

The Durham Fluent 15 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 15.

Starting the Programs.

The fluid mechanics software packages are currently available on the CIS Linux service. Detailed instructions on how to access this service are available on this website:

https://www.dur.ac.uk/cis/linux/service_facilities/

All NPCS machines in the Engineering and Computing Sciences building are “dual boot”. To access the Linux Desktop System (LDS) you will need to restart the machine and select “Linux” to run a linux system which will give you access to the fluid mechanics software. The remotely accessible service “mira” has the same versions of the software and can be accessed remotely.

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!

- Obtain a terminal window. Usually Applications → System Tools → MATE Terminal
- Start Fluent by typing *fluent* at the terminal.
- 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.
 - You should also click on the “+ Show More Options” button and set the working directory. This saves lots of hunting through menus later.
- After a short while something like Figure 1 will pop up.

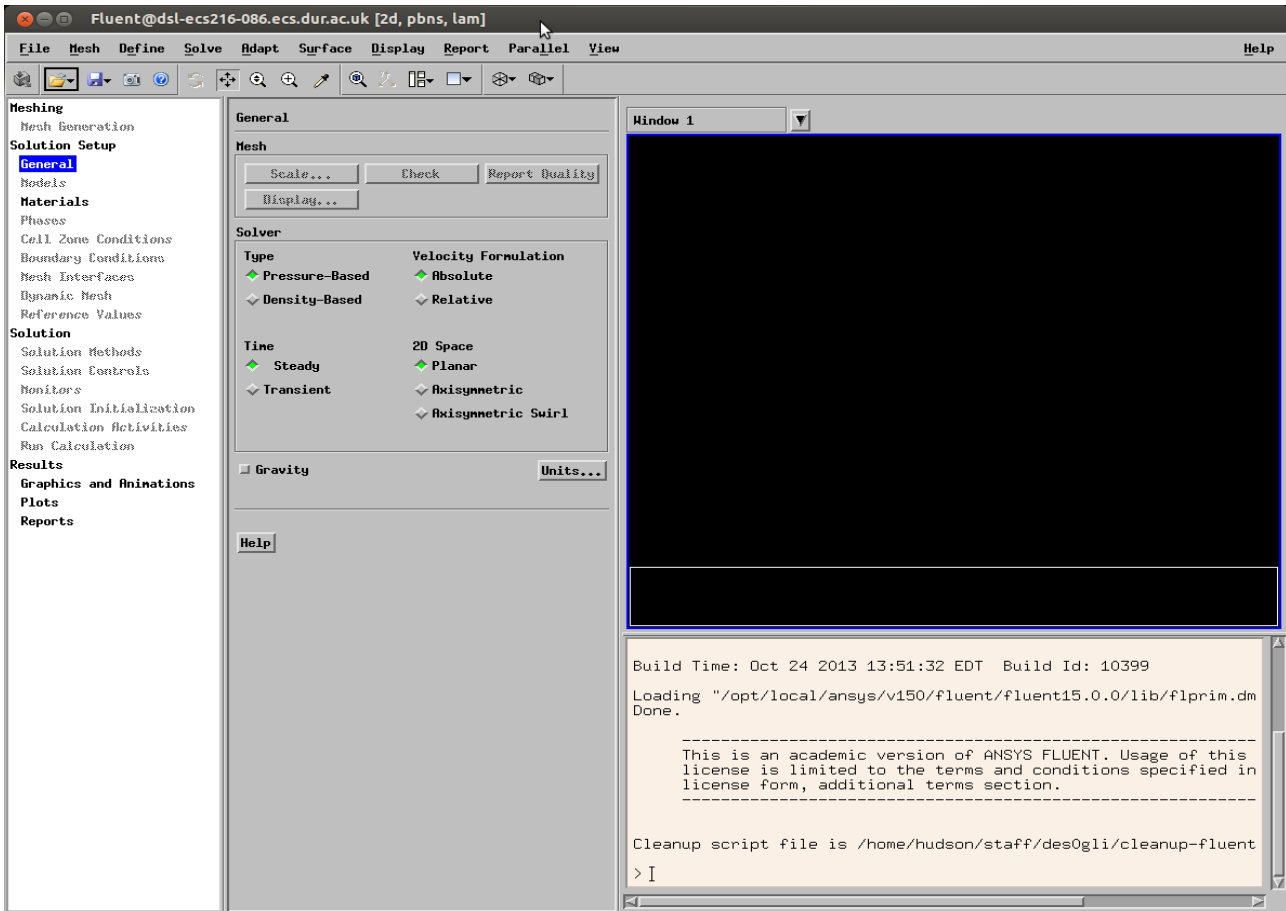


Figure 1: General View of Fluent 15

- 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 (sometimes these are called “Task Pages”) from top to bottom setting variables, boundary conditions and so on as you go.
- There are three major headings in this box (Meshing, Solution Setup, Solution and Results) with a number of subheadings. We will consider each heading in turn.

