České vysoké učení technické v Praze

Zikova 4, 166 36 Praha 6

Numerical modelling of Flow with Transition on an Airfoil

Author:
Samuel Bedford
April 2007

Supervisor:
Assoc. Prof. P.Šafařík

Assistant Supervisor:
Ing. T.Hyhlík

Introduction

There are essentially three major parts to this project; firstly, a simple geometry is created and analysed (a flat plate), then experimental data will be compared with the computational results to determine how accurate the computation is. Lastly, an airfoil will be modelled in the same computational parameters. Certain computational parameters set in Fluent, for the flat plate model, will be adjusted and compared with the experimental data, in an attempt to investigate how the adjusted parameters affect the final result. Having determined, using the above method, the best parameters to provide an accurate result, the analysing of the airfoil will be much simpler.

Initially, vital data had to be gathered regarding both the geometry of an airfoil profile and data relevant to turbulence. The data about the geometry of an airfoil prompted some discussion, as it was seen to be useful to apply the calculations to an airfoil which was required for a specific industrial purpose. To this end, the Institute of Thermomechanics (CAS, Prague) was contacted, Professor Vaclav Tesar (of the above institute) suggested calculations be applied to the NACA profile; NACA 63-418, an airfoil intended for use on horizontal axis wind turbines (HAWTs). Professor Tesar provided some images of the profile of the airfoil as well as a 51 coordinate list, to allow the creation of the shape of the airfoil in Gambit. The data relevant to turbulence was acquired from the ERCOFTAC database by Professor Jaromir Prihoda. ERCOFTAC (European Research Community on Flow Turbulence and Combustion) is a leading institute in the research about flow turbulence and combustion. Though a lot of data was provided, the data most relevant to this project were the T3a and T3a- test cases. The experiments from which this data was gathered were performed by Mr Coupland of Rolls-Royce. As well as experimental data, a detailed description of the experimental set-up and procedure were also provided from ERCOFTAC, the dimensions from the experiment were used to build the model in Gambit.

Boundary Layers

As stated in the introduction, the main purpose of this project is the modelling of transition within the boundary layer on a flat plate. Thus some basic knowledge of boundary layers and their formation is requisite.

For a fluid moving relative to a plate, a boundary layer develops. The fluid next to a wall is moving with the velocity of the wall (usually stationary, so the fluid usually has zero velocity), at a long distance from the wall, the fluid has the velocity of the free stream velocity. The boundary layer defines the region where the change between the two situations occurs. For flow over a smooth flat plate, the boundary layer will remain stably laminar up to a Reynolds number of $1x10^5$, however if the Reynolds number increases the boundary layer will become turbulent (it is almost impossible to maintain a stable laminar boundary layer at $2x10^5$) [6]. Figure 1 shows the boundary layer profile for the; laminar, transitional and turbulent regions as well as the viscous sublayer. The Figure also shows how the wall shear stress varies as distance from the leading edge of the plate increases.

