FOSSEE, IIT Bombay
OpenFOAM Case Study Project
2nd September 2020

TURBULENT FLOW OVER BACKWARD-FACING STEP

Kamal S Kumar
Fourth Year B.Tech
Swami Vivekanand University

Abstract:

Backward-Facing Step (BFS) flow is a flow separation model, in which separation of flow occurs when the flow is exposed to a sudden increase in cross-sectional area. These phenomena can be observed in internal flows in diffusers, turbines, combustors, or pipes, and external flows over aircrafts, around buildings, or over stepped channels. The flow separation after a step will introduce separation bubble formation, evolution and re-attachment process, which is dependent on the BFS geometric design, the inlet and outlet conditions, turbulent intensity, as well as heat transfer conditions. The purpose of this case study is to simulate and analyse the changes in flow properties over a BFS, and to obtain data regarding changes of flow parameters with the change in Turbulence Models and Reynolds number. This study is being performed using various FLOSS software.

Keywords: Flow Separation, Backward-Facing Step, Turbulence Modelling

1. Introduction

The Backward-Facing Step (BFS) flow is a fluid flow model used to study the changes in the flow properties or nature of flow when a fully developed flow encounters a backward facing step. It is also called as "sudden expansion flows", "circular expanding flow", "diverging channel" and so on.

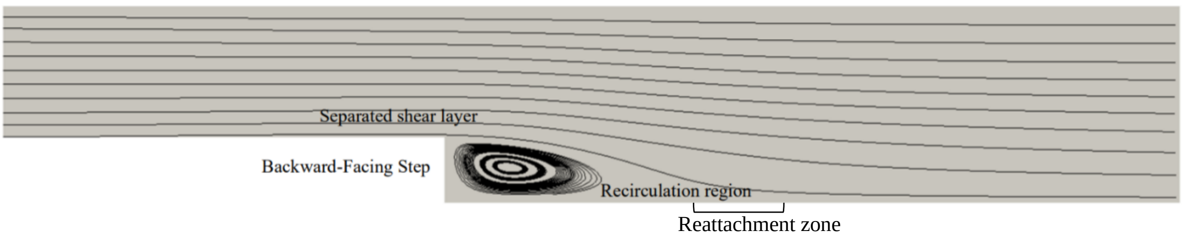

Fig.1 Flow velocity profile of BFS flow

A continuous retardation of the flow turns the wall shear stress negative and the flow reverses and a region of recirculating occurs behind the step which is mostly responsible for the creation of unsteady eddy structure or recirculation bubbles. The flow bubble/recirculation is driven by the low pressure created by the presence of vortices in the separated shear layer. Finally, as the fluid recovers from the sudden expansion, the pressure gradient becomes favourable and the fluid attaches at a position called reattachment point.

The nature of flow over a BFS, especially recirculation and reattachment of flow about the step, has intrigued many. Hence, being studied and found its application in various industries such as aviation, automobile, construction, etc.

2. Mathematical Modelling

2.1 Governing Equations

This case study is based on Reynolds-averaged Navier–Stokes equations (or RANS equations or RAS equations), are time-averaged equations of motion for fluid flow. For a stationary flow of an incompressible Newtonian fluid, these equations can be written in Einstein notation in Recirculation region Separated shear layer Backward-Facing Step Cartesian coordinates as:

$$\rho \bar{u}_j \frac{\partial \bar{u}_i}{\partial x_j} = \rho \bar{f}_i + \frac{\partial}{\partial x_j}\left[-\bar{p}\delta_{ij} + \mu\left(\frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i}\right) - \rho \overline{u_i' u_j'}\right]$$

The Navier-Stokes equation is supplemented with the incompressibility condition:

$$\nabla \cdot U = 0$$

2.2 Turbulence Models

In this case study, we simulate the working fluid through three different turbulence models -

a. kEpsilon
b. kOmega
c. kOmegaSST

kEpsilon [k-ε] Turbulence Model:

It is the most common model used in computational fluid dynamics (CFD) to simulate mean flow characteristics for turbulent flow conditions. It is a two-equation model which gives a general description of turbulence by means of two transport partial difference equations.

- The first transported variable is the turbulent kinetic energy (k)

$$\frac{\partial(\rho k)}{\partial t} + \frac{\partial(\rho k u_i)}{\partial x_i} = \frac{\partial}{\partial x_j}\left[\frac{\mu_t}{\sigma_k}\frac{\partial k}{\partial x_j}\right] + 2\mu_t E_{ij}E_{ij} - \rho\varepsilon$$

- The second transported variable is the rate of dissipation of turbulent kinetic energy (ε)

$$\frac{\partial(\rho \varepsilon)}{\partial t} + \frac{\partial(\rho \varepsilon u_i)}{\partial x_i} = \frac{\partial}{\partial x_j}\left[\frac{\mu_t}{\sigma_\varepsilon}\frac{\partial \varepsilon}{\partial x_j}\right] + C_{1\varepsilon}\frac{\varepsilon}{k}2\mu_t E_{ij}E_{ij} - C_{2\varepsilon}\rho\frac{\varepsilon^2}{k}$$

Unlike earlier turbulence models, k-ε model focuses on the mechanisms that affect the turbulent kinetic energy. The underlying assumption of this model is that the turbulent viscosity is isotropic, in other words, the ratio between Reynolds stress and mean rate of deformations is the same in all directions.

kOmega [k–ω] Turbulence Model:

In computational fluid dynamics, the k–omega (k–ω) turbulence model is a common two-equation turbulence model, that is used as a closure for the RANS equations. The model attempts to predict turbulence by two partial differential equations for two variables.

