Computational Science and Engineering

Modelling and Simulation of Turbulence

REPORT ON SIMULATION OF FLOW THROUGH RECTANGULAR CAVITY USING OPENFOAM

SUPERVISED BY

Dr.-Ing. Johann Turnow

SUBMITTED BY

Group No.:-8

Jadhav Onkar	(216100299)
Singh Atul	(216100191)
Dhaval Vithani	(215206848)
Thota Sai Rahul	(216100192)

Summary:

The present work aims to address the simulation of flow through rectangular cavity using openfoam. K-Epsilon , k-Omega SST and laminar flow models are used for running simulations with the help of SimpleFoam solver of openFoam. These models work with the help of incompressible Naviers stoke equation. The goal in developing the present simulation is to obtain a preferable model for a given geometry and suggesting the mesh resolution. Comparisons between the simulations obtained from different models leads to the conclusion that K-Omega SST model with fine mesh is preferable for the given geometry. Both pressure and velocity distributions of the k-Omega SST show a vivid point of separation at the bends, compared to coarse mesh of the same model. This shows fine mesh yields better result.

Contents

List of Figures

A   Introduction

The aim of this report is to obtain the simulation of flow over a rectangular cavity using the OpenFOAM framework. The rectangular cavity is shown in figure 1. All dimensions are in [m]. The inlet velocity is uniform with a magnitude of 0.25 $\frac{m}{s}$ and a maximum level of fluctuation is 5% at the inlet. The fluid within the domain is water with a constant temperature of 293.15K. The kinematic viscosity of the water is taken as ν=1×10^{-6}. 2D pitzDaily case is used for running the simulations with the help of 'simpleFoam' solver.

B   Pre-Processing
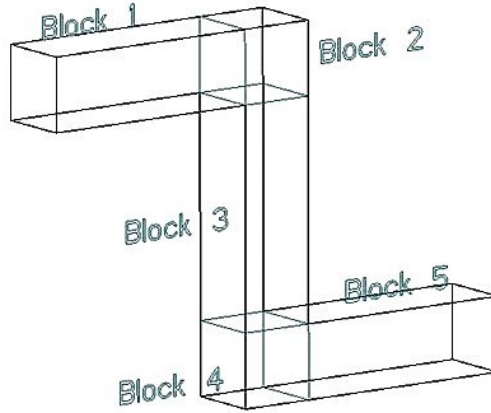
B.1   Geometry and Mesh generation

Figure 1: Rectangular cavity

The given geometry is a computational domain, orderly divided into 5 blocks as shown. Coordinates for the block are arranged in such a way that all blocks build the complete coherent geometry. Special attention is given to the bend of this geometry. This is done in order to stress the large change in pressure and velocity observed in block 2 and 4. Also, mesh resolution especially near the wall need to be considered fine. A flow near the wall is divided into four sublayers; Viscous, Inertial, Buffer sublayer and Ambient flow. The very thin layer next to the wall where viscous effects are dominant is the viscous sublayer. Next to the viscous sublayer is the buffer layer, in which turbulent effects become significant. At this layer, eddies are formed which leads flow to the turbulence. Above the buffer layer is the overlap (or transition) layer, also called the inertial sublayer, in which the turbulent effects are much more significant. Above that is the outer or turbulent layer in the remaining part of

the flow in which turbulent effects dominate over molecular viscous effects. As this turbulent and buffer layers are thin leads to the small eddies formation [2]. So, As much as possible, the mesh should be made fine enough to prevent the wall-adjacent cells from being placed in the buffer layer. For given geometry following mesh setup is used. For these simulation two types of meshing are used i.e. Fine Meshing and Coarse Meshing.

For Coarse Meshing,
hex (0 1 4 3 6 7 10 9) (50 20 1) simpleGrading (1 1 1)
hex (1 2 5 4 7 8 11 10) (20 20 1) simpleGrading (1 1 1)
hex (15 16 2 1 21 22 8 7) (20 50 1) simpleGrading (1 1 1)
hex (12 13 16 15 18 19 22 21) (20 20 1) simpleGrading (1 1 1)
hex (13 14 17 16 19 20 23 22) (50 20 1) simpleGrading (1 1 1)

The above mentioned are five blocks of the geometry. In which first bracket gives coordinates of a particular hex. Second bracket shows Mesh resolution.

For Fine Meshing,
hex (0 1 4 3 6 7 10 9) (140 50 1) simpleGrading (1 1 1)
hex (1 2 5 4 7 8 11 10) (50 50 1) simpleGrading (1 1 1)
hex (15 16 2 1 21 22 8 7) (50 140 1) simpleGrading (1 1 1)
hex (12 13 16 15 18 19 22 21) (50 50 1) simpleGrading (1 1 1)
hex (13 14 17 16 19 20 23 22) (140 50 1) simpleGrading (1 1 1)

B.2 Numerical calculations

(1)Calculation of Reynolds number(Re):-

$$R_e = \frac{U \times H}{\nu}$$

$$R_e = \frac{0.25 \times 0.1}{1 \times 10^{-6}}$$

$$R_e = 25000$$

Where,
R_e = Reynolds number
U = Maximum velocity of the object relative to the fluid (m/s)
H = Characteristic linear dimension (m)

ν = Kinematic viscosity (m^2/s)

(2)Calculation of turbulence kinetic energy(k):-

$$k = \tfrac{3}{2}(U \times I)^2$$
$$k = \tfrac{3}{2}(0.25 \times 0.05)^2$$
$$k = 2.3437 \times 10^{-4}\tfrac{m^2}{s^2}$$

Where,
R_e = Reynolds number
U = Maximum velocity of the object relative to the fluid (m/s)
I = Turbulence fluctuation

(3)Calculation of Turbulence dissipation rate(ε):-

$$\varepsilon = 0.16 \times \frac{k^{\frac{3}{2}}}{L_T}$$
$$\varepsilon = 0.16 \times \frac{0.0002343^{\frac{3}{2}}}{0.1}$$
$$\varepsilon = 5.7409 \times 10^{-6}\tfrac{m^2}{s^3}$$

Where,
k = Turbulence kinetic energy$(\tfrac{m^2}{s^2})$
L_T = Turbulent length(m)

(4)Calculation of Specific turbulence dissipation rate(ω):-

$$\omega = 1.8 \times \frac{\sqrt{k}}{L_T}$$
$$\omega = 1.8 \times \frac{\sqrt{0.0002343}}{0.1}$$
$$\omega = 0.27556\tfrac{1}{s}$$

Where,
k = Turbulence kinetic energy$(\tfrac{m^2}{s^2})$
L_T = Turbulent length(m)

B.3   Velocity distribution near the wall

The law of the wall is defined as, at a certain point, the average velocity of a turbulent flow is proportional to the logarithm of the distance from that point to the wall, or the boundary of the fluid region [12].
In the log layer the velocity profile can be estimated with the log law: $u^+ = \tfrac{1}{\kappa} ln(y^+) + B$ and close to the wall in the viscous sublayer, $u^+ = y^+$

