GMSH Workshop: Ocean

J. Lambrechts and B. Seny

Introduction

The aim of this workshop is to show how to build ocean meshes with GMSH. Here are some useful informations about the tools that are needed to perform the workshop.

- GSHHS is a high-resolution shoreline data set amalgamated from two databases in the public domain. The data have undergone extensive processing and are free of internal inconsistencies such as erratic points and crossing segments. The shorelines are constructed entirely from hierarchically arranged closed polygons. The data can be used to simplify data searches and data selections, or to study the statistical characteristics of shorelines and land-masses. It comes with access software and routines to facilitate decimation based on a standard line-reduction algorithm.
  `http://www.ngdc.noaa.gov/mgg/shorelines/gshhs.html`

- The old plugin in GMSH is broken and difficult to manipulate. Here we will use a simple python script and Inkscape for editing the SVG files (you could use other SVG editors).
  `http://inkscape.org/`

- Our python scripts are faster if ANN (A Library for Approximate Nearest Neighbor Searching) is available. But swig is needed. Go to `GMSH2011/Exercises/ocean/ann` and launch the `MakeFile` (`make -f MakeFile.osx` for MAC). You will probably have to adapt the `MakeFile`. However this library is not required for this workshop, it just accelerates some computations.
  `http://www.cs.umd.edu/~mount/ANN/`

- For the world ocean we will use stereographic coordinates.

1 Meshing the world ocean

We will start with the world ocean that is a simple case because no external boundaries have to be defined.

1.1 Create a uniform mesh

- Go to `GMSH2011/Exercises/ocean`
- `tar xjf gshhs+wdbii_2.2.0.tbz`
- Lauch the python script: `gshhs2geo.py`
- Visualize the geometry in the GMSH interface: `gmsh world.geo`
- Create a plane surface with the GMSH interface → **Geometry** → **Elementary Entities** → **Add** → **New** → **Plane Surface**. In steorographic space Antarctica is the external boundary so we will use it here. After that use CTRL drag to select all other continents as holes.
- Create a uniform coarse mesh : → **Mesh** → **2D**

1.2 Play with the Fields to locally adjust the mesh size

One way to specify the element sizes in GMSH is by using the Fields. To access the Field interface go to → **Mesh** → **Define** → **Fields**.

To have information about the different kinds of Fields you can click on the **Help** button when you create a new Field. You can combine different Fields by including them into other Fields. As an example you could define two arbitrary Fields F1 and F2 and take the maximum of these two Fields by creating a third Field F3: → **New** → **Max**. Put then Field 1 and 2 in the **FieldList**. Don't forget to click on the apply button to save the options. Among all the Fields you have to choose a unique Background Field.

1. Before defining the actual Fields we have to set up 2 options in the `.geo` file. To edit the file you can use **Geometry** → **Edit** and **Geometry** → **Reload**:
   - Integrating with a high precision the objective mesh size on the boundary to generate the 1D mesh can be expensive. It is not necessary in our case because this size is almost constant, so we reduce the integration precision.
     `Mesh.LcIntegrationPrecision = 1e-3;`

- If `Mesh.CharacteristicLengthExtendFromBoundary` is set, GMSH only computes the objective mesh size on the boundary then interpolates this size inside the domain, so we disable this option.
  `Mesh.CharacteristicLengthExtendFromBoundary = 0;`

2. First we want to increase the resolution near the coastlines to capture them more accurately. Two Fields are needed to do that:

   - **Field 1**: By default the script defined an attractor Field that computes the distance from the closest coastal node.
   - **Field 2**: Add a Treshold Field to define a refinement depending on **Field 1** (distance to the coast): → **New** → **Treshold**. Check the radio button to define it as the Background Field. You can use the following parameters:

DistMax	=	6000	km
DistMin	=	300	km
IField	=	1	
LcMax	=	1000	km
LcMin	=	100	km

3. You can now remesh the domain (do not forget to regenerate the 1D mesh also). You can try different 2D meshing algorithms: → **Tools**→ **Options**→ **Mesh** → **General** → **2D algorithm**. Once we have a mesh we can use it to visualize the Fields: select one Field and use the **Visualize** button.

4. Another other refinement criterium commonly used in oceanography is to consider the square root of the bathymetry because it prescribes the tidal wave propagation velocity. To do that we will define 4 new Fields:

   - **Field 3**: A **Structured Field** is used to load the bathymetry file `etopo2.bin`. This file contains the bathymetry data obtained from: `http://www.ngdc.noaa.gov/mgg/fliers/01mgg04.html`. The data was converted in the format described in the Field documentation.
   - **Field 4**: Create a **LonLat Field** to convert the coordinates of **Field 3** to the sphere.
   - **Field 5**: Create a **Treshold Field** to remove the depths that are smaller than 0 or greater than 5000. Use the following parameters:

DistMax	=	5000	m
DistMin	=	0	m
IField	=	4	
LcMax	=	5000	m
LcMin	=	0	m

- **Field 6**: Finally use a **MathEval Field** to take the square root of the previous Field and combine it with the distance criterium (**Field 2**).

$$(0.5 + 2 * Sqrt(F5/5000)) * F2$$

Define this Field as the background field and regenerate the 1D and 2D mesh.

