# The Durham Fluent 12 Tutorial

## *Introduction*

This provides a short step by step guide to modelling an aerofoil using the Fluent package. The aim being to introduce you the important features of Fluent in the shortest possible space of time. This tutorial is for Fluent 12 earlier tutorials used Fluent 6 which has a significantly different layout although "under the hood" things are very similar.

## *Computing the Flow*

This step is much more straightforward than meshing. In CFD modelling flow most of your time will be taken meshing the problem and then analysing the results, computation is the easy bit.

- Start Fluent by typing ***fluent*** at a terminal on the vega service.

- This pops up a window in which you can set your working directory and so on. You need to ensure that under "Dimension" the 2D radio button is checked. After a short while something like Figure 1 will pop up.
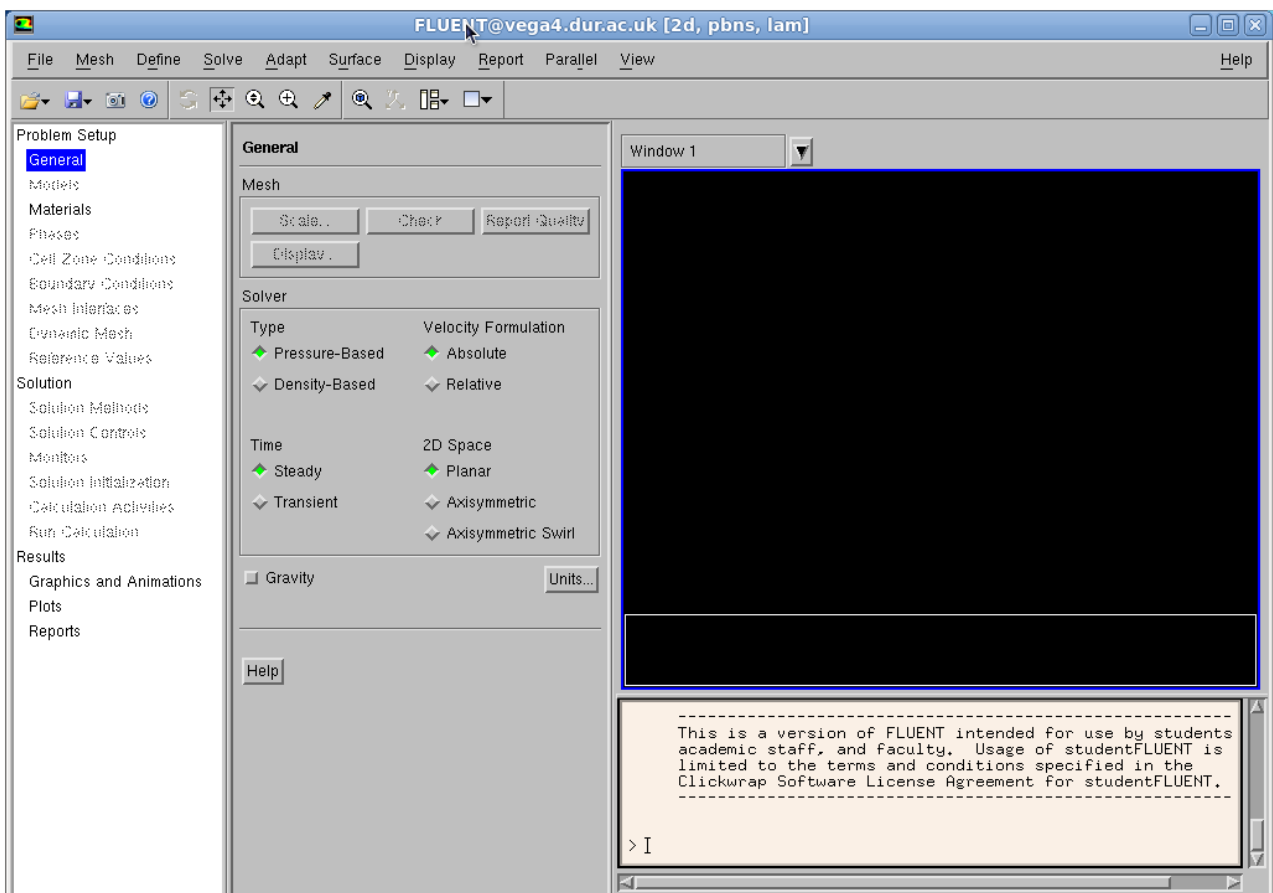


*Figure 1: General View of Fluent 12*

- The key thing to note here is the box on the left hand side. This guides you through the solution set-up. The plan is to work your way through each of the categories from top to bottom (sometimes these are called "Task Pages") setting variables, boundary conditions and so on as you go.