Where:
u^+ Dimensionless velocity
y^+ Dimensionless wall distance
κ von Karman's constant (≈ 0.41)
B Constant (≈ 5.1)

B.4 Boundary Conditions

The boundary conditions are given in zero folder of pitzDaily. Pressure and Velocity will be same for both laminar and turbulent flow. Boundary conditions need to be defined at inlet, outlet and at the walls of the geometry.

a) Inlet Boundary Condition:-
Velocity at inlet in X-direction is given as 0.25 $\frac{m}{s}$ while the velocities in Y and Z directions are zero. Pressure at inlet is given by Newmann's boundary condition i.e. Zero Gradients.

b) Outlet Boundary Condition:-
Velocity at the outlet is given by Newmann's boundary condition.
At the outlet of the model, the typical boundary condition for pressure is fixed static pressure.

c) Boundary Condition at the walls:-
At the upper and lower walls no slip condition is given which means velocities in these directions are zero.
Additionally, for turbulent models k (Turbulent kinetic Energy), ω (Specific rate of dissipation) and ε (turbulent dissipation) need to be defined which has been calculated as,[9]

k = $2.3437 \times 10^{-4} \frac{m^2}{s^2}$

$\omega = 0.27556 \frac{1}{s}$

$\varepsilon = 5.7409 \times 10^{-6} \frac{m^2}{s^3}$

C Equations of Model

C.1 Laminar Flow Model

Laminar flow is governed by Continuity equation and Navier-Stokes equation.
In fluid dynamics, the continuity equation is an expression of conservation of mass. In (vector) differential form, it is written as,

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \vec{u}) = 0.$$

where,
ρ is density,
t is time,
and $\vec{u}$ is fluid velocity.

For incompressible flow, the density remains constant so, the resulting equation is,

$$\frac{\partial u_j}{\partial x_j} = 0$$

In tensor form,

$$\nabla \cdot \vec{u} = 0$$

The left-hand side of the equation is the divergence of velocity [6].

The Navier-Stokes equations govern the motion of fluids and can be seen as Newton's second law of motion for fluids.

$$\frac{\partial u_i}{\partial t} + \frac{\partial u_i u_j}{\partial x_j} = f_i - \frac{1}{\rho}\frac{\partial p}{\partial x_i} + \gamma \nabla \frac{\partial u_i}{\partial x_j}$$

The left hand side of the equation is composed of two terms, local acceleration and convective acceleration. Local acceleration is due to change of velocity in time and the convective acceleration is due to the particle motion in a non-uniform velocity field. On the RHS, first term denotes body force acting at each point in the fluid. Second term gives the acceleration due to pressure gradient and the last term represents acceleration due to the diffusion process.

C.2 Turbulence modelling

Turbulence modelling is the construction and use of a model to predict the effects of turbulence. The following models are used for obtaining simulation results.

a] k-ϵ Model:
It is a two equation model which means, it includes two extra transport equations to represent the turbulent properties of the flow. The first transported variable is turbulent kinetic energy. The second transported variable in this case is the turbulent dissipation (ϵ). It is the variable that determines the scale of the turbulence, whereas the first variable (k), determines the energy in the turbulence.[1]
For turbulent kinetic energy k,

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_i}(\rho k u_i) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right] + P_k + P_b - \rho\epsilon - Y_M + S_k$$

For dissipation ϵ,

$$\frac{\partial}{\partial t}(\rho\epsilon) + \frac{\partial}{\partial x_i}(\rho\epsilon u_i) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_\epsilon}\right)\frac{\partial\epsilon}{\partial x_j}\right] + C_{1\epsilon}\frac{\epsilon}{k}(P_k + C_{3\epsilon}P_b) - C_{2\epsilon}\rho\frac{\epsilon^2}{k} + S_\epsilon$$

where, Turbulent viscosity is modelled as,

$$\mu_t = \rho C_\mu \frac{k^2}{\epsilon}$$

Production of k,

$$P_k = -\rho \overline{u_i' u_j'} \frac{\partial u_j}{\partial x_i}$$

Effect of buoyancy,

$$P_b = \beta g_i \frac{\mu_t}{\mathrm{Pr}_t} \frac{\partial T}{\partial x_i}$$

where Pr_t is the turbulent Prandtl number for energy and g_i is the component of the gravitational vector in the ith direction. For the standard and realizable - models, the default value of Pr_t is 0.85.
The coefficient of thermal expansion, β, is defined as,

$$\beta = -\frac{1}{\rho} \left(\frac{\partial \rho}{\partial T} \right)_p$$

Model constants,

$$C_{1\epsilon} = 1.44, \quad C_{2\epsilon} = 1.92, \quad C_\mu = 0.09, \quad \sigma_k = 1.0, \quad \sigma_\epsilon = 1.3$$

b] k-omega SST Model:
The SST k-ω turbulence model is a two-equation eddy-viscosity model. In which k is a turbulence kinetic energy and omega is the specific rate of diffusion. k-ω SST model shows its better results in adverse pressure gradients and separating flow.[1]
Kinematic Eddy Viscosity,

$$\nu_T = \frac{a_1 k}{\max(a_1 \omega, SF_2)}$$

Turbulence Kinetic Energy,

$$\frac{\partial k}{\partial t} + U_j \frac{\partial k}{\partial x_j} = P_k - \beta^* k\omega + \frac{\partial}{\partial x_j} \left[(\nu + \sigma_k \nu_T) \frac{\partial k}{\partial x_j} \right]$$

Specific Dissipation Rate,

$$\frac{\partial \omega}{\partial t} + U_j \frac{\partial \omega}{\partial x_j} = \alpha S^2 - \beta \omega^2 + \frac{\partial}{\partial x_j} \left[(\nu + \sigma_\omega \nu_T) \frac{\partial \omega}{\partial x_j} \right] + 2(1 - F_1)\sigma_{\omega 2} \frac{1}{\omega} \frac{\partial k}{\partial x_i} \frac{\partial \omega}{\partial x_i}$$

