## **Chapter 1**

## **Unstructured 2D Delaunay mesh** generation with xfig and Triangle

In this document we demonstrate how to use oomph-lib's conversion routine fig2poly (a C++-based standalone executable, generated from the source code fig2poly.cc to create \*.poly files for Jonathan Shewchuk's open-source mesh generator Triangle , based on the output from the open-source drawing program xfig.

## 1.1 Mesh generation with xfig, fig2poly and Triangle

Mesh generation with xfig, fig2poly and Triangle is extremely straightforward.

- 1. Draw the (piecewise linear) domain boundaries, using <code>xfig's</code> polyline drawing tool. Each polyline represents a distinct mesh boundary. The start and end points of the polyline should not coincide <code>fig2poly</code> will automatically "fill in" the missing line segment.
- 2. If the domain has any holes, place a single circle (drawn with xfig's circle/ellipse drawing tool) into each hole. Use the circle/ellipse drawing tool that requires the specification of the radius. The radius of the circle is irrelevant.
- 3. Save the figure as a \*. fig file.
- 4. Use fig2poly to convert the \*. fig file into a \*. fig. poly file. For instance, ./fig2poly some\_figure.fig
  will generate a file some\_figure.fig. poly.
- 5. Process the \*. fig. poly file produced by fig2poly with Triangle and use the resulting \*.poly, \*.ele and \*.node files as input to oomph-lib's TriangleMesh, as described in another example.
- 6. Done!

#### 1.1.1 Comments:

- fig2poly expects the xfig output file to conform to "Fig format 3.2" and checks for the presence of the string "#FIG 3.2" in the first line of the \*. fig file. below.
- The figure must contain only polylines and circles. The presence of any other objects will spawn an error message and cause fig2poly to terminate.
- The figure must not contain any "compound objects". Compound objects may, of course, be used while drawing but you should break them up before saving the figure.

# 1.2 Example 1: Solution of Poisson's equation on a rectangular domain with a hole

Here is a screen shot from an xfig session. The figure defines a quadrilateral domain with a quadrilateral hole.

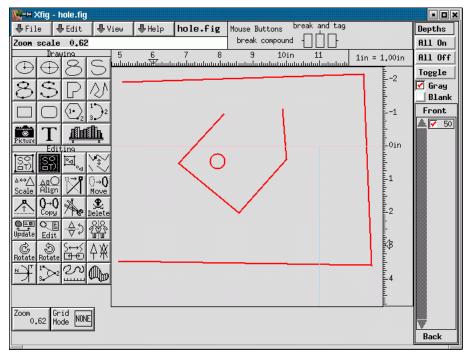


Figure 1.1 Screen shot of xfig session.

Here is a plot of the resulting mesh. It was generated by converting the file hole.fig generated by xfig, to hole.fig.poly, using fig2poly and processing the resulting file with triangle -pq -a0.02 hole.fig.poly

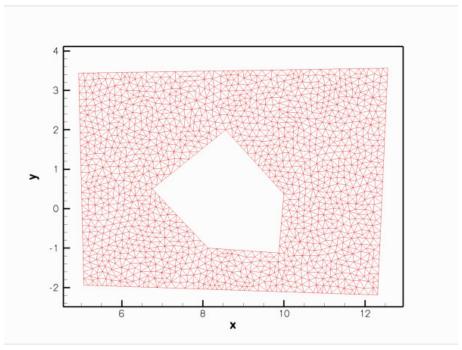


Figure 1.2 The mesh.

Finally, the figure below shows a plot of the computed solution of a Poisson equation with a unit source function, obtained with three-noded (red) and six-noded (green) triangular Poisson elements:

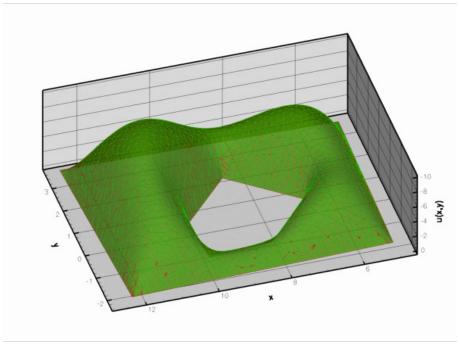


Figure 1.3 Solution of Poisson's equation on the mesh generated from the xfig output.