- There are three major headings in this box (Problem Setup, Solution and Results) with a number

of subheadings.  We will consider each heading in turn.  Firstly however you have to read in the mesh.  File → Read → Case and select your file

## Problem Setup Tab

- On the "General" task push the Check button (this should return no problems)

- The Models task is where you can select the turbulence model and so on.  For now the default laminar flow model will be fine.

- The other task pages can be left as default until you get to "Boundary Conditions"

  - Select the flow exit and change the "Type" to "pressure outlet".  By default check the gauge pressure is set to zero which is what we want.

  - Select the flow inlet and change the type to "velocity inlet" set the velocity to be 15 m/s

- You can if you wish set the reference values to appropriate values for your aerofoil.  These are the values that Fluent 12 will use to determine things like coefficients.

## Solution Tab

- The Solution Methods and Solution Controls can be left at default values.

- On the "Monitors" task page make sure that the residuals are set to Print and Plot

- On the "Solution Initialization" task page set the X velocity to 10 m/s and hit "Initalize"

- The other task pages can be left at their default values.

- On the "Run Calculation" you can hit "Check Case" to learn about some helpful suggestions ignore these for now.

- Set the number of iterations to 1000 and press Calculate to watch the computer work for you! The problem should converge in around 50 iterations.

## Results Tab

- You can then view your results using the "Graphics and Animations" task page.  Contours → Set Up.. This might give you something like  Figure 2 after some fiddling.

## Including a Turbulence Model

The inviscid flow solution is fine for bulk pressure changes, but if we wanted to examine the flow under more realistic conditions we can use this solution as the basis for the next run.

- Under Problem Setup → Models →  Viscous → Edit .  Try selecting the Spallart-Allmaras turbulence model.  The default values will be o.k. for now.