- The first variable being the turbulence kinetic energy (k)

$$\frac{\partial(\rho k)}{\partial t} + \frac{\partial(\rho u_j k)}{\partial x_j} = \rho P - \beta^* \rho \omega k + \frac{\partial}{\partial x_j}\left[\left(\mu + \sigma_k \frac{\rho k}{\omega}\right)\frac{\partial k}{\partial x_j}\right], \qquad \text{with } P = \tau_{ij}\frac{\partial u_i}{\partial x_j}$$

- The second (ω) is the specific rate of dissipation (of the turbulence kinetic energy k into internal thermal energy)

$$\frac{\partial(\rho\omega)}{\partial t} + \frac{\partial(\rho u_j \omega)}{\partial x_j} = \frac{\alpha\omega}{k} P - \beta\rho\omega^2 + \frac{\partial}{\partial x_j}\left[\left(\mu + \sigma_\omega \frac{\rho k}{\omega}\right)\frac{\partial \omega}{\partial x_j}\right] + \frac{\rho\sigma_d}{\omega}\frac{\partial k}{\partial x_j}\frac{\partial \omega}{\partial x_j}$$

kOmega SST [Shear Stress Transport] Turbulence Model:

Shear Stress Transport turbulence model, is a widely used and robust two-equation eddy-viscosity turbulence model used in CFD. The model combines the k-omega turbulence model and k-epsilon turbulence model such that the k-omega is used in the inner region of the boundary layer and switches to the k-epsilon in the free shear flow, aiming to overcome the deficiencies of the standard k-omega model w.r.t dependency on the freestream values of k and omega.

The turbulence specific dissipation rate equation is given by:

$$\frac{D}{Dt}(\rho\omega) = \nabla\cdot(\rho D_\omega \nabla\omega) + \frac{\rho\gamma G}{\nu} - \frac{2}{3}\rho\gamma\omega\left(\nabla\cdot\mathbf{u}\right) - \rho\beta\omega^2 - \rho\left(F_1 - 1\right)CD_{k\omega} + S_\omega$$

and the turbulence kinetic energy by:

$$\frac{D}{Dt}(\rho k) = \nabla\cdot(\rho D_k \nabla k) + \rho G - \frac{2}{3}\rho k\left(\nabla\cdot\mathbf{u}\right) - \rho\beta^* \omega k + S_k$$

The turbulence viscosity is obtained using:

$$\nu_t = a_1 \frac{k}{\max(a_1\omega, b_1 F_{23}\mathbf{S})}$$

3. Simulation Procedure

3.1 Geometry and Mesh

The side view of the domain being used in this case study is shown below:

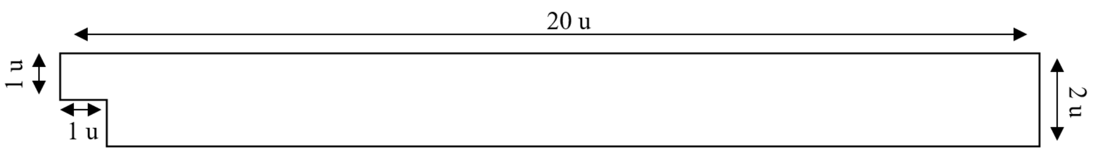

Fig.2 Side view of the domain

Here, 1 u implies 1 meter. This geometry is given a depth of 0.1 u. Geometry and mesh of this geometry is created using blockMesh utility, which can be done by creating a blockMeshDict in the case study directory or modifying any pre-existing blockMeshDict file (located in the system or constant/polyMesh folder) copied from any pre-built tutorials.

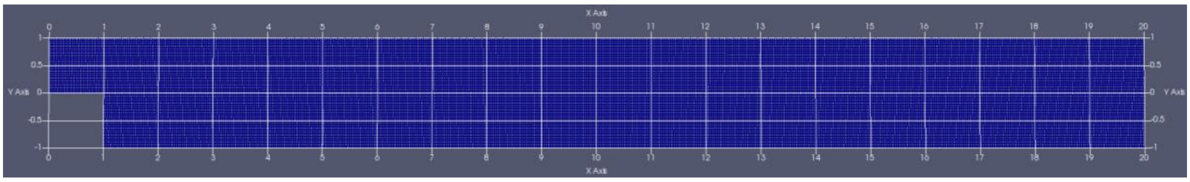

Fig.3 Mesh visualized in Paraview (Surface with Edges preview)

3.2 Initial and Boundary Conditions

- Boundary and Initial Conditions (BC/IC)
  Boundary conditions to be used in CASE A and CASE B of this case study is given below:

Flow Parameters	Inlet	Top and Bottom Walls	Front and Back	Outlet
Pressure (p)	zeroGradient	zeroGradient	empty	fixedValue
Velocity (U)	fixedValue	noSlip	empty	zeroGradient
Turbulent kinetic energy (k)	fixedValue	kqRWallFunction	empty	zeroGradient
Turbulence dissipation (ε)	fixedValue	epsilonWallFunction	empty	zeroGradient
Specific turbulence dissipation (ω)	fixedValue	omegaWallFunction	empty	zeroGradient

  The value of each flow parameters (IC) is different for each subcase and will be discussed in details in result section.

- Transport Properties
  Transport model is set as Newtonian, with kinematic viscosity (ν) equal to 1.516×10^{-5} (air at 20 ℃).

- Turbulence Properties
  For Case A (Different Turbulence Models)
  - CASE A1:
    Simulation type is set as RAS (Reynolds-Averaged Simulation) and RAS model is set as kEpsilon with turbulence on.
  - CASE A2:
    Simulation type is set as RAS (Reynolds-Averaged Simulation) and RAS model is set as kOmega with turbulence on.
  - CASE A3 and CASE B:
    Simulation type is set as RAS (Reynolds-Averaged Simulation) and RAS model is set as kOmegaSST with turbulence on. This RAS model aims to overcome the deficiencies of the standard kOmega model with respect to dependency on the freestream values of k and omega.

3.3 Solver

This study demands for an incompressible turbulent flow, we will be using simpleFoam solver, which utilizes the SIMPLE (Semi-Implicit Method for Pressure Linked Equations) algorithm. The solver follows a segregated solution strategy. This means that the equations for each variable characterizing the system (the velocity *u*, the pressure *p* and the variables characterizing turbulence) is solved sequentially and the solution of the preceding equations is inserted in the subsequent equation.

3.4 Result Extraction

The plots are generated using GNUPLOT, and the visualization of simulation and contour is done using PARAVIEW.

4. Result

The OpenFoam simulation results for BFS flow with different turbulence models and varying Reynolds number is given in this section. All the simulated cases are subjected to run for 4000 seconds without any convergence criteria assigned.