Closure Coefficients and Auxilary Relations,

$$F_2 = \tanh\left[\left[\max\left(\frac{2\sqrt{k}}{\beta^*\omega y}, \frac{500\nu}{y^2\omega}\right)\right]^2\right]$$

$$P_k = \min\left(\tau_{ij}\frac{\partial U_i}{\partial x_j}, 10\beta^* k\omega\right)$$

$$F_1 = \tanh\left\{\left\{\min\left[\max\left(\frac{\sqrt{k}}{\beta^*\omega y}, \frac{500\nu}{y^2\omega}\right), \frac{4\sigma_{\omega 2}k}{CD_{k\omega}y^2}\right]\right\}^4\right\}$$

$$CD_{k\omega} = \max\left(2\rho\sigma_{\omega 2}\frac{1}{\omega}\frac{\partial k}{\partial x_i}\frac{\partial \omega}{\partial x_i}, 10^{-10}\right)$$

$$\phi = \phi_1 F_1 + \phi_2(1 - F_1)$$

$$\alpha_1 = \tfrac{5}{9},$$

$$\alpha_2 = 0.44$$

$$\beta_1 = \tfrac{3}{40},$$

$$\beta_2 = 0.0828$$

$$\beta^* = \tfrac{9}{100}$$

$$\sigma_{k1} = 0.85, \sigma_{k2} = 1$$

$$\sigma_{\omega 1} = 0.5, \sigma_{\omega 2} = 0.856$$

C.3   Momentum equation

The linear momentum equation also known as cauchy's momentum equation describes the non-relativistic momentum transport in any continuum and given by[2],

$$\frac{\partial(\rho\vec{u})}{\partial t} + \overline{\nabla} \cdot (\rho\vec{u}\vec{u}) = \rho g + \overline{\nabla}\sigma_{ij}$$

The LHS of the equation is combined of two terms i.e. local acceleration and convective acceleration. Local acceleration is the time rate of change at fixed point. The convective acceleration is the time rate of change due to the movement of fluid element in the flow field where the flow properties are spatially different.
The RHS of the equation consists of stress tensor $(\nabla\sigma_{ij})$ and gravitational force.

C.4   Reynolds Averaging

Reynolds averaging refers to the process of averaging a variable or an equation in time. Let Φ be any dependent variable that varies in time. This variable can be decomposed into a fluctuating part, Φ' and an average part $\overline{\Phi}$ in the following way:

$$\overline{\Phi} \equiv \tfrac{1}{T}\int_T \Phi(t)dt$$

$$\Phi' \equiv \Phi - \overline{\Phi}$$

Where T is a long enough time to average out the fluctuations in Φ.[5]

C.5 Reynolds Averaged Navier Stokes (RANS) equations

To take time averaging in Reynolds NS Equations, instantaneous value is divided into two parts. The mean value and fluctuating value. The left hand side of this equation represents the change in mean momentum of fluid element due to the unsteadiness in the mean flow. The term $(-\rho\overline{u_i'u_j'})$ appearing in RANS equation is dominating part of the shear stress referred to as Reynolds stress. .[7] For stationary incompressible fluid Newtonian fluid, this equation can be written as, [14]

$$\rho\frac{D\overline{u_i}}{Dt} = F_i - \frac{\partial p}{\partial x_i} + \mu \, \triangle \, \overline{u_i} - \rho\frac{\partial \overline{u_j}\overline{u_i}}{\partial x_j}$$

• **Boussinesq Approach:** Boussinesq Approach is a basis of many practical models. The approach is to define Reynolds stress proportional to the velocity gradient of mean flow which is given as,

$$-\overline{u_i'u_j'} = \nu_t(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}) - \frac{2}{3}k\delta_{ij}$$

Where ν_t is known as turbulent eddy viscosity.
The ν_t is obtained by k-epsilon and k-omega models and this approach helps to obtain Reynolds stresses and this is how we close the term obtained in Naviers-Stokes equation [15].

D Post Processing

D.1 Results

Laminar Model for a Coarse Meshing:-
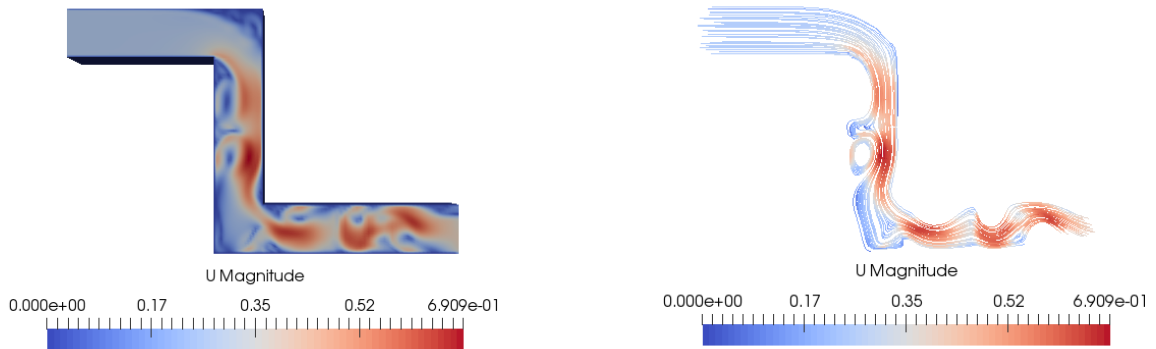
Figure 2: Laminar flow velocity for coarse mesh
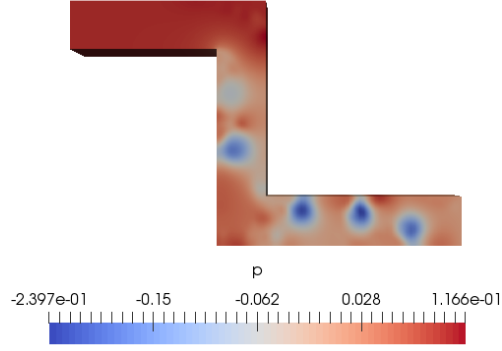
Figure 3: Laminar flow pressure for coarse mesh

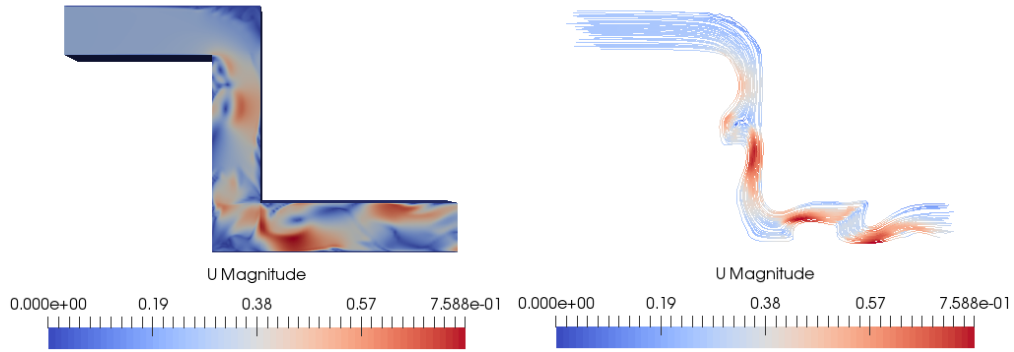**Laminar Model for a Fine Meshing:-**

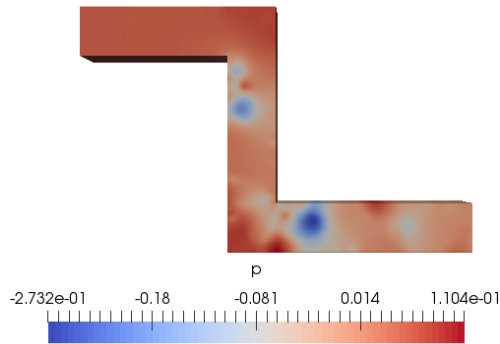
Figure 4: Laminar flow velocity for fine mesh