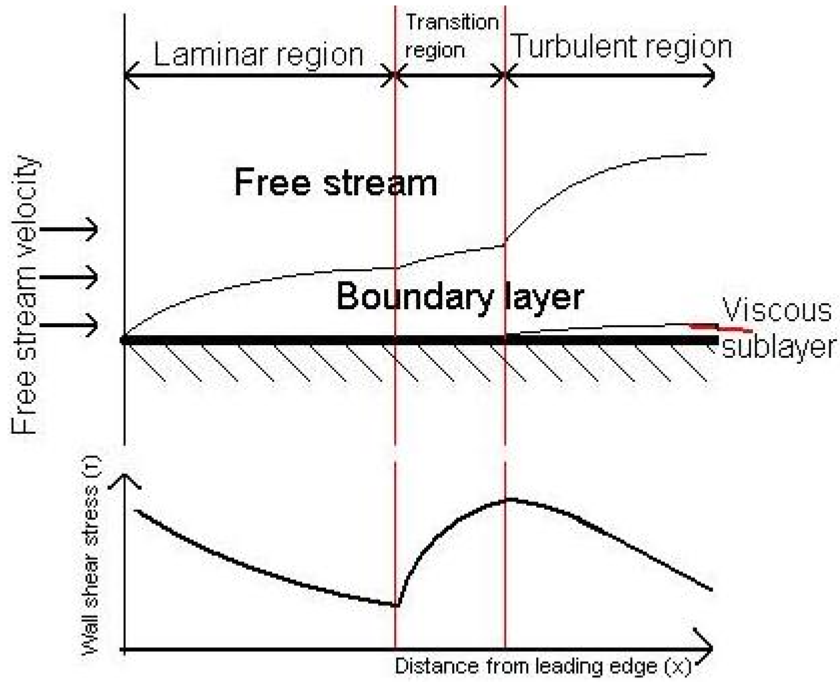

Figure 1: The profile of a boundary layer along with a graph showing how the wall shear stress varies with distance from the leading edge.

The viscous sublayer as shown in Figure 1 is an important part of the boundary layer, especially when attempting to create a mesh. It can be shown that the non-dimensionalised velocity, u/u*, is given by:

$$\frac{u}{u*} = \frac{y}{\nu \times u*} \qquad \text{(eq. 1)}$$

Where; u is the free stream velocity, ν is the kinematic viscosity, y is the distance from the wall (perpendicularly). u* is the friction velocity or shear stress velocity, it is given by $\sqrt{\tau/\rho}$ (where τ is the wall shear stress and ρ is the density), this term has units of m/s and thus can be used to non-dimensionalise a velocity term.

It has been shown, experimentally, that the laminar sublayer is formed if equation eq. 1 has a value of approximately 5. Using this value it can be shown that the sublayer thickness is given by:

$$y' = \frac{25\nu}{u} \qquad \text{(eq. 2)}$$

Where y' is the thickness of the viscous sublayer.

This will be important when meshing the model as it is important to have a low mesh density in the sublayer, i.e. it is best to have only 1 or 2 cells containing the viscous sublayer (this will be discussed further in the 'Meshing the Flat-Plate Model' section.

Turbulence Modelling

Turbulence

In Computational Fluid Dynamics the modelling of turbulence is a particularly complex problem, derived in part from the fact that there is no actual definition of turbulence. Turbulence is normally visualised as being a region of chaotic flow with molecules/regions of fluid, in relatively 'circular' motion, called eddies. The radii of these eddies forms a continuous spectrum between the smallest possible eddie (discussed more in the successive paragraphs) and the largest eddie possible, normally limited in some way by the physical geometry of the environment in which the eddies are present. The above, though useful as a method of visualisation is utterly useless from a scientific (& modelling) point of view. Turbulence is explained scientifically as having a number of characteristics; Irregularity, Diffusivity, High Reynolds Numbers, Three-dimensionality, dissipation and continuity. Turbulent flow is random and chaotic, and thus can be described as being irregular, diffusivity explains some 'spreading rate' of the boundary layer as the flow becomes turbulent. Turbulence occurs at large Reynolds numbers; for turbulence on flat plates without a pressure gradient it will occur at an Reynolds number of $\approx 10^5$. Turbulence occurs in three dimensions (although the model is only 2-d the turbulence is modelled as being 3-d). At the small end of the eddie spectrum - called the Kolmogorov length scale, the kinetic energy is transformed into internal energy due to the viscosity of the fluid, thus turbulence is dissipative. Lastly, though the turbulent region contains small eddies, these are still much larger (in diameter) than the molecular scale so the flow can be described as a continuum [4].

As discussed above, there is a continuous spectrum of eddie sizes, from the largest eddies, which are in some way limited by the geometry of the local environment to the smallest eddies which are limited by the viscosity of the fluid. The largest eddies interact with eddies which are slightly smaller, and these interact with eddies which are slightly smaller and so on. This process is called a cascade and transfers the turbulent energy of the flow into internal energy. The energy spectrum can be represented graphically, as follows;