CASE A - Different Turbulence Models

These results are obtained by simulating a BFS flow with fixed boundary and initial conditions but with varying turbulence models. The fluid has a Reynolds number ranging between 500 and 800 i.e., Re ∈ (400, 800). The value of internal pressure and velocity throughout the cross-section is zero.

- CASE A1 - kEpsilon Turbulence Model

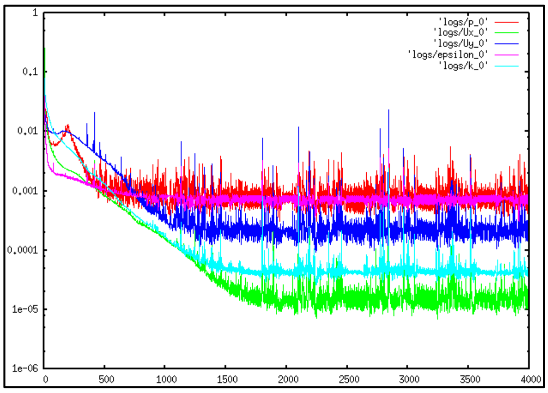

Fig.4 Monitoring convergence using residual logs

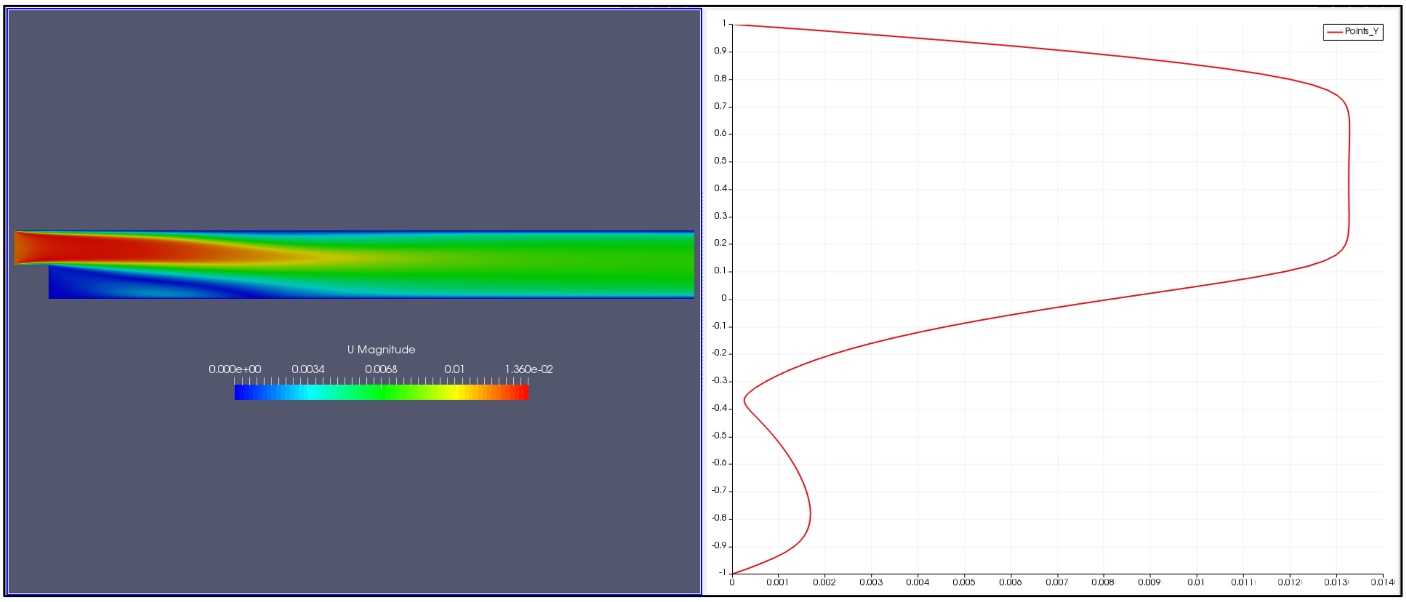

Fig.5 Velocity profile at a cross-section created by a line (3 m from the inlet)