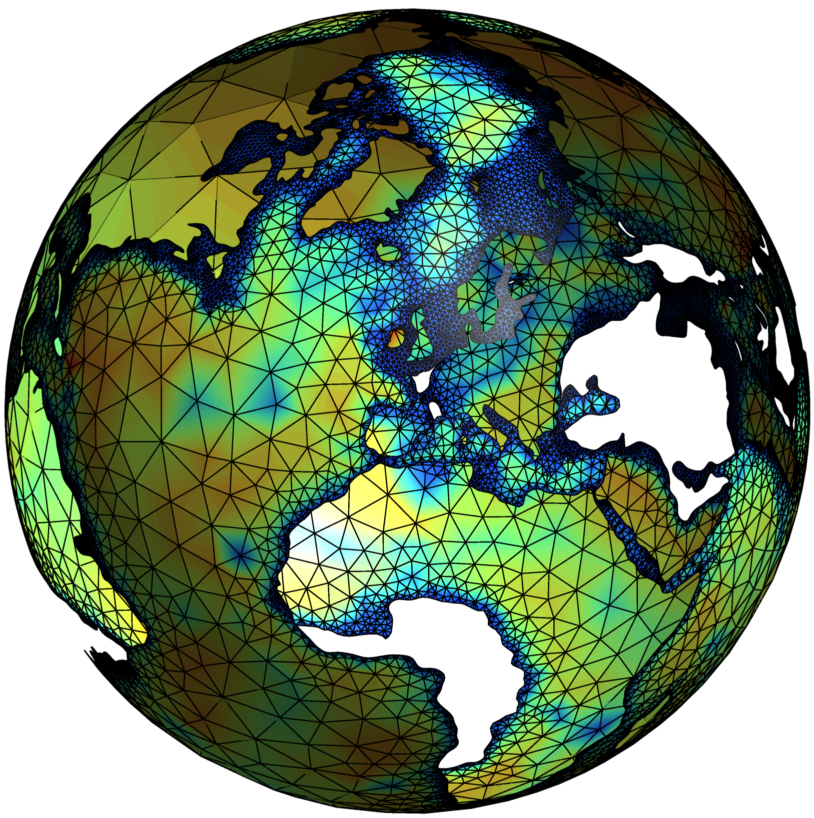


Figure 1: Mesh of the world ocean made up of 22140 triangles with the bathymetry as background field.

2 Pre-process the boundaries: remove the Panama Canal

GMSH is not so good as a CAD editor. In this second exercise we will learn how to use the vector graphics editor Inkscape to manipulate and correct the coastlines.

1. Install, configure and use Inkscape:

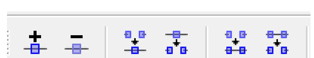
- Install Inkscape:
  - For LINUX use the package manager of your distribution.
  - For MACOSX open the `Inkscape-0.48.2-1-SNOWLEOPARD.dmg` file.
- Open `world.svg` with Inkscape. The world appears in stereographic coordinates. We can start editing the coastlines.
- The python script can only parse a very limited subset of the SVG objects. Coasts are converted in line segment, the only object in Inkscape that will be reinterpreted by the script. After that, in GMSH, they will be there converted in NURBS. Some options in Inkscape have to be set to keep the SVG file readable. For this go to → **File** → **Inkscape Preferences**

  → **SVG output**: disable "Allow relative coordinates".

  → **Tools** → **Node**: disable "Deleting nodes preserves shape"
- When two different paths share an ending node this node node is duplicated in Inkscape. Both copies have to be in the exact same location so that the script can identify them as a unique node. To do that, in the Inkscape interface, enable "Snap to cusp nodes" on the toolbar on the right of the screen and take care that the "handle to cusp node" message appears. In french "cusp nodes" is translated as "points de rebroussement". You will need to reactivate this option each time you open a new SVG file.

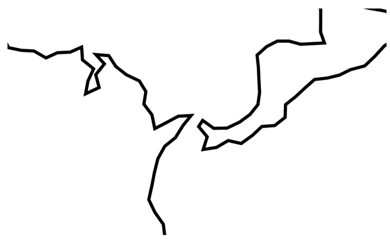
- You can select the paths and move the nodes with the "Edit paths by nodes"  tools. Use the five buttons on the upper left toolbar to insert nodes, delete nodes, join nodes, break paths, join paths or delete segments .

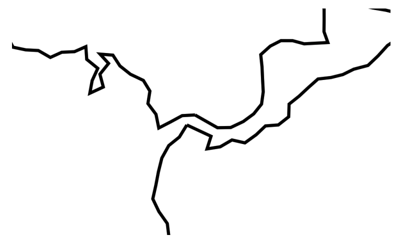
  By using shift and the mouse you can select multiple points. To add a new path use the straight line button . Take care to not create curved paths accidentally. After splitting a path in two new paths, it is necessary to put these new paths in separated objects. To do this go to the menu bar → **Path** → **Break Apart**.
- The "Title" property of the path will be interpreted as a physical tag in GMSH. You can edit these properties by going to the menu bar → **Object** → **Object Properties**.

2. Bind North and South America by removing the Panama Canal.

   As an exercise define physical tags for North and South America and remove the Panama Canal. Then save the SVG file and launch the python script `svg2geo.py`. It generates a new file `world2.geo`. Open it and check the physical tags by going to → **Tools** → **Visibility** → **Physical Groups**



(a)



(b)

3 Choose your favorite sea and mesh it!

Here we propose you to choose a sea and generate a mesh on it. Therefore you may need a higher resolution of the coastlines. You can change the GSHHS file in the `gshhs2geo.py` script. Each resolution is stored in the `/GMSH2011/Exercises/ocean/gshhs` subdirectory. There are 5 different levels of resolution sorted from the highest to the lowest: f, h, i, l, c. Once you have chosen the region of the world that you want to mesh you need to precise the corresponding minimum and maximum longitude and latitude in the `gshhs2geo.py` file in the `edgeLength` function. You can choose the LonLat coordinate system instead of the stereographic one by uncomment and comment the right lines in the `gshhs2geo.py` and `svg2geo.py` files. Use Inkscape to create open sea boundaries by cutting paths and recombine them with straight lines. After that define, as in the first exercise, appropriate Fields and generate the mesh.