Figure 5: Laminar Flow Pressure for fine mesh

1) Laminar Flow for coarse and fine meshes:- The mesh is a structured grid of $50{\times}20$ blocks along the x axis. The cluster of mesh is more dense in the bends of blocks 2 and 4. Reynolds number for the given flow is 25000, but as the solver employees its

equations for lower Reynolds number, the present simulations leads to inaccurate results. This is because the laminar flow Reynolds number is usually defined around $2300 - 2500$ which is very less as compared to the present value. Hence the deviations are observed in fig. 2 to fig. 5. Another reason being the Bousinessq term $(-\rho \overline{u'_i u'_j})$ which implies that turbulent characteristics depend on local conditions only, i.e , the Turbulence adjusts itself instantaneously along the flow domain [4]. The turbulent kinematic viscosity appearing in Bousinessq approach is present only for very large Reynolds Number.

k-Epsilon Model for a Coarse Meshing:-
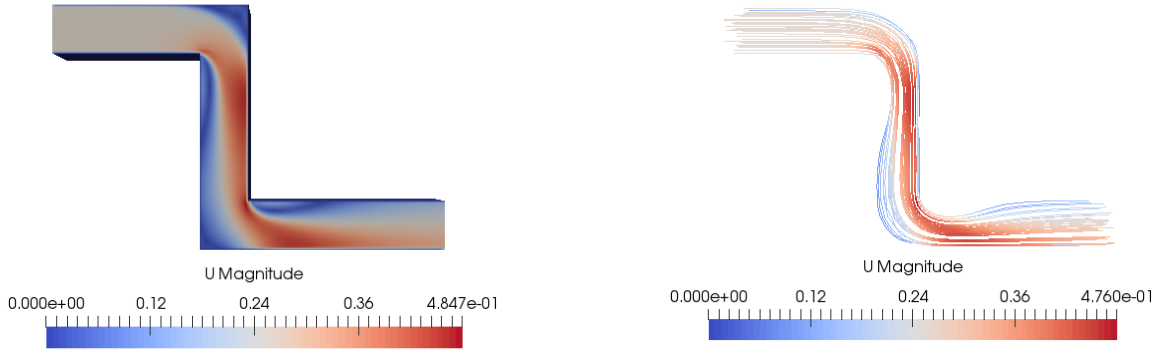
Figure 6: k-ε flow velocity for coarse mesh

Figure 7: k-ε flow pressure for coarse mesh

13
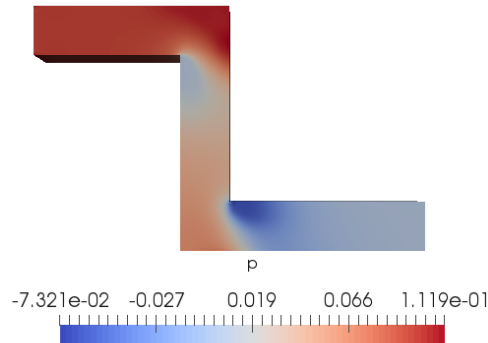
k-Epsilon Model for a Fine Meshing:-
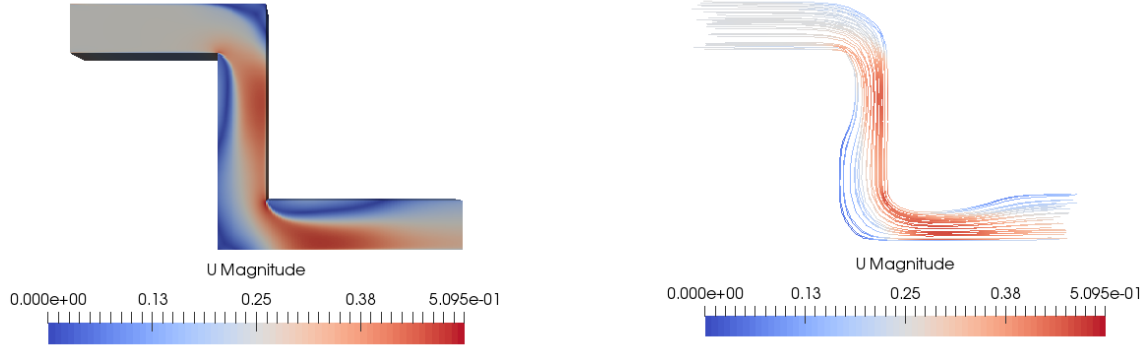
Figure 8: k-ε flow velocity for fine mesh
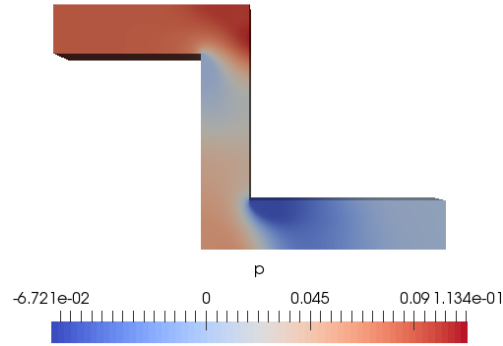
Figure 9: k-ε flow pressure for fine mesh

2) K-epsilon (K-ε) for coarse and fine meshes:- Observing the above figures (fig. 6 to fig. 9) of streamlines for K epsilon model, we do not see any reattachments and separated flows. The two equation K epsilon model is shown to be inappropriate for inlet or compressor flows as they have higher pressure gradients and this decreases the accuracy of the results [13].