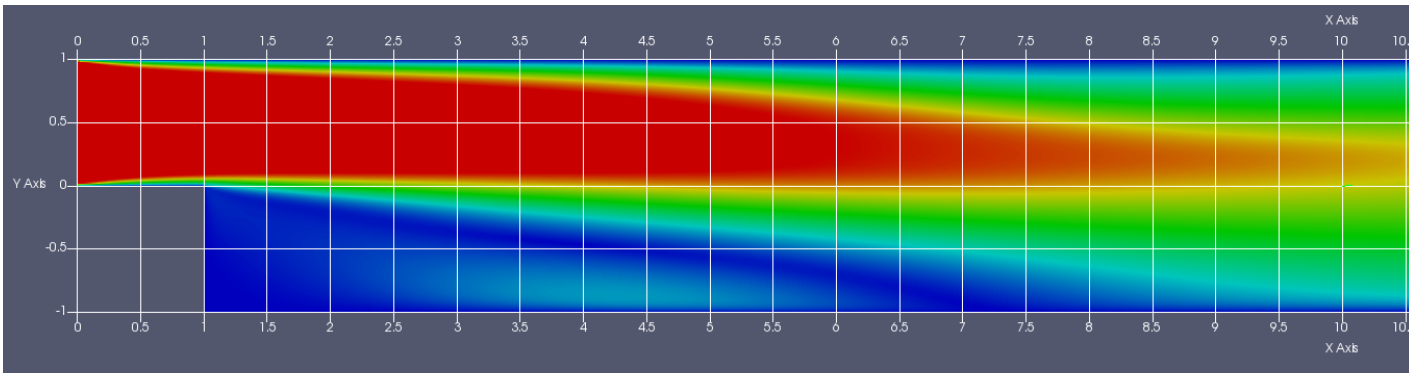

Fig.6 Velocity contour (t = 4000s) with Grids enabled

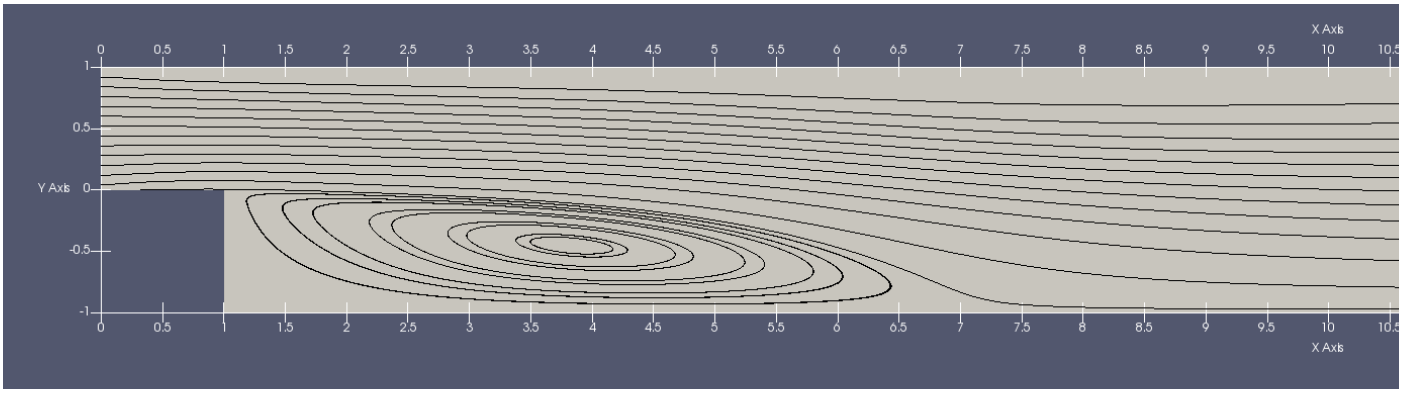

Fig.7 Flow streamlines (t = 4000s) with Grids enabled

CASE A2 - kOmega Turbulence Model

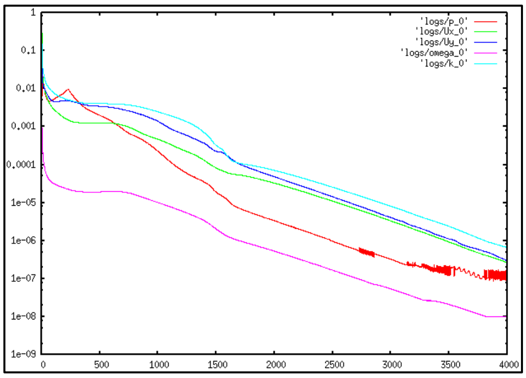

Fig.8 Monitoring convergence using residual logs

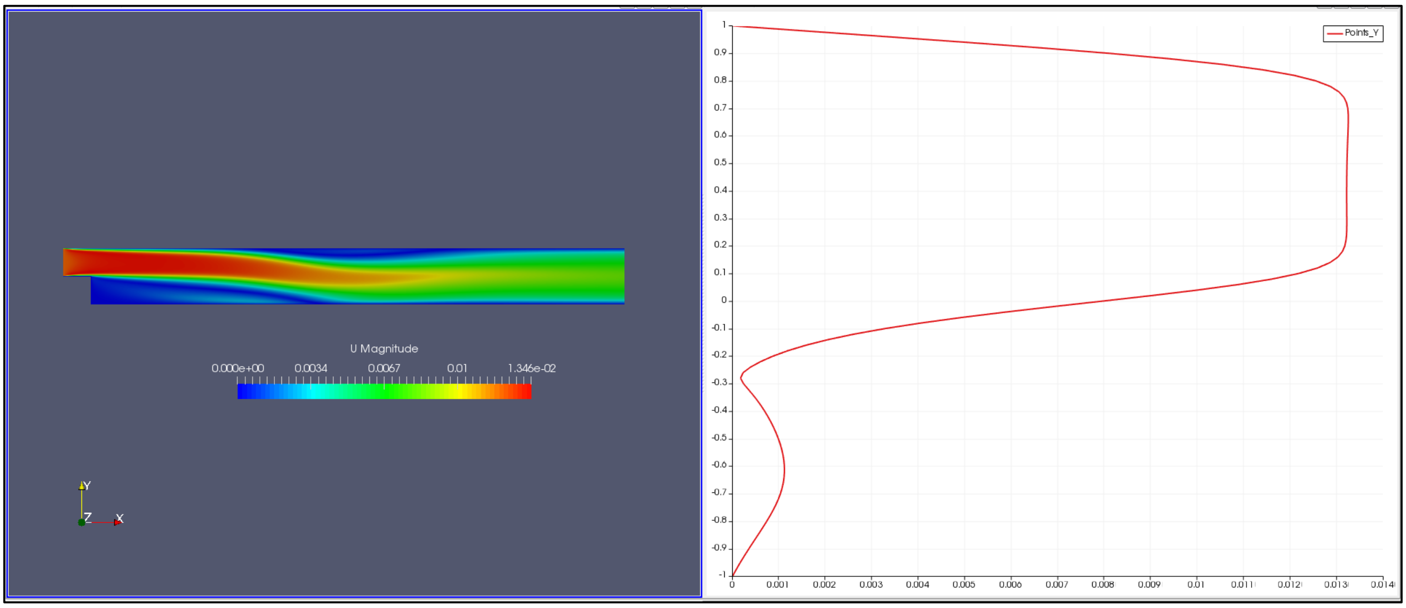

Fig.9 Velocity profile at a cross-section created by a line (3 m from the inlet)