Meshing

This tutorial assumes that you already have a mesh. Fluent 15 features some meshing features not present in previous versions but you will need to look at the manual to discover more about those features.

To load in the mesh File → Read → Mesh. If you don't have one there should be one on the web-page with this guide. This guide assumes that you will be using an aerofoil inside a square mesh. If you have a mesh created from the Pointwise tutorial - work through this document now and then check the appendix for tips on how to you use the mesh you created in Pointwise.

Solution Set Up Task Pages

- 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”. Check the gauge pressure is set to zero (the default) 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 will use to determine things like coefficients.

Solution Task Pages

- 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 select “Standard Initialization”, this picks up the fact that we have an inlet velocity of 15 m/s and suggests an X velocity of 15 m/s. This is quite a good way to start so click on “Initialize”
- 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! My example mesh no longer converges but you runs through the 1000 iterations in less than a minute. If this was a real fluids problem you would need to investigate why this isn't the case – for now we will look at some results.

Results Task Pages

- 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 Solution 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. This seemed to solve the convergence issue for me.

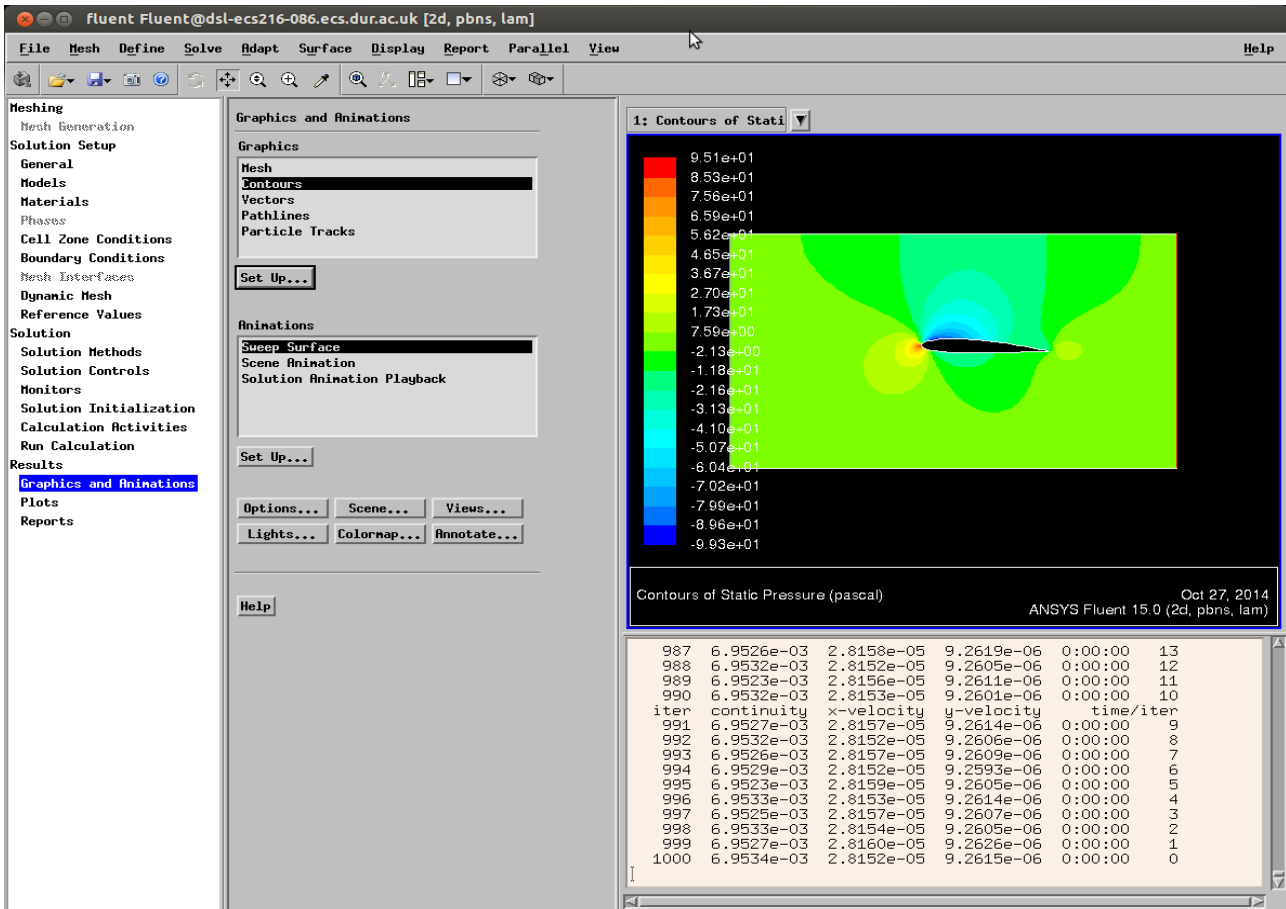


Figure 2: Some Sample Contours of Pressure

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

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! This is **not meant to represent good practise** in CFD but the mechanics of getting the solution to run.