k-Omega SST Model for a Coarse Meshing:-
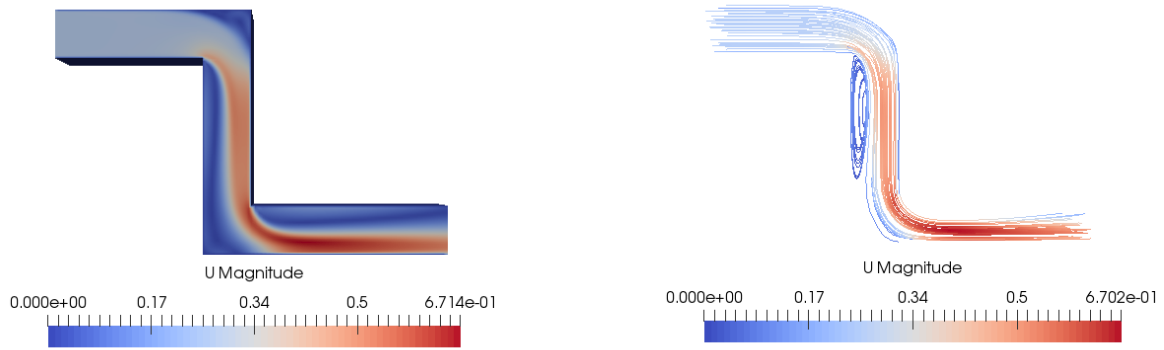
Figure 10: k-ω-SST flow Velocity for coarse mesh
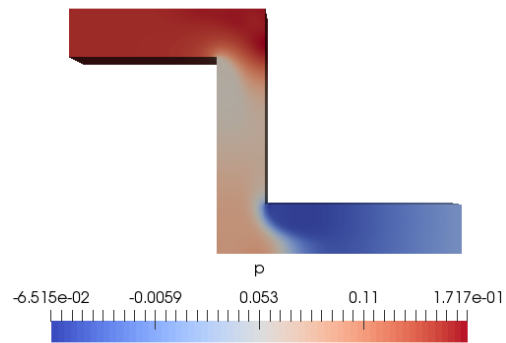
Figure 11: k-ω-SST flow Pressure for coarse mesh

k-Omega SST Model for Fine Meshing:-
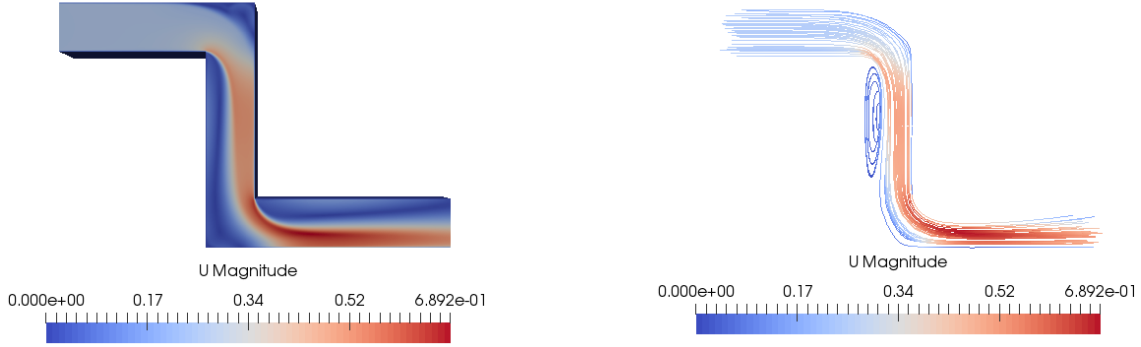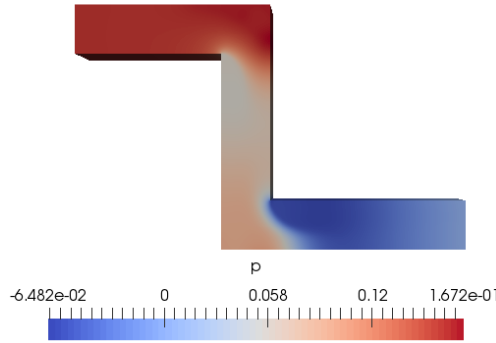

Figure 12: k-ω-SST flow Velocity for fine mesh

Figure 13: k-ω-SST flow Pressure for fine mesh

3) K-Omega SST (K-ω-SST) From figure 12 and 13, it is observed that the changes in flow directions as well as the reattachment points are better observed in K omega sst model compared to others. Also, experiments have shown superior results for wall bounded flows and high adverse pressure gradients in case of K-Omega SST model. So the k-omega SST model is preferable for the present geometry and initial conditions

D.2   Reattachment Point

The seperation point can be defined as the point where the shear stress is zero between the forward and backward flow.[10]

Separation at sharp corners observed in blocks 2, 3 and 4 represent sharply decelerating flow situations where the loss of energy in boundary layer leads to separation [8] which also suggests increase in drag. The separation of boundary layer is observed in figure 12 in block 3. This can be reasoned from the flow at the trailing edge of block 1 reversing its direction, also justified by the adverse pressure gradient observed in that region. The basic mechanism

16

for the reattachment point is that the boundary layer is unable to follow the sharp corner of the geometry. This leads to recirculation and reversal of flow direction as observed and shown in figure 12. Approximately, the co-ordinates (0.3,-0.05,0.05) of block 3 is a reattachment point in case of K-Omega SST model for fine mesh.

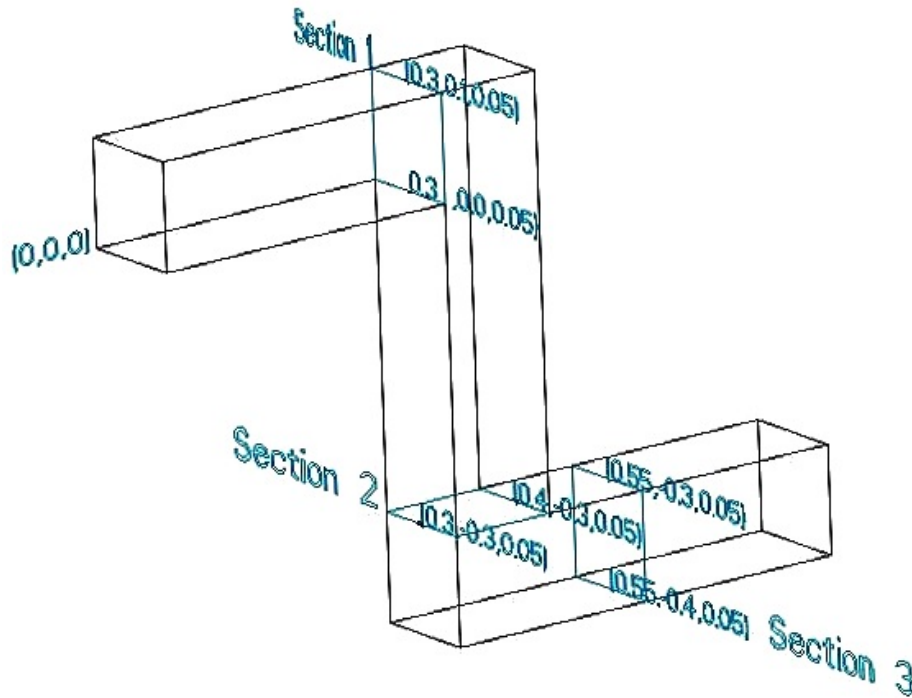## D.3   Velocity and Pressure Profile at selected Sections

