TURBULENCE PRACTICE
Report on RANS MODELLING OF COUETTE-POISEUILLE FLOW

Bipin Kumar SINGH [1]

[1]Master's Student in Turbulence
Ecole Centrale de Lille, FR 59650

bipin-kumar@master.centralelille.fr

1. Introduction:

A simulation intended to describe the critical flow parameters(velocity vectors) in a combination of couette-poiseuille flow type is described and performed. The initial adn boundary conditions taken into account for the analysis are described in the two reference paper of El Telbany and Gilliot's thesis.

By definition couette flows are the flow of a viscous fluid in the space between two surfaces, one of which is moving tangentially relative to the other. The configuration often takes the form of two parallel plates.

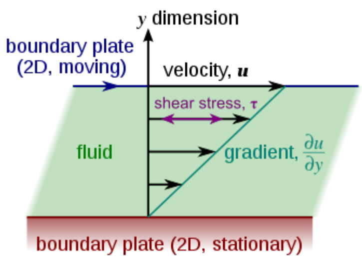


Figure 1. Velocity profile for couette flow

Similarly a poiseuille flow can be dfined as flow created between two infinitely long parallel plates, separated by a distance h h with a constant pressure gradient $G = -dp/dx = constant$ is applied in the direction of flow. The flow is essentially unidirectional because of infinite length. a velocity profile which is inherent to these kinds of flow has been shown below.

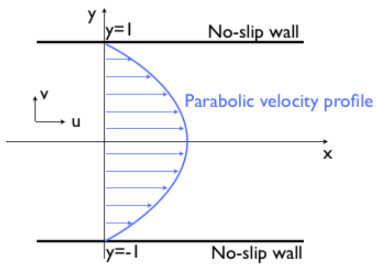


Figure 2. Velocity profile for poiseulle flow

We will see at the end of the report a detailed comparison of the output of velocity profiles of the program with the experimental ones, these will then be analysed in order to come to a conclusion.

2. Experimental Setup:

The velocity of the moving wall is in the direction of the mean flow through the channel (or directly opposed to it), and the hypothetical flow is therefore unidirectional. The flows actually studied are those in a flat channel (aspect ratio 12 to 28 in these tests) through which air is blown, and one of whose sides consists of a flat belt which can be moved either with the air blown through the channel or in the opposite direction. The experimental situation is shown schematically in figure 1. It is supposed that the motion near the mid-plane and exit from this channel, that is, in the vicinity of the access ports shown, approximates to the fully-developed unidirectional motion described above.

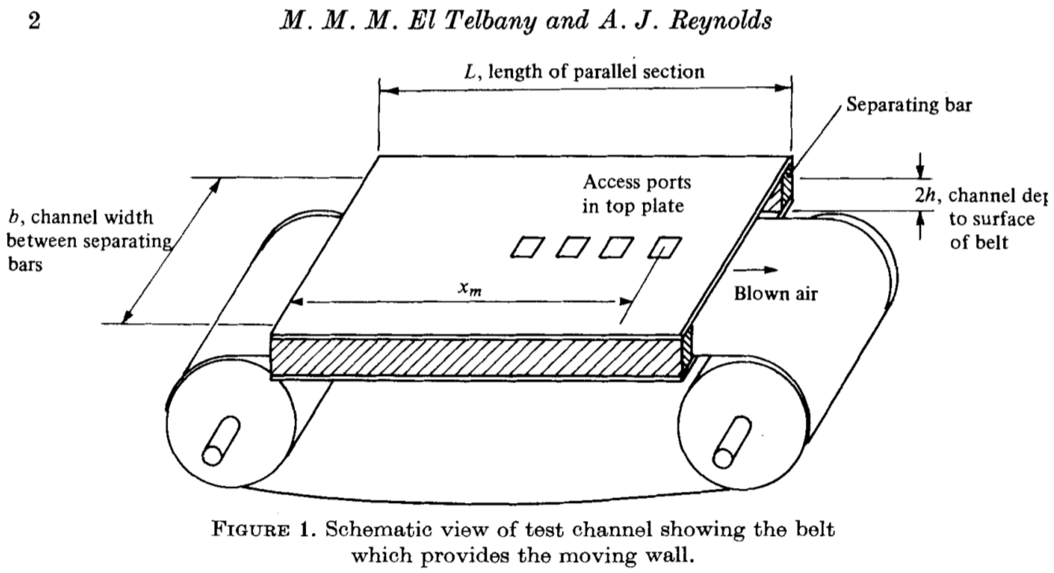


Figure 3. Experimental setup of couette-poiseulle flow

3. Derivation of flow equations:

In order to derive the flow equations and simplify in order to solve the system of equations, we have taken into account several hypotheses which are then applied to Navier-Stokes equations. Below are the hypothesis that have been taken into account:

1)Viscous-there is a constant of kinematic viscosity

$$\nu = constant$$

2)Steady State conditions -variables do not vary with time.

$$\frac{\partial}{\partial t} = 0$$

3)2-directional

$$\frac{\partial}{\partial x_k} = 0$$

4)Parallel flow-

$$\frac{\partial x_i}{\partial t} = 0$$

$$v_j = 0$$

$$v_k = 0$$

5)Isothermal-To keep the viscosity term constant

$$Temperature(t) = constant$$

The Reynolds Averaged Navier-Stokes equation below:

$$\frac{\partial \overline{u_i}}{\partial t} + \overline{u_j}\frac{\partial \overline{u_i}}{\partial x_j} = -\frac{1}{\rho}\frac{\partial \overline{p}}{\partial x_i} + \frac{1}{\rho}\frac{\partial}{\partial x_j}\left(\mu\frac{\partial \overline{u_i}}{\partial x_j} - \rho\overline{u_i' u_j'}\right)$$

the hypotheses when applied to the RANS equation yields the following equation which can then be solved in order to get the solution of mean velocity of the fluid flow:

$$\frac{\partial \overline{p}}{\partial x_i} = \frac{\partial}{\partial x_j}\left(\nu\frac{\partial \overline{u_i}}{\partial x_j} + \nu_t\frac{\partial \overline{u_i}}{\partial x_j}\right)$$

we integrate the above equation in order to reduce the equation as a function of first Order Differential Equation of u in space x_j. Hence we obtain:

$$[\frac{\partial \overline{u}_i}{\partial x_j}(\nu + \nu_t)]_S^N = \frac{\partial \overline{p}}{\partial x_i}[(x_j)]_S^N$$

The above equation will be solved using finite volume method in order the get the mean velocities of the fluid flow.

Turbulent Viscosity: the nu_t term in the above equation is also called turbulent viscosity term which is obtained from the Boussinesq's assumption that helps us replace the Reynolds stress tensor term with a another variable. this variable, Turbulent viscosity can then be modelled with the help of several tools. one of the method that can be used to model the equation is the called the MIXING LENGTH'S MODEL.

In this method, the turbulent viscosity is modelled based on the mixing lengths model of the fluid flow. for the given type of wall bounded flow the turbulent viscosity can be defined as the following:

$$\nu_t = l_m^2 * mod(du/dy)$$

where,

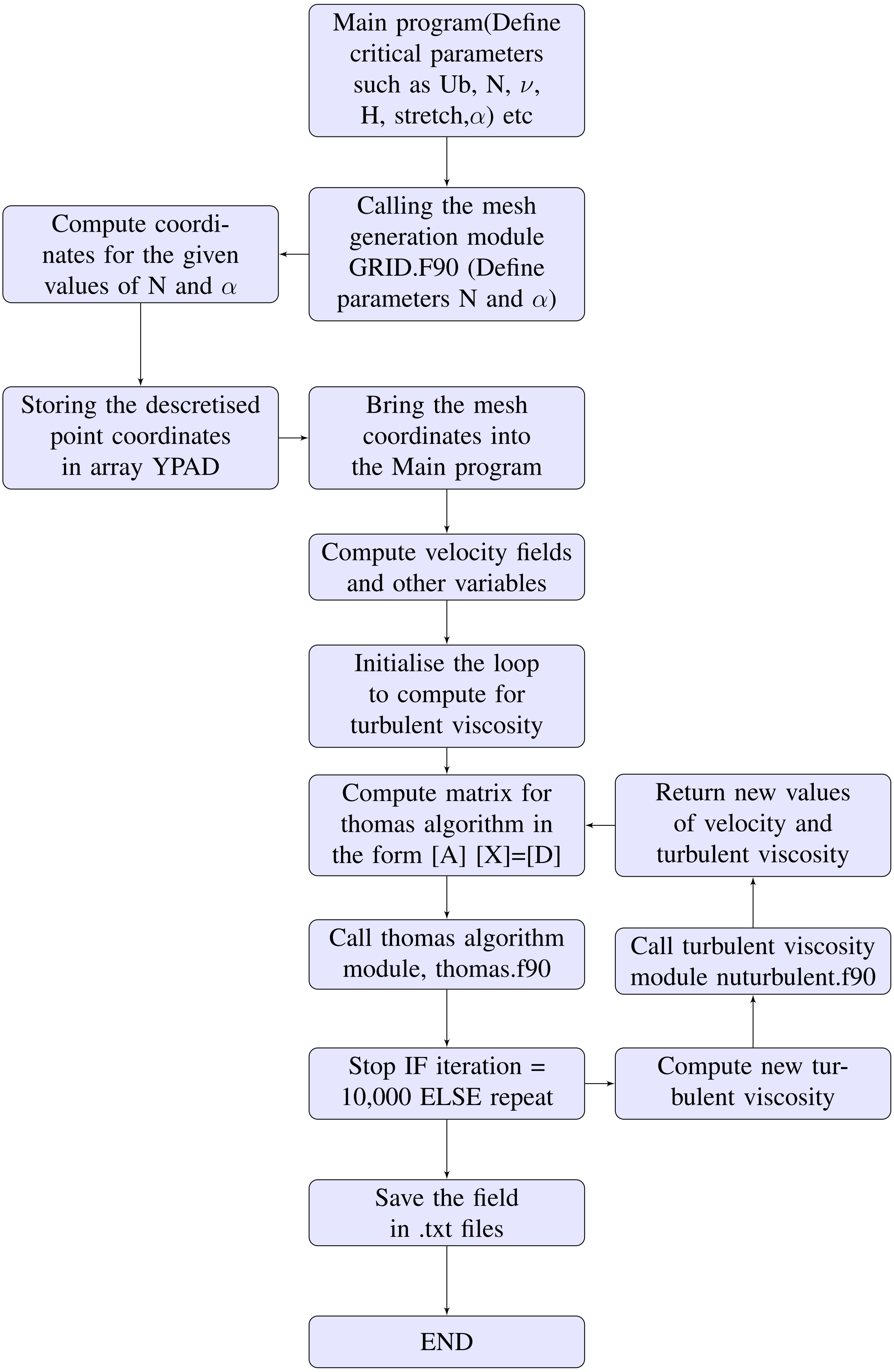
$$l_m = l_o * (1 - exp(-y/A))$$

$$2 * l_o/H = 0.21 - 0.43 * (1 - 2 * (y/H)^4) + 0.22 * (1 - 2 * (y/H))^6$$

with $A = 26 * \nu/u_t$ and $y \leq H/2$

3.1. Program structure:

In this section, we will describe the how the equations have been coded in order to compute the required velocity fields. this is explained by the flow diagram shown below that describes the role of each module to simulate the problem, below is flow diagram of the code:


Main program(Define critical parameters such as Ub, N, ν, H, stretch,α) etc
Calling the mesh generation module GRID.F90 (Define parameters N and α)
Compute coordinates for the given values of N and α
Storing the descretised point coordinates in array YPAD
Bring the mesh coordinates into the Main program
Compute velocity fields and other variables
Initialise the loop to compute for turbulent viscosity
Compute matrix for thomas algorithm in the form [A] [X]=[D]
Return new values of velocity and turbulent viscosity
Call thomas algorithm module, thomas.f90
Call turbulent viscosity module nuturbulent.f90
Stop IF iteration = 10,000 ELSE repeat
Compute new turbulent viscosity
Save the field in .txt files
END

4. Results and discussion:

4.1. Problem:1

Using the finite volume method write a program allowing to compute u(y),$u_(\tau 1)$,u$_(\tau 2) and$U$_q$ (bulk velocity), given $V_w = V_b$ (velocity of the moving wall), dp/dx, ν, ρ, and H for different grids from homogeneous (α =0) to stretched grid (α 0.05) with different number of grid points N.