Using the Mesh from the Pointwise Geometry

The steps are essentially the same except you only have one boundary condition to set.

Read the case into Fluent.

- File → Read → Case. It may complain about the high aspect ratio of your cells - if you cared about the results you ought to do something about this.
- Follow the same steps as before with the exception of the Solution Setup Task Page.
 - Models: Turn on the Spallart-Allmaras Turbulence model.
 - Materials: Select the air and hit "Create/Edit". In the drop down change the density from "constant" to "ideal gas". Leave the rest of the values at the defaults.
 - Boundary Conditions: Here you set the circular outer edge to be a "Far Field Boundary" condition - you specify a speed and direction of the far field flow. I suggest Mach 0.6, X component of 1 and Y component of 0.05.
 - You need to specify a compressible Mach number as the far field boundary condition only works with compressible flows.
- Solution Task Page:
 - Initialization: I had some success with using the Hybrid method - just hit "Initialize"
 - Run Calculation: Set the Number of Iterations to be 1000 and hit "Calculate"
- You should end up with something like Figure 3

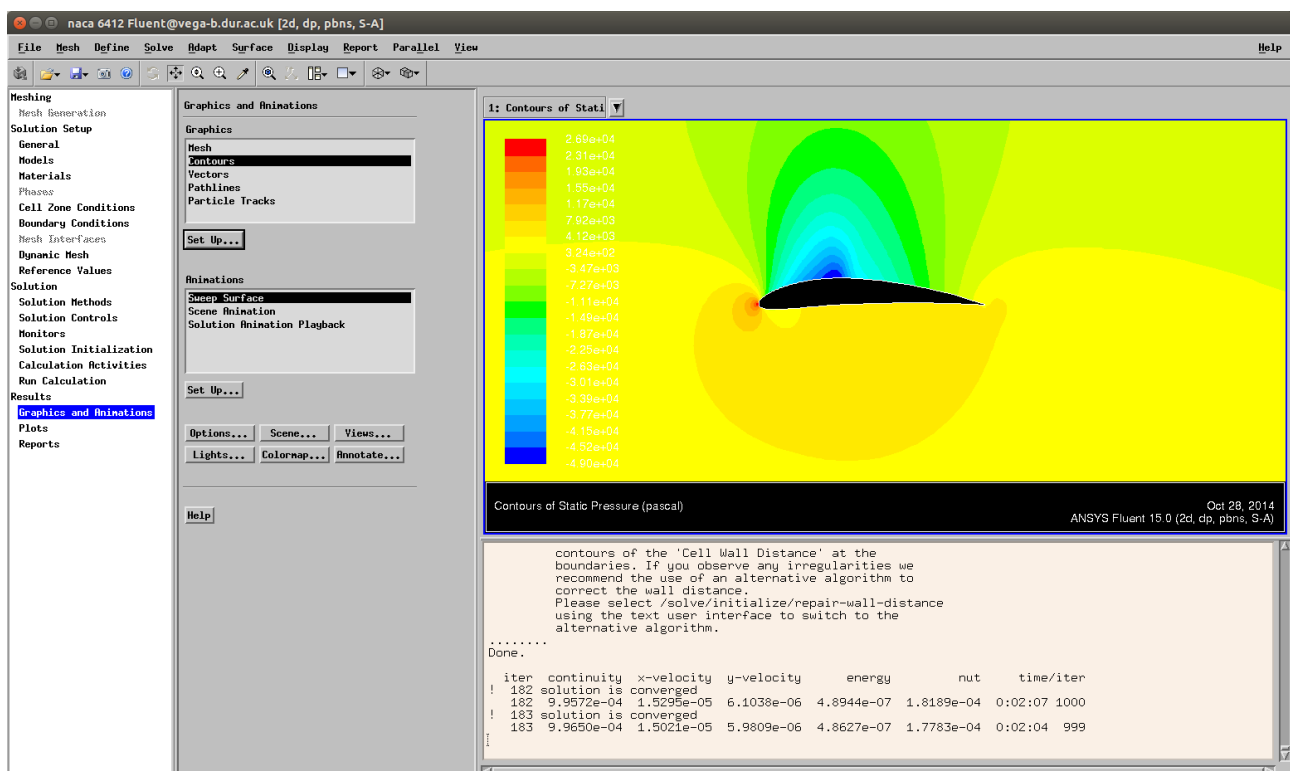


Figure 3: Solution on Pointwise Mesh

Hints and Tips on Using Fluent

- Every menu in fluent has a “Help” button. Click on it and you get LOTS of help – this should be your first port of call if you are not sure about something.
- 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 Task Pages:
 - 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, Turbulence: 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. See the first point about the “Help” button.

Grant Ingram
5 November 2015