This solution was computed with the driver code <code>mesh\_from\_xfig\_poisson.cc.</code>

## 1.3 Example 2: Finite-Reynolds-number flow past the oomph-lib logo

Here is a screen shot from another xfig session. The figure defines a quadrilateral domain containing the oomph-lib logo whose letters create holes in the domain.

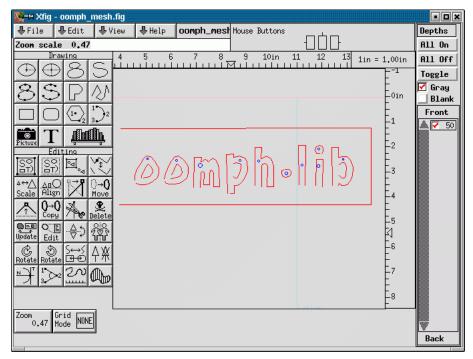


Figure 1.4 Screen shot of xfig session

Here is a plot of the mesh generated with the same procedure discussed above.

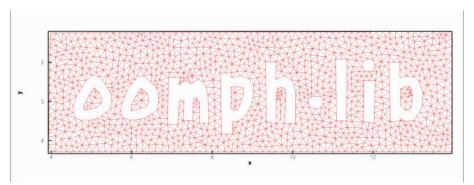


Figure 1.5 The mesh.

Finally, the figure below shows a plot of the solution of the steady Navier-Stokes equations (velocity vectors and pressure contours) in this domain. No-slip conditions were applied on all boundaries. Zero velocities were imposed on all boundaries apart from the outer bounding box (boundary 1) where we set

$$\mathbf{u} = \left( \begin{array}{c} -1 \\ -1 \end{array} \right).$$

The plot may therefore be interpreted as showing the flow field that is generated when the rigid quadrilateral box that surrounds the oomph-lib logo moves in the north-westerly direction while the logo itself remains stationary. This was computed with the driver code <code>mesh\_from\_xfig\_navier\_stokes.cc</code> which is very similar to that for the driven cavity problem, so we shall not discuss in in detail.

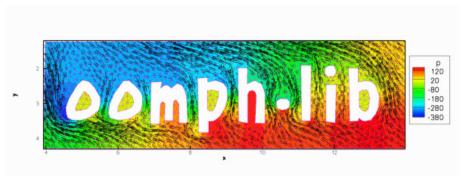


Figure 1.6 Flow past the oomph-lib logo.

### 1.4 Comments and Exercises

## 1.4.1 Numbering of the mesh boundaries

- 1. Each polyline in the xfig-generated figure represents a distinct mesh boundary. fig2poly assigns boundary numbers to these, depending on the order in which the polylines are listed in the \*. fig file. Since this is not always obvious, it is usually necessary to examine the mesh boundaries by calling Mesh :: output\_boundaries (...) before assigning boundary conditions.
- 2. Since boundaries are defined by closed polygons, all nodes that are located on a specific polygon have the same boundary number. In cases where a finer sub-division of the boundary is required (e.g. to identify inflow boundaries) some post-processing of the mesh may be required.

#### 1.4.2 Exercises

- 1. Download and install xfig (if you work in a linux environment, xfig is likely to be available already as it is part of many linux distributions) and create your own meshes.
- 2. Think of a way to modify the xfig-based mesh generation procedure so that a closed boundary can contain several sub-boundaries so that in- and outflow boundaries can be identified separately. [This is an "advanced" exercise and one to which no solution exists as yet please let us have your code if you find an easy way to do this!].

## 1.5 PDF file

A pdf version of this document is available.