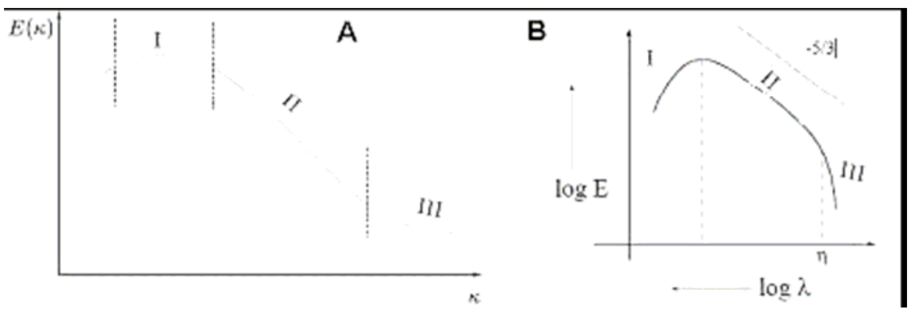

Figure 2: Two diagrams showing the energy spectrum for a turbulent region. Figure A uses linear scales and Figure B uses logarithmic scales. (From references [4] & [5] respectively)

The spectrum graphs in Figure 2 show three distinct regions. The region on the left hand side of the graphs (region I) is the large eddie region where energy is extracted from the free stream by the large eddies, these large eddies pass their energy on to slightly smaller eddies etc. The second region (region II) is the transport or cascade region where the turbulent energy is passed, by the cascade process, to the small eddies. The gradient of the line in region II is -5/3, and is described by the Kolmogorov spectrum law:

$$E(\kappa) \propto \varepsilon^{2/3} \kappa^{-5/3} \qquad \text{(eq. 3)}$$

Region III is the dissipative region, where turbulent kinetic energy is lost to internal energy due to the viscous stresses in the fluid. The eddie scale at which this happens is called the Kolmogorov length scale [4].

Computational Modelling

For simplicity, when deriving the equations used for computational modelling, velocity (U) is described in terms of mean velocity Ū and some fluctuation velocity, u. The same is done for Pressure (P), it is described in terms of mean pressure, $\bar{P}$, and some fluctuation pressure, p. This is shown as follows:

$$U_i = \overline{U} + u \qquad \text{(eq. 4)}$$

$$P_i = \overline{P} + p \qquad \text{(eq. 5)}$$

This gives velocity and pressure values without any time history and which gives a way of modelling the components without having to model each individual eddie, in other words it is a way of representing the overall picture of what the velocity and pressure are doing without being absolutely specific.

Continuity can be represented by the following equation:

$$\frac{\partial \rho}{\partial t} + \rho U_i = 0 \qquad \text{(eq. 6)}$$

Assuming that the flow is incompressible, the Navier-Stokes equation can be written as follows:

$$\frac{\partial \rho U_i}{\partial t} + (\rho U_i U_j) = -P_i + [\mu(U_{ij} + U_{ji})]_j \qquad \text{(eq. 7)}$$

Substituting equations eq. 4 and eq. 5 into the continuity equation (eq. 6) and the Navier-Stokes equation (eq. 7) gives the Reynolds-Averaged Navier-Stokes equations, as follows:

$$\frac{\partial \rho}{\partial t} + \rho \overline{U}_i = 0 \qquad \text{(eq. 8)}$$

$$\frac{\partial \rho \overline{U}_i}{\partial t} + (\rho \overline{U}_i \overline{U}_j) = -\overline{P}_i + [\mu(\overline{U}_{ij} + \overline{U}_{ji}) - \rho \overline{u_i u_j}]_j \qquad \text{(eq. 9)}$$

The above substitution produces a new term, $\overline{u_i u_j}$, called the Reynolds Stress Tensor function, it is a term arising as a result of the existence of turbulence - it is a correlation between the fluctuating velocities. This term is unknown but is required to enable the equation to be used for modelling. This is called the closure problem, there are four main types of computational models which can be used to solve this problem; the Algebraic models, the One-equation models, the Two-equation models and the Reynolds stress models.

In this project two-equation models will be used. In these models, two transport equations are derived to describe the transport of two scalars (normally either k and ε or k and ω). The Reynolds stress tensor is then calculated using an assumption that it relates to the velocity gradients and an eddy velocity - the eddy velocity being obtained from the two transport scalars. The Two-equation models are perceived to offer a good compromise between complexity (and thus accuracy) and computational time required [4].

The assumption used in the two-equation models (as well as others) is that that Reynolds stress tensor is related to the velocity gradients by using the turbulent viscosity - This is called the **Bossinesq assumption**. For preliminary work the k-ω model will be used, this model was proposed by Wilcox and is gaining popularity.