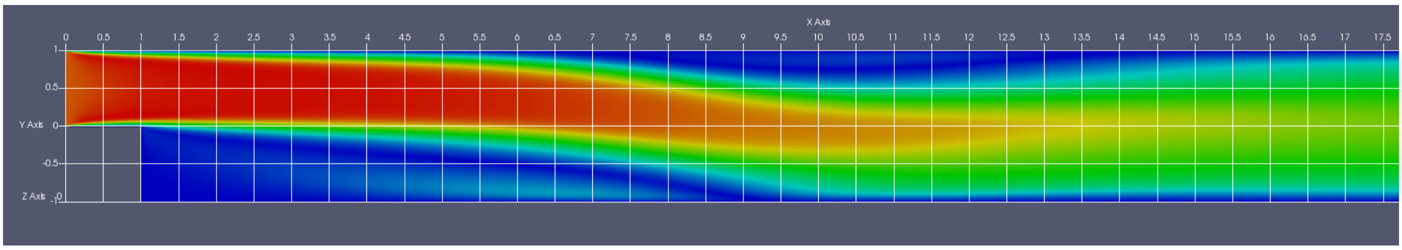

Fig.10 Velocity contour (t = 4000s) with Grids enabled

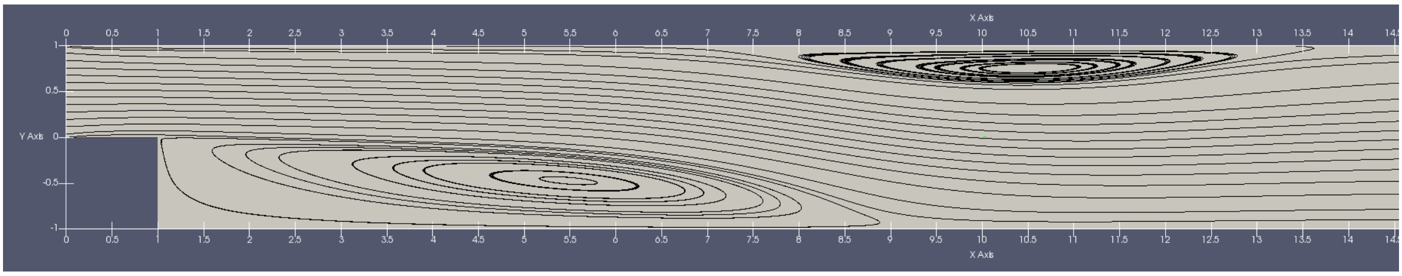

Fig.11 Flow streamlines (t = 4000s) with Grids enabled

- CASE A3 - kOmegaSST Turbulence Model

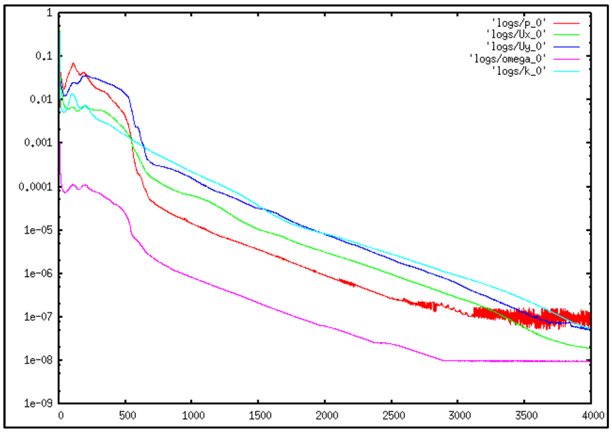

Fig.12 Monitoring convergence using residual logs

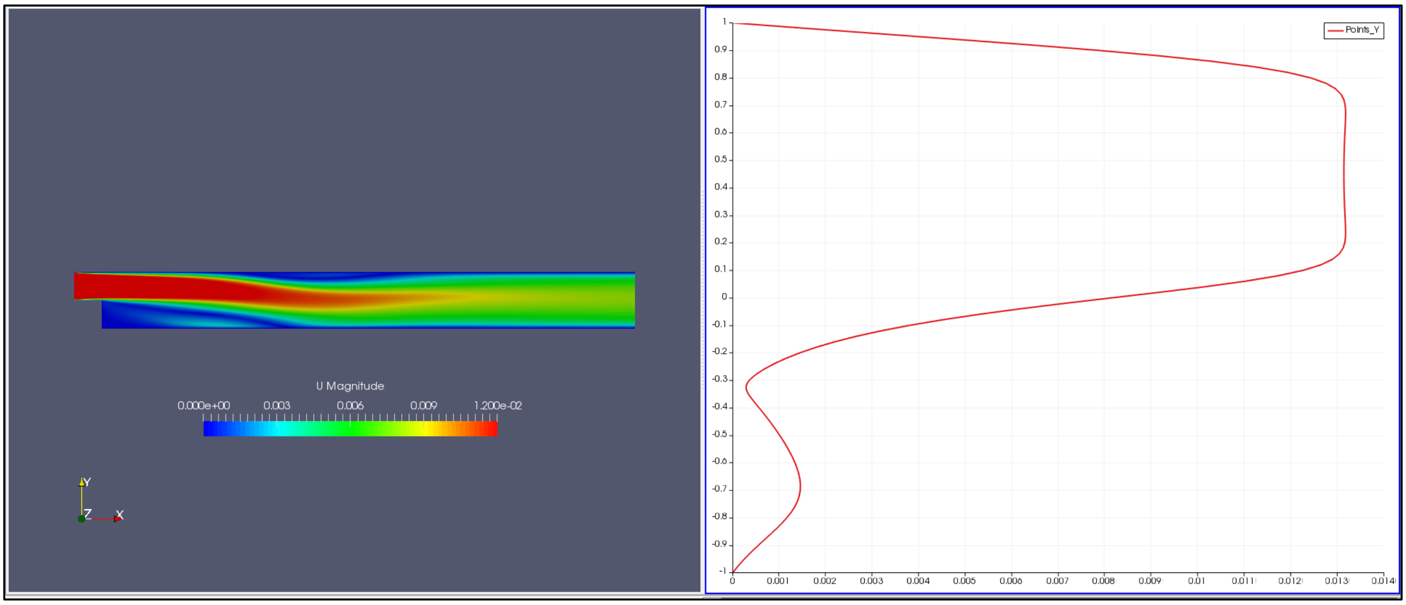

Fig.13 Velocity profile at a cross-section created by a line (3 m from the inlet)