Solution: the program has been coded and is attached along with report to compute all the specified variables.

4.2. Problem:2

Make the computation for the 15 cases of the El Telbany and Reynolds paper (Case 1 to 15):

Solution: All the computation has been done for the specified cases and outputs have been saved as .txt files, these have also been attached along with the original program.

4.3. Problem:3

Make the computation for the 3 cases of Table 4.3 of the thesis of Anne Gilliot (Cases 16 to 18):

Solution: Even these simulations were performed with the code and output files have been attached for review.

4.4. Problem:4

Plot u(y) and compare it to experimental results for all cases.

Solution: The data stored as an output of the earlier questions were then plotted using python tool. below are the output images for the plots.

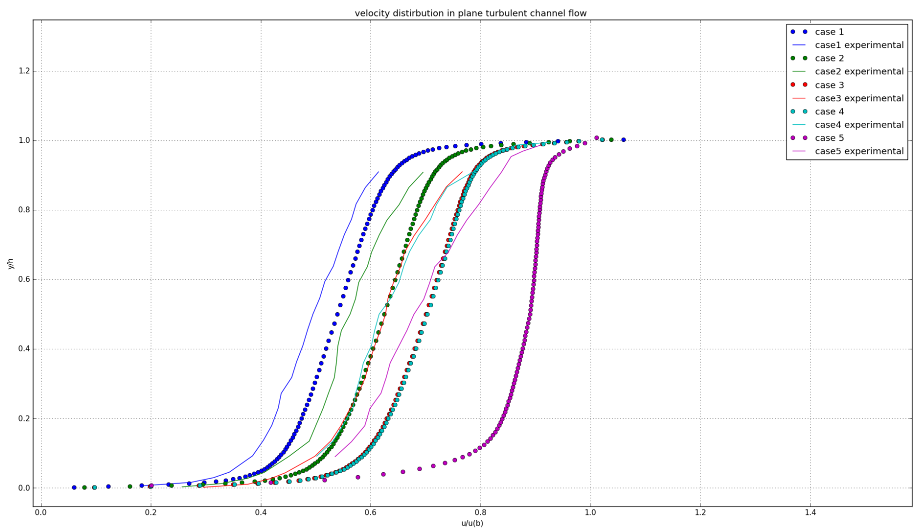


Figure 4. U(y) plot for experimental and simulation for Reynolds case 1-5

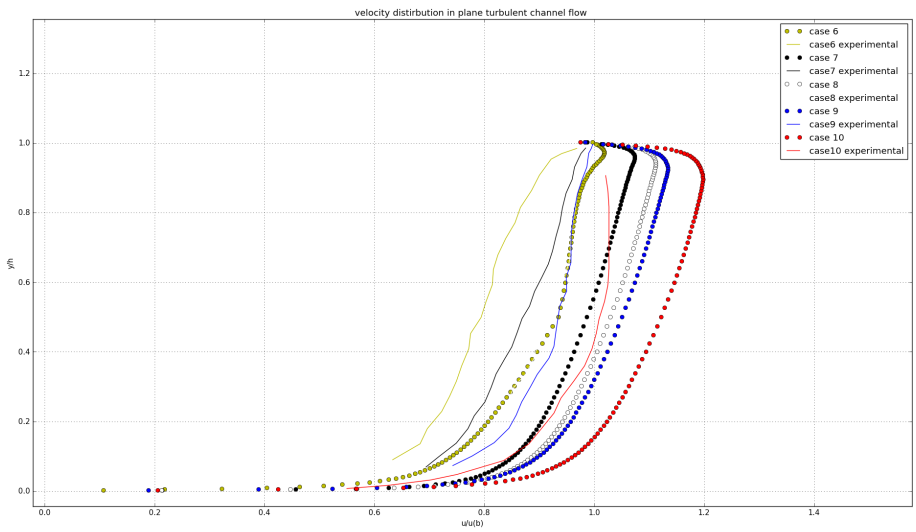


Figure 5. U(y) plot for experimental and simulation for Reynolds case 6-10

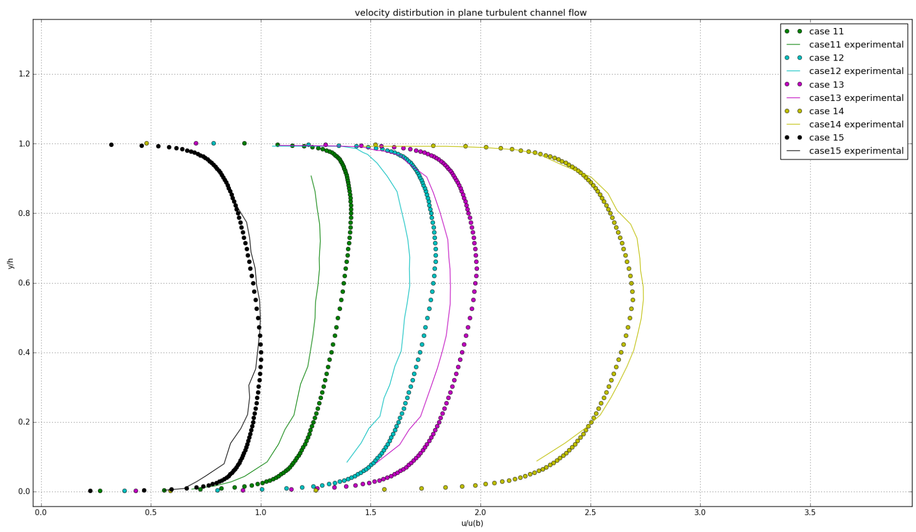