The Airfoil

The profile to be used in this project is part of the 6 series, a series which was derived theoretically by NACA; it is created by first specifying the desired pressure distribution across the wing, then using the Blasius equation (among many others) to derive the optimum geometric shape. The principle idea of the 6 series is to increase the distance over which the air flow is laminar up to the theoretical maximum. The 6 series is only designed to be operated in a small range of lift coefficients, but will have very low drag when operated within this region. The code for the six series defines the airfoil's exact geometry and characteristics. The first number (6) represents the series (the 6-series), and thus indicates that it is an airfoil designed for increased laminar characteristics. The second number (3) represents the location, from the leading edge, of the minimum pressure, measured in tenths of chords (i.e. in this case the minimum pressure is at 0.3 chords from the leading edge). The third number (first after the dash), 4, represents the designed lift coefficient. The remaining two numbers (18) represent the thickness of the airfoil as a percentage of the chord. A subscript number is sometimes placed after the first two numbers to represent the number of tenths either side of the specified lift coefficient at which low drag is maintained. Figure 3 is an image of the NACA 63-418 airfoil. [2]

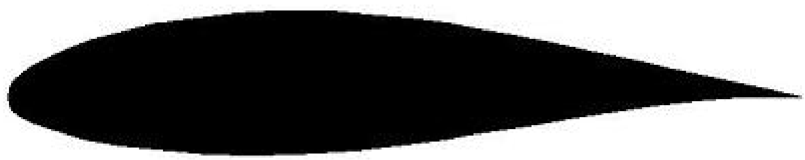

Figure 3: The Profile of the standard NACA 63-418 airfoil

Meshing the Flat-Plate Model

Gambit was used to create a model of the flat plate within the computational domain. This set-up is very simple, the dimensions as well as the computational boundary conditions are shown in Figure 4.

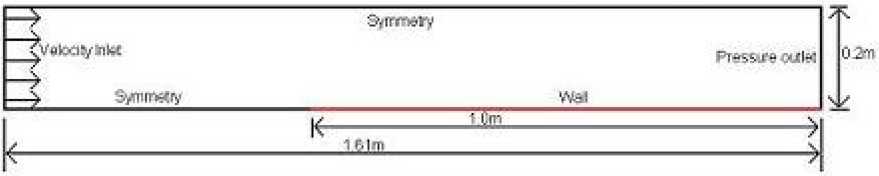

Figure 4: A schematic of the computational domain including dimensions and computational boundary conditions

Meshing this model, though simple, is a non trivial problem. This is mainly due to the two opposing requirements; rapid computation (few cells) and high mesh density in the region of interest - the boundary layer (requiring a very high mesh density). This problem was solved, in part, by using graded edge meshes, when using this function, a grading ratio is specified as well as an interval count, the grading ratio specifies the amount larger each successive node spacing is.

As the primary objective of this project is to investigate transition a fine mesh was required, both at the leading edge of the wall and close to the wall along its length. In arriving at the final mesh shown in Figure 5, approximately 10 meshes were created, each mesh was inappropriate for some reason; either it was not accurate enough when compared with experimental data or it required too much computational time.

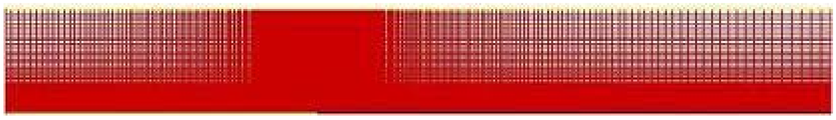

Figure 5: The final mesh, created using graded edge meshes with low grading ratios

This mesh has 38659 cells, the cells close to the wall at the leading edge have very low aspect ratios (i.e. they are nearly square) as this was found to be a requirement for an accurate mesh. The cells adjacent to the wall have a height, perpendicular to the wall, of 0.0338 mm, and the second cell has a height of 0.03571 mm, this means that the first two cells form a strip 0.0709 mm wide which is enough to completely contain the viscous sublayer (see the 'boundary layer' section).