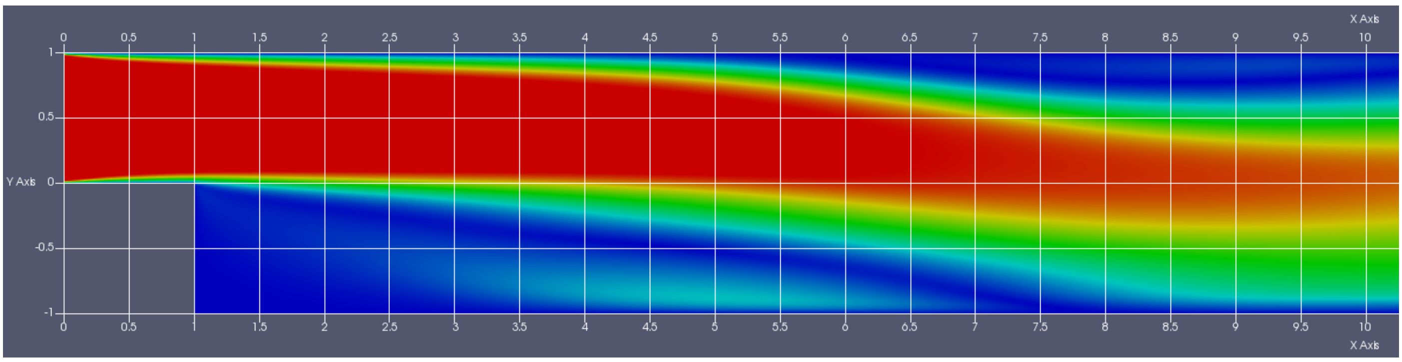

Fig.14 Velocity contour (t = 4000s) with Grids enabled

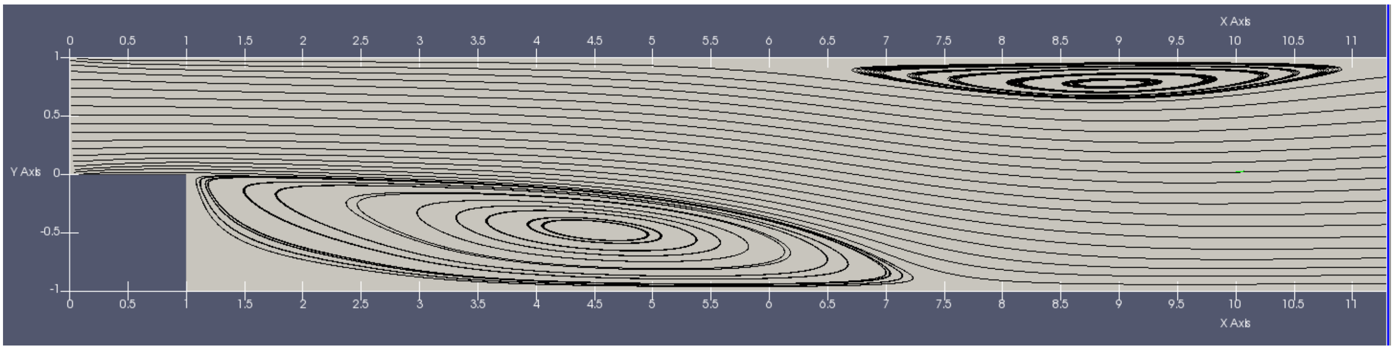

Fig.15 Flow streamlines (t = 4000s) with Grids enabled

CASE B - Varying Reynolds number

These results are obtained by simulating a BFS flow with kOmegaSST turbulence model, with varying Reynolds number. The fluid parameters changes with change in Reynolds number (inlet/freestream velocity) and are calculated separately for each case. The value of internal pressure and velocity throughout the cross-section is zero.

- CASE B1 (200 < Re < 500)

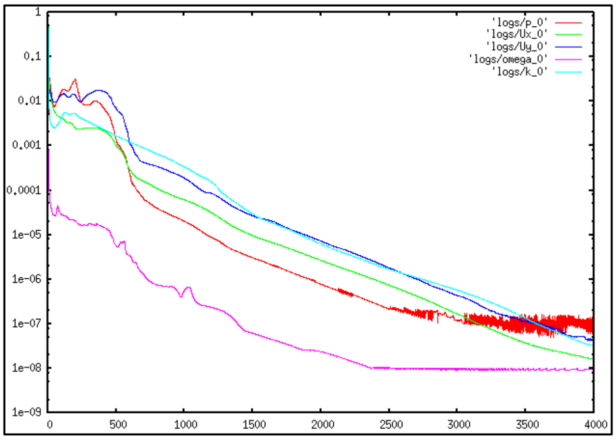

Fig.16 Monitoring convergence using residual logs

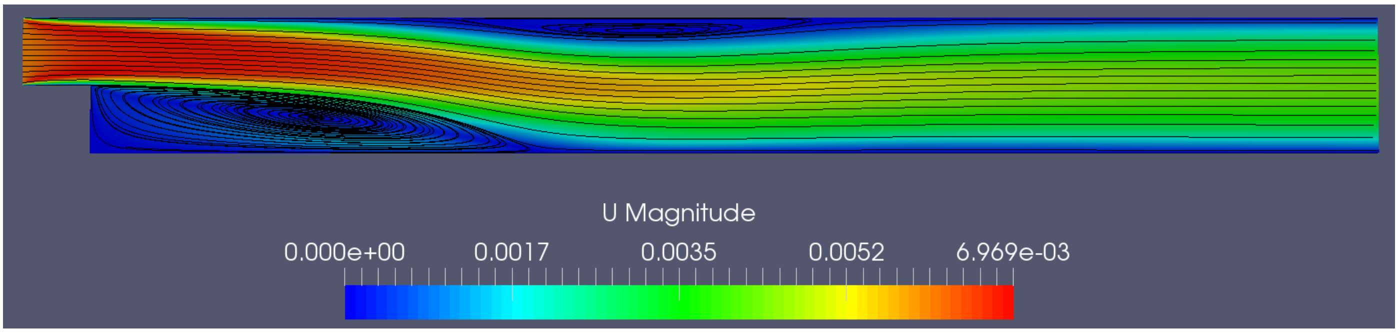

Fig.17 Flow streamlines (t = 4000s)