- Got back to Solution → Run Calculation.  Then uses the inviscid velocity data etc. with the new turbulence model.
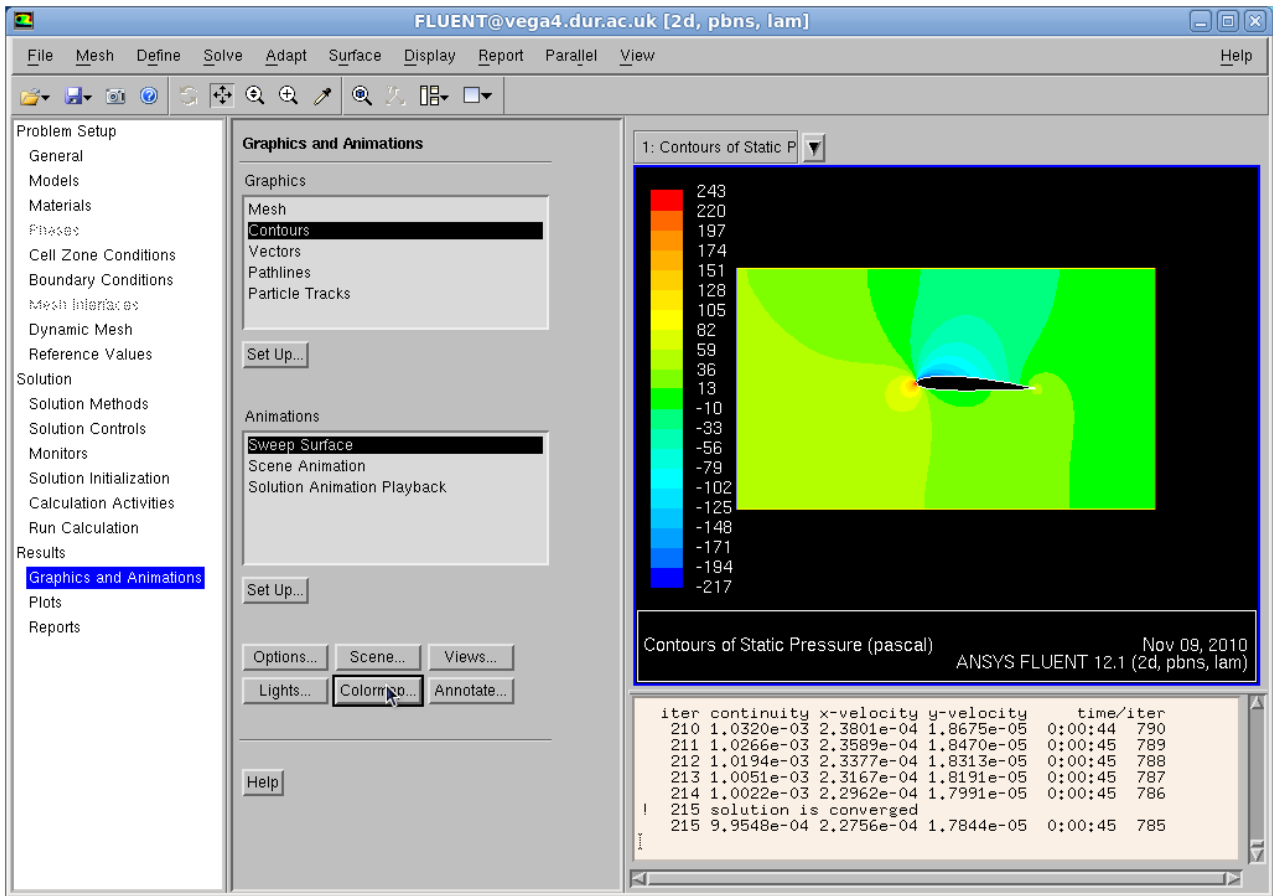
*Figure 2: Some Sample Contours of Pressure*

## *Summary*

You have now successfully meshed up and calculated a basic geometry. Other things you could try with the data:

- Add a turbulence model

- Change the residuals from their default values to much smaller ones

- Extend the problem into 3D

- Tilt the aerofoil and examine the prediction of separation

- Investigate the effect of using a much finer grid


Note that this is a tutorial on how to use Fluent to show you the mechanics of running a solution. The flow model of the aerofoil produced captures bulk flow effects but for anything more refined you will have to carefully consider how to model the scenario!

## *Hints and Tips on Using Fluent*

- Check the grid when you first read it in – if it reports errors fix them!

- The default viscous model is Laminar flow you probably want to change this to something else later.  Work out a Reynolds number to see if the flow is turbulent.

- You do not need to solve the energy equation unless you are working with hot flow or compressible flow.

- Spalart-Allmaras is an efficient and apparently good turbulence model but requires a reasonably good grid near the wall.

- Standard wall functions are appropriate if the near wall grid is not too fine, but are only available with certain turbulence models.

- A symmetry boundary condition can be used to create a frictionless wall.

- Reference Values are not used in the solution but are values used in calculating coefficients etc which you might want to display (ie: pressure coefficient will be based on the velocity in "reference values" not on the velocity associated with a velocity inlet etc).

- Solution Tab:

  ◦ First Order Upwind will be most likely to converge but is least accurate, use this for first iterations but then change to get better solution.

  ◦ Aim to switch to:  Pressure: second order, Momentum: QUICK, Turblence: QUICK

  ◦ Pressure Velocity Coupling stays at SIMPLE unless doing an unsteady run (or perhaps something hot / compressible?)

  ◦ Start with default under-relaxation factors. Reducing will be more likely to converge but it will take longer. Reduce by up to factor of about 3 from defaults if required.

- Print and Plot the residuals but also watch convergence on other values (eg: force on a model, pressure on a surface)

- Don't forget to save the file.  If you are doing an unsteady run you need to set the simulation to write out solutions explicitly after each time-step.

- You can plot contours of any quantity such as residuals to look for where problems occur in your solution.

- Read the manual!  A lot of my "assisting students" involves me reading the documentation and telling people what is in it.

Grant Ingram

2 September 2011