Analysis Parameters for the Flat-Plate Model

When the meshes had been created in Gambit they were exported to Fluent. In Fluent, the analysis parameters had to be set. As the purpose of this project is the examination of transitional flows, it was important that the correct functions were 'turned on' to allow for transition to be modelled. To this end the most important setting was to turn on 'transitional flows'. Other important settings included setting the viscous model to k-omega and the free stream to 'intensity and length scale'. The free stream conditions were set for the inlet in accordance with the ERCOFTAC data, i.e. velocity of 5.4 ms^{-1}, turbulence intensity set to 3 % and a length scale of 0.04 m. Lastly the solver was set to perform 'second order upwind' calculations.

The computation converged in 300000 iterations and took approximately 6 hours. Fluent was set to plot a graph of the residuals of; x-velocity, y-velocity, k, omega and continuity, as the iteration was performed. This graph is shown as Figure 6.

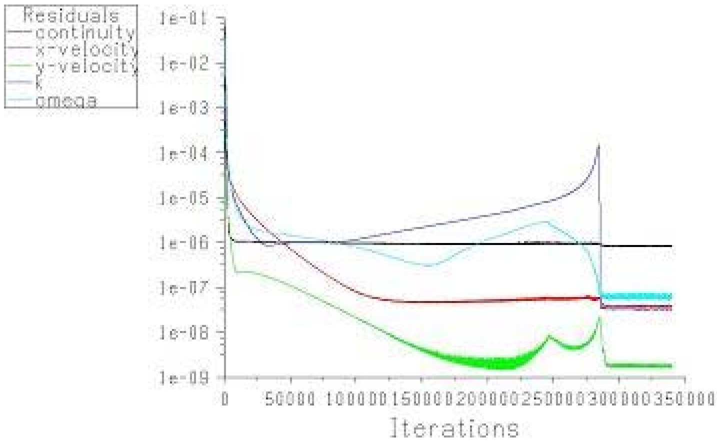

Figure 6: A graph showing the residuals as the iteration progressed. It clearly shows that the model converged at approximately 300000 iterations.

Results for the Flat-Plate Model

Using the results from the computation two graphs have been plotted; the first shows the coefficient of friction against Reynolds number, the second shows non-dimensionalised velocity as a function of non-dimensionalised distance (wall units). As well as the computational results, ERCOFTAC data and theoretical results have been plotted on the same axis. The theoretical values were calculated as follows:

Coefficient of skin friction (laminar flow) = $0.664\,\mathrm{Re}_x^{-\frac{1}{2}}$

Coefficient of skin friction (turbulent flow) = $0.370\log(\mathrm{Re}_x)^{-2.584}$

Reynolds number, $\mathrm{Re}_x = \dfrac{U_0 x}{\nu}$

U_0 = free stream velocity,
x = distance along the wall from the leading edge,
ν = the kinematic viscosity