Figure 14: Sections at which Pressure and Velocity are observed

The section 1 is defined at (0.3 0.0 0.05) and (0.3 0.1 0.05)
Section 2 is defined at (0.3 -0.3 0.05) and (0.4 -0.3 0.05)
Section 3 is defined at (0.55 -0.4 0.05) and (0.55 -0.3 0.05)
The graphs of "Kinematic Pressure vs Distance along x or y axis" as well as "Mean velocity Vs Distance along x or y axis" is obtained for each of the three models on these line segments which lie at the middle of the named sections.
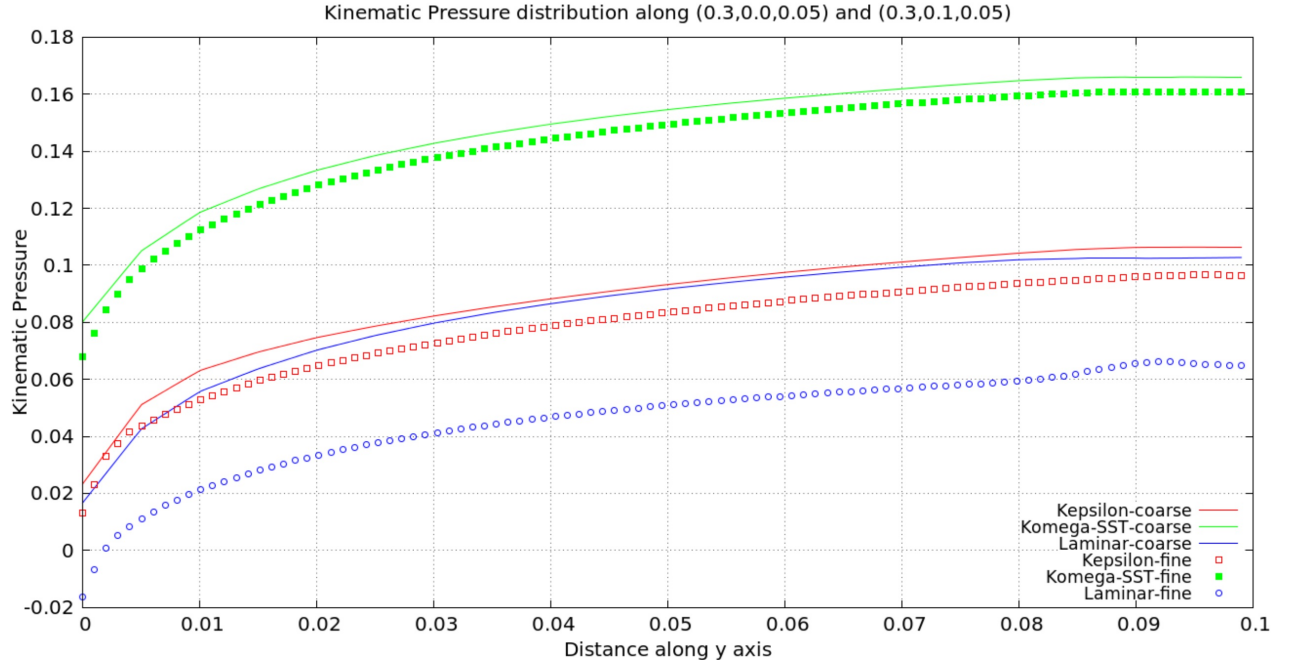
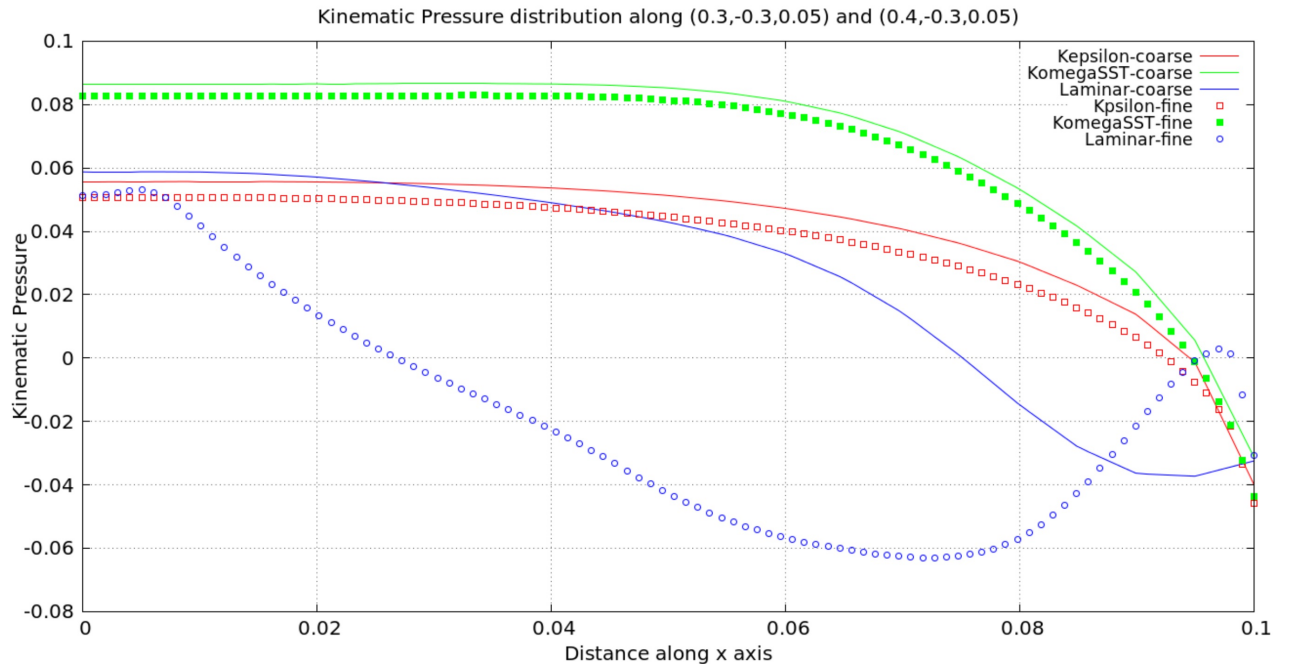Figure 15: Kinematic Pressure at section 1
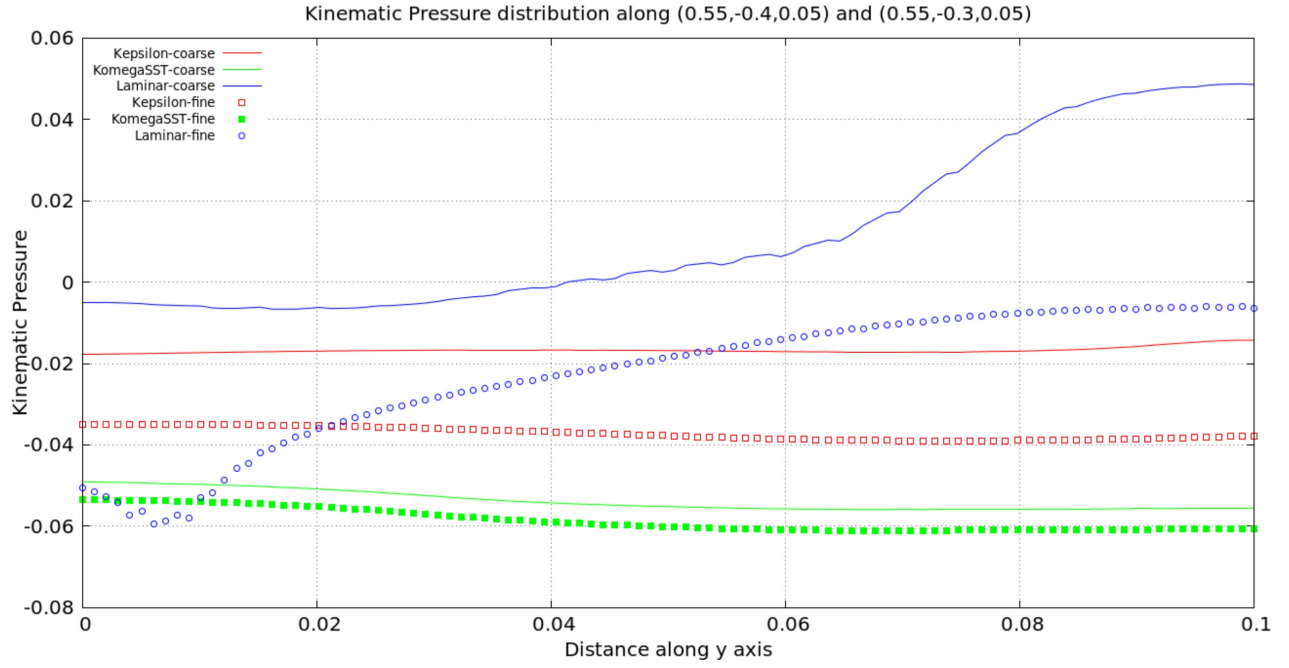

