how to build the geometry.

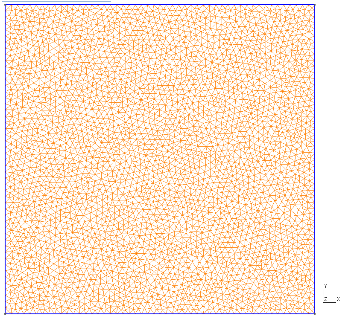
Viewing and meshing a 3D geometry: The various files

- annulus.geo : Stores the geometry as an ASCII script file.
  - Can be edited (try cat annulus.geo at your terminal).
- annulus.msh : Stores the mesh
  - Also contains tags (numerical ID) on element vertices, edges and faces that we can use to assign boundary conditions.
  - Can be ASCII or binary.

- Fluidity can also read meshes in “triangle” format
  - Issue /path/to/fluidity/bin/gmsh2triangle annulus.msh
  - This will generate annulus.edge, annulus.ele, annulus.node files.
  - Issue ls -l at your terminal.
- Fluidity can also read meshes in ExodusII format.

Generating and meshing a 2D geometry: Aim

- To generate a 2–D, structured mesh on a $1,000$km $\times 1,000$km square.
- Typical element size (edge length): 20km.
- Mesh will be used in subsequent simulations/examples in this training event.



Generating & meshing a 2D geometry: Getting started

Close your existing Gmsh instance and start a new one from your terminal:

bzr cat lp:fluidity/training/gmsh/gmsh_tutorial_presentation/2d-example-tutor.geo>2d-example-tutor.geo (This fetches a file to be used by the tutors when helping you!)

gmsh 2d-example.geo & (The ampersand is important)

We proceed by defining:

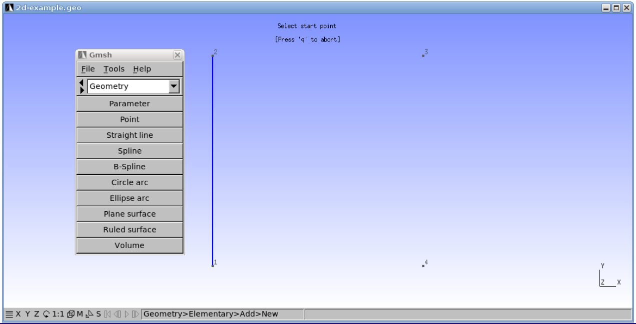
1. Points
2. Lines
3. Surfaces

Generating & meshing a 2D geometry: Creating points

- Geometry (G) > Elementary Entities > Add > New > Point
- The Contextual Geometry Definitions window will appear.
- Enter the point coordinates and click "Apply".
  - Do not move cursor outside Contextual Geometry Definitions window while entering coordinates! Hold shift down if you have to.
  - Always look at the instructions shown in the graphic window.
  - Point 1: [0.0 , 0.0 , 0.0] with characteristic length of 2e4
  - Point 2: [0.0 , 1.e6, 0.0] with characteristic length of 2e4
  - Point 3: [1.e6, 1.e6, 0.0] with characteristic length of 2e4
  - Point 4: [1.e6, 0.0 , 0.0] with characteristic length of 2e4
- Press 'q' and close Contextual Geometry Definitions window.

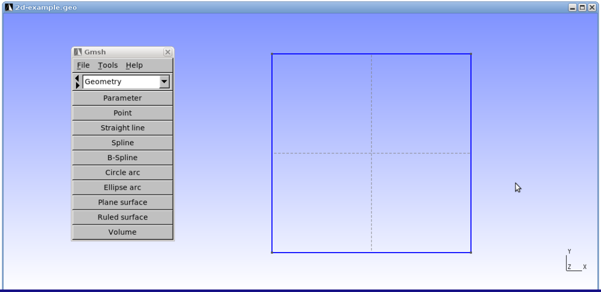
Generating & meshing a 2D geometry: Creating lines

- Geometry > Elementary Entities > Add > New > Straight Line
  - Draw a line between two points by selecting the points.
  - Join points (1,2), (2,3), (3,4), (4,1)
  - Once all lines are drawn, press ‘q’
  - Always look at the instructions shown in the graphic window.



Generating & meshing a 2D geometry: Declaring a plane surface.

- Geometry(G) > Elementary Entities > Add > New > Plane Surface
- Click on any of the sides, all sides should be highlighted in red.
- Press ‘e’ then ‘q’.
- Gmsh will highlight the surface with grey, dash-lines.



Generating & meshing a 2D geometry: Declaring physical groups.

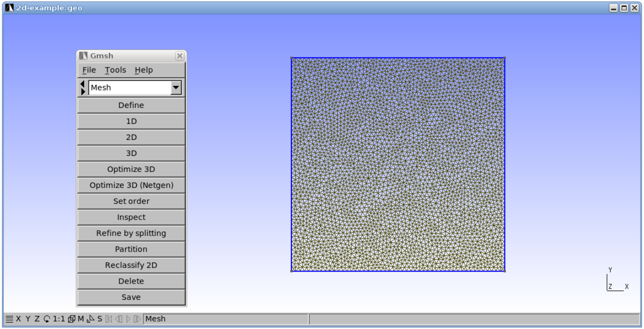
In order to specify regions and boundaries in Fluidity, they must first be defined as “Physical Groups” in Gmsh:

- Assign “Physical Line” ID’s to the domain boundaries.
  - Geometry (G) > Physical Groups > Add > Line
  - Select bottom side and press ‘e’.
  - Select right side and press ‘e’.
  - Select top side and press ‘e’.
  - Select left side and press ‘e’.
  - Once you have done all sides press ‘q’.