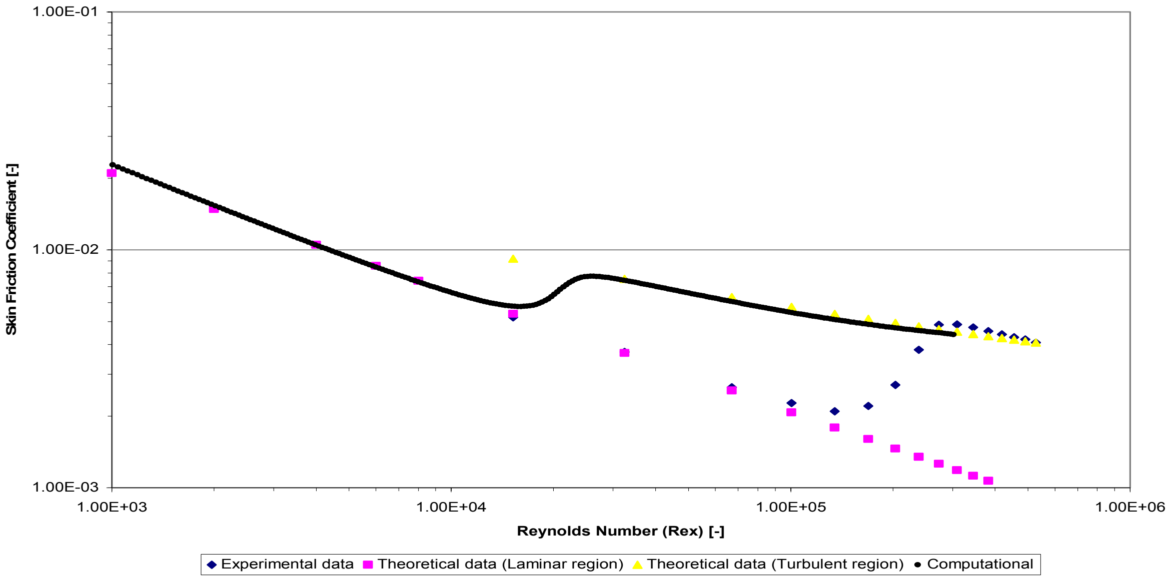

Figure 7: A graph showing the coefficient of skin friction as a function of the Reynolds number for computational, experimental and theoretical data. The graph utilises log-log scales to accurately represent the data.

Clearly the theoretical data cannot show transition, it shows the laminar and turbulent regions separately. However, outside of the transitional region the experimental data fits the theoretical data very well. The computational data is also a very close fit in the laminar and turbulent regions. The computation predicts that transition will occur at a lower Reynolds number than that which occurs in reality, this is shown on the graph in Figure 7 where transition for the computation occurs at a Reynolds number of $\approx 2 \times 10^4$, whereas the experimental data shows transition occurring at $\approx 2 \times 10^5$ - a difference of 1 order of magnitude. This is a known problem with the k-omega model; it tends to predict transition 'earlier' than actually occurs experimentally [7].

The second graph which has been plotted is that of U^+ (non dimensionalised velocity) as a function of Y^+ (non-dimensionalised distance - wall units), it shows how the velocity changes on a line perpendicular to the wall. Again, computational, theoretical and experimental data has been plotted. The theoretical data was calculated using the following formula.

(Laminar region) $U^+ = Y^+$

(Turbulent region) $U^+ = \dfrac{1}{\kappa} \ln y^+ + B$

Where κ and B are constants with values of 0.41 and 5.2 respectively

The experimental data can only be used for the laminar region as the ERCOFTAC data provided was of a low Reynolds number flow. It clearly shows the laminar region and turbulent region, both of which fit the theoretical (and experimental) data very. The computed results also show the transitional region where the data doesn't fit the theoretical results (either laminar or turbulent).

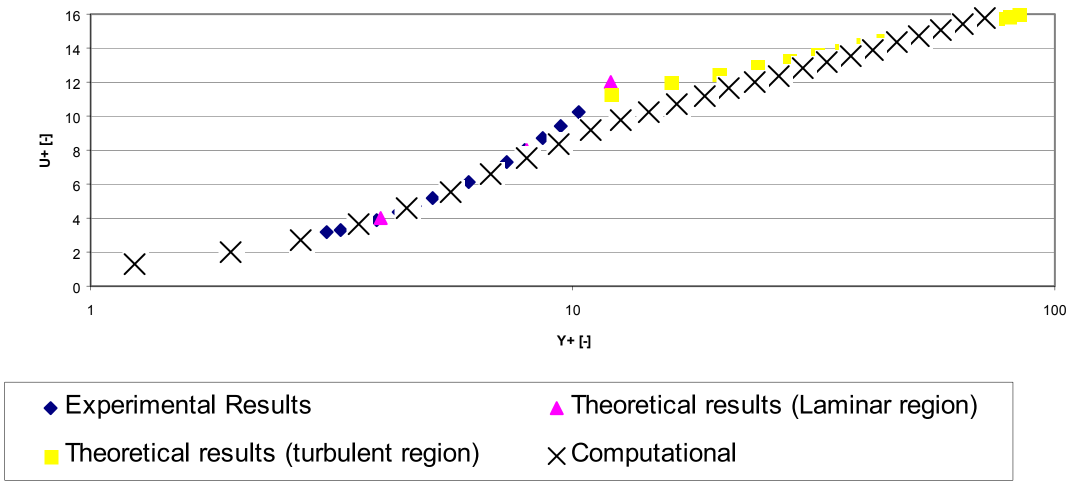

Figure 8: A graph showing non-dimensionalised velocity as a function of wall units.