Figure 16: Kinematic Pressure at section 2

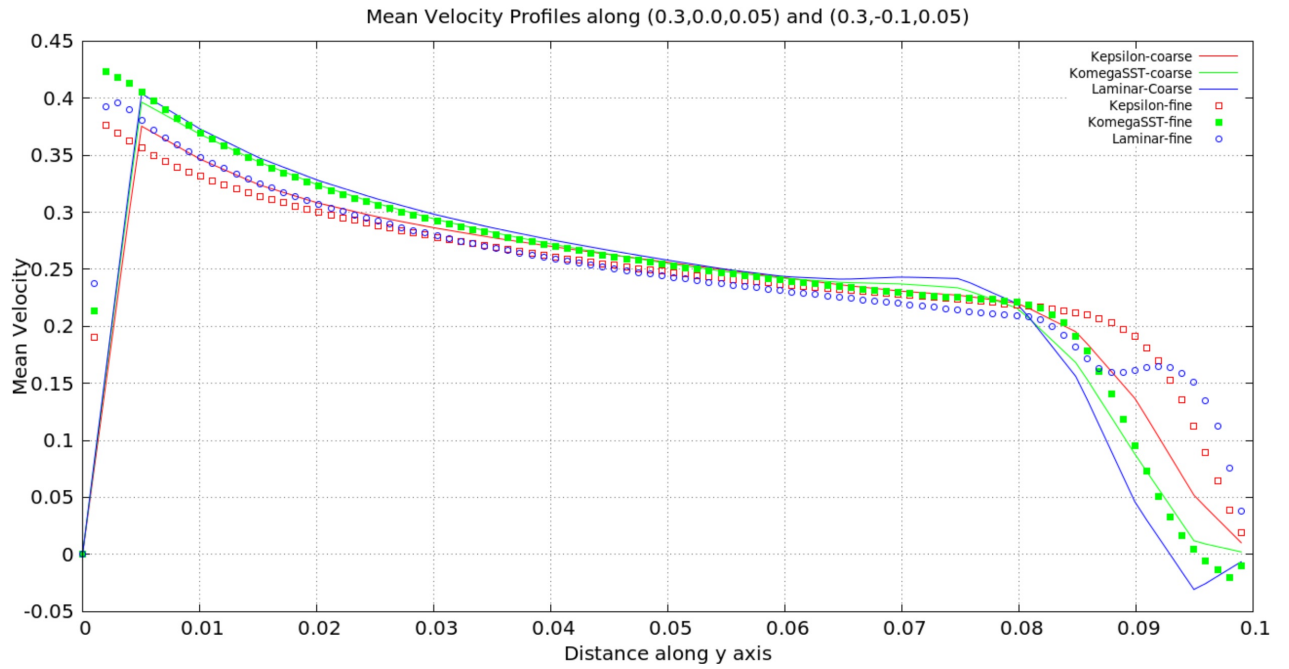Figure 17: Kinematic Pressure at section 3
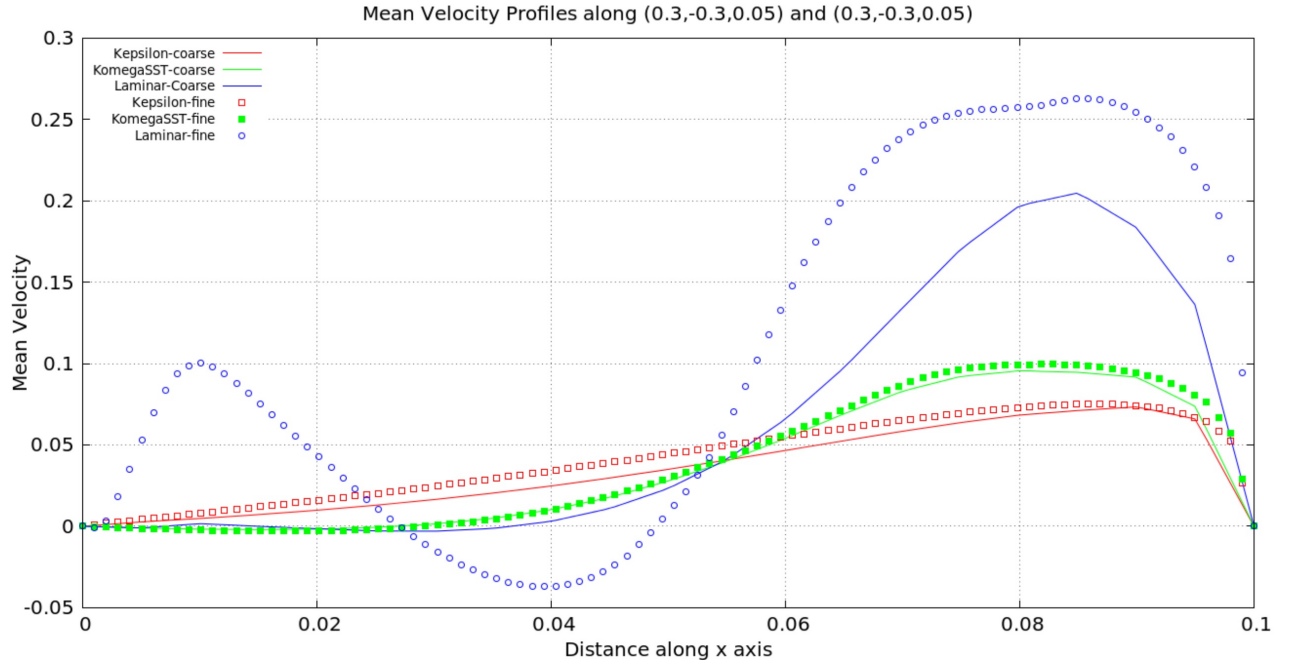