Figure 6. U(y) plot for experimental and simulation for Reynold's case 11-15

Looking at fig 4,5,6, we can conclude that the program gives a good approximation of the flow profiles for all the cases discussed in Reynold and telbany's experiment and the simulation is satisfactory.

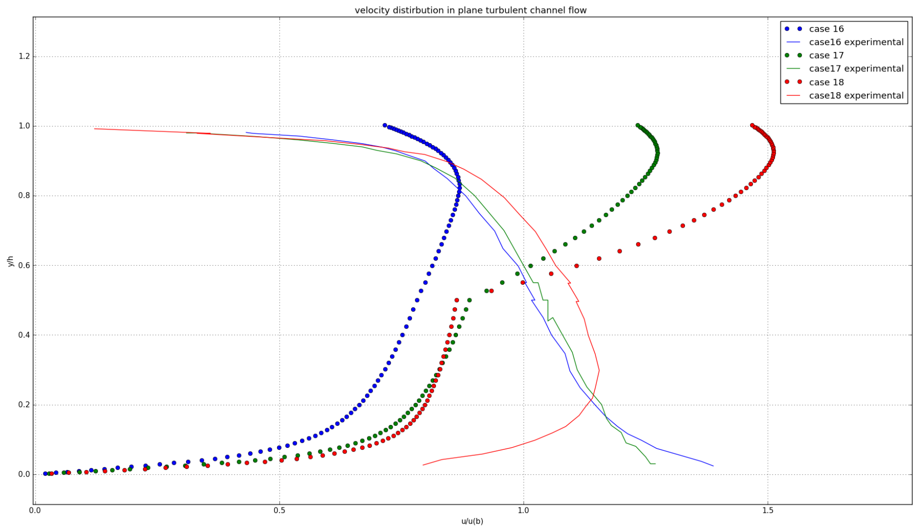


Figure 7. U(y) plot for experimental and simulation for Gilliot's case 1-3

In Fig. 7, we can notice a complete inversion of flow profiles between the experimental ones and the computed ones. this can be explained as a results of different wall in motion compared to the one programmed. in the program we took into consideration that the top wall is in motion whereas in the case on A. Gilliot's experiment the wall under motion is the bottom one. therefore there is a descrepancy in the flow profile in the above plot.

4.5. Problem:5

Compare $u_\tau 1$,$u_\tau 2$ and U_q to experimental results for all cases.

Solution: In figure 8 we see the comparison of $u_\tau 1$ and $u_\tau 2$ computed by the program with experimental values. the profile seems to be almost good and we can see a good variation in the curve compared to the experimental ones.