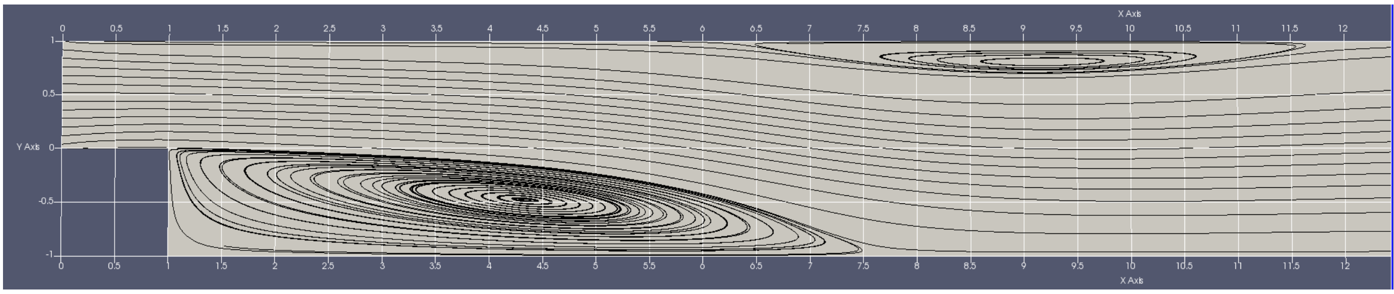

Fig.18 Flow streamlines (t = 4000s) with Grids enabled

- CASE B2 (500 < Re < 800)

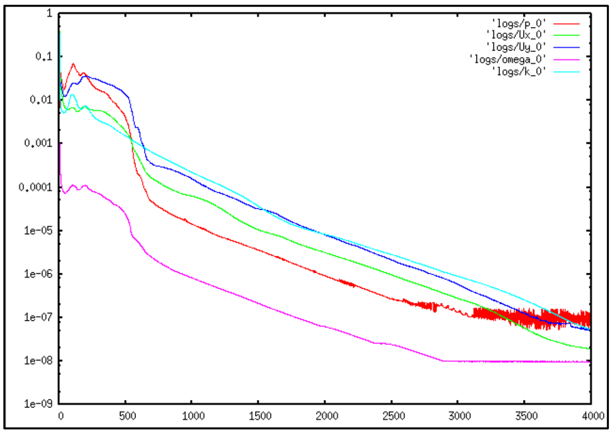

Fig.19 Monitoring convergence using residual logs

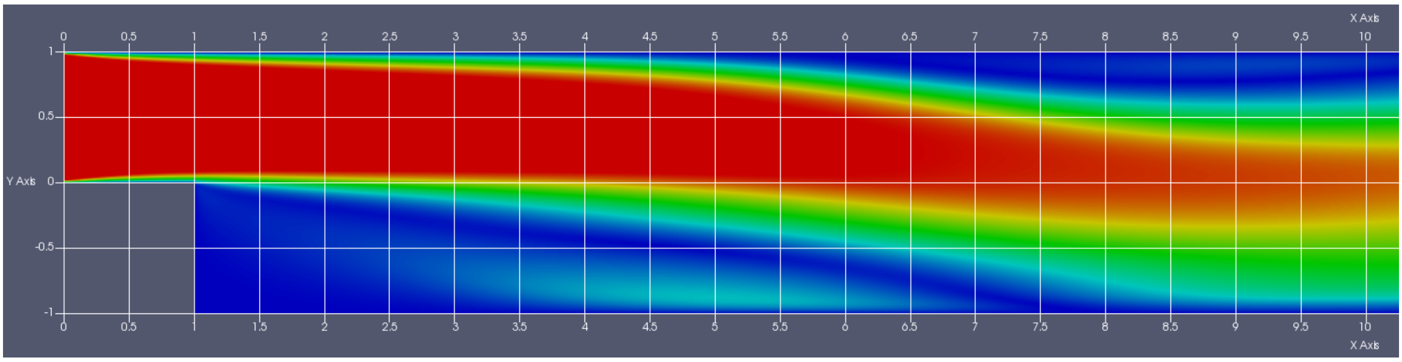

Fig.20 Velocity contour (t = 4000s) with Grids enabled

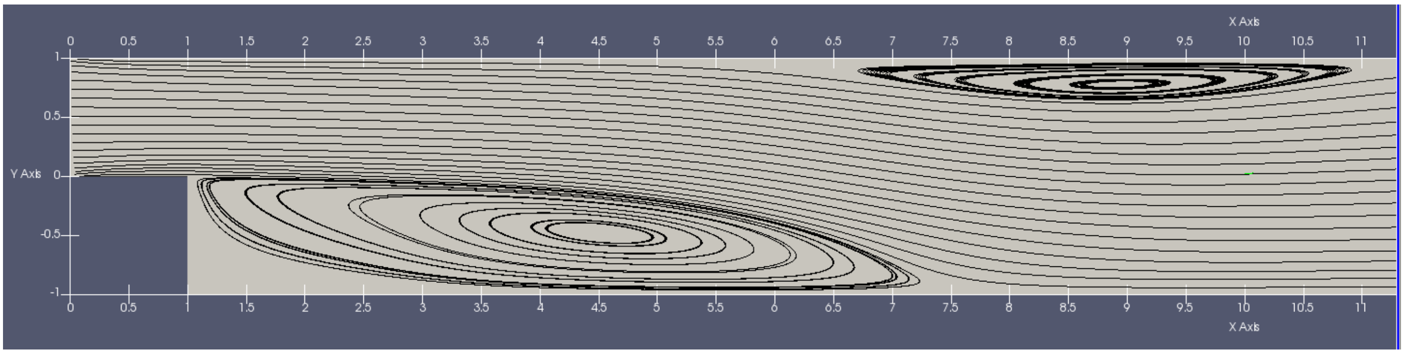

Fig.21 Flow streamlines (t = 4000s) with Grids enabled

- CASE B3 (800 < Re < 2000)