- Assign “Physical Surface” ID to the plane surface.
  - Geometry (G) > Physical Groups > Add > Surface
  - Select the highlighted surface: Click on the grey dash lines.
  - Press ‘e’ then ‘q’.

Generating & meshing a 2D geometry: Producing a mesh

- To produce a 2–D mesh: Mesh (M) $>$ 2D
- To save the mesh: Click on File (menu window) and select Save Mesh.
- Convert to triangle format: /path/to/fluidity/bin/gmsh2triangle 2d-example.msh



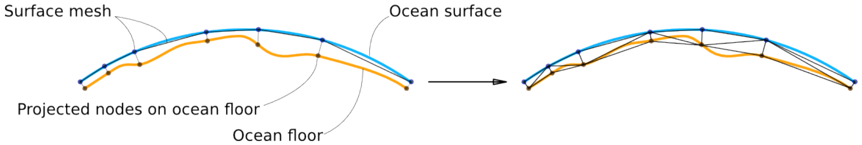
Meshing realistic ocean domains: Key points

- A Gmsh user typically has to specify two essential parts:
  - Domain shape.
  - Characteristic element size.

- In domains representative of oceans:
  - The geometry is very complex, boundaries are fractal–like
  - Ideal characteristic element size can also be dependent on many parameters, for example:
    - Depth
    - Ocean floor topography
    - Explicit requirement in order to resolve tidal turbine array, etc.

Meshing realistic ocean domains: Our approach so far

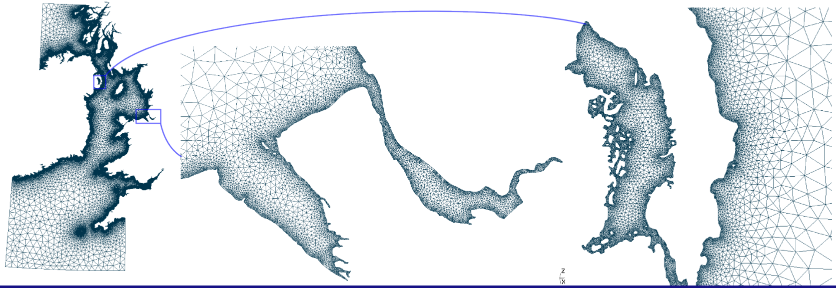
- Domain geometry & Mesh are constructed on a spherical shell, representative of the Earth's surface geoid.
- See fetched Gmsh tutorial document.
  - Coastlines extracted from GSHHS dataset, via Gmsh plug–in.
  - Open boundaries are drawn, usually constant longitude & constant latitude lines.
  - Element size is defined using "attractor" and "threshold" fields available in Gmsh.
  - 2–D surface mesh is extruded along the radial direction within Fluidity.
- Mesh is "vertically extruded" within Fluidity.



Shortcomings – Domain boundaries.

A simple CAD engine is insufficient:

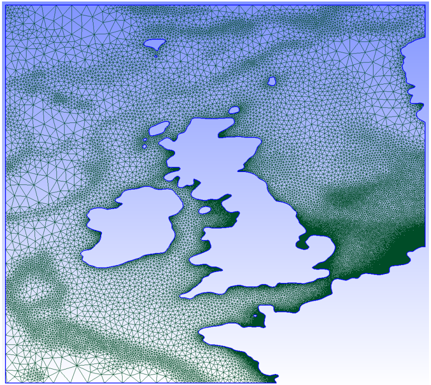
- Shorelines are geometrically very complex.
- The Gmsh GSHHS plug–in fits a spline through the GSHHS points, leading to intersecting shorelines (support for Gmsh GSHHS plugin now dropped).
- Drawing arbitrary lines as open boundaries –e.g. contour at a given depth– not easily done.



Shortcomings – Mesh size

The spatial variation of the mesh element size could be complex:

- Mesh element size must usually be fine near the coastlines to capture their structure.
- Mesh element size must usually be fine in areas of steep ocean floor topography and in shallow areas: Not easily done with current approach.



Meshing realistic domains – Our proposed approach

Use Geographical Information Systems to extract domain boundaries and prescribe mesh metric size.

- Existing GIS software capable of reading databases in popular formats.
- Capability of extracting contours from field-type databases is usually available. ($\rightarrow$domain boundaries)
- Capability of generating field-type databases is usually available. ($\rightarrow$mesh size metric)
- A project bringing together GIS software with mesh generation software has been registered by AMCG:
  - `https://launchpad.net/meshing`
  - Publication coming soon.
  - Watch this space!

Further reading

- AMCG, A Gmsh tutorial
- C. Geuzaine and J.-F. Remacle, Gmsh: a three-dimensional finite element mesh generator with built-in pre- and post-processing facilities.. International Journal for Numerical Methods in Engineering, Volume 79, Issue 11, pages 1309-1331, 2009.
- C. Geuzaine and J.-F. Remacle, Gmsh Reference Manual.. Available at, `http://geuz.org/gmsh/#Documentation`.
- A. Avdis and S.L.Mouradian, A Gmsh tutorial, available over launchpad.

Questions?

AMCG:

`http://amcg.ese.ic.ac.uk/`

Fluidity:

`http://amcg.ese.ic.ac.uk/Fluidity`

Fluidity on launchpad:

`https://launchpad.net/fluidity`