Figure 18: Velocity at Section 1
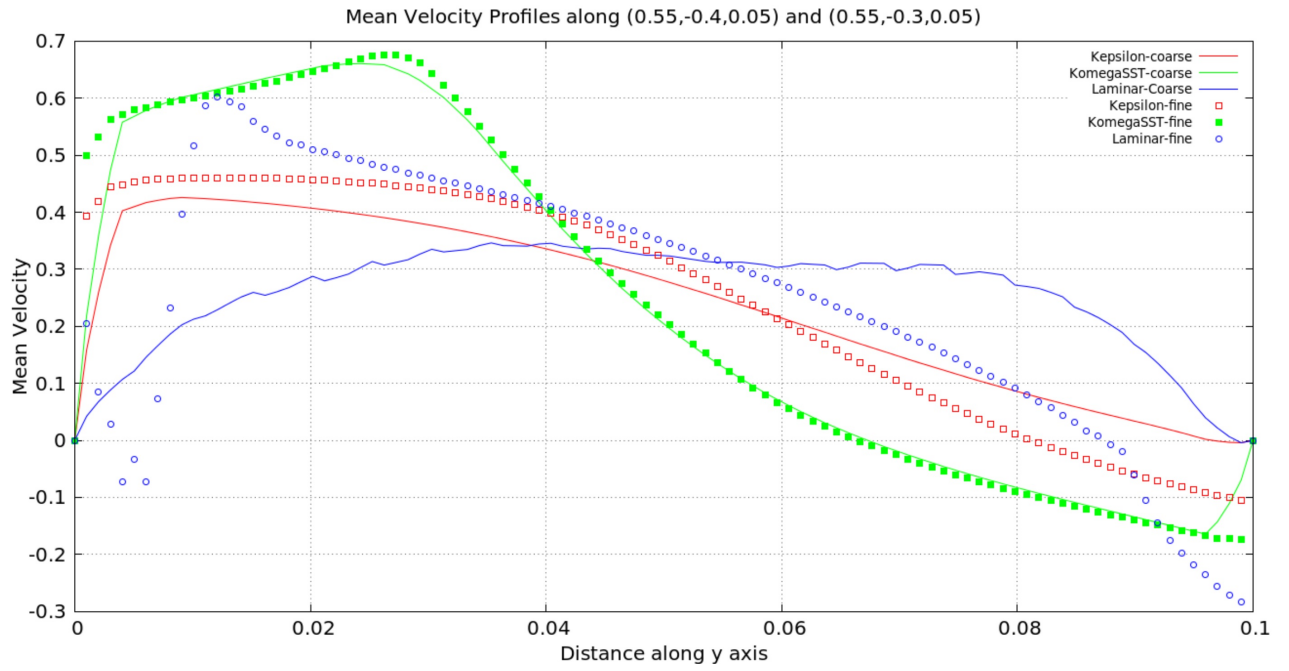
19

Figure 19: Velocity at Section 2

Figure 20: Velocity at Section 3

E   Conclusions

- The above graphs (from fig. 15 to fig. 20) represent the velocity changes in the three selected sections. Namely 1, 2 and 3. The ordinate of these graphs represents the mean velocity and kinematic pressures while the abscissa is the distance along y axis for sections 1 and 3 and distance along x axis for section 2. The extremities of these sections and that of the x axis on the graphs is the walls of the geometry.

- From the figures 12 and 13 of the K-omega SST model for fine mesh, both pressure and velocity distributions shows a vivid point of separation at the bends, compared to coarse mesh of the same (figures 10 and 11). This indicates a fine mesh yields a better and an accurate results.

- From figures 15 , 16 and 17 general observations indicate a nearby deviations of coarse mesh graphs (indicated by dots) from the fine mesh graphs (indicated by lines) for K-epsilon model while that for K-omega SST model are nearly of same nature showing very less deviations. Also, the above discussion conclude that K-Omega SST model is preferable for the present geometry.

- From the conducted simulations the k-omega SST model will be used for further investigations and improvements because it gives better results.

F   References

1] Prof. Dr.-Ing. habil. Nikolai Kornev and Prof. Dr.-Ing. habil. Irina Cherunova . Lectures on computational fluid dynamics and heat transfer with applications to human thermo-dynamics.2014.

2] YUNUS A.ÇENGEL and JOHN M.CIMBALA fluid mechanics fundamentals and applications. Third Edition. 3] F Moukalled and L. Mangani. The Finite Volume Method in Computational Fluid Dynamics: An Advanced Introduction with OpenFOAM and Matlab. Springer, 2015.

4] H. Versteeg and W. Malalasekera. An Introduction to Computational Fluid Dynamics: The Finite Volume Method. Pearson Education, 2007.

F.1   Web References

5]$http://www.cfd-online.com/Wiki/Reynoldsaveraging.$

6]$http://www.cfd-online.com/Wiki/Continuityequation.$

7]$https://en.wikipedia.org/wiki/Reynolds-averagedNavier\%E2\%80\%93Stokes_equations$

8]$http://www.bakker.org/dartmouth06/engs150/11-bl.pdf$

9]$http://www.computationalfluiddynamics.com.au/convergence-and-mesh-independent-study/\%20mesh\%20resolution$

10]$https://en.wikipedia.org/wiki/Flowseparation$

11]$http://www.cfdsupport.com/OpenFOAM-Training-by-CFD-Support/node85.html$

12]$http://www.cfd-online.com/Wiki/Lawofthewall$

13]$https://en.wikipedia.org/wiki/K-epsilonturbulencemodel.$

14]$http://daad.wb.tu-harburg.de/fileadmin/BackUsersResources/Flood_Probability/2D/Steffi-2D/pdf/Reynolds_average_Navier-Stokes_equation.pdf$

15]$http://daad.wb.tu-harburg.de/fileadmin/BackUsersResources/Flood_Probability/2D/Steffi-2D/pdf/Approaches_for_turbulent_modelling.pdf.$