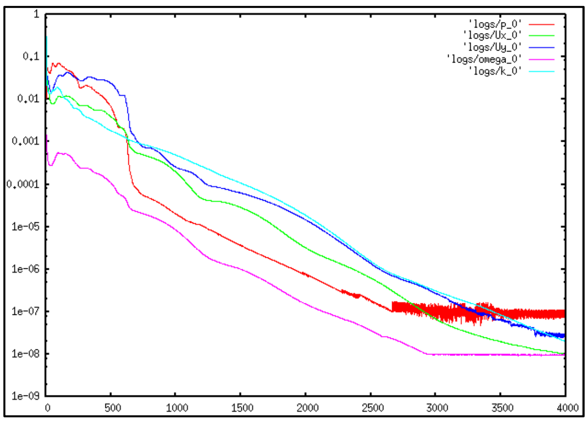

Fig.22 Monitoring convergence using residual logs

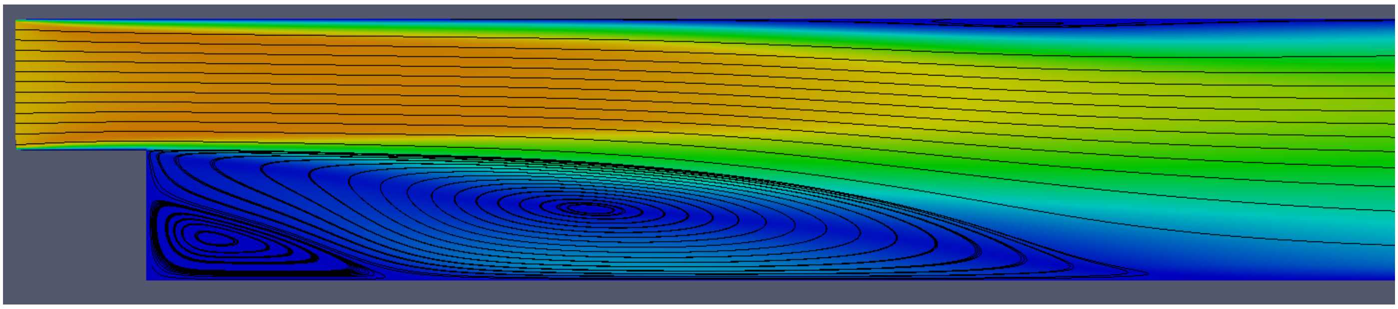

Fig.23 Flow streamlines above velocity contour (t = 4000s)

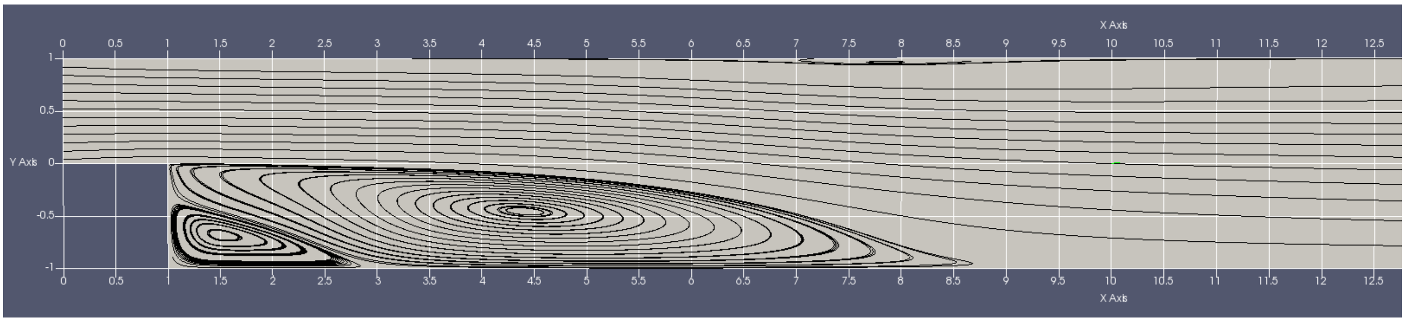

Fig.24 Flow streamlines (t = 4000s) with Grids enabled

Conclusion

From the simulations conducted in Case A, it is evident that all three turbulence model produces quite different results and selection of particular turbulence plays a huge role in obtaining desired results since these heavily influence the outcome of the simulation.

The results of Case A1 do not show the presence of a secondary vortex or recirculation bubble whereas the later cases do. This shows that the k-epsilon turbulence model is not an appropriate choice for similar CFD case studies. In case of reattachment of flow, the flow in Case A1 (6.5 - 7.5 m) re-attaches earlier than both Case A2 (8.5 - 9.5 m) and Case A3 (7.0 - 8.0 m). Case A2 predicts the near-wall regions better than either cases but performs poorly in far-wall regions.

The k-omega and k-epsilon are two-equation models. These are semi-analytic models. The k-epsilon model predicts well far from the boundaries or walls and the k- omega model predicts well at near-wall regions. The SST model being a combination of these, shows similar characteristics in the near-wall regions to k-omega model, and it shows similar characteristics in regions far from the wall regions to k-epsilon model. The selection of a model depends on the specific problem; no turbulence model is suited for every case. Fine-tuning is required to get the desired output.

From the simulations conducted in Case B, it is evident that with an increase in Reynolds number (inlet velocity) the flow takes longer to reattach and the reattachment zone is shifted backward (7 - 8 m in Case B1, 7.5 - 8.5 m in Case B2 and 8.5 - 9.5 m in Case B3). Also, Case B shows very intriguing recirculation region variations. In cases of Case B1 and Case B2, a secondary recirculation region can be observed at the top wall region, which seems to cease with the increase in Reynolds number (inlet velocity) of the flow, as it becomes almost negligible in size in Case B3. In Case B3, the secondary recirculation region is shrinking and a tertiary recirculation region formation takes place, such that it shares an interface with the primary recirculation region.

Reference

1. https://www.openfoam.com/documentation/
2. https://www.cfd-online.com/
3. https://openfoamwiki.net/
4. Lin Chen et al. - A review of Backward-Facing Step (BFS) flow mechanisms, heat transfer and control
5. Christopher Chettri - Numerical solution of a 2D flow over a backward facing step
6. An introduction to computational fluid dynamics : the finite volume method Author: Henk Kaarle Versteeg and Weeratunge Malalasekera
7. Formulation of the k-w Turbulence Model Revisited, David C. Wilcox
8. Ten Years of Industrial Experience with the SST Turbulence Model: F. R. Menter 1, M. Kuntz 1 and R. Langtry

System and Software

This study was conducted on the system whose hardware specification is listed as follows:

Device	Acer Nitro 5
Processor	Intel i5-7300HQ
GPU	HD Graphics 630 GTX 1050
RAM	8 GB
Drive Type	HDD
OS	CAELINUX (Case Study) Windows 10 (Documentation)

Software used for Case Study:

- Geometry and Mesh- Gmsh v3.0.6
- Simulation- OpenFOAM v4.1
- Post-processing- Paraview v5.0.1 & Gnuplot v4.6