For the above computation, the solver was set to 'second order upwind', by way of investigation the same model was computed with the solver set to 'QUICK'. When the model had re-converged with the new settings, data about the coefficient of friction was extracted and plotted against the Reynolds number on a graph along with the data from Figure 7. From this graph it was not possible to determine if there was any difference in the results when using Second Order Upwind or QUICK methods. In an attempt to discover it there was any difference between the results, a graph was plotted of the percentage deviation against the Reynolds number; this is shown as Figure 9

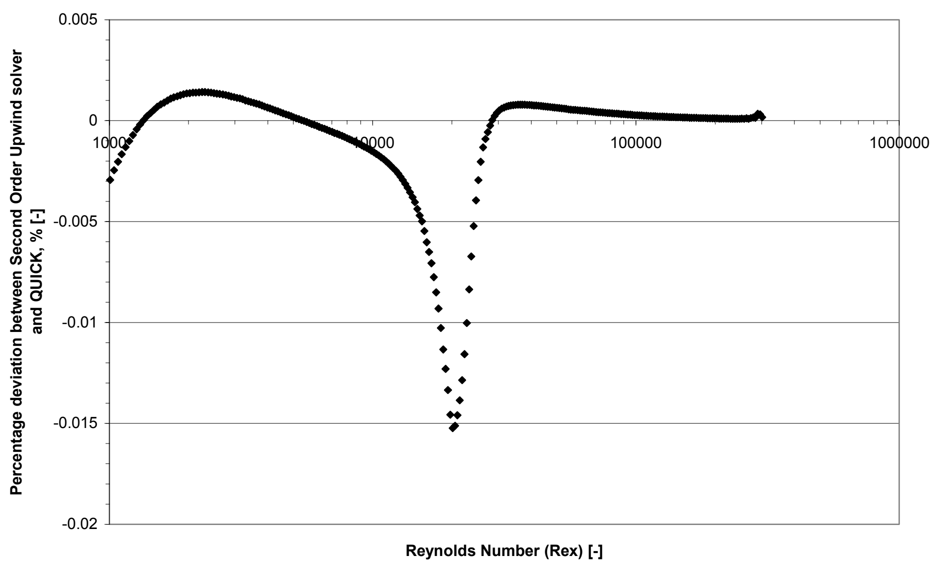

Figure 9: A graph showing the percentage difference between the coefficient of friction for two types of solver (Second Order Upwind and QUICK) as a function of the Reynolds number.

Clearly the largest difference is at the transitional region, however this is still less than one fiftieth of a percent so it is insignificantly small. For successive calculations the original solver will be used (Second Order Upwind).

Conclusion

Thus far, a simple geometry has been created in Gambit and analysed in Fluent. The results have been compared with theoretical (calculated) values and with data from the ERCOFTAC database. The model has been found to be reasonably accurate at representing the laminar and turbulent region, it also predicts transition but predicts it at a Reynolds number approximately one order of magnitude lower than that which occurs in reality. Two solver methods have been used and found to produce very similar results, so 'second order upwind' will be used from here on. The geometry and domain of the airfoil have been created and are being meshed at present.

Acknowledgements

I would like to thank the CVUT (Czech Technical University) for allowing me to study in Prague as part of the Socrates-ERASMUS program. I am particularly grateful to Prof. P. Safarik and Ing. T. Hyhlik, both of whom are supervising this project. I would like to thank Prof. V. Tesar and Prof. J. Prihoda who have provided essential data for this project. I would also like to acknowledge the grants provided by the IMECHE and by the Socrates-ERASMUS program, both of which were crucial in enabling me to come to Prague. Lastly I would like to thank Ing. A. Sladek who has given up his personal computer and office space to allow me to carry out this project.

References

[1] Javafoil - http://www.mh-aerotools.de/airfoils/javafoil.htm
[2] Aerospaceweb - http://www.aerospaceweb.org/
[3] Numerical Computations of Aerodynamic Performance of Wind Turbine Rotor Sections - Using General Purpose Software - Vaclav Tesar
[4] An Introduction to Turbulence Models - Lars Davidson (www.tfd.chalmers.se/~lada)
[5] Course notes for module 3211 (department of Engineering, Exeter University) - Dr Gavin Tabor
[6] Fluid Mechanics (4^{th} ed) - John F. Douglas, Janusz M. Gasiorek, John A. Swaffield
[7] Simulation of Transition with a Two-Equation Turbulence Model - David C. Wilcox
Fluent Inc. - Fluent User's Guide, Fluent Inc., 1998