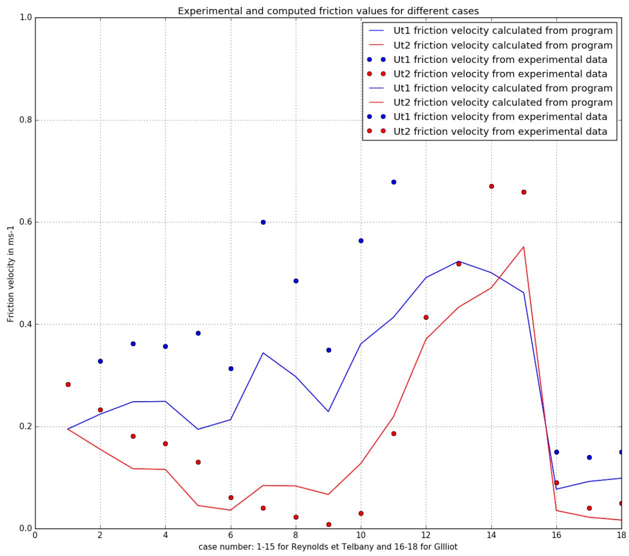


Figure 8. Comparison of $u_\tau 1$ and $u_\tau 2$ between experimental and computed with the program for all the 18 cases

4.6. Problem:6

Discuss the results and the influence of the parameters of the grid(α and N)

Solution: there were different analysis done for various values of N and α. below are the results and plots for those analysis.

Influence of N:

Effect of N on residual.

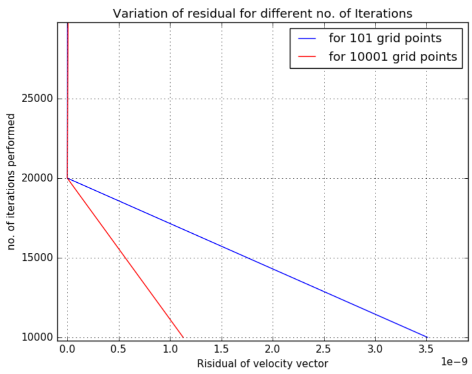


Figure 9. Effect of Number of descretised points on residual for differnt number of iterations for case 1

In the above figure we can see that the residual values for two same number iterations are different for 101 points and 1001 points. it seems to be less for 1001 points for 10000 iterations than for 101 points. this is because of local refinement and since the velocity fields are computed more accurately. however, we should also point out that the time taken for computation was a lot more than 101 points for 1001. we also notice that both the simulation have the same residual for 20,000 iterations after which both of them tend to converge.

Influence of α:

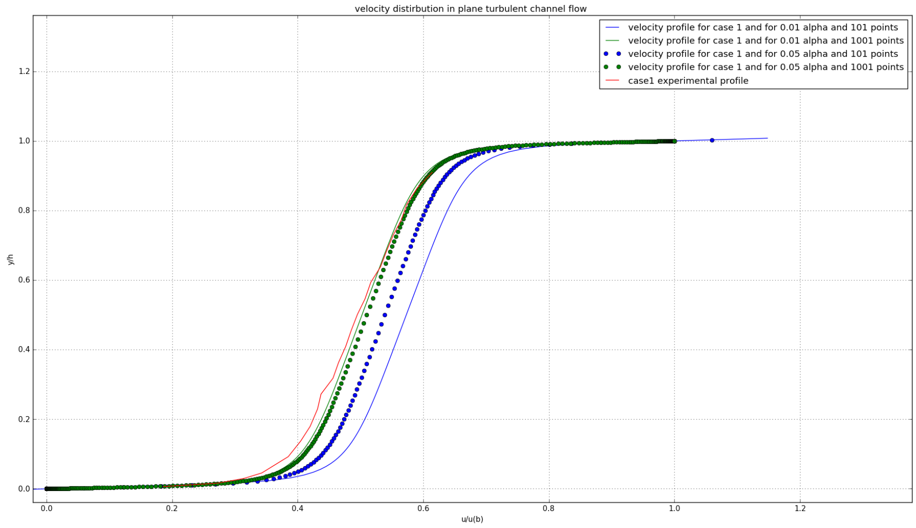


Figure 10. Velocity profiles for different alpha and number of descretised points for case 1 of reynolds

From the above plot we can conclude that for less fine mesh higher stretching helps obtain better values of velocity profile compared to the ones for more refined mesh. this can be explained as for more refined mesh a higher stretching adds more error in the refinement hence the computed values of velocity fields have more error than with lesser stretching.

——————————-End of document—————————