

# 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 `xfig`'s polyline drawing tool. Each polyline represents a distinct mesh boundary. The start and end points of the polyline should not coincide – `fig2poly` 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 [mesh\\_from\\_xfig\\_poisson.cc](#).

### 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

$$\mathbf{u} = \begin{pmatrix} -1 \\ -1 \end{pmatrix}.$$

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 `mesh_from_xfig_navier_stokes.cc` which is very similar to that for the `driven cavity problem`, so we shall not discuss 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.